

Drawing Environment

Overview

This lesson describes the AutoCAD® Civil 3D® (AutoCAD®) software drawing environment. Whether refreshing your knowledge or learning for the first time, these exercises will help you develop familiarity with basic entity creation and modification, entity properties, layers, blocks, layouts, and template files. A strong knowledge of these basics will enable you to work with the AutoCAD Civil 3D lessons and software more efficiently.

This lesson is intended to help students become familiar with the basic drawing environment of Civil 3D, which is based on AutoCAD. Students who already have a strong working knowledge of this topic should proceed to Lesson 2, AutoCAD Civil 3D Interface.

Objectives

After completing this lesson, you will be able to:

- Navigate the Civil 3D drawing environment, zoom and pan to view objects, use the command window, use the Help system, and explore environment settings and function keys.
- Configure fundamental drawing settings and options such as Scale, Grid, Snap, Object Snap, file paths, and display colors.
- Create basic AutoCAD objects such as lines, polylines, circles, arcs, and polygons using menus, palettes, keyboard commands, mouse controls, coordinates, and object snaps.
- Modify AutoCAD objects using multiple techniques including grip editing and object properties
- Use layers to control object display.
- Navigate and view objects in 3D.
- Create reusable blocks.
- Use externally-referenced drawings in the current drawing.
- Draw objects in paper space (layouts) and configure one or more viewports in a layout.
- Create a properly formatted layout with required elements such as a title block, north

arrow, border, and scale.

- Configure the page setup and plot layouts to an engineering scale.

Exercises

The following exercises are provided in a step-by-step format in this lesson:

1. Navigate the Civil 3D Drawing Environment
2. Review Options and Drawing Settings
3. Create Objects
4. Modify Objects
5. Navigate the 3D Drawing Environment
6. Create Blocks and Use External References
7. Work with Layouts and Viewports
8. Plot an Engineering Drawing to Scale

Civil 3D Drawing Environment

AutoCAD is the drawing environment used by AutoCAD Civil 3D design software. The software enables you to design in 3D, therefore, it is critical for you to learn the basics of the graphic environment prior to continuing through the rest of this curriculum. In the early years of CAD, design was performed separately from the drafting and production of final drawings. Civil 3D has changed this paradigm so that design and production are performed simultaneously. Understanding the drawing environment is crucial to design using the intelligent objects of Civil 3D. Feeling comfortable with creating and modifying objects, layers, viewing in 3D, using blocks, and being able to print to scale will help you be more productive as you learn the design tools of Civil 3D.

These exercises cover many basic concepts and the novice user is strongly encouraged to work through each one of the exercises to gain a full understanding of the graphic environment of Civil 3D.

A few key terms are:

Key Terms

Grips	Small squares and triangles that appear on selected objects. After selecting the grip, you edit the object by dragging it with the pointing device instead of entering commands.
UCS Icon	An icon that indicates the orientation of the UCS (User Coordinate System) axes. (UCSICON).
Grid and Snap	A grid is an area covered with regularly spaced dots or lines to aid drawing. The grid spacing is adjustable and the grid dots are never plotted. Snap settings relate to an invisible grid that locks the pointer into alignment with the grid points according to the spacing settings.

	Snap grid does not necessarily correspond to the visible grid.
Object Snap (Osnap)	Methods for selecting important geometric points on an object while you create or edit a drawing. Examples of object snaps include endpoint, midpoint, and center.
Layer	Layers are used to organize drawing data. Every drawing object is assigned to a layer. Objects can adopt the layer visibility settings, including on\off, freeze\thaw, color, and linetype.
Selection Window	A selection window is used to select one or more objects that a command can act upon at the same time. Implied windowing means using a right to left window to select items the window touches and a left to right window to select items entirely within the window.
Named View	A view, or graphical orientation, that can be saved and restored.
Block	A generic term for one or more objects that are combined to create a single object.
External Reference	A drawing file referenced by another drawing.
Layout	The environment in which you create and design paper space layout viewports to be plotted. Multiple layouts can be created within each drawing. Contract drawing borders are usually created on layouts.
Viewport	A bounded area that displays some portion of the model space of a drawing. A viewport is created on a layout.
Paper Space	One of two primary spaces in which objects reside. Paper space is used for creating a finished layout for printing or plotting, as opposed to doing design or drafting work that is performed in model space.
Model Space	One of the two primary spaces in which objects reside. Typically, a geometric model is created in a three-dimensional coordinate space called model space. A final layout of specific views and annotations of this model is placed in paper space.
Drawing Template Files	Drawing template files (DWT files) contain standard AutoCAD settings, layer definitions, linetypes, symbols, paper space layout definitions, dimension styles, and text style definitions. In addition, template files can include Civil 3D drawing information in either the Settings tree (including Civil 3D settings, object styles, label styles, tables, description keys, and point import\export formats), or the Prospector tree (including Civil 3D objects such as point group and surface definitions).

Exercise 1: Navigate the Civil 3D Drawing Environment

In this exercise, you navigate the Civil 3D drawing environment.

There are many tools available for viewing, creating, and modifying objects.

Recognizing these tools and knowing how to use them quickly will help as you move into the later lessons. Customizing the drawing environment is frequently done to optimize your interaction.

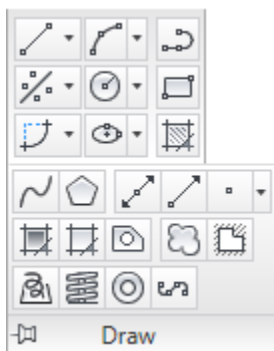
No visible changes will be made to the drawing during this exercise.

For this exercise, start AutoCAD Civil 3D. A new drawing, *drawing1.dwg*, is opened.

1. View the Home tab. Note the three panels: Draw, Modify, and Layers. These panels display commands that are used for more generic drawing purposes.

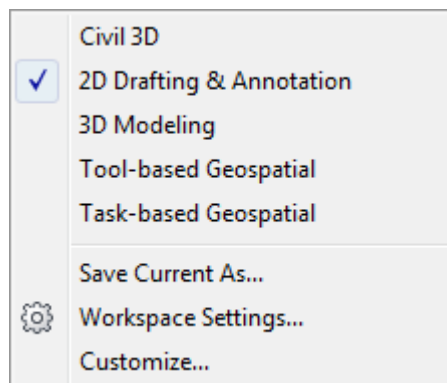


2. Click the panel name to see more commands that are available.



More drafting type commands can be found in a different interface setup, called a workspace. A workspace is the configuration of the user interface.

3. On the status bar at the bottom of the screen, click the Workspace Switching button. Click 2D Drafting & Annotation.

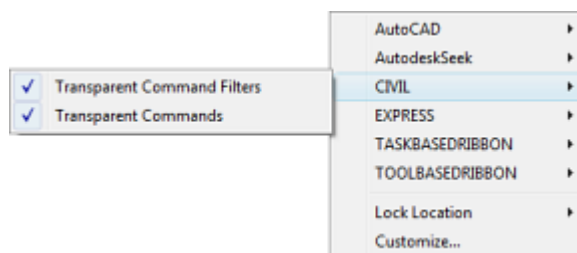


Note that your Workspace settings are used to determine which tabs, panels, and palettes are available. The 2D Drafting & Annotation workspace contains classic tools for drafting.

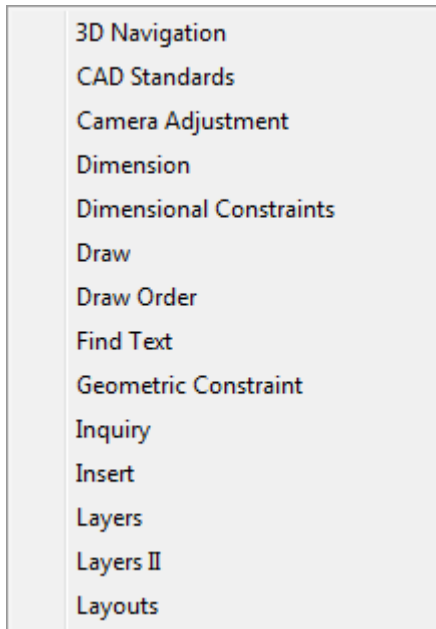
4. Try the other available workspaces to notice the different tabs and panels that are available for your use. When finished, return to the Civil 3D workspace.

There are numerous additional toolbars that can be used while in AutoCAD Civil 3D. The Civil 3D workspace has the Transparent Commands and Filters toolbars turned on.

5. Right-click the right side beneath the Transparent Commands toolbar. Click Civil. View the two toolbars that are displayed.

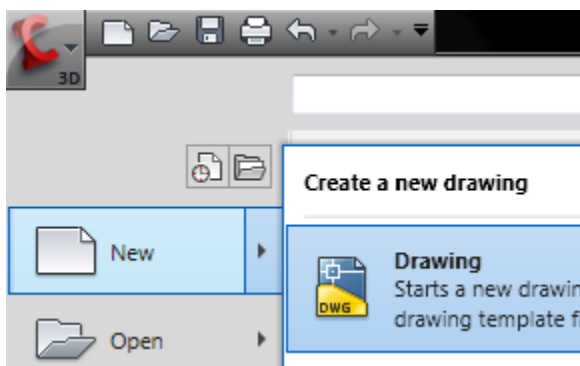


6. Move to the AutoCAD toolbar selection and view the available toolbars; there are many that can be used.



Next, you create a new drawing and begin to create and modify AutoCAD objects.

7. Click the application menu > New > Drawing.

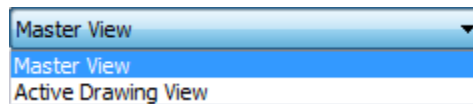


It is recommended that you use a drawing template file (*.dwt) when creating a new drawing (*.dwg) file. Template files contain predefined layers, blocks, styles, and settings to give the new drawing consistency with a company or client standard. When Civil 3D is started, it uses the *_AutoCAD Civil 3D (Imperial) NCS.dwt* template file to begin a new drawing file. When you open a new drawing file, you can select any template file.

8. Click the *_AutoCAD Civil 3D (Imperial) NCS.dwt* file in the list of templates. Click Open.

A second drawing named *Drawing2.dwg* is opened and has inherited the standardized items from the template file.

9. In the Prospector tab of Toolspace, switch to Master View.



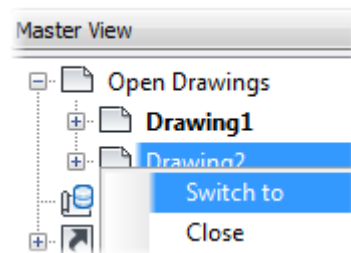
Notice that two drawings are open at the same time.

10. Right-click Drawing1. Click Switch To.

You are now viewing *Drawing1.dwg*

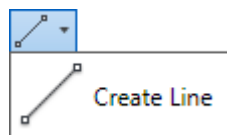
11. Right-click Drawing2. Click Switch To.

You are now viewing *Drawing2.dwg*



Draw Objects and View Properties

12. On the Home tab, Draw panel, click Line > Create Line.



Note: You can also begin this command by entering **line** or **l**, as a shortcut, and pressing ENTER. You can also open the AutoCAD Draw toolbar and click the Line button to begin this command.

13. Click anywhere in the drawing area to begin the line. Move the cursor and click again to end the line.

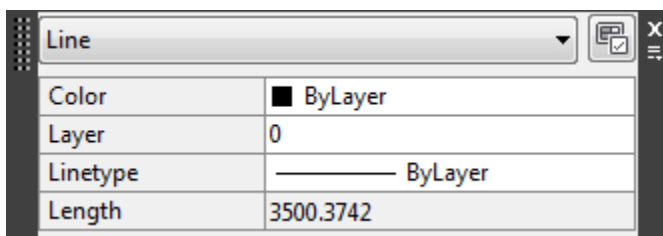
AutoCAD treats individual lines as single entities, but will automatically begin the next line where the previous line ended. The line is created on the current layer. In a new drawing, the current layer is named 0.

14. Press ENTER to end the Draw Line command.

15. In the drawing area, click the line.

Blue grips are shown on key geometric locations, different for each type of entity. These grips show you that one or more objects are *selected* in a *selection set*. You can perform commands on the selection set in a noun\verb sense where the selected objects are the noun(s) and the command is the verb. The line is now selected.

Depending on your settings, you may have a Quick Properties palette that displays on your screen and presents a short summary of the line's properties.



You can choose to have this palette automatically open by toggling the Quick Properties button on the status bar.

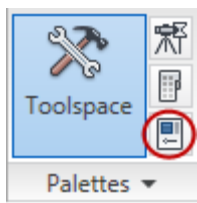
16. Click Quick Properties on the status bar to open and close this window. Leave the palette closed.

Note that the status bar is at the bottom of the drawing area.



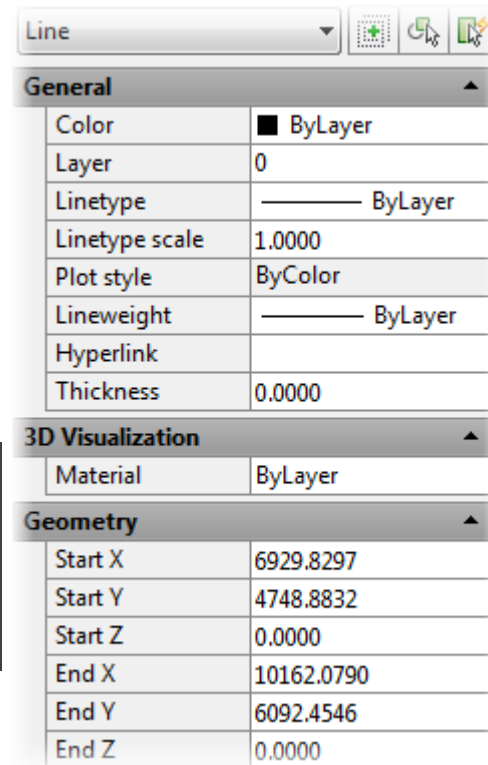
A more complete Properties palette is available.

17. Click the Properties button on the Palettes panel.



The Properties palette displays, showing the properties of the selected entity. Examine

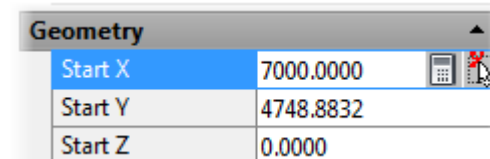
the properties in the palette. All of the properties with a white background can be edited in the palette by clicking in the cell and changing the value. The shortcut key for opening this palette is CTRL-1.



18. Close the Properties palette by clicking the 'x'. Press CTRL-1 to reopen.

19. Click in the Start X value cell.

20. Highlight the existing value and modify it. Press TAB.



The starting X position of the line changes to the new value. Notice the two small icons to the right of the cell. Clicking the Calculator icon enables you to use a calculator to enter a formula for the property. Clicking the Pointer icon enables you to click in the drawing area with your cursor.

21. Close the Properties palette.

22. Press ESC.

The grips disappear from the line, meaning that the line is no longer selected.

23. Draw a second line crossing the first line.

Zoom and Pan

24. If you have a cursor wheel, roll the wheel forward and backward. Hold the wheel down and move the cursor.

Rolling the wheel automatically uses the Zoom function and moving the cursor with the wheel pressed uses the Pan function. There are several methods to accomplish the same task using AutoCAD Civil 3D.

25. On the View panel, click Pan.



26. Click and drag the mouse to pan the drawing.

27. Right-click. Click Exit to end the command

You should always look at the command window to see the command options or to see what the program is expecting you to do. Be sure to expand the command window to be able to view at least three lines.



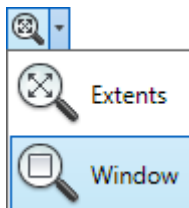
During the Pan command, the command window told you that you could use the ENTER key, the ESC key, or right-click for a shortcut menu to end the command.

28. Enter **P** on the keyboard. Press ENTER.

The keyboard shortcut for the Pan command is the letter P. Many commands have handy keyboard shortcuts.

29. Click and drag the mouse to pan the drawing. Press ESC.

30. On the View panel, click the down arrow next to Zoom. Click Zoom Window.



You need to click one corner, move the cursor around the lines, and click the opposite corner of a bounding rectangle. The drawing area is resized to fit the bounding rectangle.

The keyboard shortcut for Zoom is Z. The command window shows a number of options as shown below. You enter the capital letter of the option desired to start the command. Your keyboard entry is not case sensitive.

[All/Center/Dynamic/Extents/Previous/Scale/Window/Object] <real time>:

31. Click one corner. Move the cursor. Click the opposite corner of the window.

32. Enter **Z**. Press ENTER. Enter **P**. Press SPACEBAR.

P is the key for the previous option of the Zoom command. The view prior to the Zoom command is restored. Notice that you can use the spacebar on the keyboard for ENTER.

Civil 3D also has shortcut commands for zoom options.

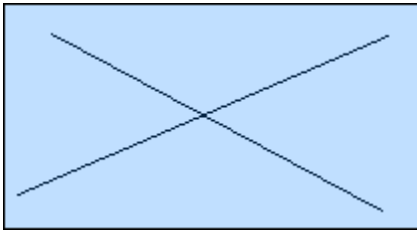
33. Enter **ZE**. Press SPACEBAR.

This executes the Zoom Extents command. AutoCAD Civil 3D automatically zooms to include the extents of all visible objects.

Select Objects

One very common procedure is to select objects in order to perform an operation on them. To select objects, you either click them or click a box around or touching them.

34. Click to the left of the lines, and drag a box entirely around both lines.



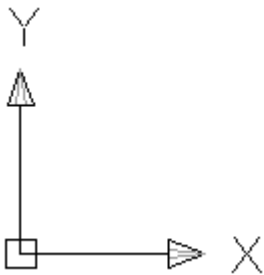
35. Click again.

When you drag a selection box from left to right, all entities that are entirely within the box are selected (window selection). When you drag a selection box from right to left, any entity touched by the box is selected (crossing selection). Try this feature to see how it works. This is called *implied windowing*.

36. Press ESC.

UCSICON

The Cartesian (X,Y) coordinate system is represented in the drawing area by an XY axis icon, called the UCS (User Coordinate System). This icon can be turned on and off.



37. On the command line, enter **UCSICON**.
Enter **Off**. Press ENTER.
38. On the command line, enter **UCSICON**.
Enter **On**. Press SPACEBAR.

Pressing ENTER or SPACEBAR repeats the last command.

39. Press SPACEBAR again.

Notice the command window presents options for the UCSICON command.

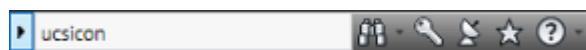
40. Enter **Off**. Press SPACEBAR.

View Help

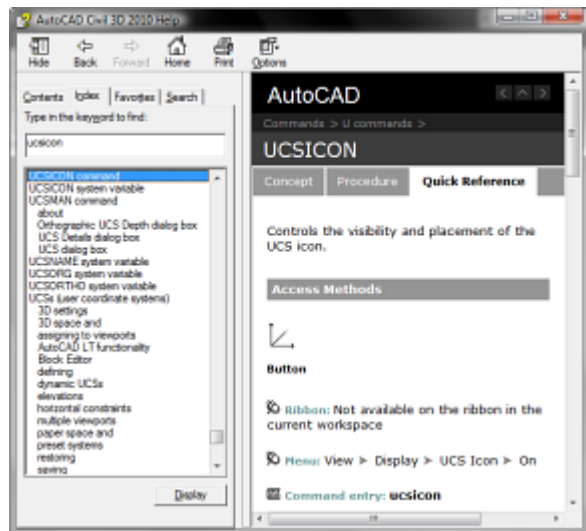
Most commands are available with menus or keyboard shortcuts. Help for the UCSICON command is available by searching the InfoCenter Help.

41. Enter **ucsicon** in the search window of InfoCenter.

InfoCenter will search for appropriate topics and give you a list.



You can also click the question mark to open the full Civil 3D Help program and enter **ucsicon** into the Index. F1 also opens the Help window.



42. Close the drawing and do not save the changes.

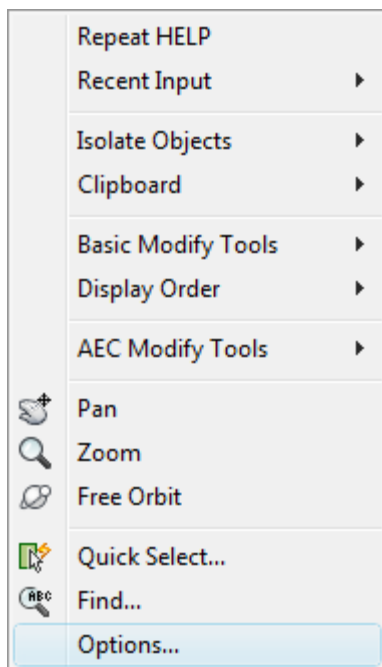
Exercise 2: Review Options and Drawing Settings

In this exercise, you explore the various settings and options to customize AutoCAD Civil 3D.

No visible changes will be made to the drawing during this exercise.

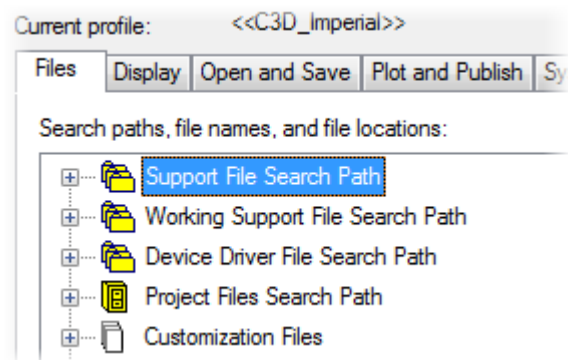
Either continue from the previous exercise or start AutoCAD Civil 3D. A new drawing, *drawing1.dwg*, is opened.

1. Right-click anywhere in the Drawing Area. Click Options.

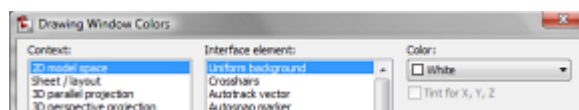


2. Click the Files tab.

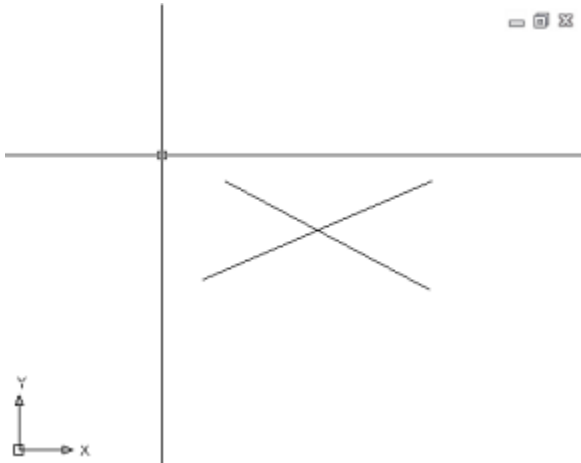
Each of the listed names and paths can be modified. To view more information on any of the options, click to highlight a tree and look at the tooltip. You can also click Help at the lower right for help on the entire page.



3. Click the Display tab.
4. Click the Colors button in the Window Elements section.
5. In the Drawing Window Colors dialog box, under Context, select 2D Model. Under Interface Element, click Uniform Background. Under Color, click the down arrow. Select White.



- Note:** Many other colors can be modified using the Colors button. Review the other options on the Display tab.
6. Click Apply & Close to close the dialog box.
 7. In the Crosshair Size section of the Display tab, drag the slider bar all the way to the right so the size is 100.
 8. Click OK.



9. Right-click anywhere in the drawing area. Click Options. Return the crosshair size to the original size if desired.

10. Click the Open and Save tab.

Review the various sections on the tab that can be modified. There are many tabs and settings as part of the Options dialog box. You should review these because the settings affect your drawing environment.

11. Click the Drafting tab.

The AutoSnap and AutoTrack settings are frequently modified by users.

12. Drag the Aperture Size slider bar slightly to the right.



13. Click the Selection tab.

You can modify the grip size and color, turn grips on or off, and modify many other options for selecting entities.

14. Click the Profiles tab.

All settings in Options can be saved to a profile that can be applied anytime. If you want to try this, click Add to List. Assign the

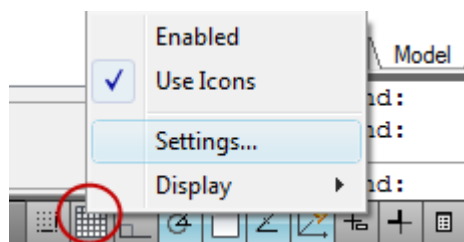
profile a name and description. Click Apply & Close.

15. Click OK to close the Options dialog box.

Drafting Settings

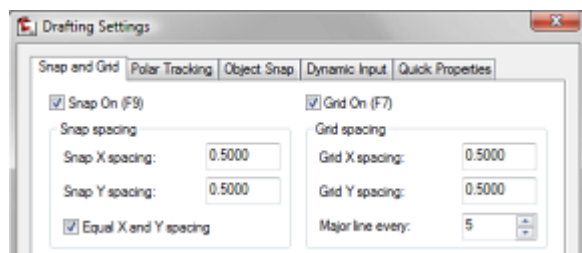
Drafting Settings include several important features to help you to be more accurate and efficient in creating objects. Using snap and grid settings can be helpful.

16. On the status bar, right-click Grid Display. Click Settings to open the Drafting Settings dialog box.



You can turn on the Snap and Grid options from the buttons on the status bar or inside the Drafting Settings dialog box. The spacing for each can be modified here.

17. Select the Snap On check box and the Grid On check box.



18. Click OK.

The grid is a series of dots set at the specified spacing interval over the area of the drawing limits. The snap setting only allows the crosshair to locate on the unit increments set in the previous dialog box. The Grid and Snap settings are independent of each other.

19. Move the cursor and watch the coordinates of the crosshair as reported in the lower left corner of the status bar.

Notice that the coordinate values only change in 0.5 unit increments. The Snap and Grid settings can be turned off and on using the status bar also.

20. On the status bar, click Snap to turn it off.



There are many other settings that can quickly be turned off or on using the status bar as well as the function keys. You can also right-click these buttons to turn them on or off or modify the settings. The Snap toggle is the F9 key. Other function keys are programmed to toggle settings on and off. Refer to the Help topic of Function Keys to learn more about Ctrl, Alt, and Shift key shortcuts.

- F1 Displays Help
- F2 Toggles Text Window
- F3 Toggles OSNAP
- F4 Toggles TABMODE
- F5 Toggles ISOPLANE
- F6 Toggles UCSDETECT
- F7 Toggles GRIDMODE
- F8 Toggles ORTHOMODE
- F9 Toggles SNAPMODE
- F10 Toggles Polar Tracking
- F11 Toggles Object Snap Tracking
- F12 Toggles Dynamic Input

21. Press F9.

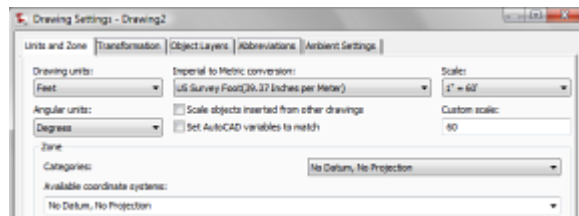
Notice the message in the command window turning Snap off.

22. Click the Grid button on the status bar to turn off the grid. 

Drawing Settings and Drawing Units

There are two important dialog boxes that should be inspected prior to beginning a project: Drawing Settings and Drawing Units.

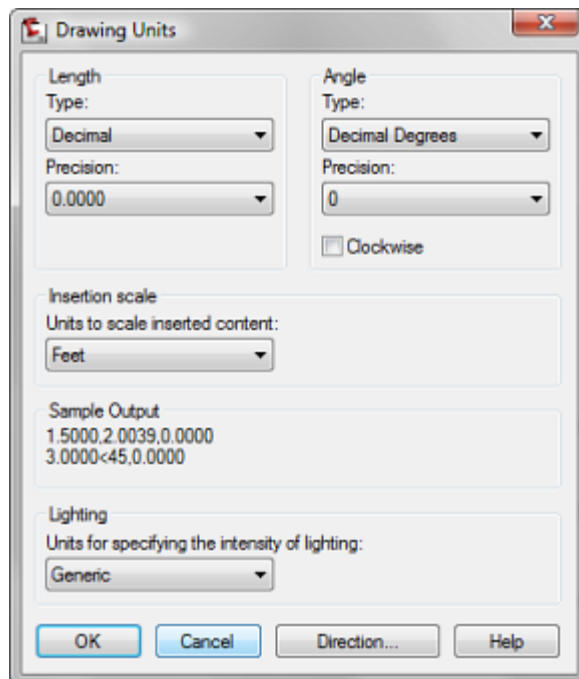
23. Click application menu > Drawing Utilities > Drawing Settings.



This dialog box sets the length distance measurement, unit conversion factors, drawing setup scale in the finished drawings, angular units, and the real-world coordinate system settings. There are multiple tabs and settings that should be reviewed.

24. Click Cancel.

25. Click application menu > Drawing Utilities > Units.



It is typical to work in decimal length and decimal degrees for angles, but these settings can be modified. The Precision setting controls only the reported number of decimal places, not the number of places stored in AutoCAD. Review the options by clicking the down arrows on these settings.

26. Click Cancel.

27. Close the drawing and do not save the changes.

Exercise 3: Create Objects

In this exercise, you create basic objects using menus, toolbars, cursor techniques, and keyboard shortcuts.

Either continue from the previous exercise or start AutoCAD Civil 3D. A new drawing, *drawing1.dwg*, is opened.

Use Object Snaps

1. Right-click the right side of the graphic screen, beneath an existing toolbar. Open the AutoCAD Draw toolbar.
2. Click Line to begin a line command. Click the drawing area to begin a line. Move the cursor and click to end the line. Press ENTER.

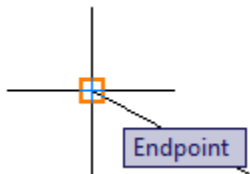


3. Press SPACEBAR.

The Draw Line command starts again.

4. Move the cursor near the line just drawn.

Notice the orange object snap boxes when your cursor is near the end of the line. If you cannot see the Osnap box, click the Osnap button on the status bar to turn it on.



Object snaps, or osnaps, are specific important geometric points of entities to which you can snap for the purposes of creating or modifying entities. Their positions are similar to the grip positions. Click when you see the Osnap box.

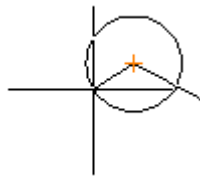
The line starts at the Osnap position.

5. Move the cursor and click to finish the line. Press ESC.
6. Click the Circle button on the Draw toolbar.



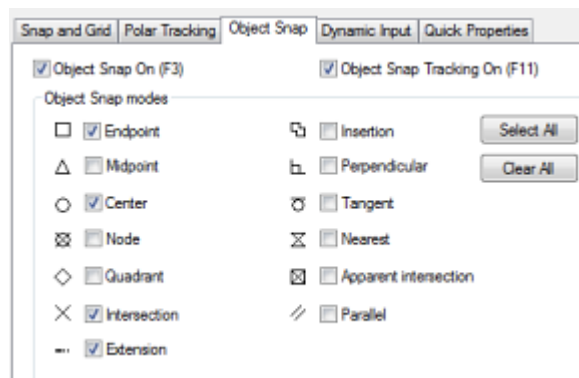
Always watch the command window for messages, instructions, and options during a command. You are prompted to specify the center point for the circle.

7. Move the cursor near the end of a line. Click when the Endpoint Osnap box appears.
8. Move the cursor to expand the diameter. Click to draw the circle.



You can control which object snap types are automatically detected in the Drawing Area.

9. On the status bar, right-click Osnap. Click Settings.



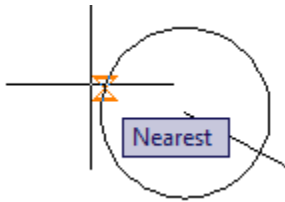
These are called running object snaps and can be modified at any time.

10. Click OK.

Object snaps are also available during a drawing command using the SHIFT+right-click combination.

11. Enter **L**, the shortcut key for drawing a line. Press SPACEBAR.
12. Hold SHIFT and right-click. On the Object Snap menu, click Nearest.
13. Move the cursor close to the edge of the circle.

The object snap symbol appears and tracks along the edge of the circle as the cursor moves.



14. Click to start the line.
15. Enter **cen**. Press SPACEBAR.
16. Move the cursor near the edge or the center of the circle.

Notice that the object snap at the center of the circle is on.

17. Click to draw the line. Press SPACEBAR to end the command.

Use Coordinates

You can also enter coordinates while drawing or modifying objects using the keyboard. There are two types of coordinate input methods, absolute and relative. When entering a position using coordinate values with the keyboard, the first x, y, z point is the absolute value in the Cartesian system. The default entry mode for the second and subsequent points is also absolute; that is, the x, y, and z values are the exact values of the coordinate system, not relative unit distances away from the previous point. You can enter relative unit distances by including @ prior to the x, y, and z values of the next point.

18. On the Draw toolbar, click the Polyline button.



A *polyline* is a single object that has multiple vertices, whereas a line is one object with only start points and endpoints.

19. Click anywhere in the drawing area to begin the polyline.

Notice the rubber-banding effect from the start point (x,y) to the crosshair.

20. Enter **500,500**. Press ENTER.

The first segment of the polyline is drawn with the second point located at absolute values of 500 x and 500 y.

21. Enter **u**. Press SPACEBAR to undo that segment.

22. Enter **@500,500**. Press SPACEBAR.

The segment of the polyline is drawn with the second point located at relative values of 500 x and 500 y.

23. Hold the cursor over the second point to view the absolute coordinates in the bottom left section of the status bar.

24. Press ESC.

AutoCAD can create objects in 3D using the z axis. The Line command can be 3D, but the Polyline command cannot. Instead, you must use the 3D Polyline command.

25. Enter **L**. Press SPACEBAR.
26. Click in the drawing area to begin the line.
27. Enter **@500,500,100**. Press SPACEBAR twice.

The second point was located using 500, 500, 100 as relative values.

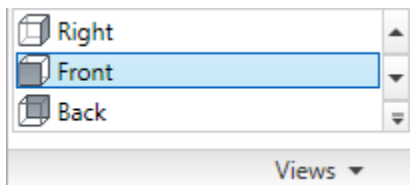
28. Click the line to select it.
29. Press CTRL-1 to open the Properties palette.

Geometry	
Start X	684.3514
Start Y	5974.7059
Start Z	0.0000
End X	1184.3514
End Y	6474.7059
End Z	100.0000

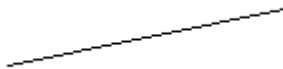
Notice the start and end X, Y, Z values.

30. Close the Properties palette.

31. Click the View tab. In the Views panel, use the arrows and select Front.



The view shifts to the front of the objects. Notice the line elevates from 0 to 100 units on the z axis.



32. In the Views panel, click Top.

Use Layers

In the following steps, you learn about layers. Objects are organized in a drawing by placing them on different layers.

33. On the Home tab, Layers panel, click the Layer Properties Manager button.



The layer named 0 is the current layer for this drawing, but the template drawing for AutoCAD Civil 3D contains many different layers.

Next create a new layer.

34. Click the New Layer button.



35. For Layer Name, enter **Test**. Press ENTER.

36. Click the Color value for the Test layer.



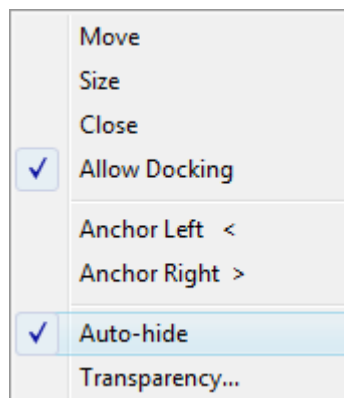
37. On the Index Color tab, in the Select Color dialog box, select Red (color 1). Click OK.

Any object drawn while this layer is current adopts the layer's properties, and is therefore red.

38. With the Test layer highlighted, click Set Current.



39. Right-click the palette sidebar. Click Auto-hide.



Turning Auto-hide on causes the palette to disappear if the cursor is not hovering on top of the palette.

40. On the Draw toolbar, click Rectangle.



41. In the drawing area, click to set one corner of the rectangle. Move the cursor and click to draw the rectangle.

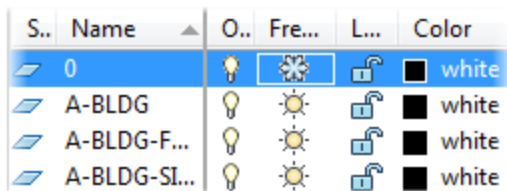


The rectangle is drawn on the Test layer and displays red. Drawings frequently use many layers to organize features. For example, a storm sewer layer contains the storm sewer pipes, manholes, and catch basins. Layer naming standards are used by all companies to give consistency to their drawings. Layer visibility is controlled in the Layer Properties Manager.

42. Click the sidebar of the Layer Properties Manager palette.

43. On the 0 layer, click the Freeze icon.

All objects on the 0 layer will not be visible.



44. Close the palette.

View the drawing area to notice which objects are visible. Other layer settings can be modified in the same manner.

45. On the Draw toolbar, click Text.



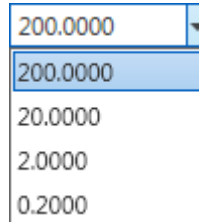
46. Click to set the location of one corner of the text box. Move your cursor and click to set the opposite corner.

A Contextual tab entitled Text Editor opens. This tab contains panels and commands that can be used with multiline text.

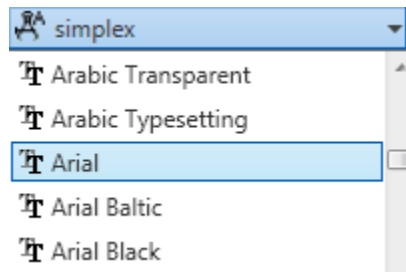
47. In the Style panel, highlight the size of 0.2. Enter **2**. Press ENTER.

48. Repeat the process with values of **20** and **200**.

49. Use the drop-down arrow to verify that multiple choices of size are now available. Select 200 as the current size.



50. On the Formatting panel, click the Bold button. Select Arial as the font.



51. Enter **Manhole**. Click the Close Text Editor button.

Manhole

52. Close the drawing and do not save the changes.

Exercise 4: Modify Objects

In this exercise, you modify basic objects using menus, toolbars, cursor techniques, and keyboard shortcuts.

No visible changes will be made to the drawing during this exercise.

Either continue from the previous exercise or start AutoCAD Civil 3D. A new drawing, *drawing1.dwg*, is opened.

1. On the Home tab, Layers panel, click Layer Properties.

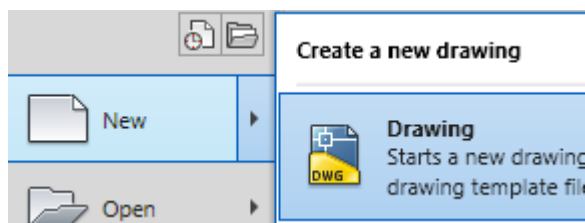


2. The default template file used with Civil 3D contains 172 layers. Review the names of these layers.

S..	Name	O..	Fre...	L...	Color
✓	0	☹	☼	🔒	wh
	A-BLDG	☹	☼	🔒	whi
	A-BLDG-FPRT	☹	☼	🔒	whi
	A-BLDG-SITE	☹	☼	🔒	whi
	A-BLDG-UTIL	☹	☼	🔒	whi
	A-PROP-LINE	☹	☼	🔒	whi
	C-ANNO	☹	☼	🔒	whi
	C-ANNO-MTCH	☹	☼	🔒	whi
	C-ANNO-MTCH-H...	☹	☼	🔒	whi
	C-ANNO-MTCH-TE...	☹	☼	🔒	150
	C-ANNO-TABL	☹	☼	🔒	red
	C-ANNO-TABL-PATT	☹	☼	🔒	whi
	C-ANNO-TABL-TEXT	☹	☼	🔒	150
	C-ANNO-TABL-TITL	☹	☼	🔒	150
	C-ANNO-TABL-TTBL	☹	☼	🔒	blu
	C-ANNO-VFRM	☹	☼	🔒	150
	C-ANNO-VERM-TEXT	☹	☼	🔒	11

In order to simplify this exercise, create a new drawing using the *acad.dwt* template file, which contains only one layer named 0.

3. Click Application Menu > New > Drawing.



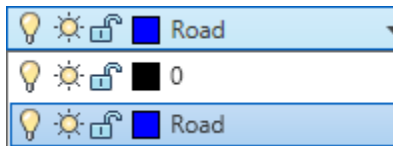
4. Select *acad.dwt*. Click Open.

A new drawing file opens using the *acad.dwt* file as a template.

5. Create a new layer named **Road** to the drawing. Change the layer color to blue.
6. Draw several lines, polylines, and circles, and add text to the drawing.

Put some of these objects on the 0 layer and the Road layer.

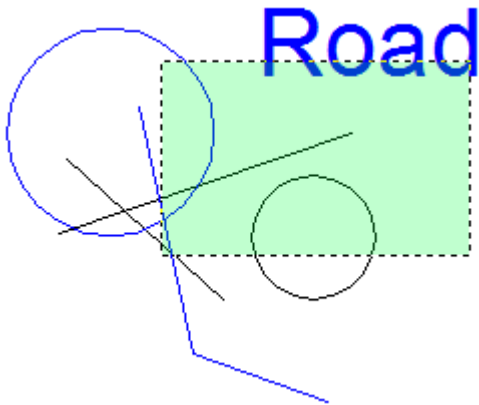
7. With the 0 layer current, click Line on the Draw toolbar.
8. In the drawing area, click to start the line. Move the cursor and click again to end the line. Press SPACEBAR.
9. Press SPACEBAR again to start the Line command again.
10. Draw another random line.
11. On the Draw toolbar, click Circle.
12. Click to set the center of the circle. Move the cursor and click again to complete the circle.
13. Click the down arrow on the Layer toolbar in the Layers panel. Select the Road layer to set it as current.



14. Repeat the previous steps to draw objects on the Road layer. Include a text object (the size will need to be smaller in the *acad.dwt* template).

15. Use a right to left selection window to select several objects.

Remember that a right to left selection window will include any object that is included or intersects its edge.



Grips display on the objects. You can add to the selection set by holding the SHIFT key and clicking other objects.

16. On the command line, enter **erase**. Press SPACEBAR.

The selected objects disappear from the screen. Actions in AutoCAD are logged and can be undone. You can also use E as the keyboard shortcut for erase.

17. On the command line, enter **U**. Press SPACEBAR.

The Undo command is frequently used.

Grip Editing

18. Click one of the lines.

Grips appear on the line. Grip editing is a powerful method to graphically modify objects.

19. Click one of the grips. Move the cursor.

The grip changes color and becomes the active grip. Editing commands use this grip as the focus. The line is stretching from one point to the active grip on the crosshair. Review the command window. You have automatically initiated the Stretch grip-edit

command, the default editing command for grips.

20. Click to stretch the line.

21. Click the grip again.

22. Press SPACEBAR. Move the cursor.

The spacebar changed the active grip command to Move. As you move the cursor the line moves.

23. Move the cursor until an object snap from another object is highlighted.

24. Click to move the line.

25. Click the grip again. Press SPACEBAR twice. Move the cursor.

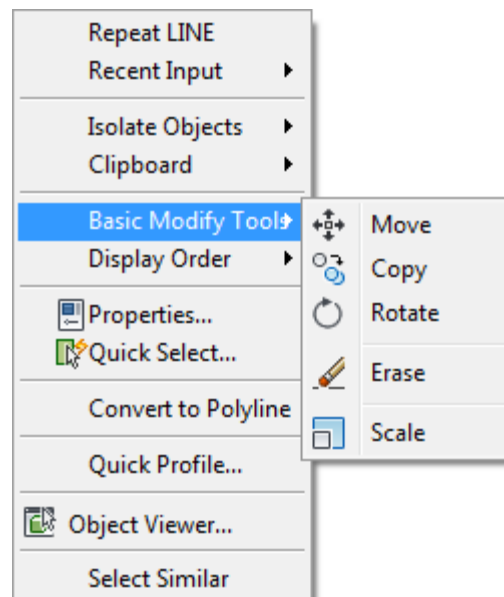
The Rotate command is active.

26. Click to rotate the line. Press ESC.

There are five grip editing commands available: Stretch, Move, Rotate, Scale, and Mirror that toggle as you press SPACEBAR.

Basic Modify Tools

Another commonly used method is to select the object(s), then right-click and use the available commands.



27. Click to select an object on the 0 layer.

28. Press Ctrl-1 to open the Properties palette.

The Properties palette for the selected object is displayed.

29. Click the Layer value cell. Click the down arrow.

30. Click Road. Press ESC. Close the Properties palette.

The object's layer changes to Road and the color changes.

Objects are also easy to copy and paste.

31. Click to select a text object. Right-click. Under Basic Modify Tools, click Copy.

Always watch the command window for messages.

32. Click to select the base point.

33. Move the cursor and click. Press ENTER.



A copy of the text object is inserted.

Information about a particular object is easily retrieved with the List command.

34. Click a polyline. On the command line, enter **list**. Press SPACEBAR.

An AutoCAD text window opens, showing you the results of your query about the polyline's properties. This window is a larger view of the command window and can be toggled on and off with the F2 key.

35. Press F2.

The Trim, Offset, and Polyline Edit (PEDIT) commands are commonly used.

36. Create two lines that cross each other.

37. On the Home tab, Modify panel, click the Trim.

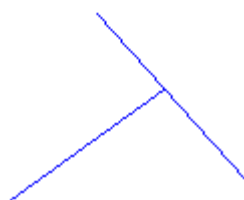


The message in the command window asks you to select cutting edge(s).

38. Click one of the crossing lines. Press ENTER.

The next prompt asks you to select the object(s) to trim.

39. Click the line you want to trim on the side that you do not want to keep. Press ENTER.



40. On the Modify toolbar, click Offset.



41. Enter **5** as the offset distance. Press ENTER.

42. Click to select a line.

43. Click to the right of the line. Press ENTER.

The line is offset 5 units on the right side.

44. Click a polyline (not a line).

45. Right-click. Click Polyline Edit.

The command window shows the options.

46. To alter the width, enter **W**. Press SPACEBAR.

47. For the width, enter **2**. Press SPACEBAR twice.



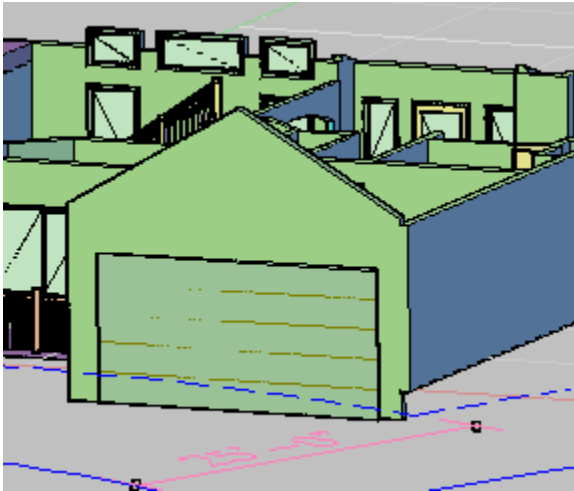
There are several important editing commands, many of which have a button

on the Modify panel or the AutoCAD Modify toolbar.

48. Close the drawing and do not save the changes.

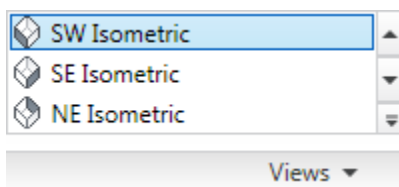
Exercise 5: Navigate the 3D Drawing Environment

In this exercise, you learn the basic commands and views for interacting with Civil 3D in 3D.

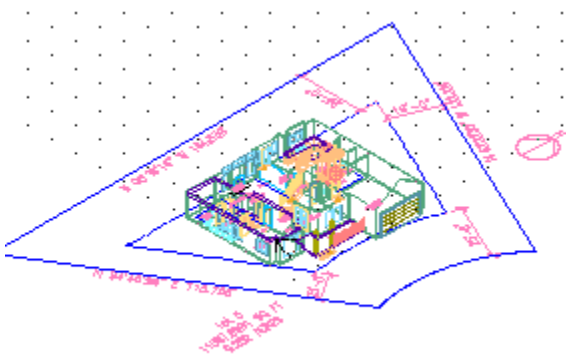


For this exercise, open
..._C3DDrawingEnvironment-EX5.dwg.

1. On the View tab, Views panel, click SW Isometric.



Your crosshair displays a third line and the drawing is shifted so it appears that your view is in the southwestern sky.



2. Use the cursor wheel to zoom and pan.

3. Right-click the toolbar area. Click AutoCAD > 3D Navigation.

The 3D Navigation toolbar opens.

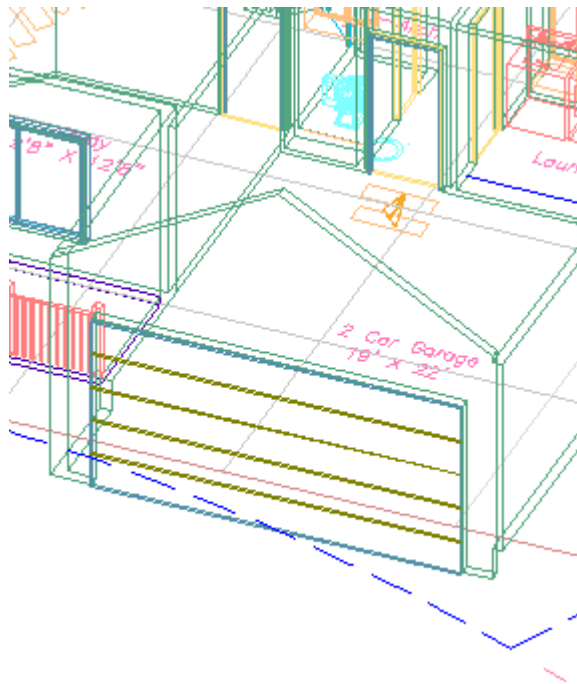


4. On the new toolbar, click Constrained Orbit.



5. Click (and hold) and drag the cursor.

You can adjust the view position while you zoom and pan.



6. Right-click. Click Exit.
7. On the toolbar, click Swivel.



8. Click and drag the mouse.

Note the difference in navigation.

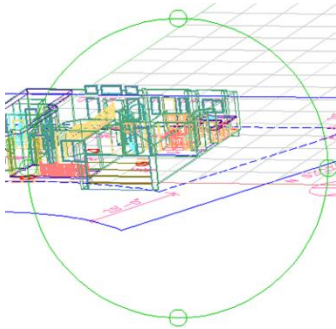
9. Right-click. Click Exit.
10. On the 3D Navigation toolbar, click and hold the Constrained Orbit button. Click Free Orbit.



A green circle appears as a navigation aid.

11. Click and drag both inside and outside of the circle.

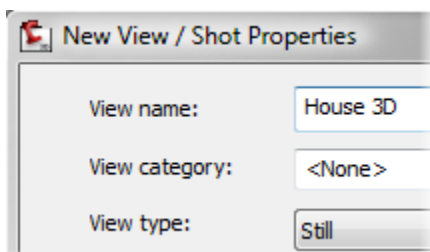
Carefully examine how the view is moved.



12. Right-click. Click Other Navigation Modes > Continuous Orbit.
13. Click and drag a direction. Let go of the mouse.

The model is continuously orbiting around the direction you supplied. Try other commands on this toolbar.

14. Right-click. Click Exit.
15. In the Views panel, click Named Views. Click New.
16. Enter **House 3D** as the name of the view.



17. Click OK.

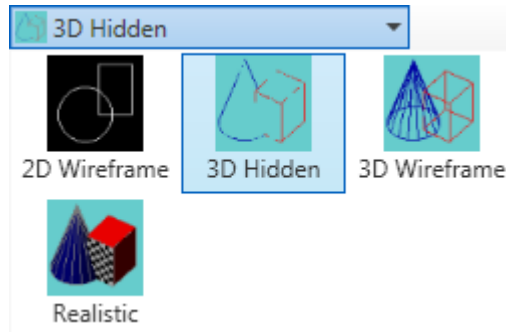
The current view is saved as House 3D. The View Manager organizes named views, which are very useful when you have a view that you need to revisit often. The View is also listed in the box in the Views panel.

18. Click OK.
19. Click Top in the Views panel.

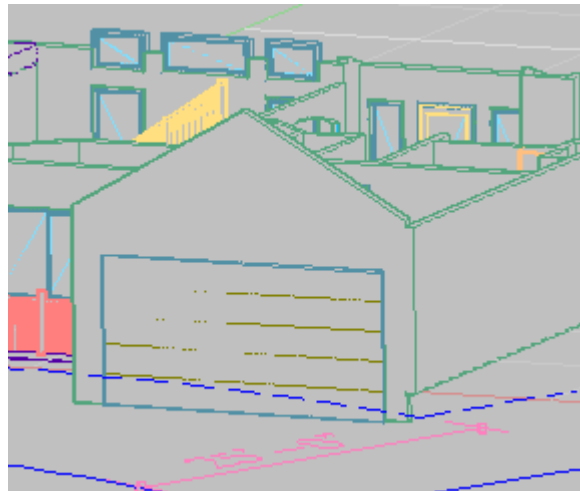
The plan view is restored.

20. Click House 3D in the Views panel box.

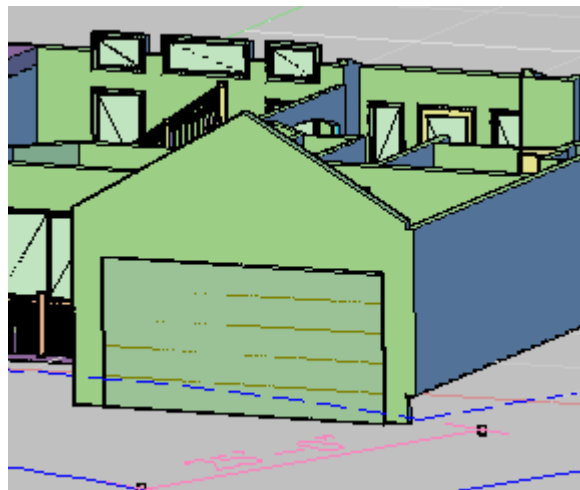
21. Click the Views panel name. Click the down arrow to change the visual style to 3D Hidden.



Objects are hidden depending on the view perspective.



22. Repeat the step and change the Visual Style to Conceptual.



Other visual styles are useful to view different features.

23. Return to the 2D Wireframe Visual Style.

24. Click Top in the Views panel box.

25. Close the drawing without saving.

Exercise 6: Create Blocks and Use External References

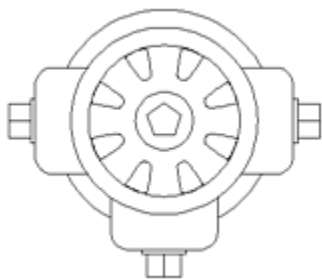
In this exercise, you create your own blocks and modify existing blocks. You also use external references to help visualize background information.

Blocks are an efficient method used to group a set of objects together and can be reused within one or many drawings. It is convenient to create custom symbols by drafting objects to form the desired shape, and then create a block. Internal block definitions are stored in the DWG file in which they were created. The Writeblock (WBLOCK) command can be used to save a selection set to a separate DWG file.

You start creating a block by drawing various objects. In this exercise, you use several drawings. Begin by opening the file ... \FireHydrant.dwg.

1. View the FireHydrant drawing.

The objects making up this shape are all individual polylines, lines, arcs, or circles.



2. Click an arc. Enter **LI (or LIST)**. Press ENTER.

The Text window describes the arc.

3. Press F2.
4. On the ribbon, Insert tab, Block panel, click Create.

The Block Definition dialog box opens.

5. Enter **Fire Hydrant** as the block name.

6. In the Base Point section of the dialog box, click Pick Point.

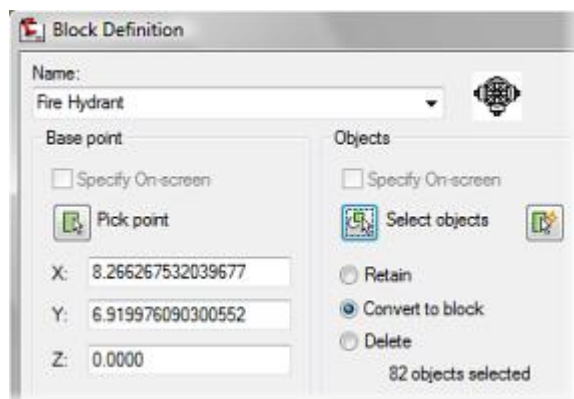
The base point is the point used for placing the block into the drawing.

7. Hold SHIFT and right-click for object snap options. Select Center.
8. Click at the center of the hydrant object.
9. In the Objects section of the dialog box, click Select Objects.

This enables you to select the objects that will be used to make the block.

10. Use a window to select all of the objects. Press ENTER.

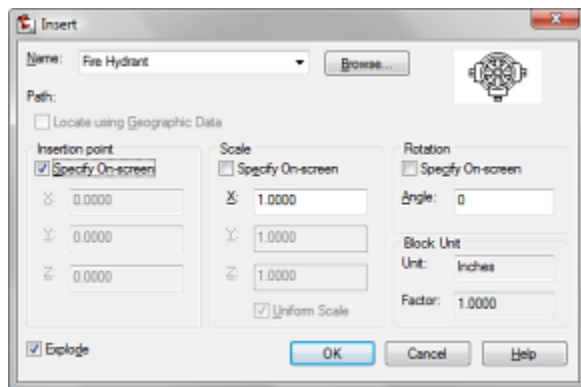
There are 82 individual objects that are incorporated into a block. The radio buttons named Retain, Convert to block, and Delete indicate how the current objects are handled after creating the block. Convert to Block is the default choice that replaces all 82 individual objects with one block.



11. Click OK.
 12. Click the block. Enter **LI**. Press ENTER.
- The Text window identifies the object as a block.
13. Press F2.
 14. Click the block. Press DELETE.

The definition of the block is retained in the DWG file and can be recalled and inserted at multiple locations. Note that using blocks saves file space. The block definition is stored only once and each subsequent insertion stores only the insertion point coordinates.

15. On the ribbon, Insert tab, Block panel, click Insert.



Because Fire Hydrant is the only block in the drawing, it displays in the Name box. If other blocks were available, they would display in the drop-down list. Review the settings in the box.

16. Be sure to select Specify On-Screen for the Insertion Point. Do not select the Explode check box. Click OK.

17. Enter **10, 10**. Press SPACEBAR.

The block is placed in the drawing, with the center point at coordinates X=10, Y=10.

18. Enter **ZE**. Press SPACEBAR to view the fire hydrant.

19. Repeat the above steps to insert a second block graphically, placing it next to the previous block.

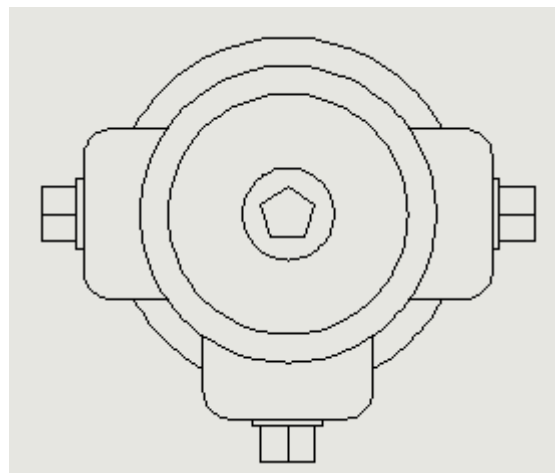
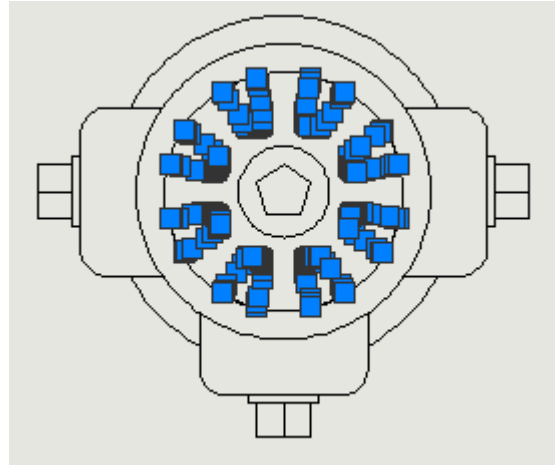
Edit the Block

Block definitions can be edited graphically and the changes will be updated for all blocks.

20. Click the block. Right-click. Click Block Editor.

The Block Editor screen opens.

21. Use editing commands to remove the objects pointing towards the middle of the hydrant.



22. Click Close Block Editor. Click Yes to save changes.

Notice that both blocks are updated to the new definition.

23. On the Home tab, Modify panel, click Explode .



24. Click one of the hydrant blocks. Press SPACEBAR.

Use the cursor and the List command to verify that the block has been exploded back into its original objects. The other block remains unexploded.

25. Close *FireHydrant.dwg* without saving.

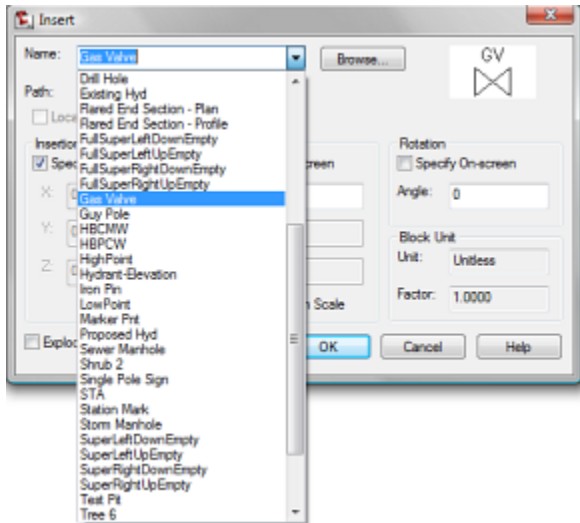
Insert and Write a Block

26. Create a new drawing using the *_AutoCAD Civil 3D (Imperial) NCS.dwt* template file.

27. On the Insert tab, Block panel, click Insert.

28. Click the down arrow under Name. Select Gas Valve.

Notice that the template file used to create the new drawing has embedded blocks.



29. Click OK. Click to pick an insertion point.

30. Enter **ZE**. Press ENTER to Zoom Extents.



31. Enter **WBLOCK**. Press ENTER.

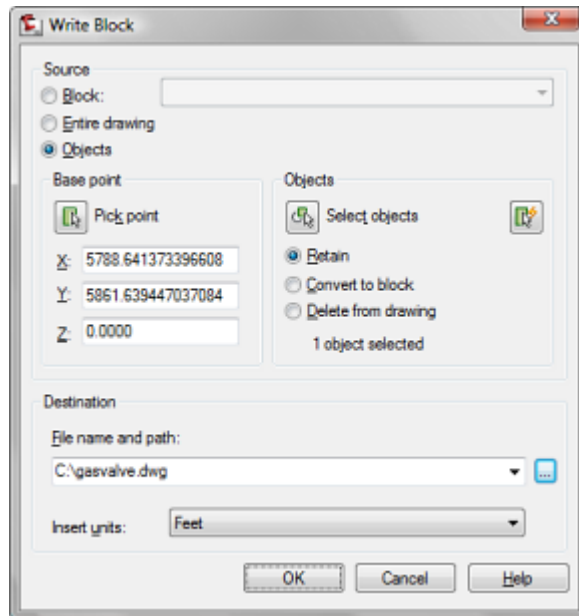
The Write Block command enables you to select objects in the drawing and save them to a separate DWG file.

You can also use the shortcut **W**.

32. In the Write Block dialog box, click Pick Point. Click any point as the base point for the Gas Valve objects.

33. Click Select Objects. Click the block. Press ENTER.

34. Click Navigate. Browse to a suitable folder. Enter **gasvalve**. Click Save.



35. Click OK. Click No to the Export AutoCAD Map Information query.

36. Open Windows Explorer and navigate to the folder to verify that the new *gasvalve.dwg* file was written.

37. Close the drawing without saving.

External References

There are frequently times when you are working on a drawing and have a good reason to want to view another drawing at the same time, either to put the first drawing in context or to view additional information when objects are not available in the first drawing. External Referencing (xref) is the method to use for this purpose.

Xrefs are for sharing graphical data. For example, a base plan can be shared into multiple engineering drawings. This way, if the base plan changes, the multiple

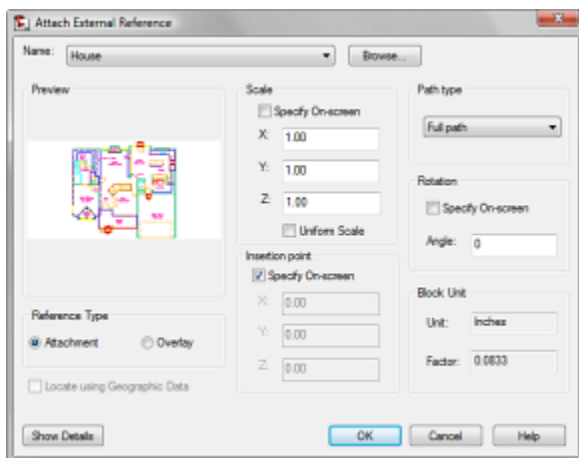
engineering drawings that share this xref are automatically updated.

38. Open the file *Subdivision.dwg*.



39. On the ribbon, Insert tab, Reference panel, click Attach.

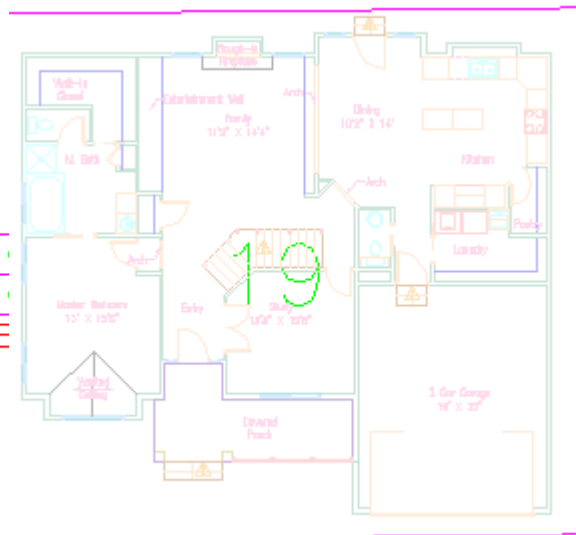
40. Navigate to the file *House.dwg*. Click Open.



41. Click OK for the default settings.

42. Move the cursor over lot 19 in the northeast section of the subdivision.

43. Click to place the xref.



It appears that the architect and the engineer need to talk about lot size versus house size. There does not appear to be sufficient setback from the lot lines to the house. You can now detach the drawing.

44. Use a right crossing window to select the House xref in the drawing. Right-click. Click Detach.

45. Close the drawing and do not save the changes.

Exercise 7: Work with Layouts and Viewports

In this exercise, you work with model and layout tabs, add and modify viewports, and prepare a layout.

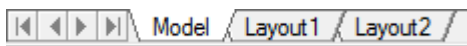
In general, model space (Model tab) is used for designing and working, while a paper space (Layout tab) is used for plotting. Although it is possible to plot from model space, it is most common to plot from a layout.

Viewports are a window into model space and are created in paper space on a Layout tab. You can control the number, size, and scale of the viewports on any layout.

For this exercise, open
... \I_C3DDrawingEnvironment-EX7.dwg.

1. View the drawing.

Look closely at the bottom of the drawing area and notice the Model tab and the two default Layout tabs.



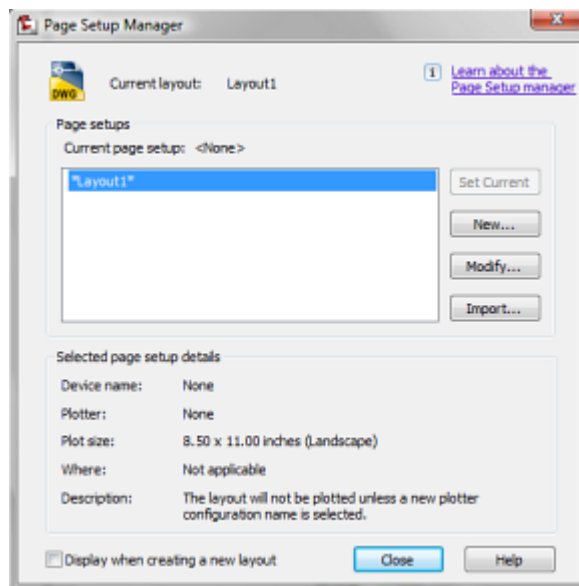
2. Click Layout1.

A default layout opens.

3. On the ribbon, Output tab, Plot panel, click Page Setup Manager.

You can store your favorite page setups in this dialog box. You can also select one if it has been previously saved. In this example, no page setups have been saved.

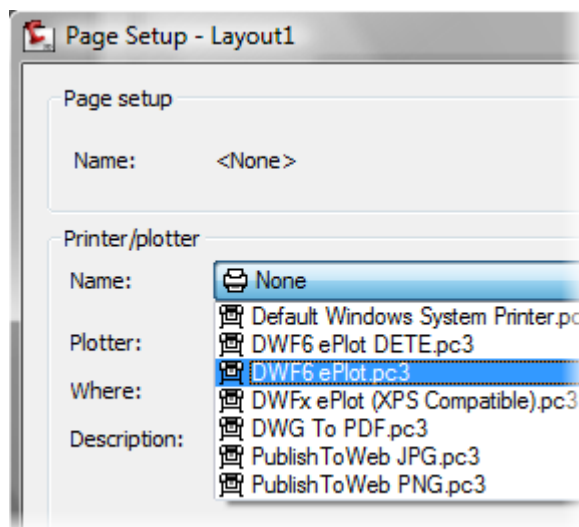
There is only one current page setup named Layout1. The details are shown in the bottom of the dialog box. Without a plotter specified, the layout will not be plotted.



4. Click Modify.

The Page Setup dialog box enables you to specify items such as paper size, printer or plotter name, drawing orientation, and scale. The page setup can be different for each layout.

5. Click the down arrow under Printer Name. Select DWF6 ePlot.pc3.



The available paper sizes are dependent upon which printer is chosen. The default paper size for the selected printer is ANSI Expand A (8.50 x 11.00 Inches).

6. Click the down arrow under Paper size to inspect the available sizes. Select ANSI Expand A, the default.

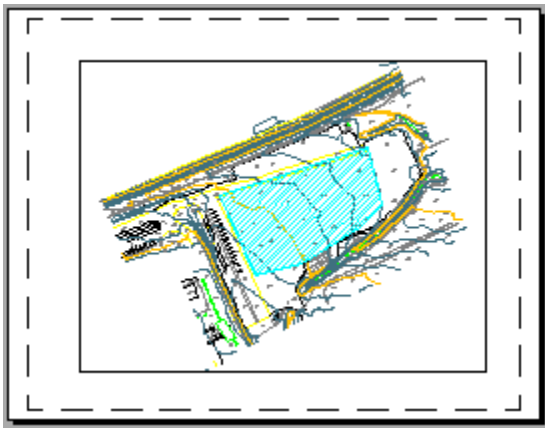
Note that metric sizes are included in the list.

7. Ensure that Layout is selected in the Plot area section.
8. Click the down arrow for Scale to inspect the options.
9. Select 1:1 as the scale.

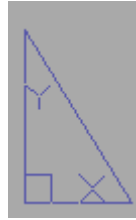
This setting applies to the paper scale, not the scale of the drawing. The 1:1 setting means that 1 inch on the paper is equal to 1 AutoCAD unit in the layout (paper space).

10. Click OK to accept these settings. In the Page Setup Manager dialog box, click Close.

The layout is displayed. The dashed lines indicate the printable area for the selected printer. The solid rectangular frame is an automatically generated viewport. A viewport is a layout object that windows an area of model space. When you are in paper space, a viewport can be moved, resized, or deleted. You can also change the scale of a viewport.



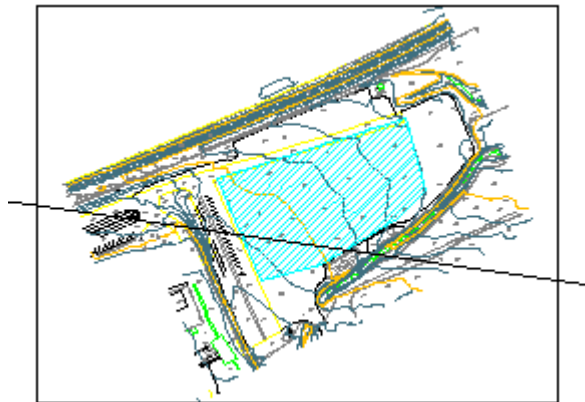
The current mode is paper space. You can tell by inspecting the icon in the lower left corner of the drawing area.



You can draw additional objects and text, zoom, and pan while in paper space. You also can get into model space while in the Layout tab.

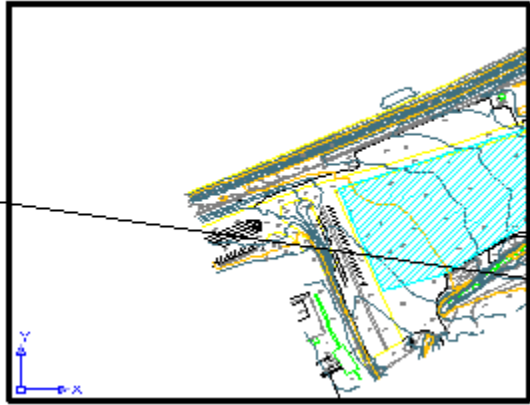
11. On the Draw toolbar, click Line.
12. Click the left side of the paper. Move the cursor to the right side of the paper. Click again to draw a line crossing the viewport. Press SPACEBAR.

While in paper space, the lines drawn are only in paper space, not in model space.



13. Double-click inside the viewport.
- You have now changed to model space while on a layout tab. Notice the button in the status bar and the UCS icon inside model space.

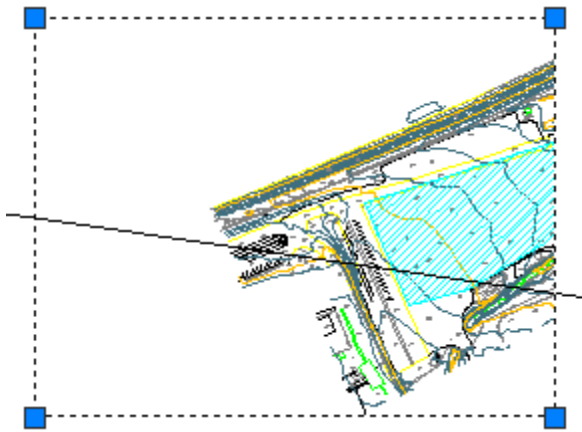
14. Pan the drawing within the viewport.
- Notice that the line does not move. You can zoom and pan, and run any other command as you would in the Model tab. If you try to select the line across the viewport, you cannot, because it is not in model space.



15. Double-click outside of the viewport.

You are now in paper space. Try to pan and zoom.

16. Click the viewport frame to select it.



17. Press DELETE.

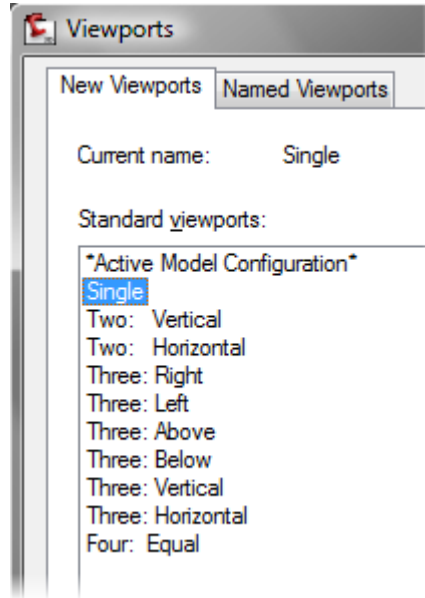
The viewport disappears, but the line stays.

18. Click the line and press DELETE.

You can create your own viewports. You can use the Viewports panel on the View tab; enter **VPORTS** at the command line; or open the Viewports toolbar.

19. Enter **VPORTS**. Press ENTER.

The Viewports dialog box enables you to choose one or more viewports in several different configurations. View the different options by selecting them.



20. Select Single viewport. Click OK.

The command line has several options. You can click a rectangle on the paper or choose to fit the viewport to the printable area.

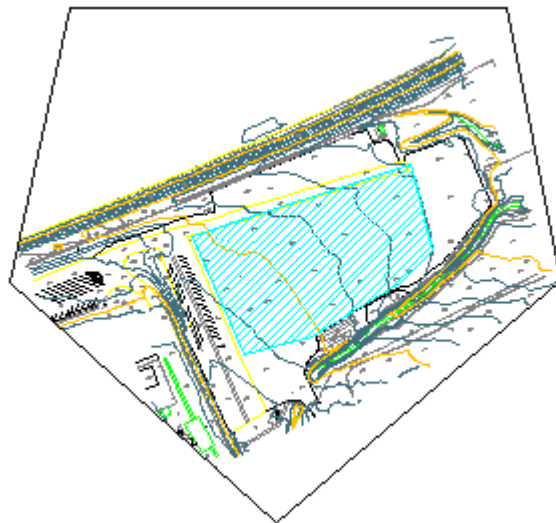
21. Press ENTER.

The Single viewport appears.

22. Select the viewport. Press DELETE.

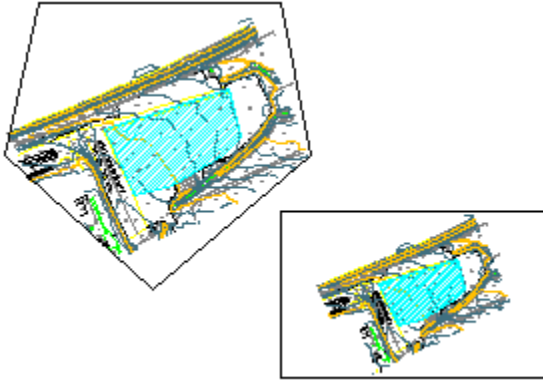
23. On the View tab, Viewports panel, click Create Polygonal.

24. Click an irregular shape on the paper. Enter **C**. Press ENTER to close.



Add a second viewport.

25. On the Viewports panel, click New.
26. Select Single Viewport in the Viewports dialog box. Click OK.
27. Click to outline the new viewport.



Viewports are powerful layout objects that can have different shapes and scales, and can show different visible layers based on settings in the Layer Properties Manager dialog box.

28. Close the drawing and do not save the changes.

Exercise 8: Plot an Engineering Drawing to Scale

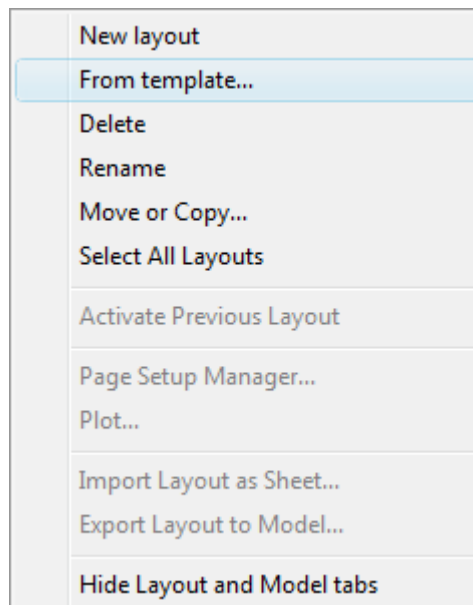
In this exercise, you use a template to create a new layout; create a scale bar; insert a north arrow; set the viewport scale; and preview and adjust the drawing and layout prior to plotting. While this exercise uses only one viewport, realize that you can use more than one viewport, and that each viewport can have its own scale and shape.

For this exercise, open
... \I_ C3DDrawingEnvironment-EX8.dwg.

1. View the drawing.

There are a number of drawing elements that should be included on every plotted sheet. Examples include a title block, north arrow, border, legend (if desired), drawing name, date, and so on. One of the easiest methods for putting a title block into a layout is to use a drawing template that already contains a title block. Another method is to insert a DWG file of a title block as a block into paper space.

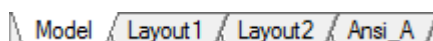
2. Right-click one of the layout tabs. Click From Template.



3. Select the *Map Book Template - 8.5x11 Classic.dwt* template file. Click Open.

4. Click OK to accept the Layout name.

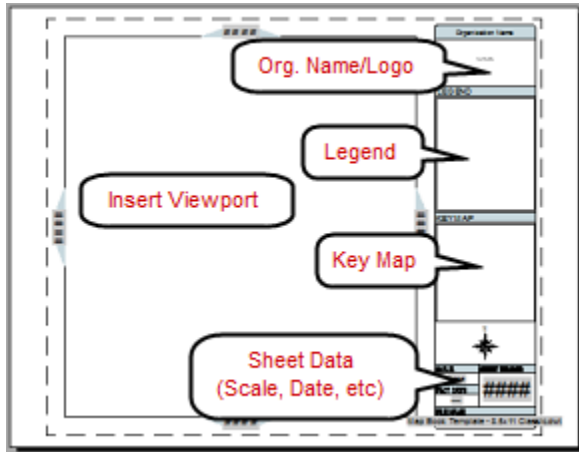
The new layout tab ANSI A Title Block is added.



5. Click the new layout tab. Zoom to the extents.

This layout contains a predefined sheet format with several embedded blocks that can be edited. A block can have attached attributes, or data tags, that are automatically inserted. You will edit these attributes.

You do not see the model in this layout yet because a main viewport is not part of this template. Next, you add a main viewport.



6. Enter **VPORTS**. Press ENTER.

Note: You can also use the New command in the Viewports panel of the View tab to create viewports.

7. Select Single in the dialog box. Click OK.
8. Use Endpoint object snaps to click two opposite sides of the box on the left of the title block.

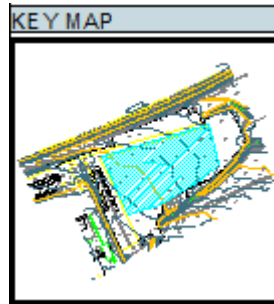
Your model is now displayed in the title block.



Edit the Title Block

There are two viewports already inserted on the right side of the title block for a legend and a key map.

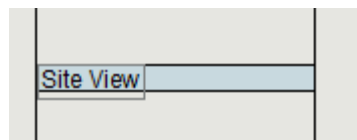
9. Double-click in the Key Map viewport to enter model space.
10. Enter **ZE**. Press SPACEBAR to zoom extents.



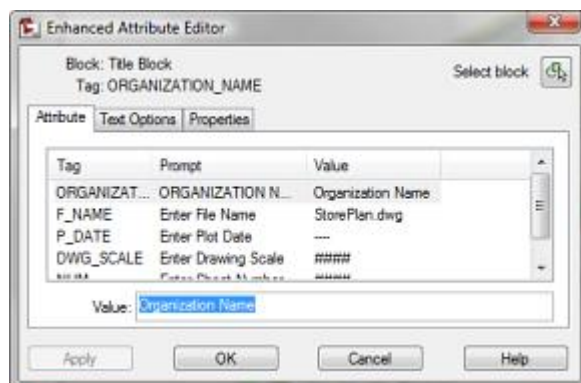
You can make your own Legend in the Model tab and then use the same technique as above to zoom into the legend in the Layout tab.

The rest of the right side of the template is one block that has individual attributes.

11. Double-click outside of a viewport to return to paper space.
12. Click the title block to select it.
13. Right-click. Click Block Editor.
14. Double-click the text object Key Map. Enter **Site View**.



15. Click Close Block Editor. Click Yes to save changes.
16. Again, click the block on the right to select it.
17. Right-click. Click Edit Attributes.



There are five attributes contained in this title block. You can edit all of them and the

values will be automatically placed on the layout.

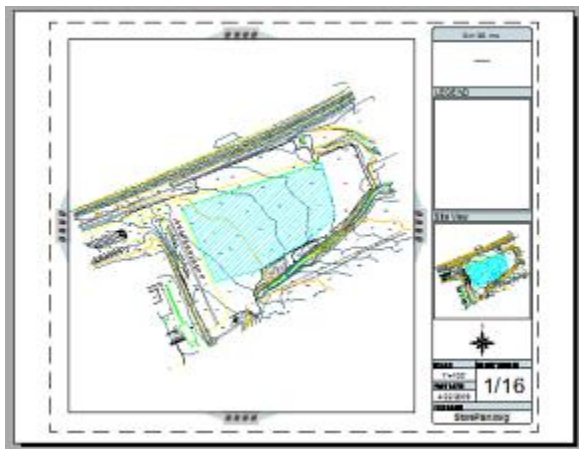
18. Click the ORGANIZATION_NAME attribute. Enter **Civil 3D, Inc.** in the Value box.
19. Click the P_DATE attribute. Enter **today's date**.
20. Click the DWG_SCALE attribute. Enter **1" = 100'**.
21. Click the NUM attribute. Enter **1\16** as if this was the first sheet of 16 total.

Attribute		
Text Options Properties		
Tag	Prompt	Value
ORGANIZAT...	ORGANIZATION N...	Civil 3D, Inc.
F_NAME	Enter File Name	StorePlan.dwg
P_DATE	Enter Plot Date	4/20/2009
DWG_SCALE	Enter Drawing Scale	1"=100'
NUM	Enter Sheet Number	1/16

22. Click OK.

View the changes to the layout. You can also use the Block Editor to modify the objects and text elements of the block.

There are four blocks on the edges of the main viewport box to enable labeling of adjoining plots. Use the Attribute Editor to modify the block attributes if desired, or delete them.



Now you draw a line of known length to verify the scale on paper. This is a simple

scale bar. You can add some additional vertical lines and hatch to make it fancy.

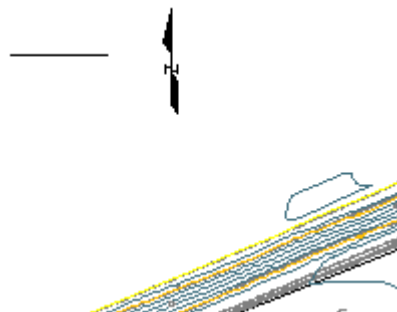
23. Double-click in the main viewport.
24. Enter **L**. Press SPACEBAR.
25. Click in the drawing to start the line.
26. Enter **@100,0**. Press SPACEBAR twice.

This draws a line 100 units in length.



One of the required elements of a drawing is a north arrow.

27. On the Insert tab, Block panel, click Insert.
28. In the Insert dialog box, click Browse.
29. Navigate to and select *NorthArrow.dwg*. Click Open.
30. Select the check boxes to specify both the Insertion Point and Scale On-screen. Click OK.
31. In model space, click in the drawing area to place the north arrow to the right of the line you drew. For a scale factor, enter **20**. Press ENTER twice.



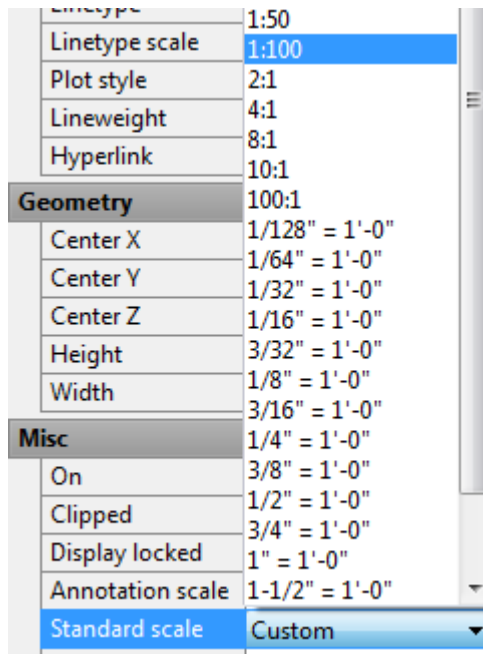
It is good practice to add these two elements in model space, not paper space. That way, the scale bar will adjust to the scale of the viewport. And the north arrow will rotate if the drawing is rotated. Most of the other drawing elements such as a title block, drawing name, company name, and legend should be added in paper space.

32. Double-click outside the viewport to enter paper space.

Change the Viewport Scale

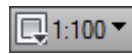
Now you change the scale of the viewport.

33. Click to select the main viewport. Right-click. Click Properties.
34. In the Misc section of the Properties palette, click in the Standard scale cell. Click the down arrow. Select 1:100. Close the Properties palette.



AutoCAD does an automatic conversion in the Imperial system for inches to feet. The 1:100 scale means that 1 inch on the paper in this viewport will be 100 feet (units) in the drawing. This is checked when the layout is plotted.

Another method to set the viewport scale is to use the down arrow for VP scale on the status bar.



35. Double-click inside the viewport. Pan the drawing so you can see the line and north arrow you drew. Move them if necessary so that the blue shaded area is shown in the viewport along with the north arrow and the line.

Do not zoom while in model space after you set the scale or the scale will change.

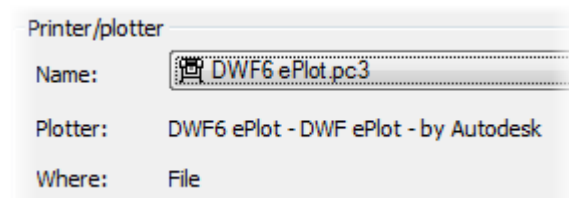
36. Double-click outside the viewport to enter paper space.

Plot the Drawing

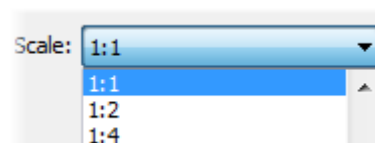
At this point, you are ready to plot the drawing. You can modify or add text inside the title block using the Block Editor if desired. You can also add other objects directly within paper space.

37. On the Output tab, Plot panel, click Plot.

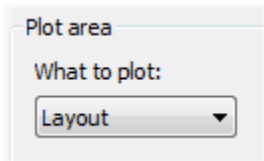
Notice that DWF6 ePlot.pc3 printer has been selected. This printer will produce a DWF plot file, rather than send the plot to your connected printer.



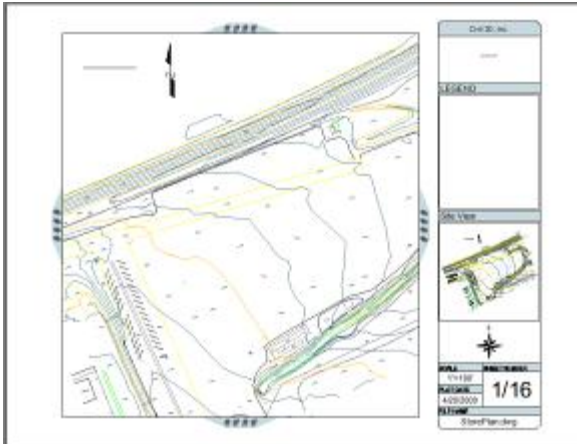
38. Click the down arrow under Scale. Select 1:1.



39. Under Plot Area, select Layout.



40. Click Apply to Layout. Click Preview.



41. From the Preview screen, click Plot.



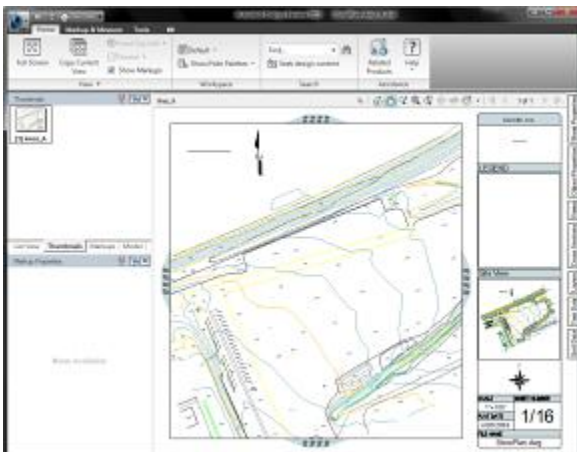
42. Name the plot file *StorePlan-Ansi_A.dwf*. Click Save.

A DWF file can be viewed and printed from Autodesk® Design Review.

43. Open Windows Explorer and navigate to the saved DWF file.

44. Double-click the file name.

Autodesk Design Review 2010 will open the file.



45. Click the Print button and inspect your printer settings.



46. Click OK to send the plot file to your local printer.

47. Close Autodesk Design Review.

48. Check the paper copy with your engineering scale to make sure the line in the viewport is 1 inch long.

49. Close the drawing in AutoCAD Civil 3D and do not save the changes.

Assessment

Challenge Exercise

Instructors provide a challenge exercise for students to do based on this lesson.

Questions

1. What is a selection set?
2. Where can you adjust the background color of the graphic screen?
3. What is an object snap?
4. What is the difference between relative and absolute coordinates?
5. How does a line differ from a polyline?
6. What are the five modes of a grip edit?
7. What is meant by exploding a block?
8. Is a zoom in paper space the same as a zoom in model space? Why or why not?
9. Can you have multiple viewports in a layout?
10. Should you place a title block in paper space or model space?

Answers

1. A selection set is a "selected" object or group of objects in a drawing. Once selected, the object(s) can have a command executed upon them in a noun/verb context.
2. The background color of the graphic screen can be adjusted by right-clicking the graphic screen and selecting Options, and then the Display tab.
3. An object snap specifies a precise location on an object. You can use an object snap whenever drawing to precisely locate a position such as the midpoint or endpoint of a line.
4. When drawing objects, you can use coordinates to enter a location. Absolute coordinates refer to the x, y, z coordinates of your active coordinate system. Relative coordinates are the coordinate *difference* from the current location.
5. A line is one object with a connected start and endpoint. A polyline is one object that contains a start and endpoint, but can also have multiple vertices in between, so it can appear to be multiple lines, but is actually only one object.
6. A grip edit is done when you select an object and highlight the grips and then select one particular grip to be "active." The five modes include Stretch, Move, Rotate, Scale, and Mirror, which you can toggle through using ENTER or SPACEBAR.
7. A block is composed of objects that can be drawn on different layers with various colors, linetypes, and lineweight properties. Exploding a block reduces the block to its elementary objects.
8. No. A zoom command in paper space changes the view of the paper, not the model. If in model space while zooming, the scale of the model will change.
9. Yes, and the viewports can be a different size and shape.
10. Since the title block does not change with the scale of the drawing, it should be placed in paper space; that way, its size is constant.

Lesson Summary

This lesson focused on developing familiarity with the AutoCAD Civil 3D drawing environment. This included basic object creation and modification, entity properties, layers, blocks, layouts, and template files. A strong knowledge of these basics will enable you to work with the AutoCAD Civil 3D lessons and software more efficiently.

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