

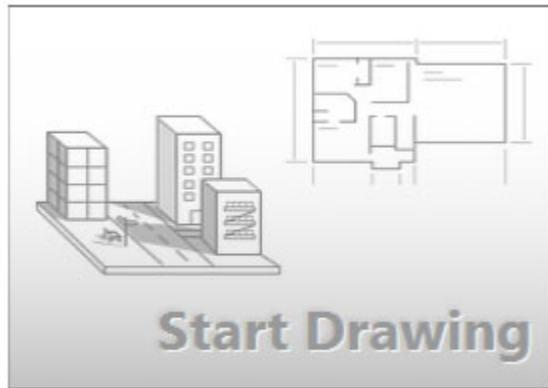
Basics of AutoCad

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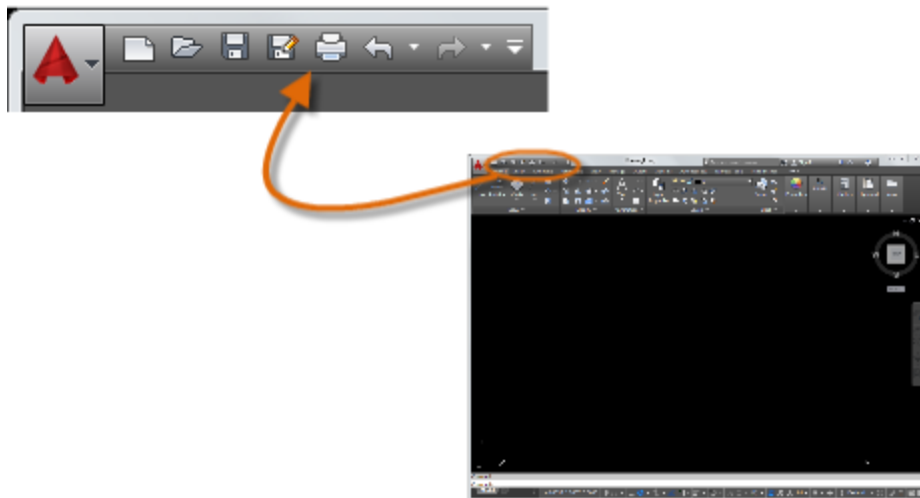
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Basics

After you launch AutoCAD, click the **Start Drawing** button to begin a new drawing.



AutoCAD includes a standard tabbed ribbon across the top of the drawing area. You can access nearly all the commands presented in this guide from the Home tab. In addition, the Quick Access toolbar shown below includes familiar commands such as New, Open, Save, Print, Undo, and so on.



Note: If the Home tab is not the current tab, go ahead and click it.

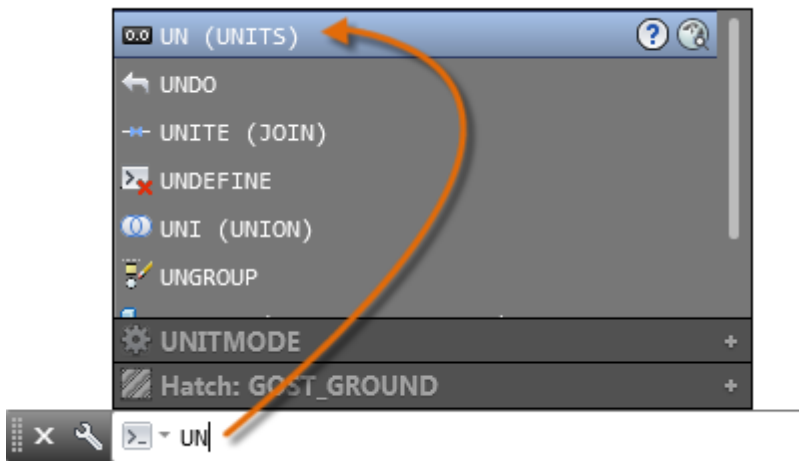
The Command Window

At the heart of AutoCAD is the Command window, which is normally docked at the bottom of the application window. The Command window displays prompts, options, and messages. **The Command window is opened by Ctrl+9**



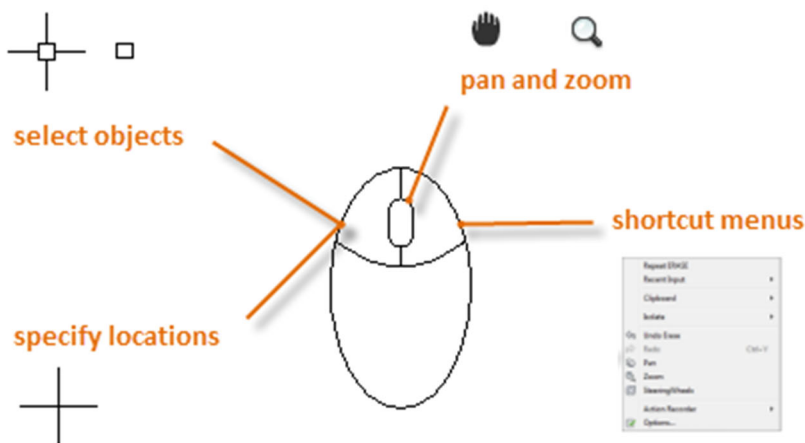
You can enter commands directly in the Command window instead of using the ribbon, toolbars, and menus. Many long-time AutoCAD users prefer this method.

Notice that as you start to **type a command**, it is completed automatically. When several possibilities are available such as in the example below, you can make your choice by clicking it or using the arrow keys and then pressing Enter or the Spacebar.



The Mouse

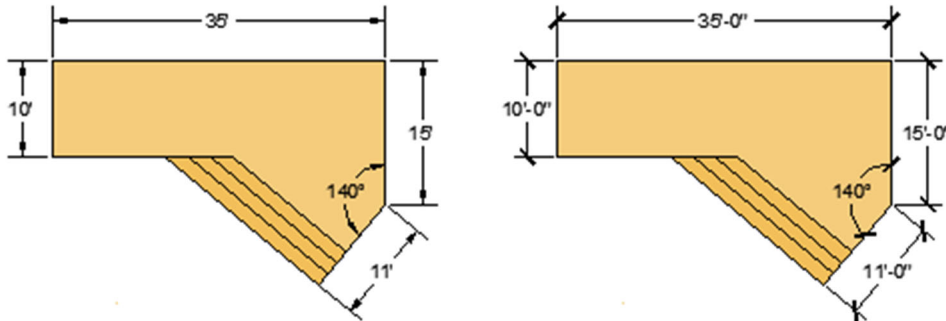
Most people use a mouse as their pointing device, but other devices have equivalent controls.



Tip: When you look for an option, try right-clicking. Depending on where you locate your cursor, different menus will display relevant commands and options.

New Drawings

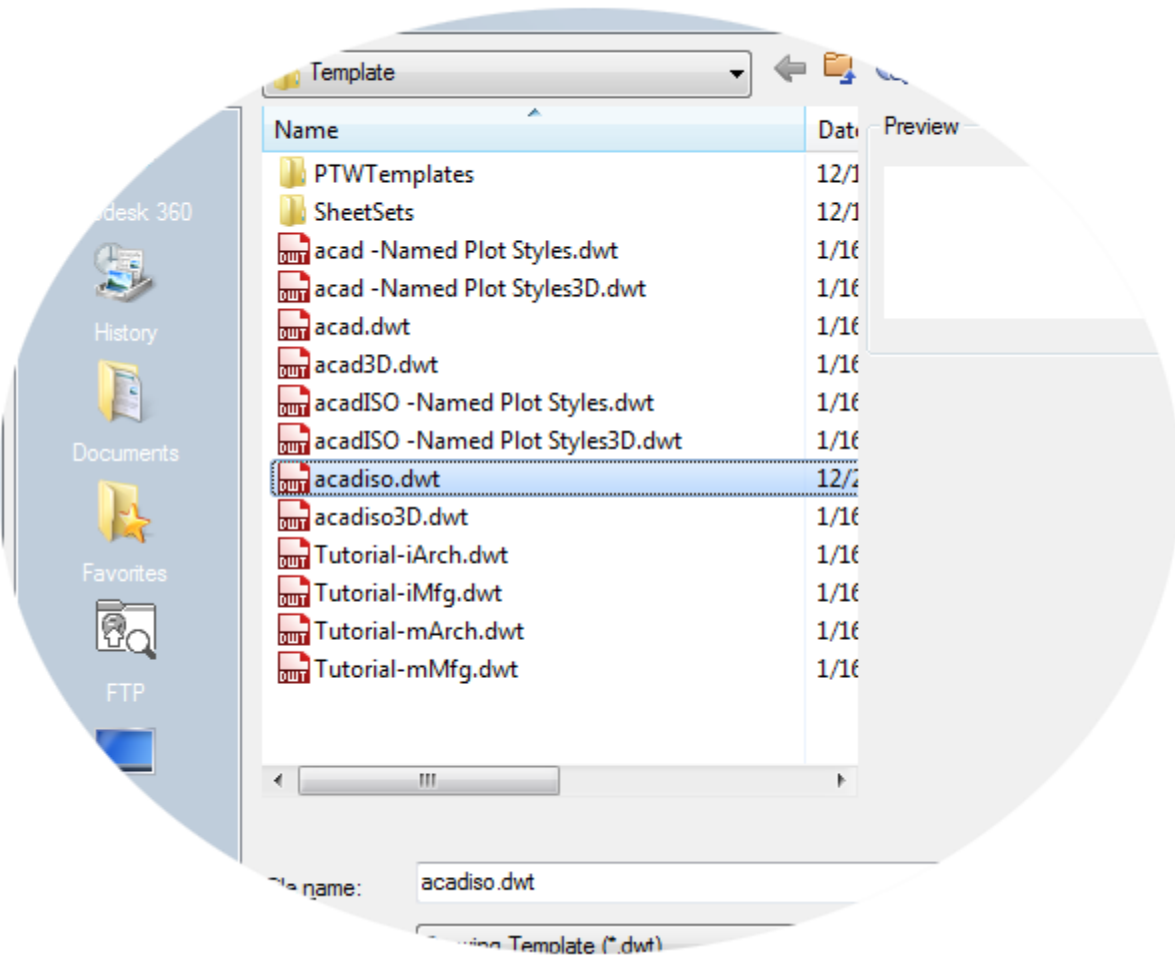
You can easily conform to industry or company standards by specifying settings for text, dimensions, linetypes, and several other features. For example, this backyard deck design displays two different dimension styles.



All these settings can be saved in a **drawing template** file. Click **New** to choose from several drawing template files (*.dwt):



- For **imperial drawings** that assume your units are inches, use *acad.dwt* or *acadlt.dwt*.
- For **metric units** that assume your units are millimeters, use *acadiso.dwt* or *acadltiso.dwt*.

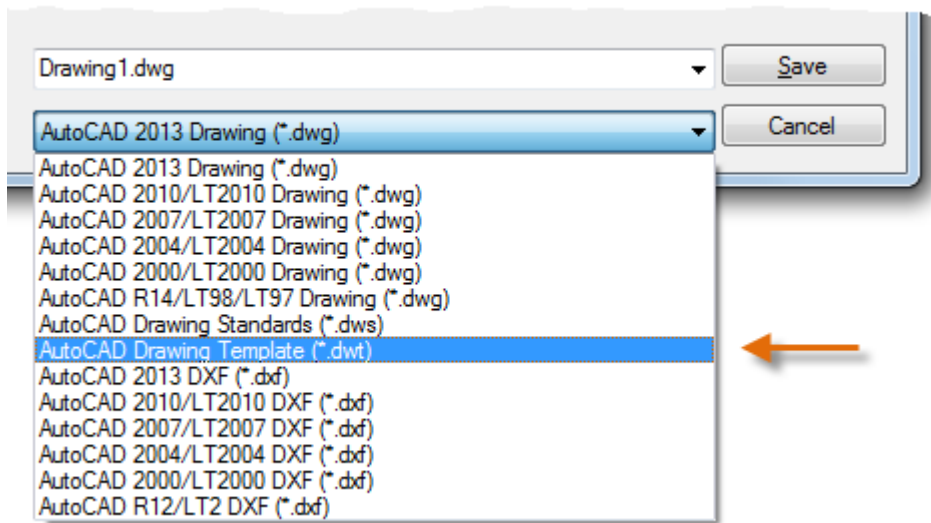


The "Tutorial" template files in the list are simple examples for the architectural or mechanical design disciplines with both imperial (i) and metric (m) versions. You might want to experiment with them. <C:\Program Files\Autodesk\AutoCAD 2019\UserDataCache\en-us\Template>

Most companies use drawing template files that conform to company standards. They will often use different drawing template files depending on the project or the client.

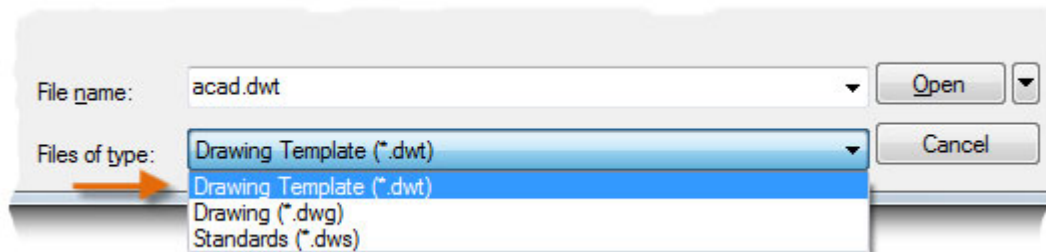
Create Your Own Drawing Template File

You can save any drawing (.dwg) file as a drawing template (.dwt) file. You can also open any existing drawing template file, modify it, and then save it again, with a different filename if needed.



If you work independently, you can develop your drawing template files to suit your working preferences, adding settings for additional features as you become familiar with them.

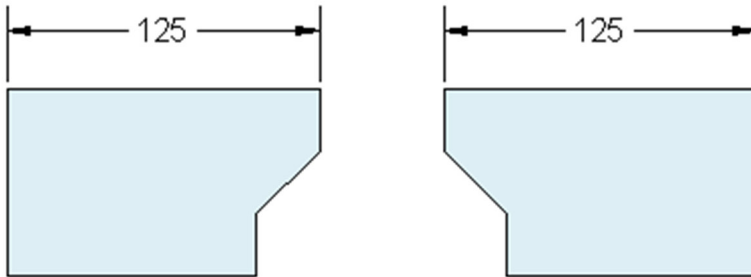
To modify an existing drawing template file, click **Open**, specify **Drawing Template (*.dwt)** in the Select File dialog box, and choose the template file.



Important: If your company has already established a set of drawing template files, check with your CAD manager before modifying any of them.

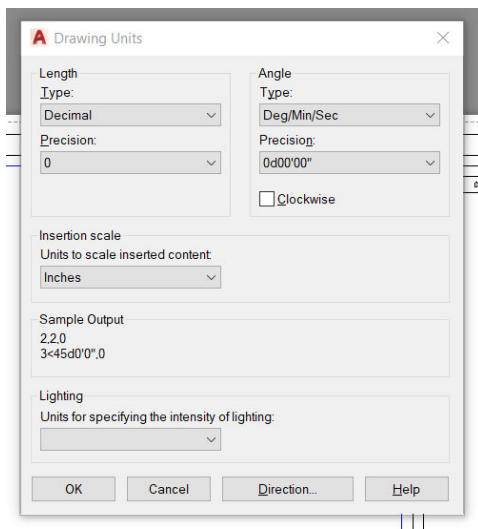
Units

When you first start a drawing, you need to **decide what the length of one unit represents**—an inch, a foot, a centimeter, a kilometer, or some other unit of length. For example, the objects below could represent two buildings that are each 125 feet long, or they could represent a section from a mechanical part that is measured in millimeters.



Unit Display Settings

After you decide what unit of length that you want to use, the **UNITS command** lets you control several unit display settings including the following:



- Format (or Type). For example, a decimal length of 6.5 can be set to display as a fractional length of 6-1/2 instead.
- Precision. For example, a decimal length of 6.5 can be set to display as 6.50, 6.500, or 6.5000.

If you plan to work in feet and inches, use the UNITS command to set the unit type to Architectural, and then when you create objects, specify their lengths in inches. **If you plan to use metric units, leave the unit type set to Decimal.** Changing the unit format and precision does not affect the internal precision of your drawing. It affects only how lengths, angles, and coordinates are displayed in the user interface.

Tip: **If you need to change the UNITS settings, make sure that you save the drawing as a drawing template file. Otherwise, you will need to change the UNITS settings for each new drawing.**

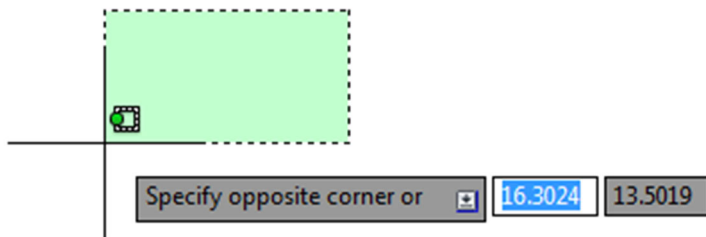
Model Scale

Always create your models at full size (1:1 scale). The term *model* refers to the geometry of your design. A *drawing* includes the model geometry along with the views, notes, dimensions, callouts, tables, and the title block displayed in the *layout*.

You can specify the scaling that is necessary to print a drawing on a standard-sized sheet later, when you create the layout.

Recommendations

- To open **Help** with information about the command in progress, simply **press F1**.
- To **repeat** the previous command, press **Enter or the Spacebar**.
- To see various options, select an object and right-click, or right-click a user interface element.
- To **cancel a command** in progress or if you ever feel stuck, **press Esc**. For example, if you click in the drawing area before entering a command, you will see something like the following:



Press Esc to cancel this preselection operation.

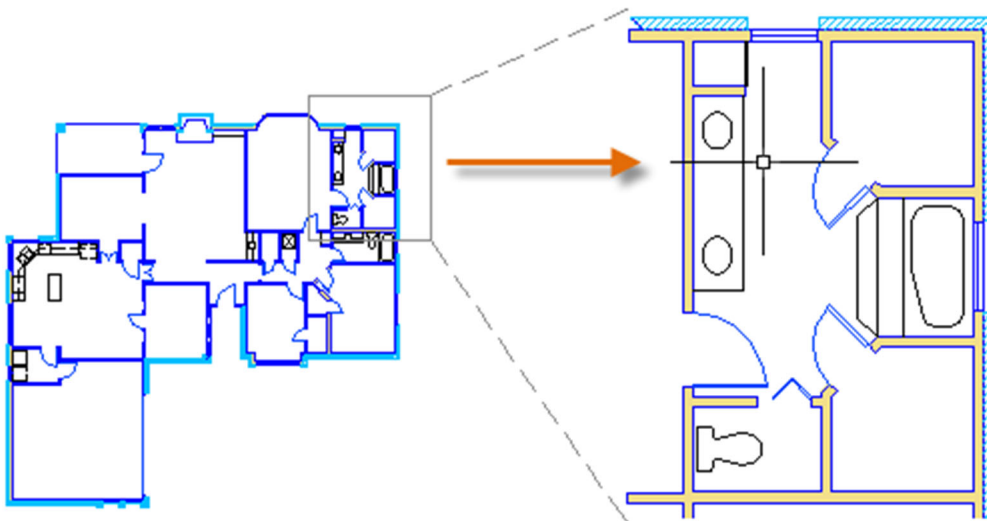
Viewing

Pan and zoom in a drawing, and control the order of overlapping objects.

The easiest way to change your view is by using the wheel on your mouse.

- Zoom in or out by rolling the wheel.
- Pan a view in any direction by holding the wheel down and then moving your mouse.
- Zoom to the extents of your model by clicking the wheel twice.

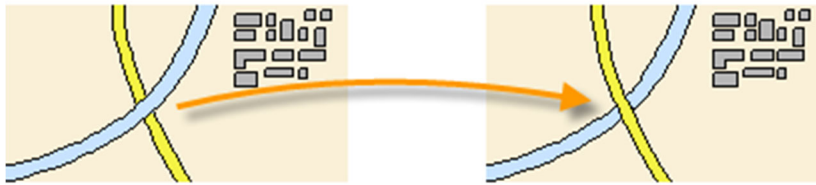
Tip: When you zoom in or out, the location of the cursor is important. Think of your cursor as a magnifying glass. For example, if you position the cursor in the upper-right area of the floor plan as shown below, zooming in magnifies the dressing room without shifting the view.



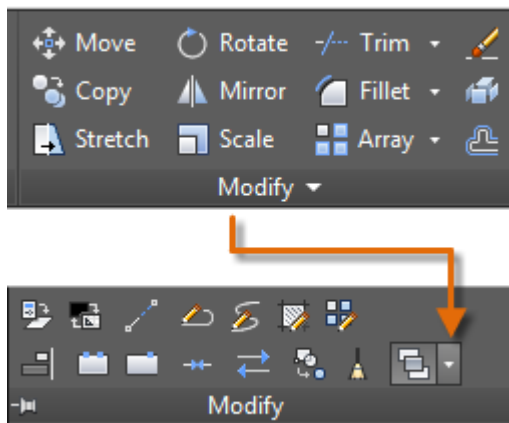
Note: If you cannot zoom or pan any more, type **REGEN** in the Command window and press Enter. This command regenerates the drawing display and resets the extents available for panning and zooming.

Overlapping Objects

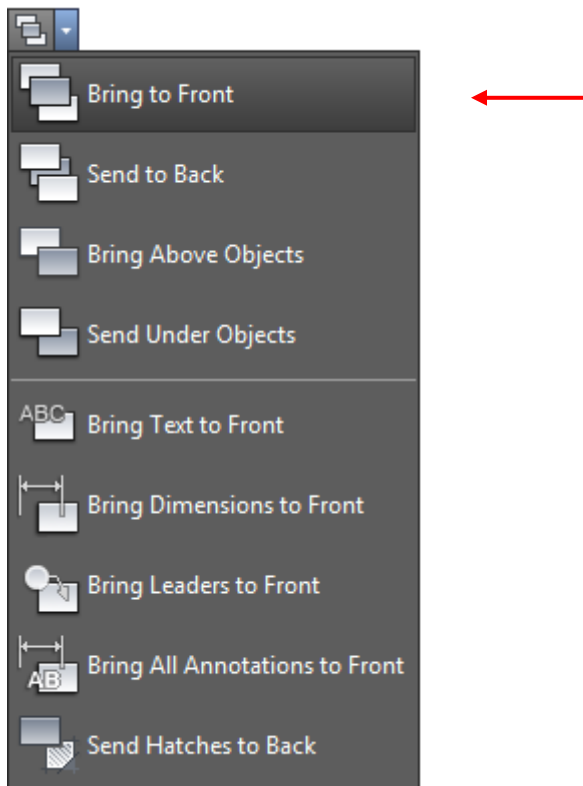
If you create objects that overlap each other, you might need to change which objects are displayed on top or in front of the others. For example, if you want the yellow highway to cross the blue river rather than the other way around, use the **DRAWORDER** command to reorder the objects.



You can access several **draw order** options from the Modify panel on the ribbon. Click to expand the **Modify panel**, and then click the down-arrow as shown below.



The draw order options that are listed include sending all hatches to the back, all text to the front, and so on.



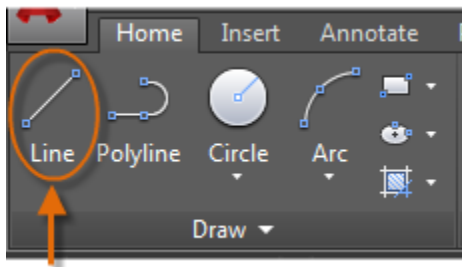
Geometry

You can create a lot of different types of geometric objects in AutoCAD, but you only need to know a few of them for most 2D drawings.

Tip: If you want to simplify the display while you create geometric objects, press F12 to turn off dynamic input.

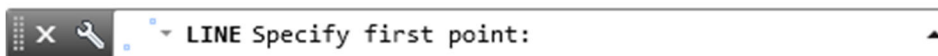
Lines

The line is the most basic and common object in AutoCAD drawings. To draw a line, click the **Line tool**.

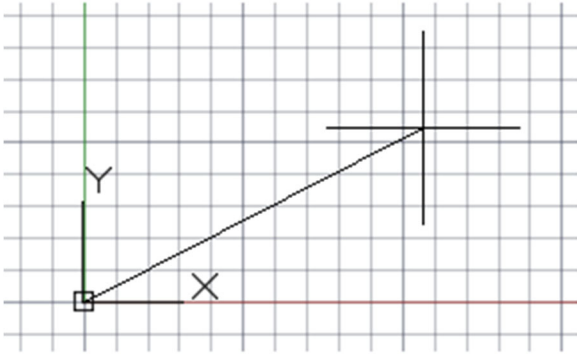


Alternatively, you can type **LINE or just L** in the Command window, and then press **Enter or the Spacebar**.

Notice the prompt in the Command window for a point location.



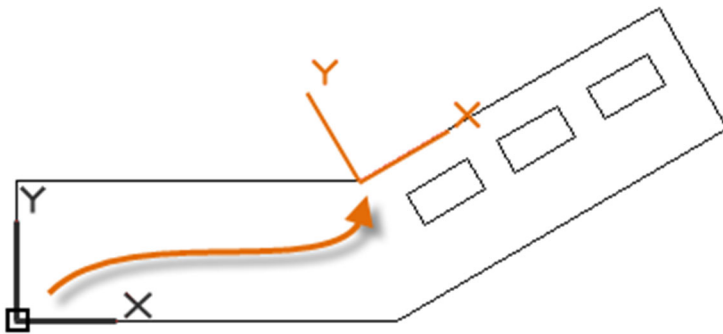
To specify the starting point for this line, you would **type in the coordinates 0,0**. It is a good idea to locate one corner of your model at 0,0, which is called the origin point. To locate additional points, you could **specify additional X,Y coordinate locations** in the drawing area, however more efficient methods for specifying points are available, and will be presented in the Precision topic.



After you specify the next point, the LINE command automatically repeats itself, and it keeps prompting you for additional points. **Press Enter or the Spacebar to end** the sequence.

The User Coordinate System

The **user coordinate system (UCS)** icon indicates the direction of the positive X and Y axis for any coordinates that you enter, and it also defines the horizontal and vertical directions in a drawing. In some 2D drawings, **it can be convenient to click, drag, and rotate the UCS to change the origin point**, and the horizontal and vertical directions.



Grid Display

To display the grid, type **grid** in the command line.

Select **On**

To snap to grid, select **Snap**

Some people like working with grid lines as a reference, while others prefer working in a blank area. To **turn off** the grid display, **press F7**. Even with the grid turned off, you can force your cursor to snap to grid increments by pressing F9.

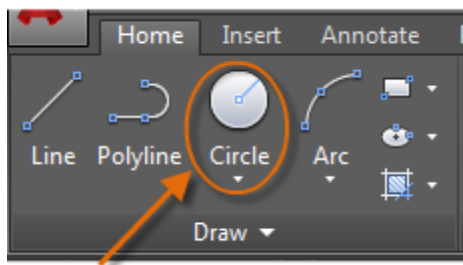
Lines as Construction Aids

Lines can serve as reference and construction geometry such as

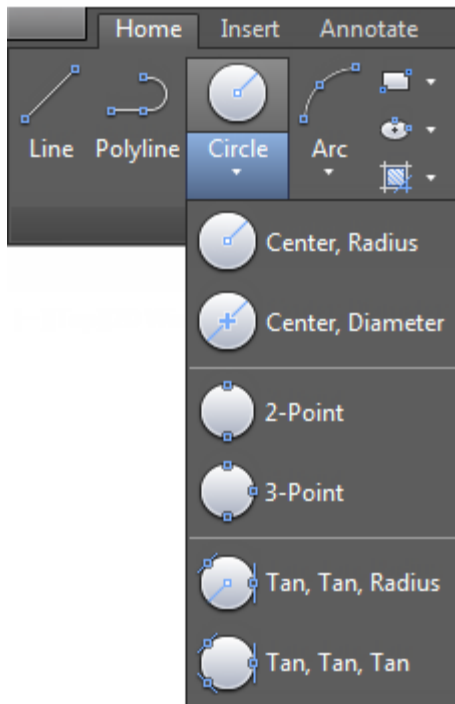
- Property line setbacks
- The mirror line of a symmetrical mechanical part
- Clearance lines to avoid interferences
- Traversal path lines

Circles

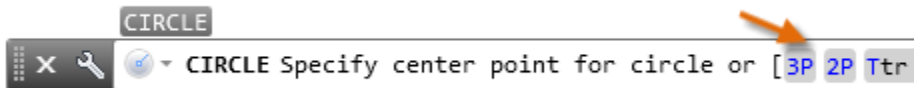
The default option of the **CIRCLE command** requires you to specify a center point and a radius.



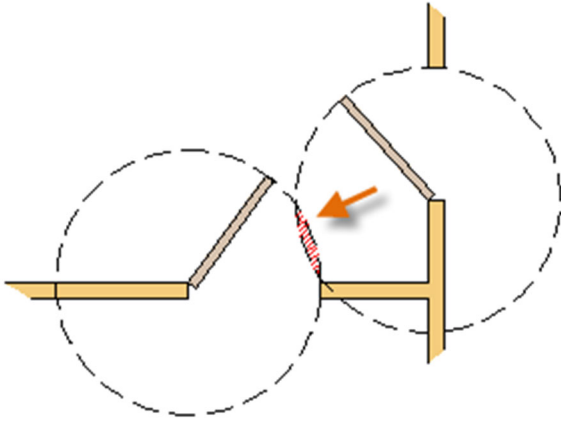
The other circle options are available from the drop-down:



Alternatively, you can also enter CIRCLE or just C in the Command window and click to choose an option. If you do, you can specify a center point, or you can click one of the highlighted command options as shown below.

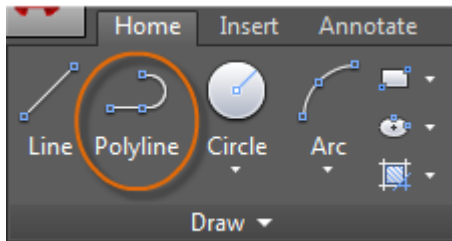


Circles can be useful as reference geometry. For example, you can see that the two doors in the illustration can interfere with each other.



Polylines and Rectangles

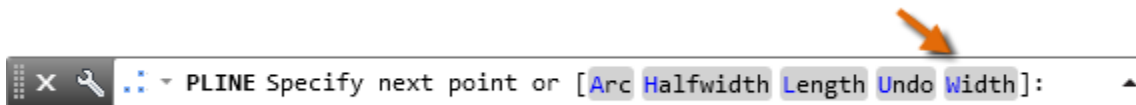
A polyline is a connected sequence of line or arc segments that is created as a single object.



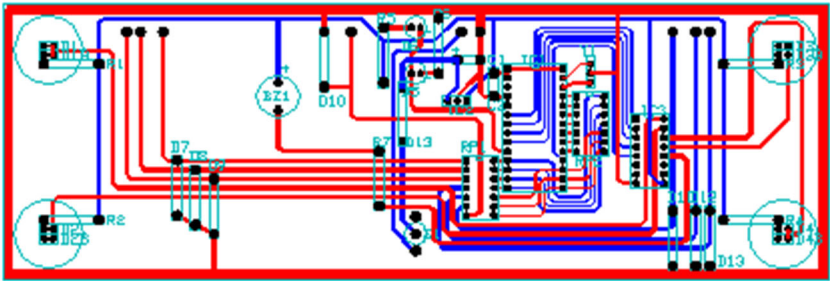
Use the PLINE command to create open or closed polylines for

- Geometry that needs to have fixed-width segments
- Continuous paths for which you need to know the total length
- Contour lines for topographic maps and isobaric data
- Wiring diagrams and traces on printed circuit boards
- Process and piping diagrams

Polylines can have a constant width or they can have different starting and ending widths. After you specify the first point of the polyline, you can use the Width option to specify the width of all subsequently created segments. You can change the width value at any time, even as you create new segments.



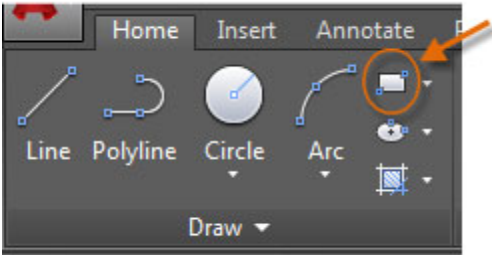
Here is an example of a printed circuit board in which the traces were created with wide polylines. The landing pads were created with the DONUT command.



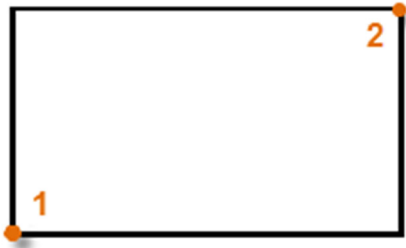
Polylines can have different starting and ending widths for each segment as shown here:



A fast way to create closed rectangular **polylines** is to use the **RECTANG** command (enter REC in the Command window).

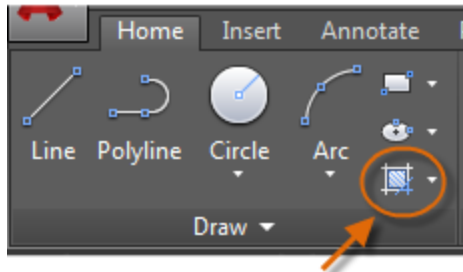


Simply click two diagonal points for the rectangle as illustrated. If you use this method, turn on grid snap (F9) for precision.



Hatches and Fills

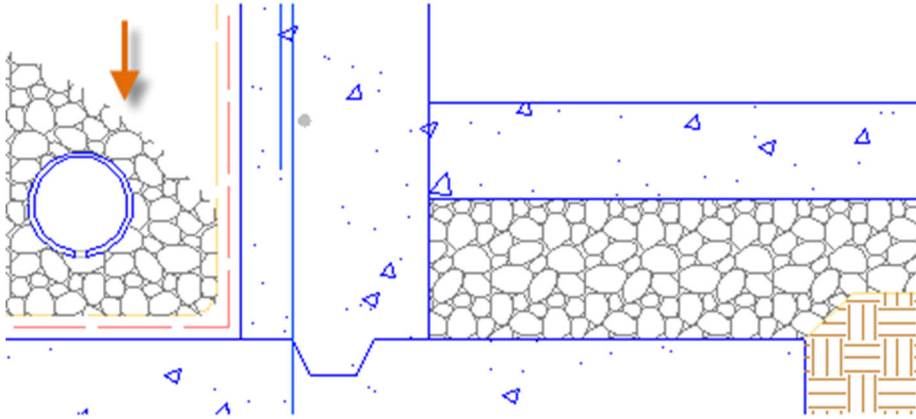
In AutoCAD, a hatch is a single, compound object that covers a specified area with a pattern of lines, dots, shapes, a solid fill color, or a gradient fill.



When you start the **HATCH command**, the ribbon temporarily displays the Hatch Creation tab. On this tab, you can choose from more than 70 industry-standard imperial and ISO hatch patterns, along with many specialized options.

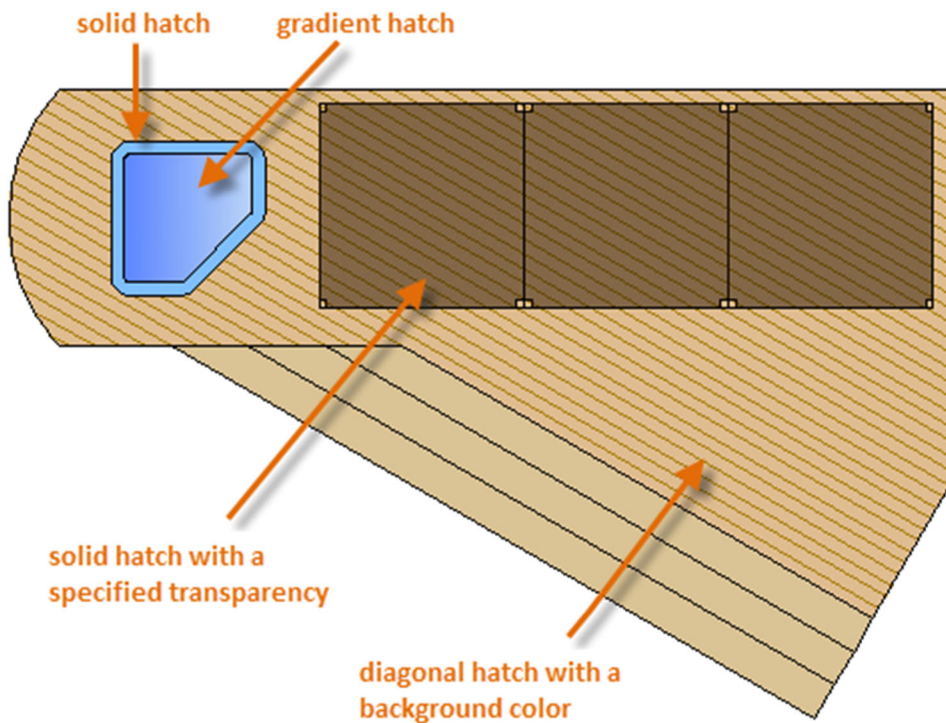
The simplest procedure is to **choose a hatch pattern and scale from the ribbon**, and **click within any area that is completely enclosed by objects**. **You need to specify the scale factor for the hatch to control its size and spacing.**

After you create a hatch, you can move the bounding objects to adjust the hatch area, or you can delete one or more of the bounding objects to create partially bounded hatches:

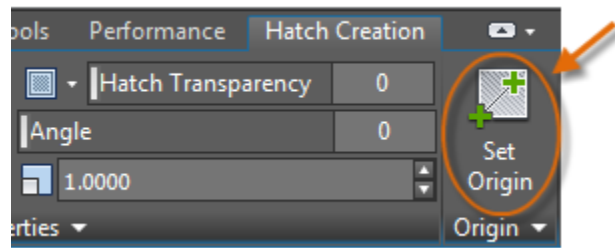


Tip: If you set a hatch pattern to be a solid or gradient fill, also consider setting a transparency level on the Hatch Creation tab for interesting overlap effects.

Here are some examples of how you can use solid-fill hatches:



Tip: If you need to align the pattern in a hatch, which might be the case with the decking boards above, use the Set Origin option to specify an alignment point.



Note: If an area is not completely enclosed, red circles display to indicate places to check for gaps. Enter REDRAW in the Command window to dismiss the red circles.

Precision

Ensure the **precision required for your models.**

There are several precision features available, including

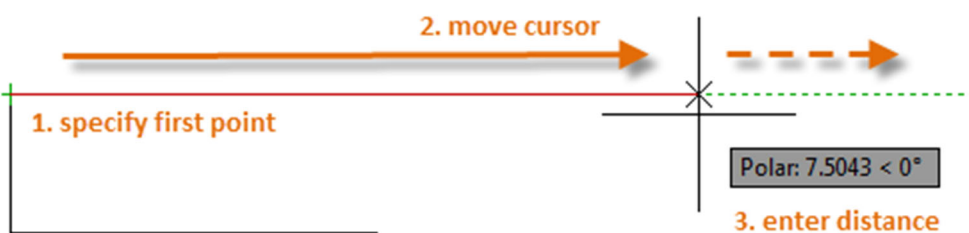
- **Polar tracking.** Snap to the closest preset angle and specify a distance along that angle.
- **Locking angles.** Lock to a single, specified angle and specify a distance along that angle.
- **Object snaps.** Snap to precise locations on existing objects, such as an endpoint of a polyline, the midpoint of a line, or the center point of a circle.
- **Grid snaps.** Snap to increments in a rectangular grid.
- **Coordinate entry.** Specify a location by its Cartesian or polar coordinates, either absolute or relative.

The three most commonly used features are polar tracking, locking angles, and object snaps.

Polar Tracking

When you need to specify a point, such as when you create a line, you can use polar tracking to guide the movement of your cursor in certain directions.

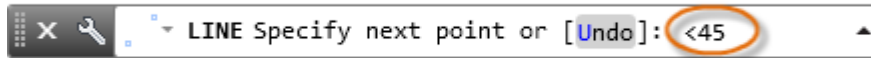
For example, after you specify the first point of the line below, move your cursor to the right, and then **enter a distance in the Command window to specify a precise horizontal length for the line.**



By default, polar tracking is turned on and guides your cursor in a horizontal or vertical direction (0 or 90 degrees).

Locking Angles

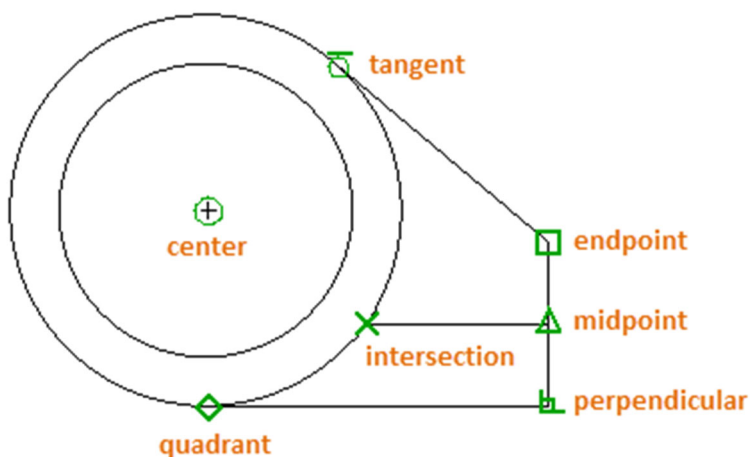
If you need to draw a line at a specified angle, you can lock the angle for the next point. For example, if the second point of a line needs to be created at a 45 degree angle, you would enter <45 in the Command window.



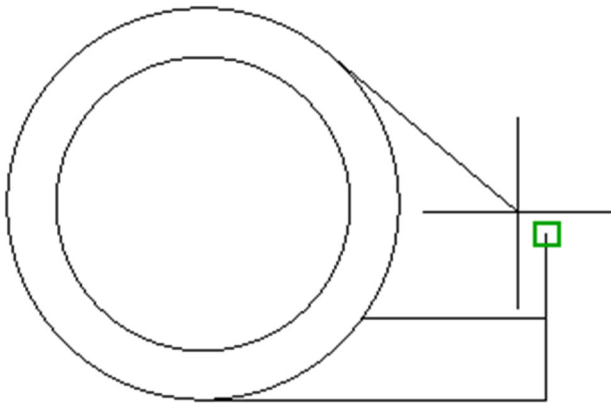
After you move your cursor in the desired direction along the 45-degree angle, you can enter the length of the line.

Object Snaps

By far, the most important way for you to specify precise locations on objects is to use object snaps. In the following illustration, several different kinds of object snaps are represented by markers.

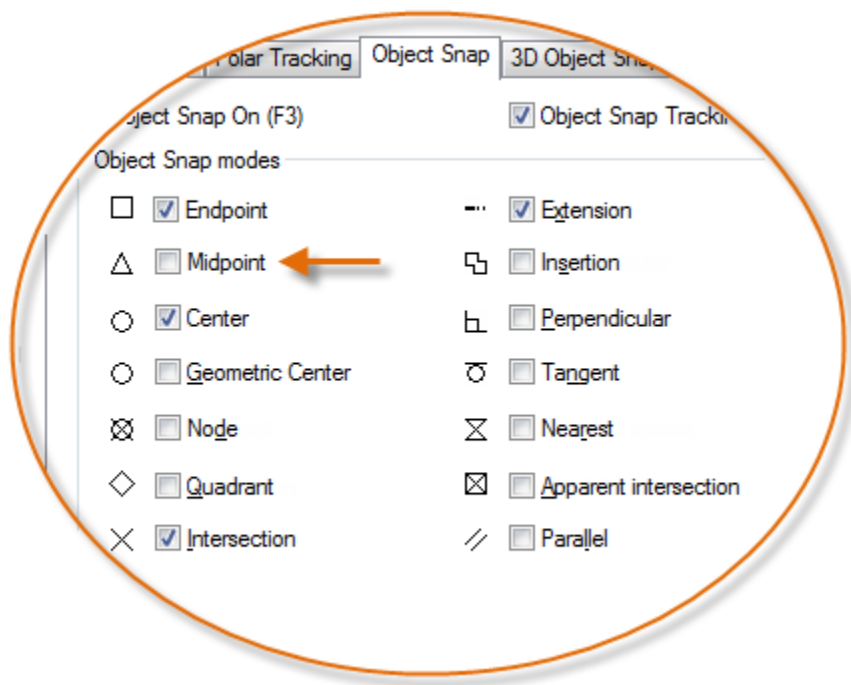


Object snaps become available during a command whenever AutoCAD prompts you to specify a point. For example, if you start a new line and move your cursor near the endpoint of an existing line, the cursor will automatically snap to it.



Set Default Object Snaps

Enter the **OSNAP** command to set the default object snaps, which are also called "running" object snaps. For example, you might find it useful to turn on the **Midpoint** object snap by default.



Recommendations

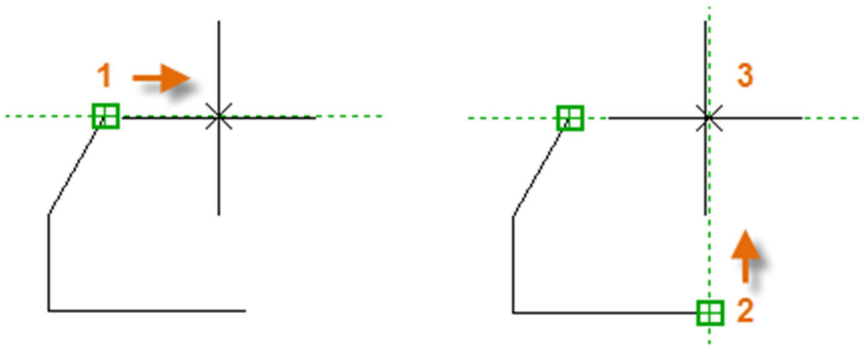
- At any prompt for a point, you can specify a single object snap that overrides all other object snap settings. You hold down Shift, right-click in the drawing area,

and choose an object snap from the Object Snap menu. Then move the cursor to select a location on an object.

- Make sure that you zoom in close enough to avoid mistakes. In a densely populated model, snapping to the wrong object will result in an error that can propagate throughout your model.

Object Snap Tracking

During a command, you can align points both horizontally and vertically from object snap locations. In the following illustration, you first hover over endpoint 1 and then hover over endpoint 2. When you move your cursor near location 3, the cursor locks into the horizontal and vertical location shown.



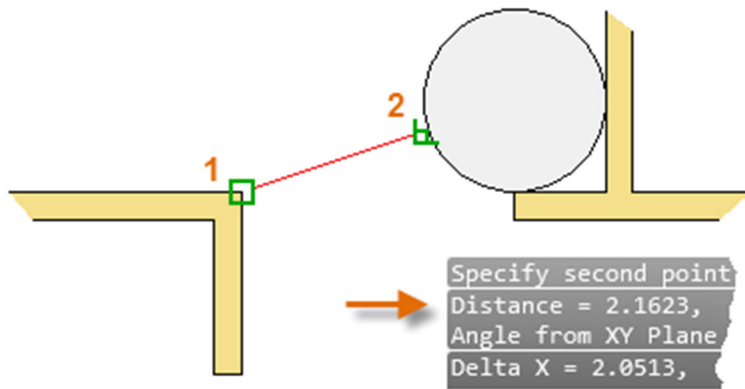
You can now finish creating the line, circle, or other object that you were creating from that location.

Verify Your Work

Recheck your geometry to catch mistakes early. Enter the DIST command (or just DI) to measure the distance between any two points in your model.

For example, you might need to find the clearance between two points shown, which might represent the corner of a wall and a small table, or perhaps a 2D section of a plastic part and a wire.





After you enter DIST, click the endpoint on the corner (1). Next, hold down Shift as you right-click, and then choose Perpendicular from the object snap menu. Finally, click the circle (2).



The number of decimal places and unit style displayed in the result is controlled by the UNITS command.

Handy Function Key Reference

The keyboard function keys all have assignments in AutoCAD. The ones that are most commonly turned on and off are indicated with a key.

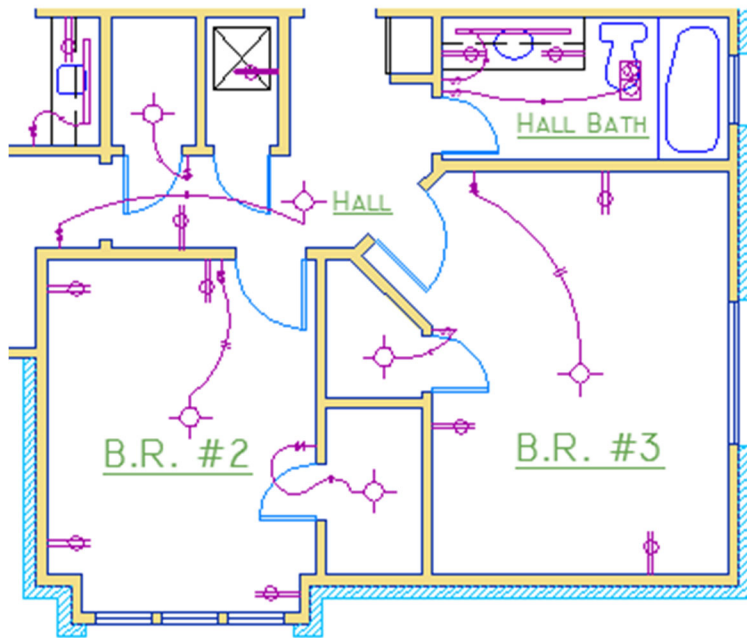
Key	Feature	Description
F1 	Help	Displays Help for the active tooltip, command, palette, or dialog box.
F2	Expanded history	Displays an expanded command history in the Command window.
F3	Object Snap	Turns object snap on and off.
F4	3D Object Snap	Turns on additional object snaps for 3D elements.
F5	Isoplane	Cycles through 2-1/2D isoplane settings.
F6	Dynamic UCS	Turns on UCS alignment with planar surfaces.
F7	Grid display	Turns the grid display on and off.
F8 	Ortho	Locks cursor movement to horizontal or vertical.
F9	Grid Snap	Restricts cursor movement to specified grid intervals.
F10 	Polar Tracking	Guides cursor movement to specified angles.
F11	Object Snap Tracking	Tracks the cursor horizontally or vertically from object snap locations.
F12 	Dynamic Input	Displays distances and angles near the cursor and accepts input as you use Tab between the fields.

Note: F8 and F10 are mutually exclusive—turning either one on will turn the other one off.

Properties

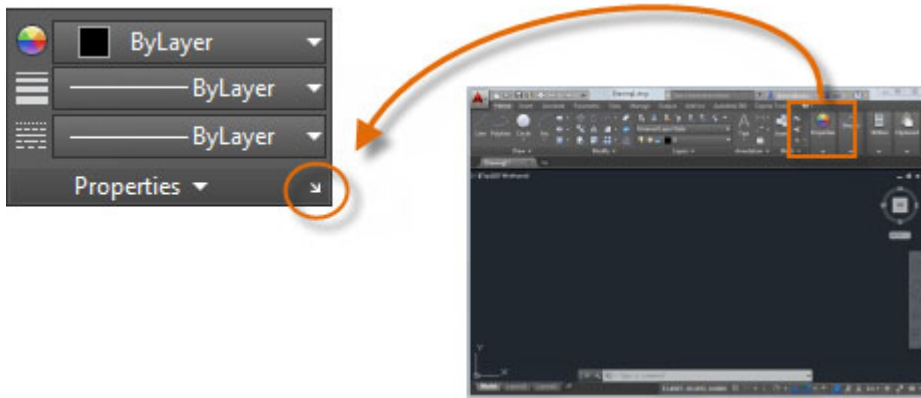
You can assign **properties such as color and linetype** to individual objects, or as default properties assigned to layers.

In the following drawing, the walls, exterior stone facing, doors, fixtures, cabinetry, HVAC, electrical, and text were created using different colors to help differentiate between them.

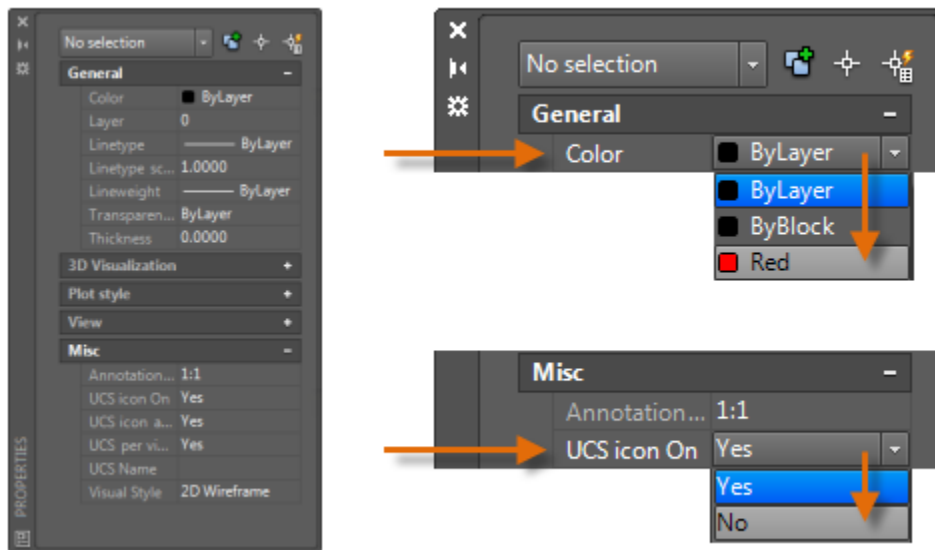


The Properties Palette

The Properties palette is an essential tool. You can open it with the **PROPERTIES** command (enter PR in the Command window), you can press **Ctrl + 1**, or you can click the tiny arrow in the Properties panel on the Home tab—whichever you prefer.

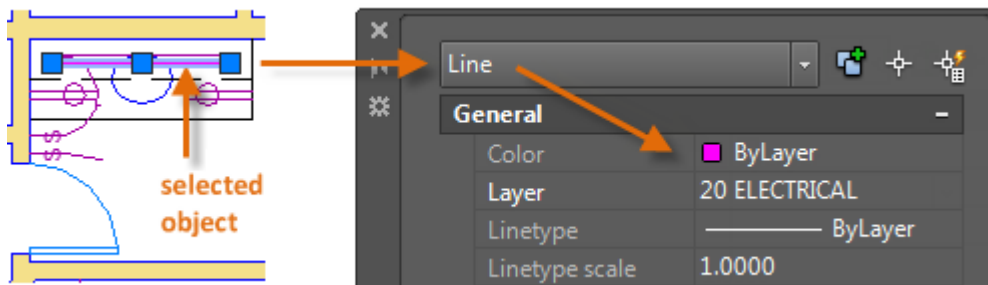


The Properties palette displays a list of all the important property settings. You can click any of the available fields to change the current settings. In the following example, if no objects are selected, the current color will be changed from ByLayer to Red and the UCS icon will be turned off.



Verify and Change Object Properties

You can use the Properties palette to verify and change property settings for selected objects. If you click an object in your drawing to select it, here is what you might see in the Properties palette.



Notice that the current properties for the selected object are displayed in the palette. You can change any of these properties by clicking and changing the setting.

A property that is set to "ByLayer" inherits its setting from the layer. In the previous example, the objects that were created on the 20 ELECTRICAL layer are purple because that is the default color of the objects on that layer.

If you select several objects, only their common properties are listed in the Properties palette. If you change one of these properties, all the selected objects will change in one operation. Selecting objects is covered in more detail in the Modifying topic.

Note: To clear the current selection, press Esc.

Quick Access to Property Settings

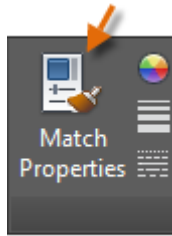
The Properties palette takes up a lot of space. For quick access to the most common properties, use the Properties panel. As you can see in this example, the listed properties will all be determined by the current layer.



The Properties panel works the same way as the Properties palette. When you select an object, the current property settings are replaced by the properties assigned to the selected object, and you can use this panel to easily change the properties of one of more selected objects.

Match the Properties of Objects

For a fast way to copy the properties of a selected object to other objects, use the Match Properties tool, or enter MATCHPROP or MA in the Command window.

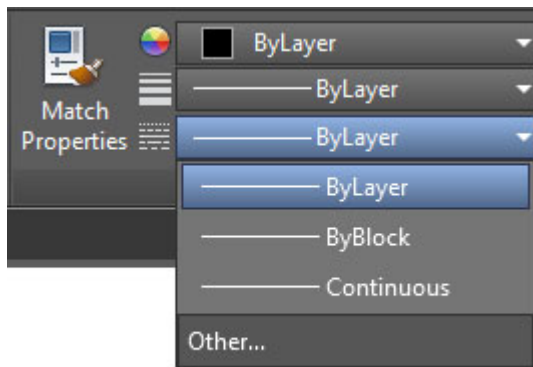


Select the source object, and then select all of the objects that you want to modify.

Linetypes

Dashed and other non-continuous linetypes are assigned from the Properties panel. You must first load a linetype before you can assign it.

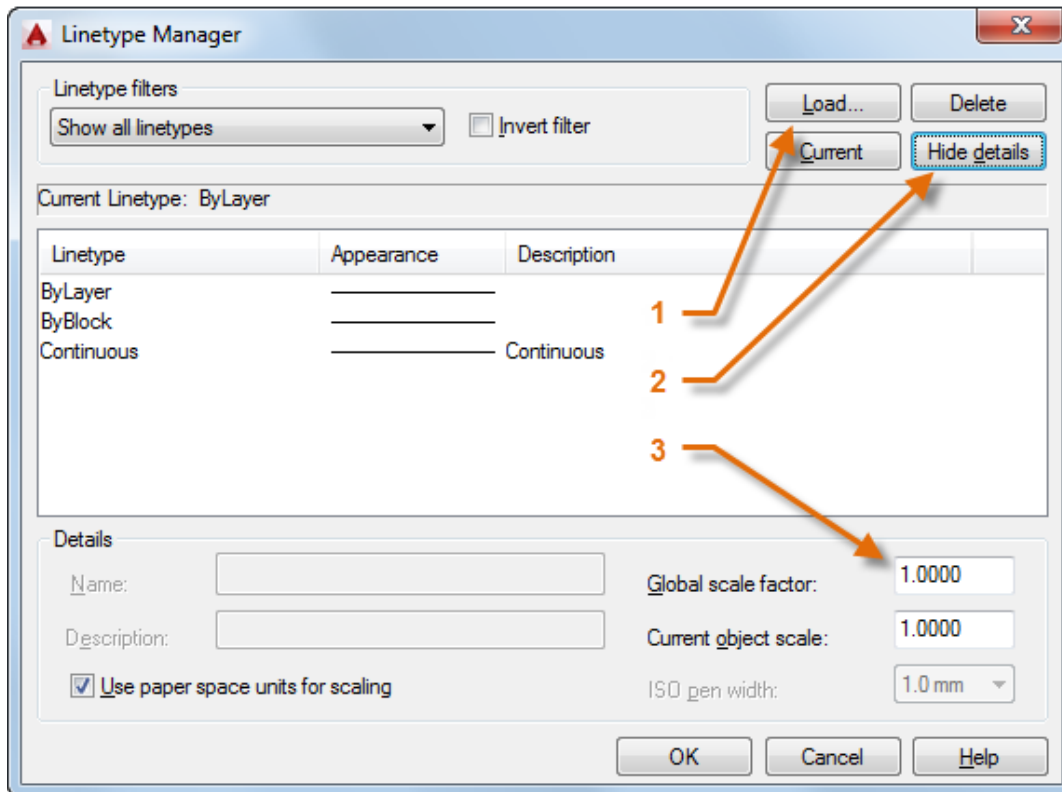
In the Linetype drop-down list, click Other.



This action displays the Linetype Manager dialog box.

Perform the following steps in order:

1. Click Load. Choose one or more linetypes that you want to use. Notice that dashed (non-continuous) linetypes come in several preset sizes.
2. Click Show/Hide details to display additional settings.
3. Specify a different "global scale factor" for all linetypes—the larger the value, the longer the dashes and spaces. Click OK.

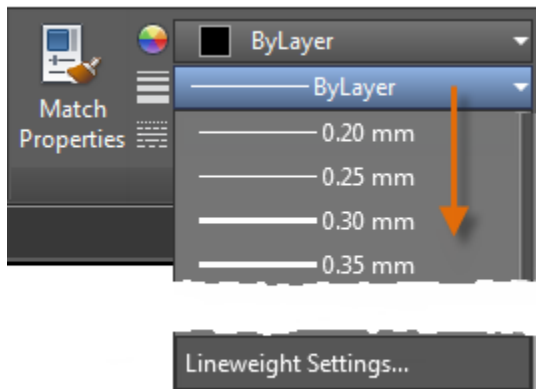


Once you've loaded the linetypes that you plan to use, you can select any object and specify a linetype from the Properties panel or the Properties palette. Alternatively, you can specify a default linetype for any layer in the Layer Properties Manager.

Lineweights

The Lineweight property provides a way to display different thicknesses for selected objects. The thickness of the lines remain constant regardless of the scale of the view. In a layout, lineweights are always displayed and printed in real-world units.

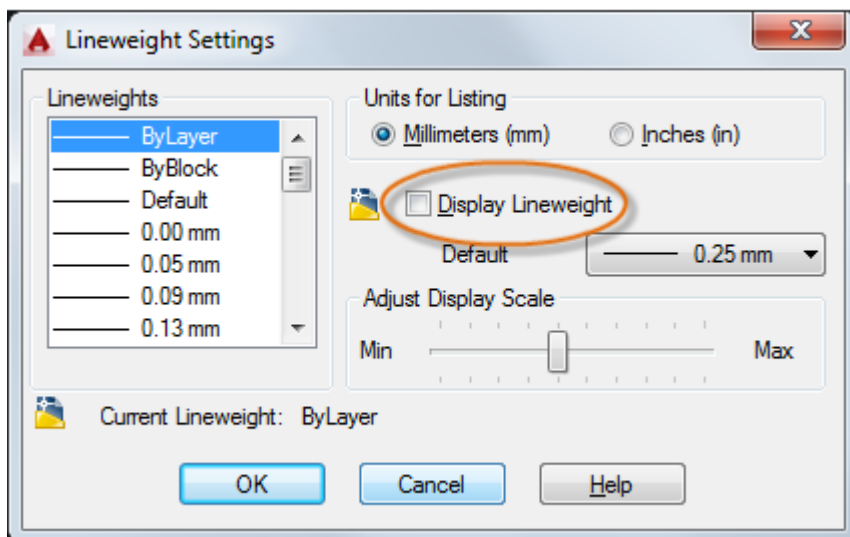
Lineweights can also be assigned from the Properties panel.



You can leave the lineweight set to ByLayer, or you can specify a value that overrides the layer's lineweight. In some cases, the lineweight previews look the same because they are displayed in approximated pixel widths on a monitor. However, they will print at the correct thickness.

Tip: It's usually best to leave lineweights turned off while you work. Heavy lineweights can obscure nearby objects when you use object snaps. You might want to turn them for checking purposes just before you print.

To control the display of lineweights, click the Lineweight Settings button at the bottom of the lineweight list. In the Lineweight Settings dialog box, you can choose whether you want to display or hide lineweights.

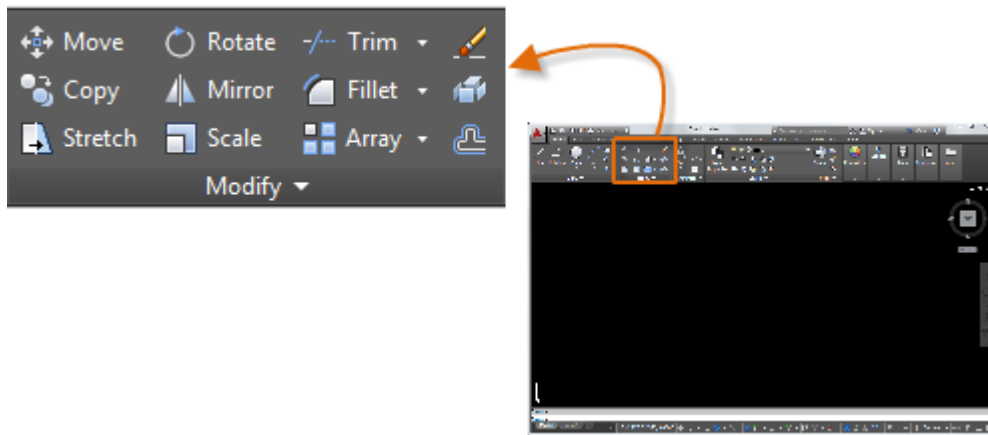


Regardless of the display setting, lineweights will always be printed at the correct scale.

Modifying

Perform editing operations such as erase, move, and trim on the objects in a drawing.

The most common of these tools are located on the Modify panel of the Home tab. Take a minute to look through them.



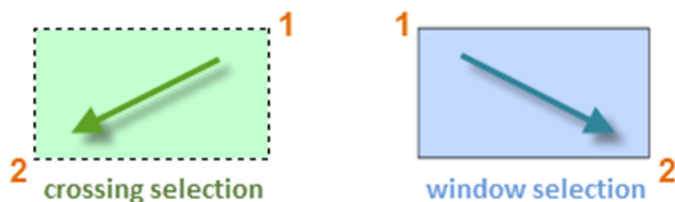
Erase

To erase an object, use the ERASE command. You can enter E in the Command window, or click the Erase tool. When you see the cursor change to a square *pickbox*, click each object that you want to erase, and then press Enter or the Spacebar.

Note: Alternatively, before you enter any command, you can select several objects and then press the Delete key. Experienced users often use this method as well.

Select Multiple Objects

Sometimes you need to select a large number of objects. Instead of selecting each object individually, you can select the objects in an area by clicking an empty location (1), moving your cursor right or left, and then clicking a second time (2).



- With a *crossing selection*, any objects within or touching the green area are selected.
- With a *window selection*, only the objects completely contained within the blue area are selected.

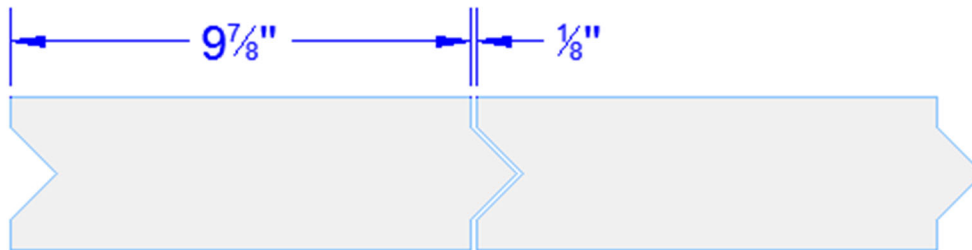
The result is called the **selection set**, the set of objects that will be processed by a command.

Tip: You can easily remove objects from the selection set. For example, if you select 42 objects, and two of them should not have been selected, hold down Shift and then select the two that you want to remove. Then, press Enter or the Spacebar, or right click to end the selection process.

Note: Clicking and dragging results in a different selection method called *lasso selection*.

Move and Copy

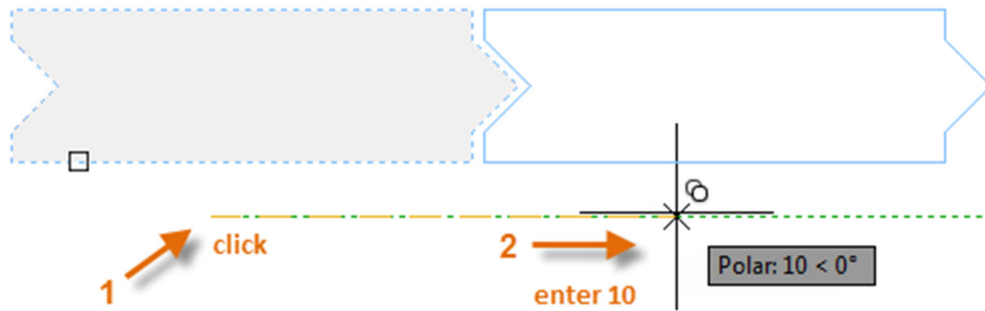
Here's how you would use the COPY command to lay out a row of decorative tiles. Starting with a polyline that represents its shape, you need to make copies that are 1/8" apart.



You click the Copy tool or enter CP in the Command window to start the command. From here, you can choose between two methods, depending on what's more convenient. You will use these two methods frequently.

The Distance Method

The second tile needs to be a total of $9\text{-}7/8" + 1/8" = 10"$ to the right of the original tile. So, you select the tile, press Enter or the Spacebar to end your selection, and click anywhere in the drawing area (1). This point does not have to be located on the tile.

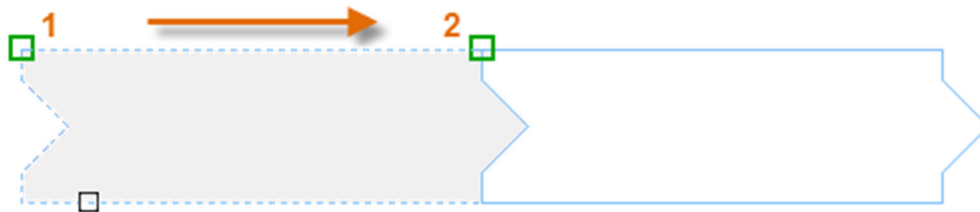


Next, you move your cursor to the right, relying on the polar tracking angle to keep the direction horizontal, and then enter 10 for the distance. Press Enter or the Spacebar a second time to end the command.

The specified distance and a direction from a point (1) is applied to the tile that you selected.

The Two Points Method

Another method, one that you will often use when you don't want to add numbers together, requires two steps. You start the COPY command and select the tile as before, but this time you click the two endpoints as shown. These two points also define a distance and direction.



Next, to add the 1/8" space between the tiles, click the Move tool or enter M in the Command window. The MOVE command is similar to the COPY command. Select the newly copied tile, and press Enter or the Spacebar. As before, click anywhere in the drawing area and move your cursor to the right. Enter 1/8 or .125 for the distance.

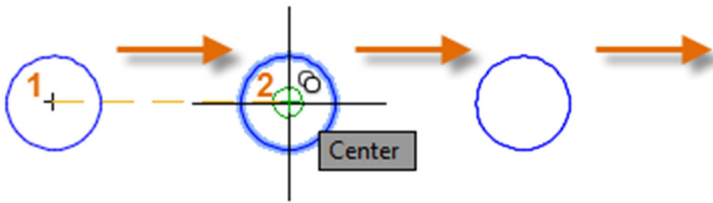
Tip: The two points that define the distance and direction don't need to be located on the object that you want to copy. You can use two points specified anywhere in your model.

Create Multiple Copies

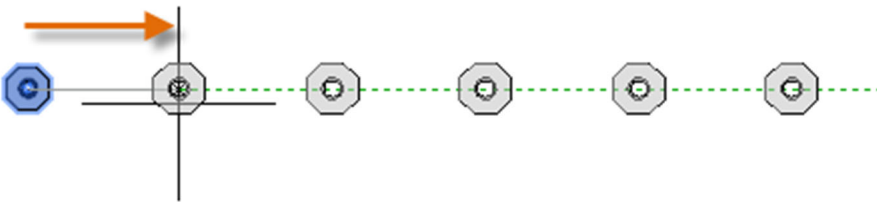
You can use the two-points method as a repeating sequence. Let's say that you want to make more copies of the circle at the same horizontal distance. You start the COPY command and select the circle as shown.



Then, using the Center object snap, click the center of circle 1, followed by the center of circle 2, and so on.

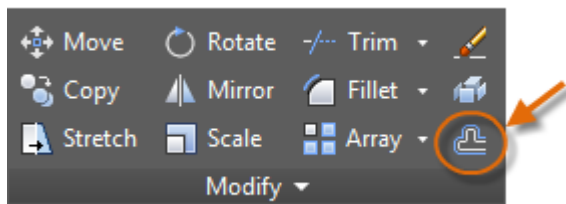


For larger numbers of copies, try experimenting with the Array option of the COPY command. For example, here's a linear arrangement of deep foundation piles. From a base point, you specify number of copies and the center-to-center distance.

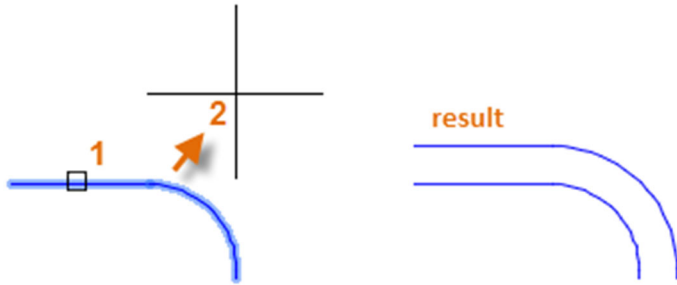


Offset

Most models include a lot of parallel lines and curves. Creating them is easy and efficient with the OFFSET command. Click the OFFSET tool or enter O in the Command window.



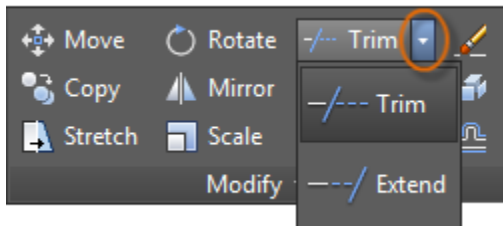
Select the object (1), specify the offset distance, and click to indicate on which side of the original that you want the result (2). Here is an example of offsetting a polyline.



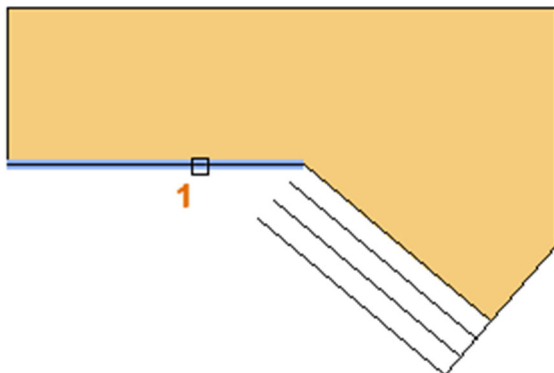
Tip: A fast way to create concentric circles is to offset them.

Trim and Extend

A popular technique is to use the OFFSET command in combination with the TRIM and EXTEND commands. In the Command window, you can enter TR for TRIM or EX for EXTEND. Trimming and extending are some of the most commonly used operations.



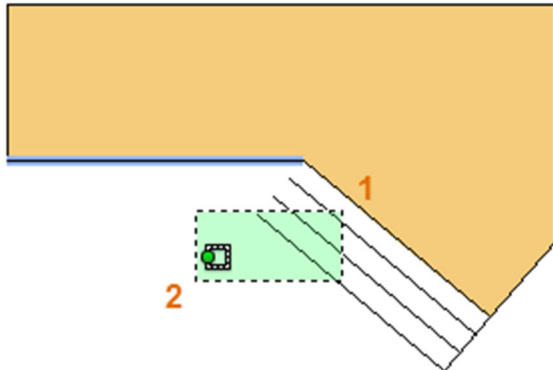
In the following illustration, you want to extend the lines that represent the steps for this deck. You start the Extend command, select the boundary, and then press Enter or the Spacebar.



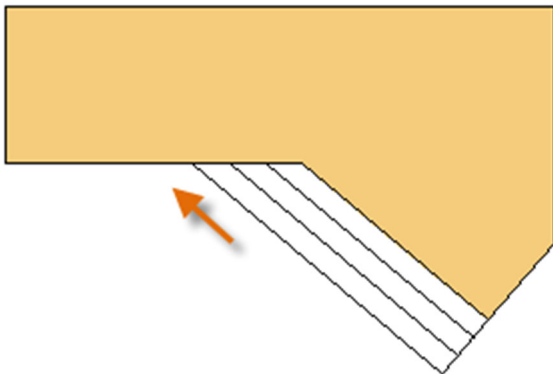
Pressing Enter or the Spacebar indicates that you've finished selecting the boundaries, and that you're now ready to select the objects to be extended.

Tip: A faster method is to press Enter or the Spacebar right away instead of selecting any boundary objects. The result is that all objects are available as possible boundaries.

Next, you select the objects to be extended (near the ends to be extended), and then you press Enter or the Spacebar to end the command.



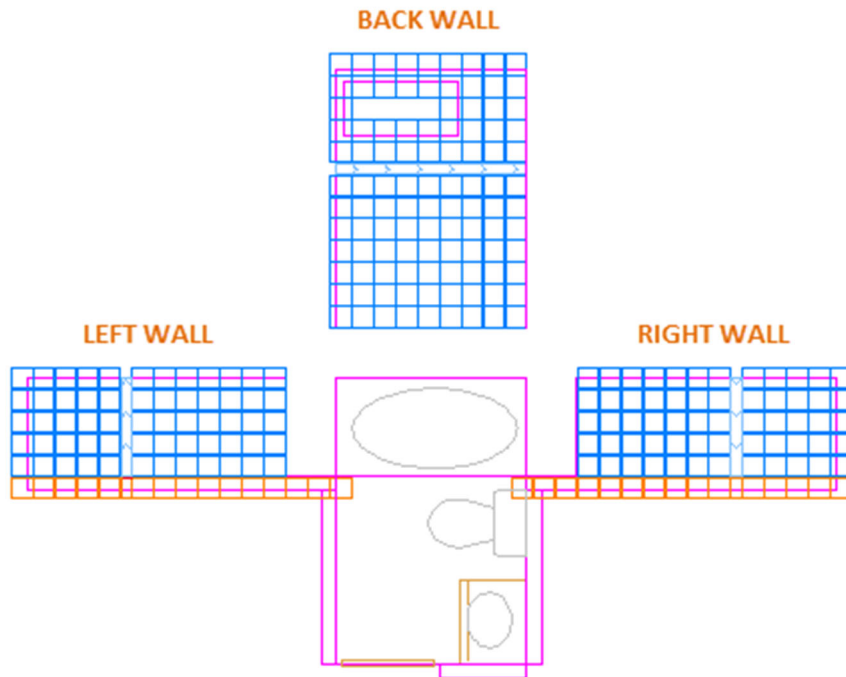
The result is that the lines are extended to the boundary.



The TRIM command follows the same steps, except that when you select the objects to trim, you select the portions to trim away.

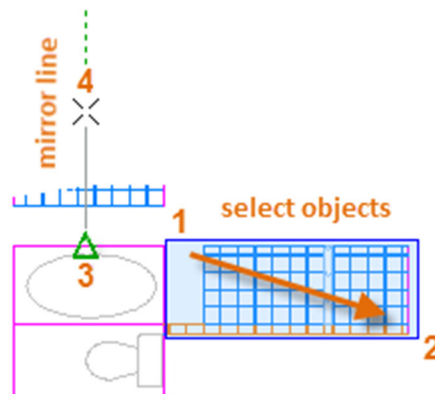
Mirror

The following illustration comes from a tile project. The walls in this residential bathroom are flattened out to be able to lay out the tile pattern and estimate the number of tiles needed.

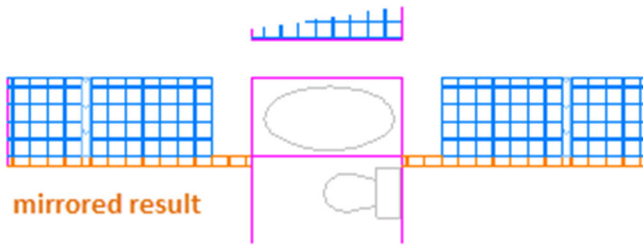


You can save a lot of work by taking advantage of the symmetry between the left and right walls. All you need to do is create the tiles on one wall and then mirror the wall across the center of the room.

In the example below, you start the MIRROR command (or enter MI in the Command window), use window selection (1 and 2) to select the geometry on the right wall, press Enter or the Spacebar, and then specify a mirror line (3 and 4) corresponding to the centerline of the bathroom.



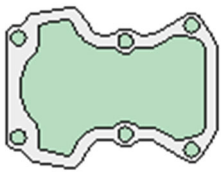
Finally, decline the option to "Erase source objects" by pressing Enter or the Spacebar.



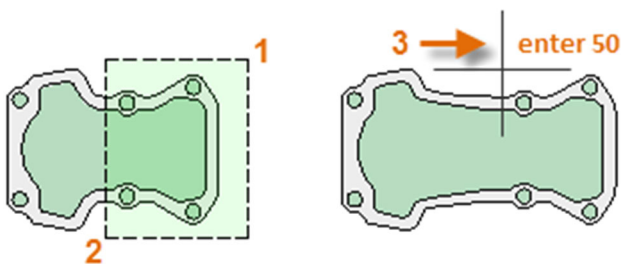
Tip: Always look for symmetry to save yourself extra work, even if the symmetry is not 100% identical.

Stretch

You can stretch most geometric objects. This lets you lengthen and shorten parts of your model. For example, this model might be a gasket or the design for a public park.



Use the STRETCH command (or enter S in the Command window) and select the objects with a crossing selection as shown below (1 and 2). The crossing selection is mandatory—only the geometry that is crossed by the crossing selection is stretched. Then click anywhere in the drawing area (3), move the cursor to the right, and enter 50 as the distance. This distance might represent millimeters or feet.



To shorten the model by a specified amount, you'd move your cursor to the left instead.

Fillet

The FILLET command (enter F in the Command window) creates a rounded corner by creating an arc that is tangent to two selected objects. Notice that the fillet is created relative to where you select the objects.



You can create a fillet between most types of geometric objects, including lines, arcs, and polyline segments.

Tip: If you specify 0 (zero) as the radius of the fillet (imagine a circle shrinking to a radius of 0), the result trims or extends the selected objects to a sharp corner.

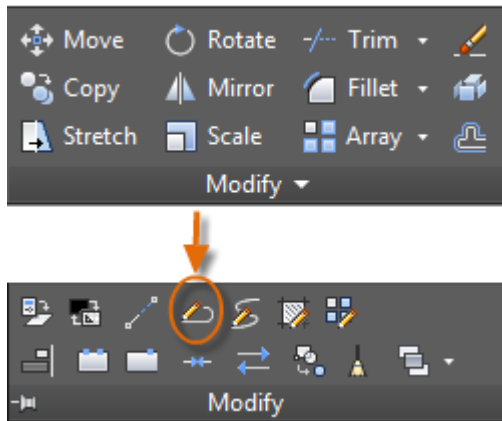
Explode

The EXPLODE command (enter X in the Command window) disassociates a compound object into its component parts. You can explode objects such as polylines, hatches, and blocks (symbols).

After you explode a compound object, you can modify each resulting individual object.

Edit Polylines

You can choose from several useful options when you want to modify a polyline. The PEDIT command (enter PE in the Command window) is located on the drop-down list of the Modify panel.



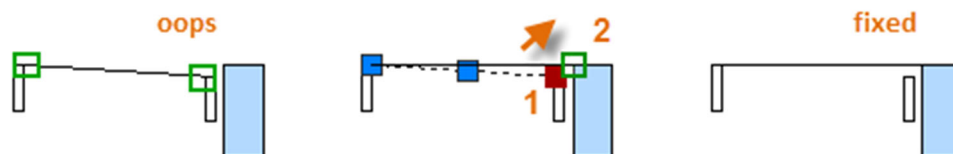
With this command, you can

- Join two polylines into a single polyline if they share a common endpoint
- Convert lines and arcs into a polyline—simply enter PEDIT and select the line or arc
- Change the width of a polyline

Tip: In some cases, the easiest method to modify a polyline is to explode it, make the modifications, and then turn the objects back into a polyline using the Join option of the PEDIT command.

Grips

Grips are displayed when you select an object without starting a command. Grips are often handy for light editing. For example, the line below accidentally snapped to the wrong endpoint. You can select the misaligned line, click on a grip and then click to specify the correct location.



By default, when you click a grip, you automatically start in ****STRETCH**** mode as indicated in the Command window. If you want to explore other ways of editing objects with grips, press Enter or the Spacebar to cycle through several other editing modes. Some people perform most editing operations using grips.

Parent topic: [The Hitchhiker's Guide to AutoCAD Basics](#)

Previous topic: [Properties](#)

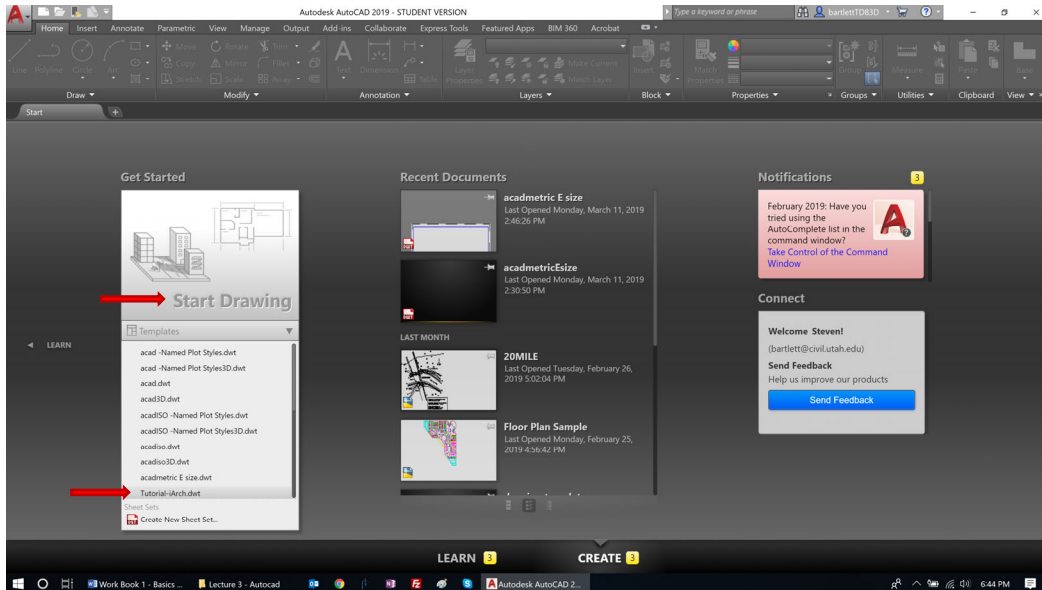
Next topic: [Blocks](#)

Related Reference

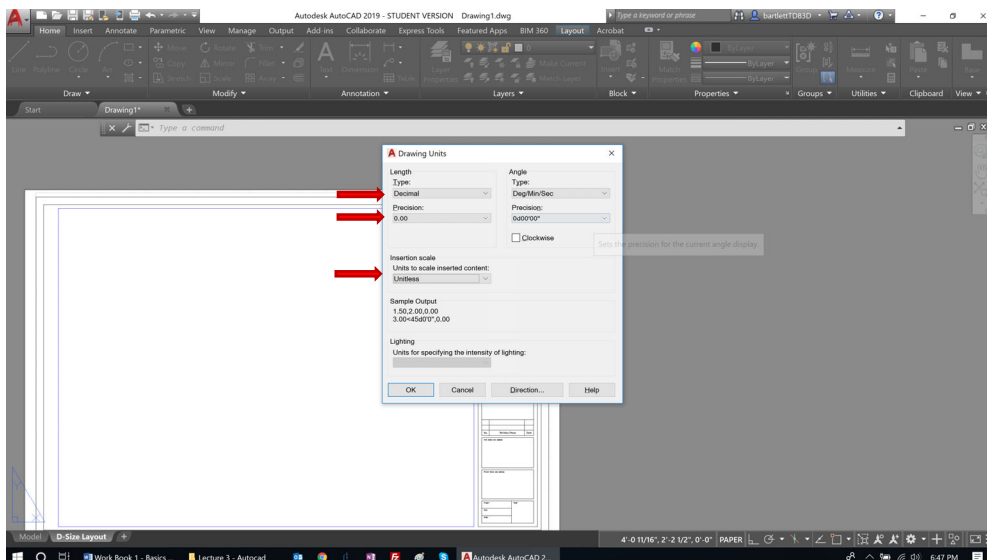
- [ALIGN \(Command\)](#)
- [COPY \(Command\)](#)
- [ERASE \(Command\)](#)
- [EXTEND \(Command\)](#)
- [EXPLODE \(Command\)](#)
- [FILLET \(Command\)](#)
- [MIRROR \(Command\)](#)
- [MOVE \(Command\)](#)
- [OFFSET \(Command\)](#)
- [PEDIT \(Command\)](#)
- [ROTATE \(Command\)](#)
- [STRETCH \(Command\)](#)
- [TRIM \(Command\)](#)

Preparing Drawing Space for Coordinates

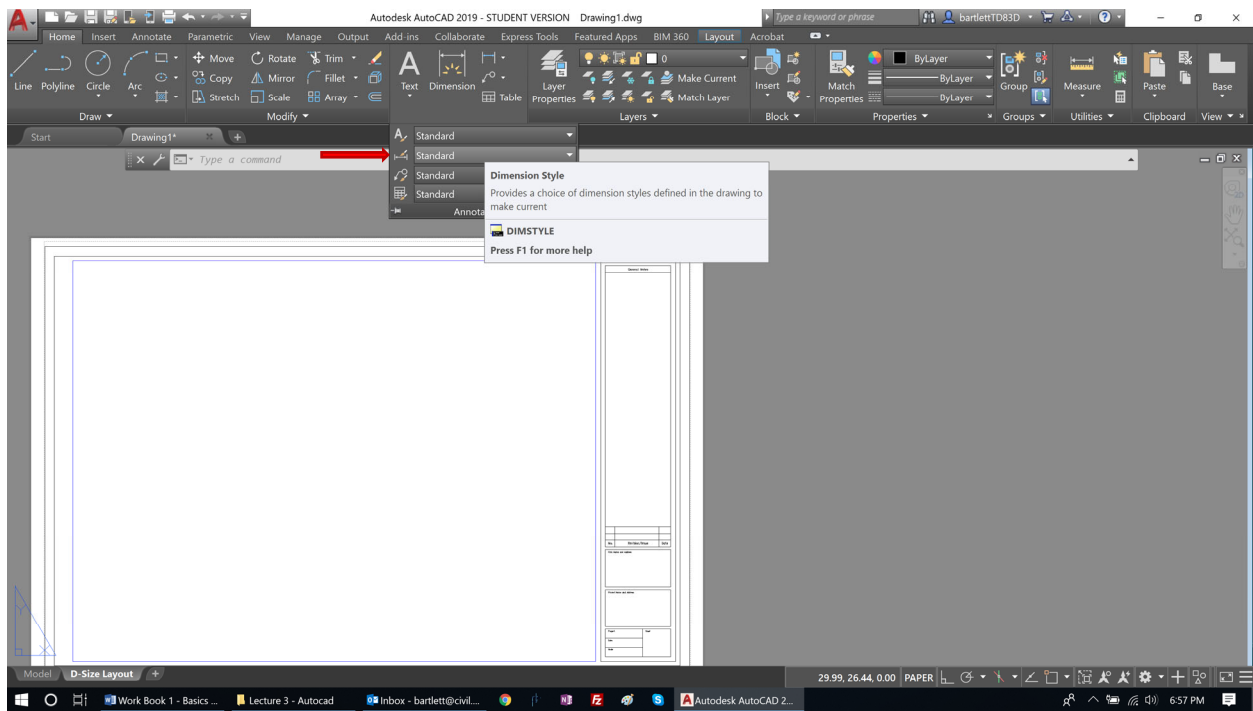
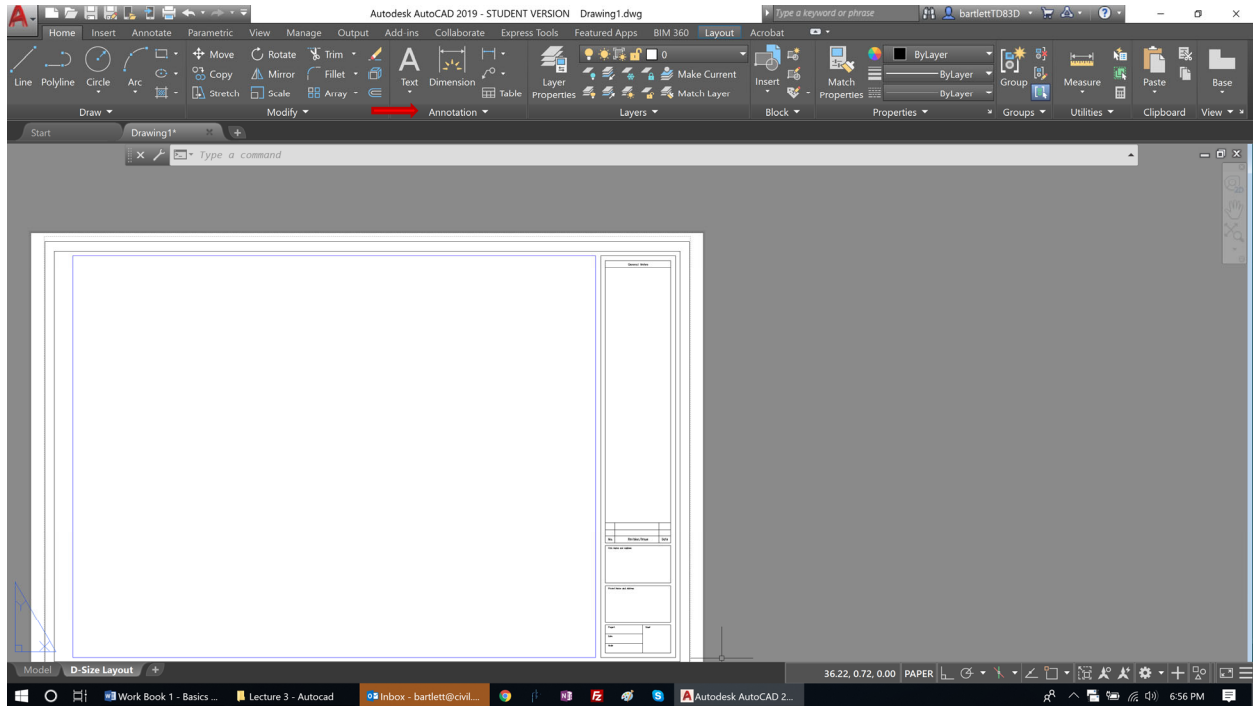
In this example, we will prepare the drawing space for a floor plan that has been measured in meters. A coordinate system will be established to aid in the drawing. Start this lesson by starting a new drawing.

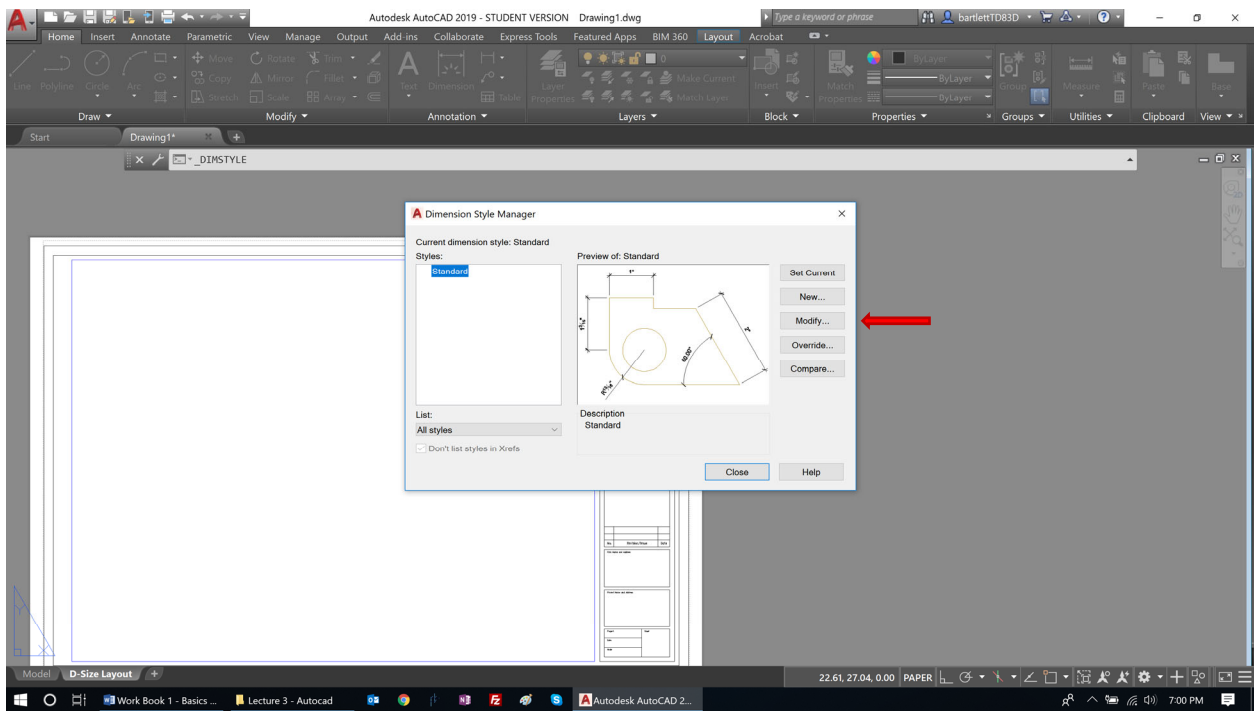
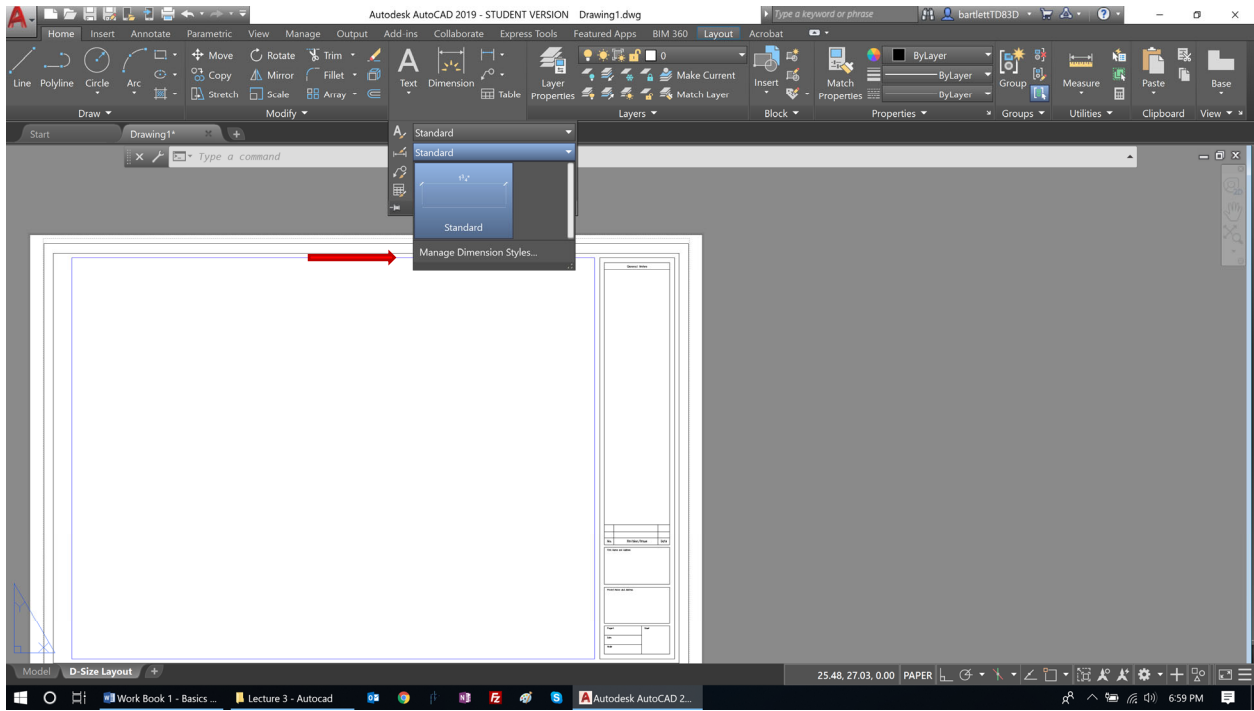


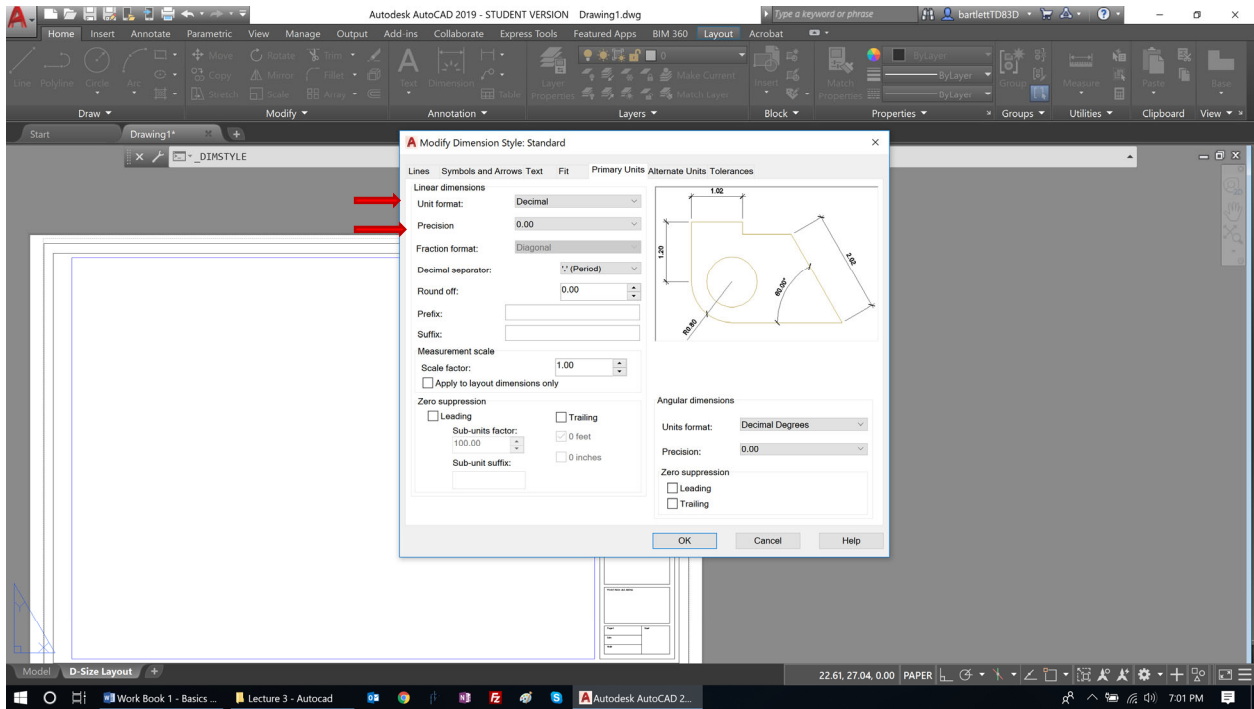
In the command line, type units and the below screen will appear. Change the **Type** to **Decimal** and **Precision** to **0.00** and the **Insertion Scale** to **Unitless**.



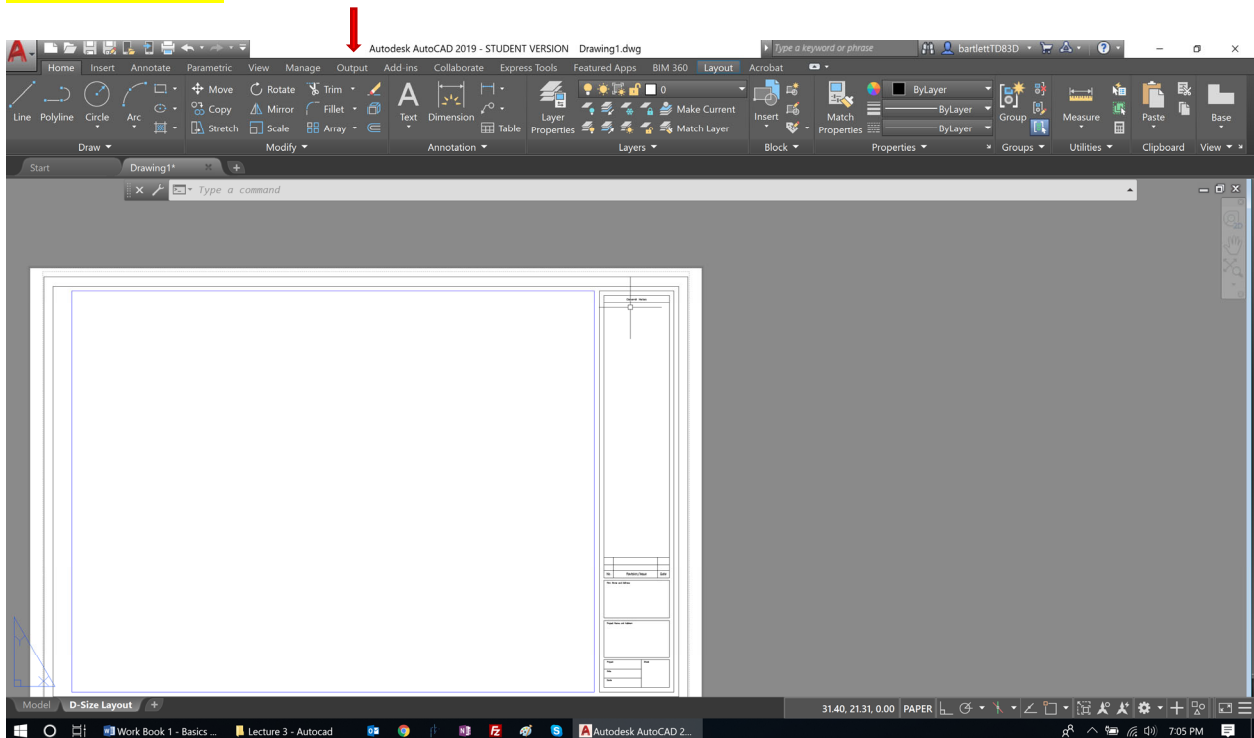
We are need to change the **Annotation** to match the units we have selected.

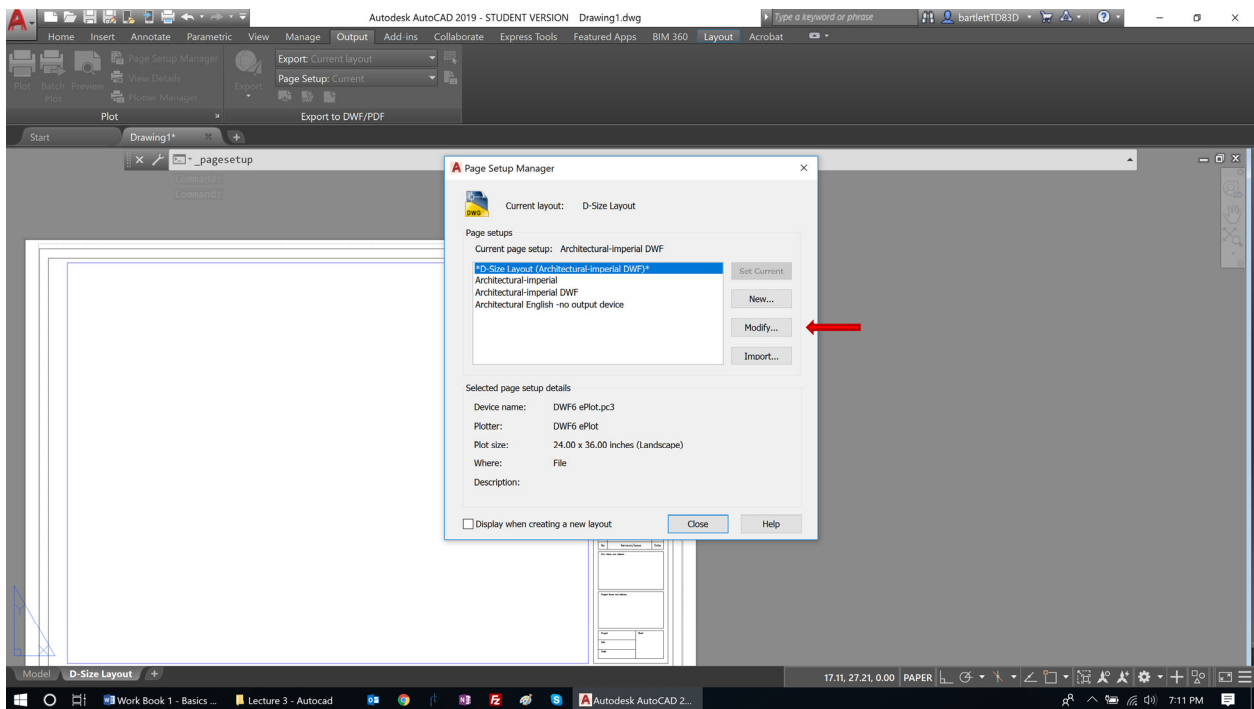
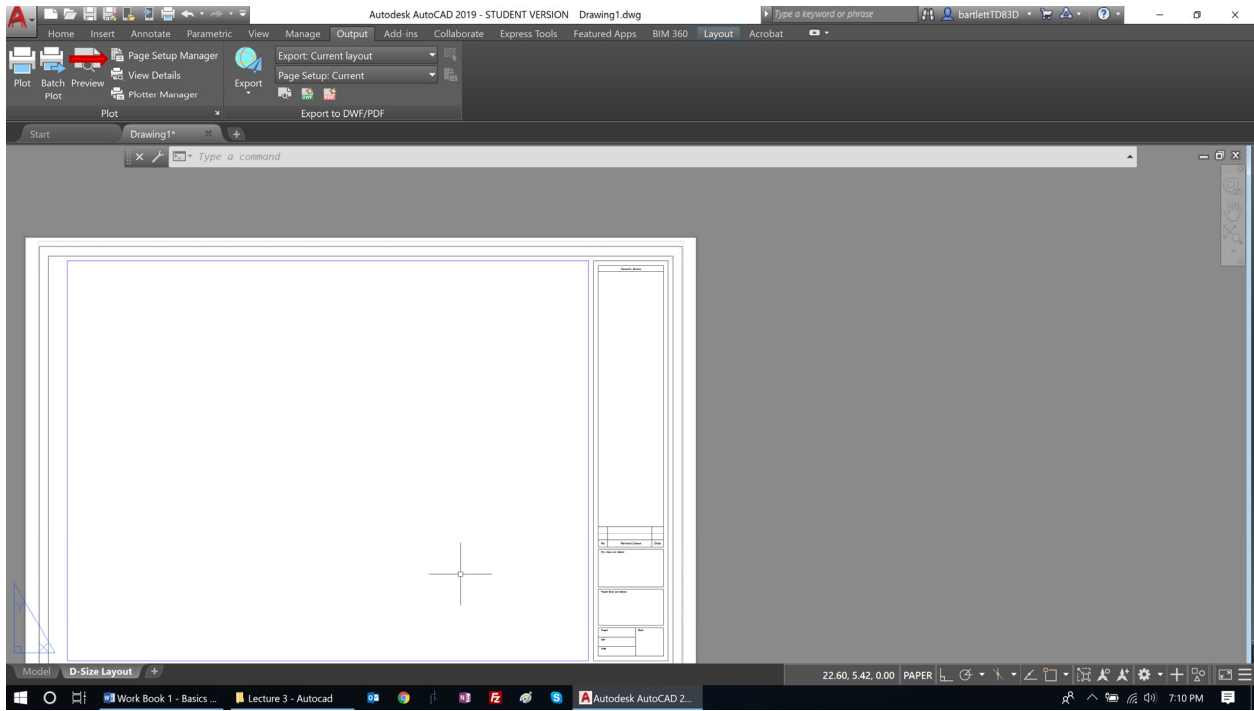


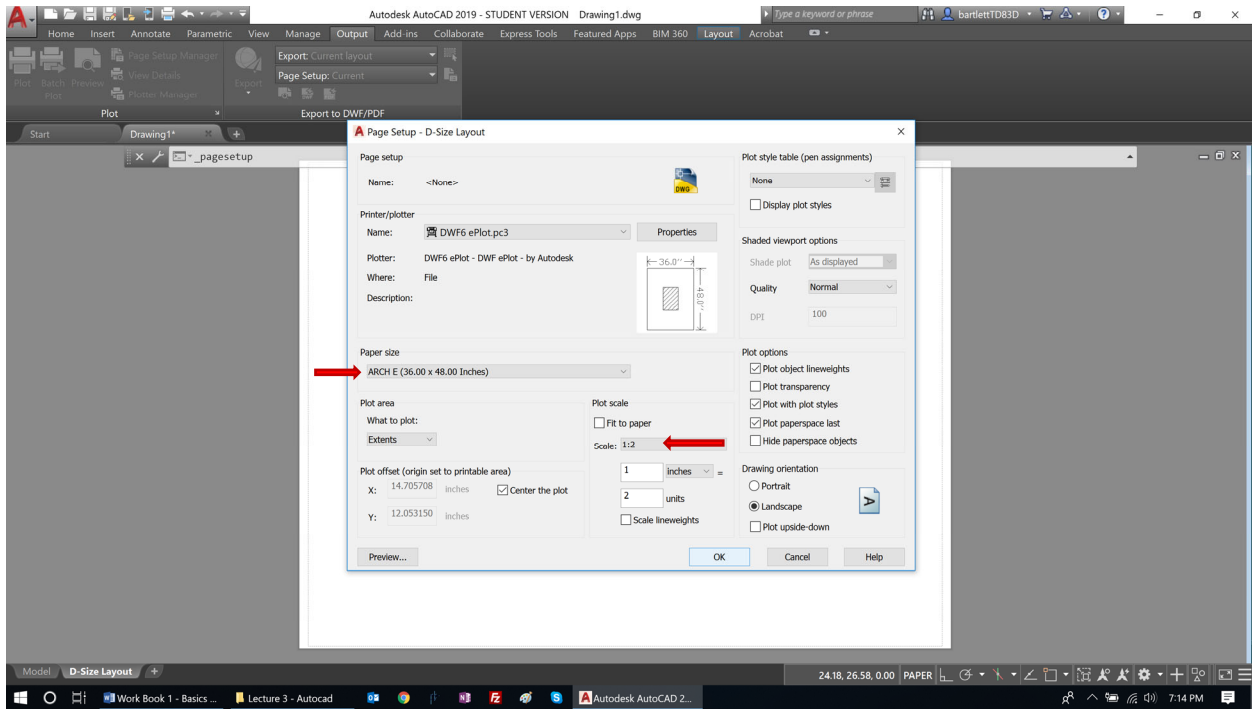




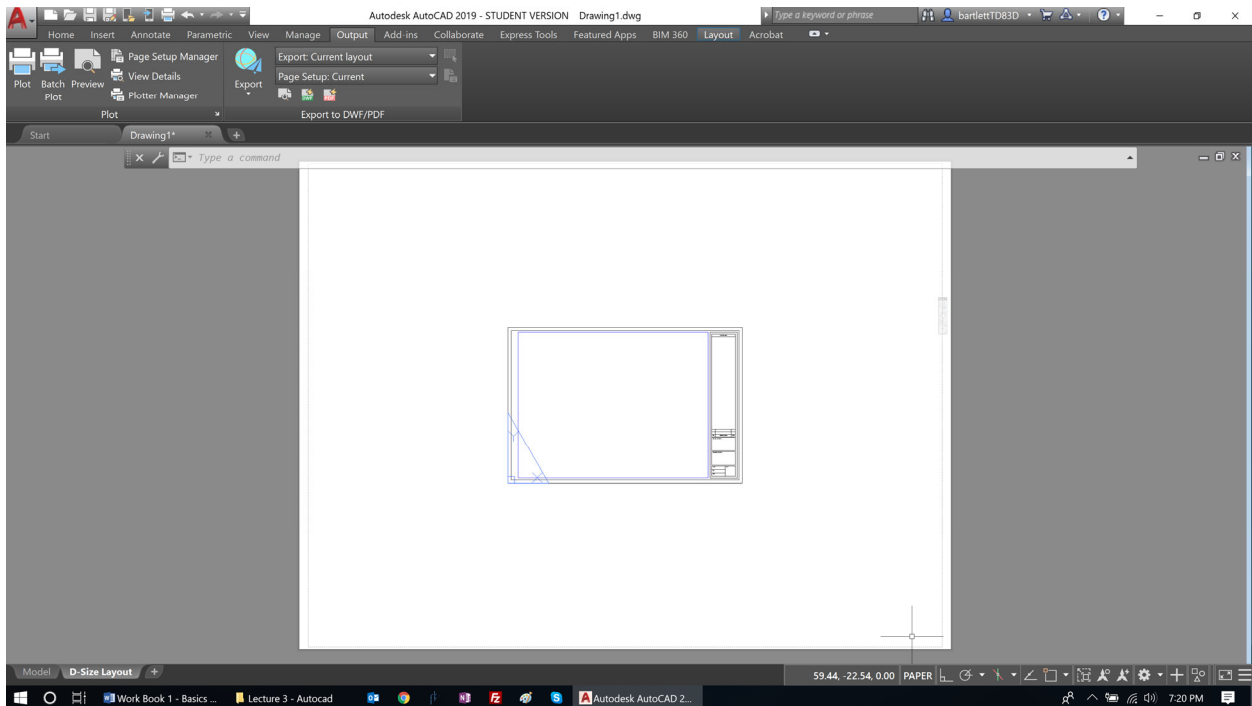
Increase the drawing size to a “E” size drawing using the **Output Command** and change the scale to 1:2



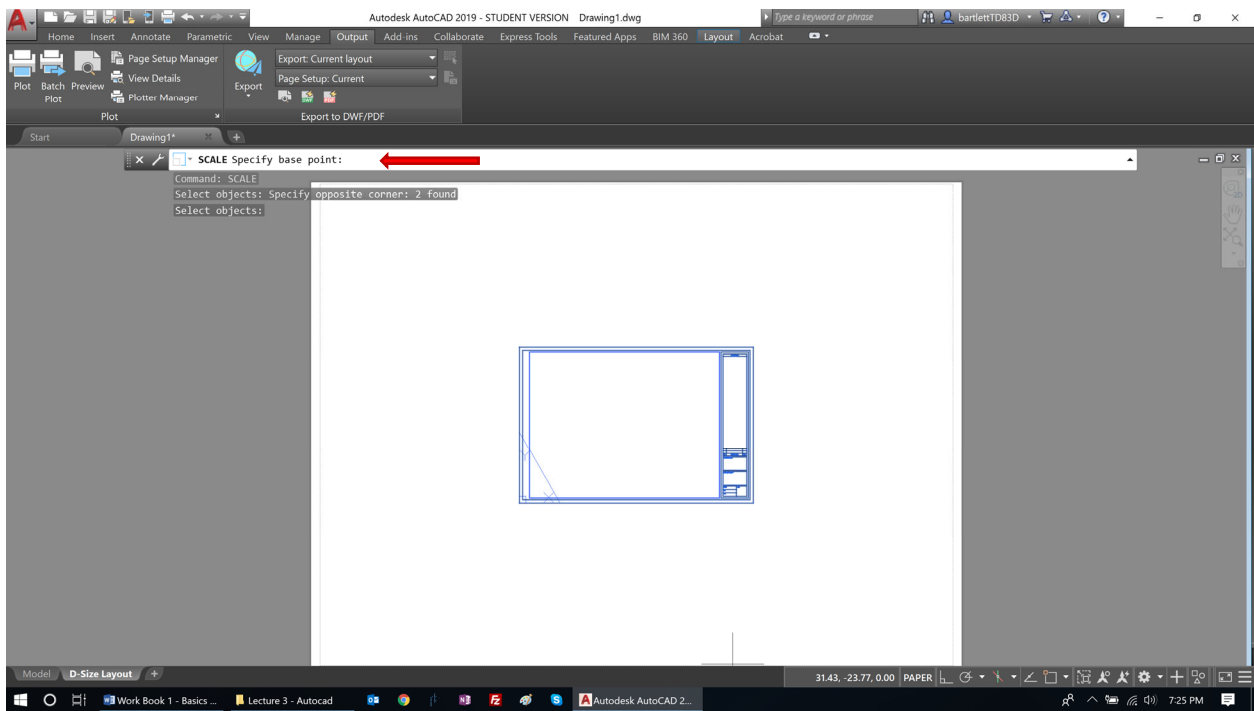
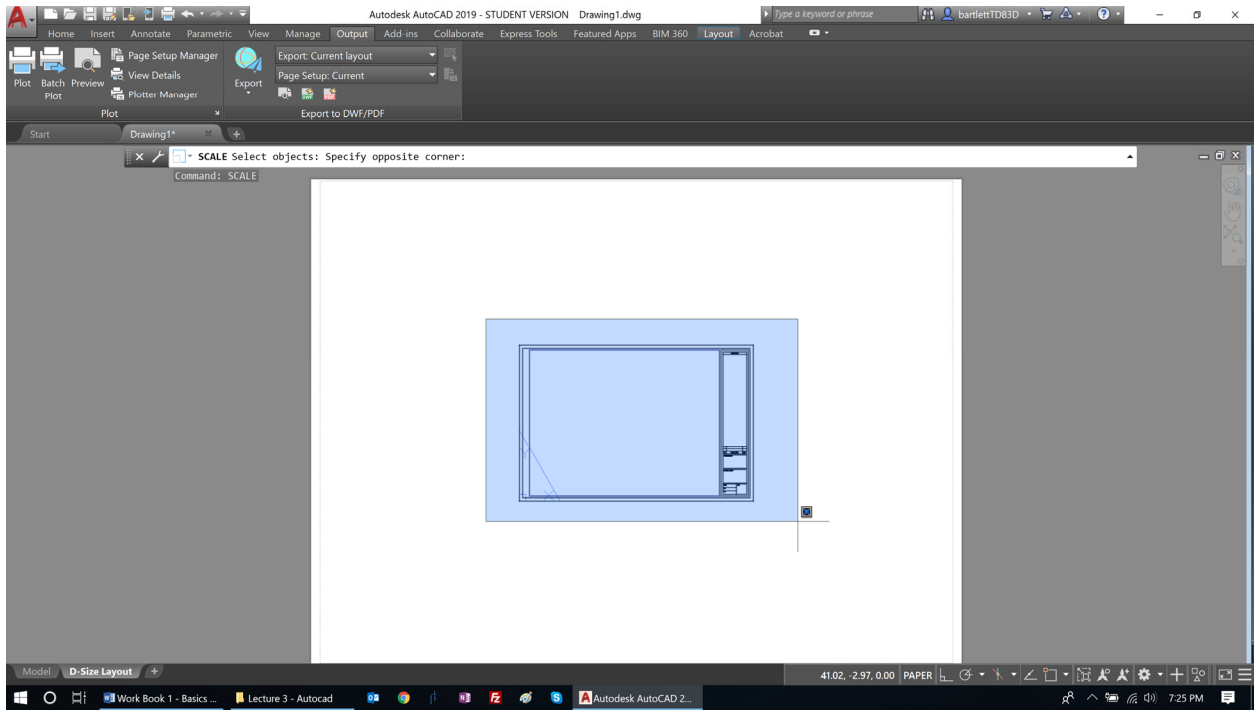


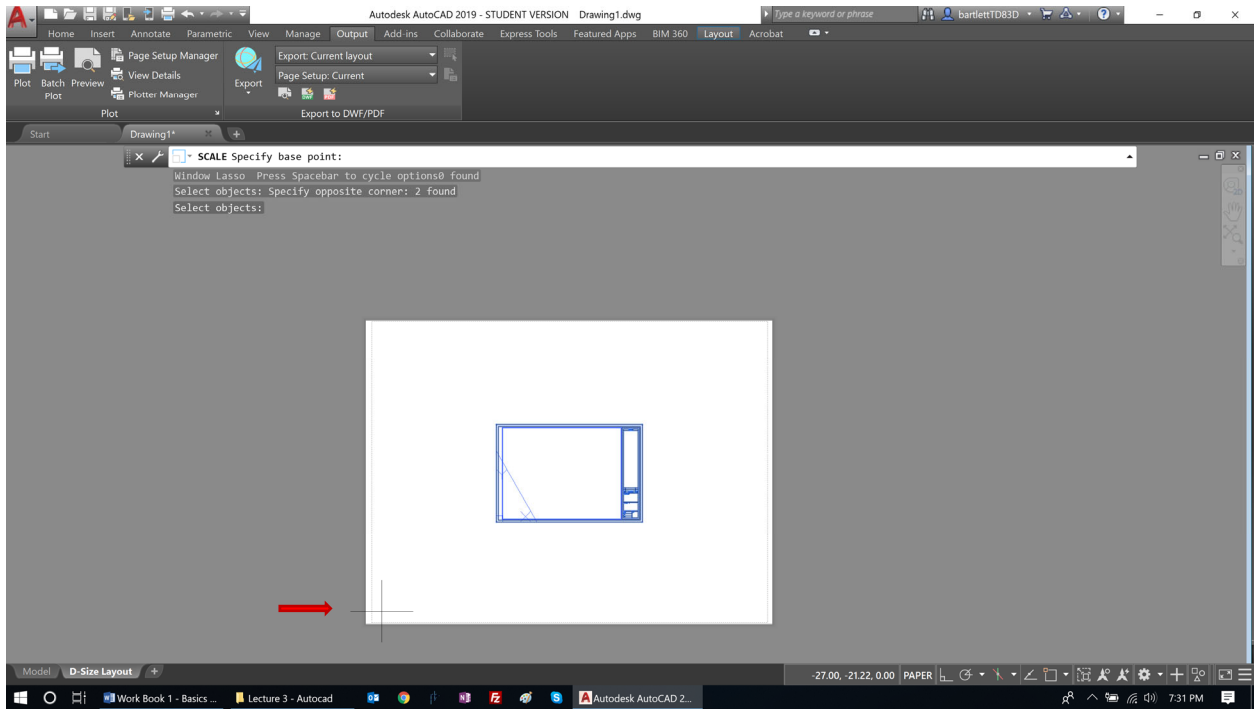


The **Scale 1:2** allows for large drawing to be placed on the “E” size drawing.

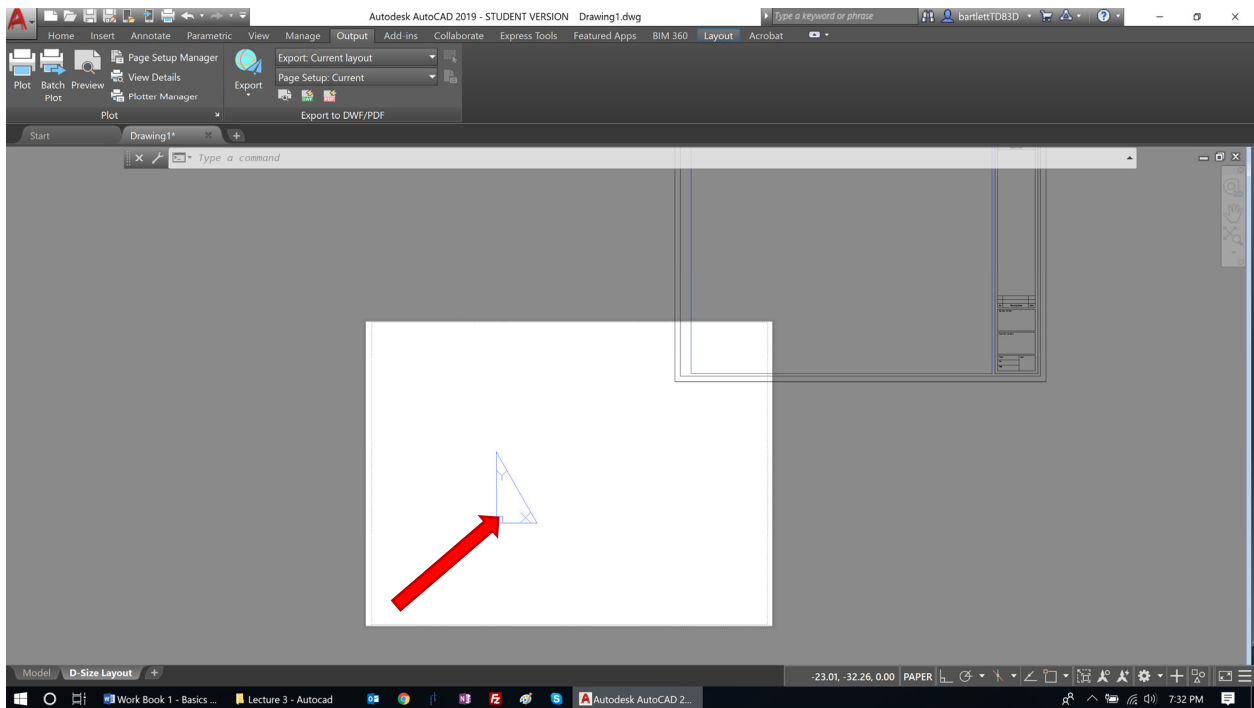


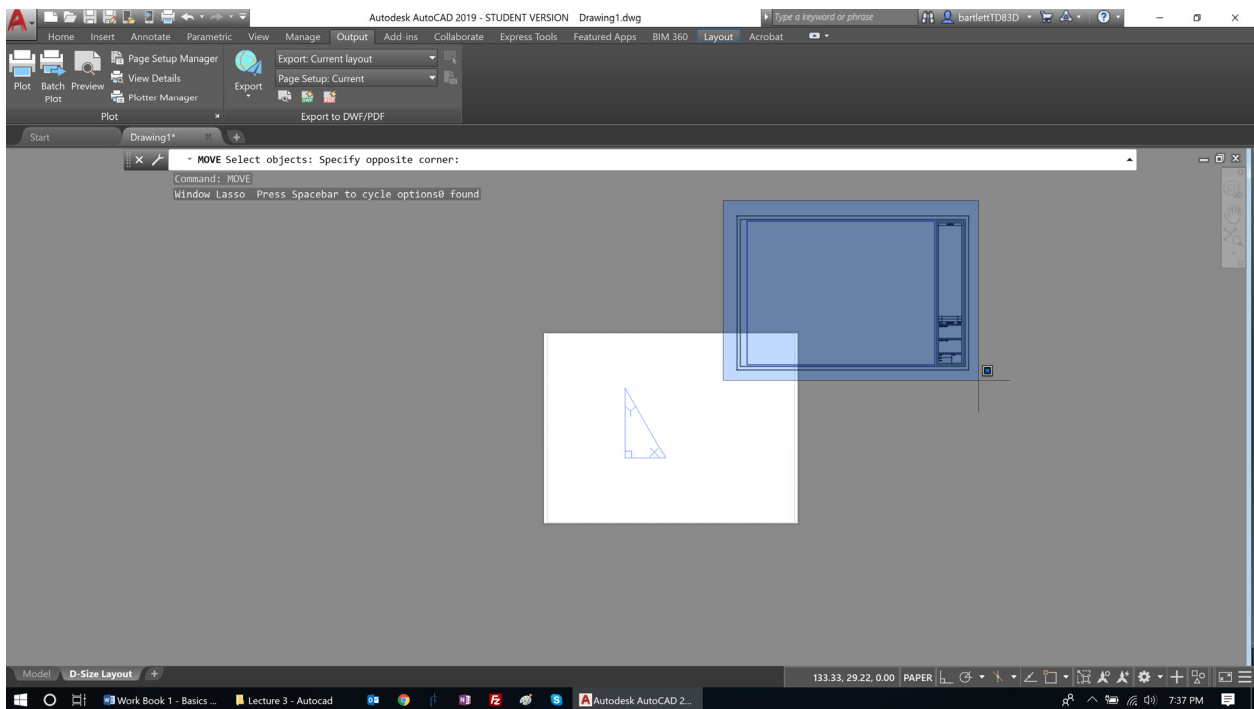
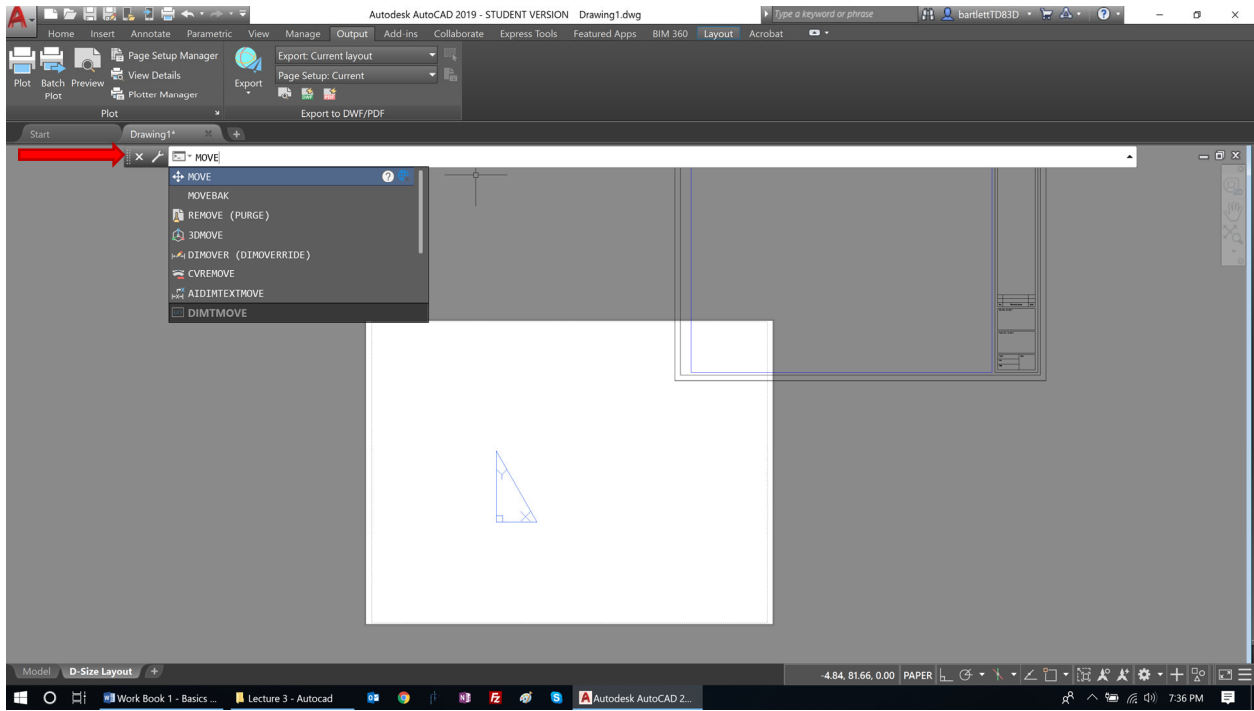
Note that the title block is too small and needs to be enlarged. Do this by typing **Scale** in the **command box** and select the title block.



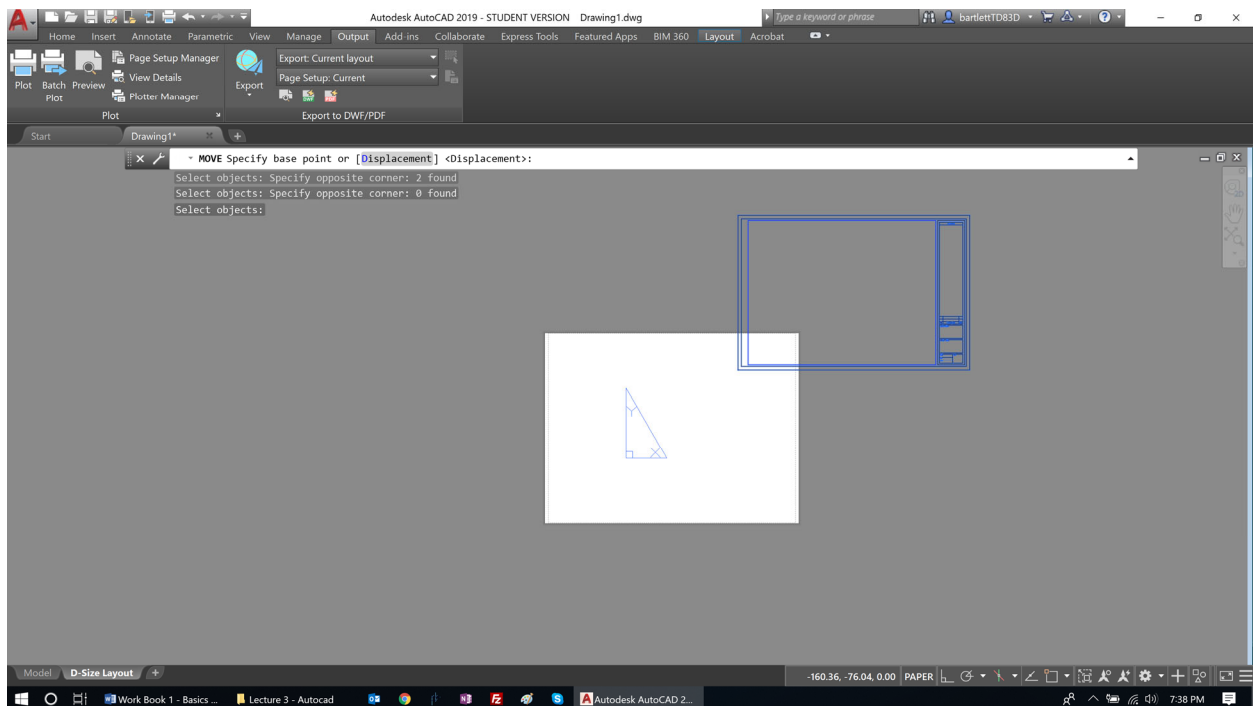
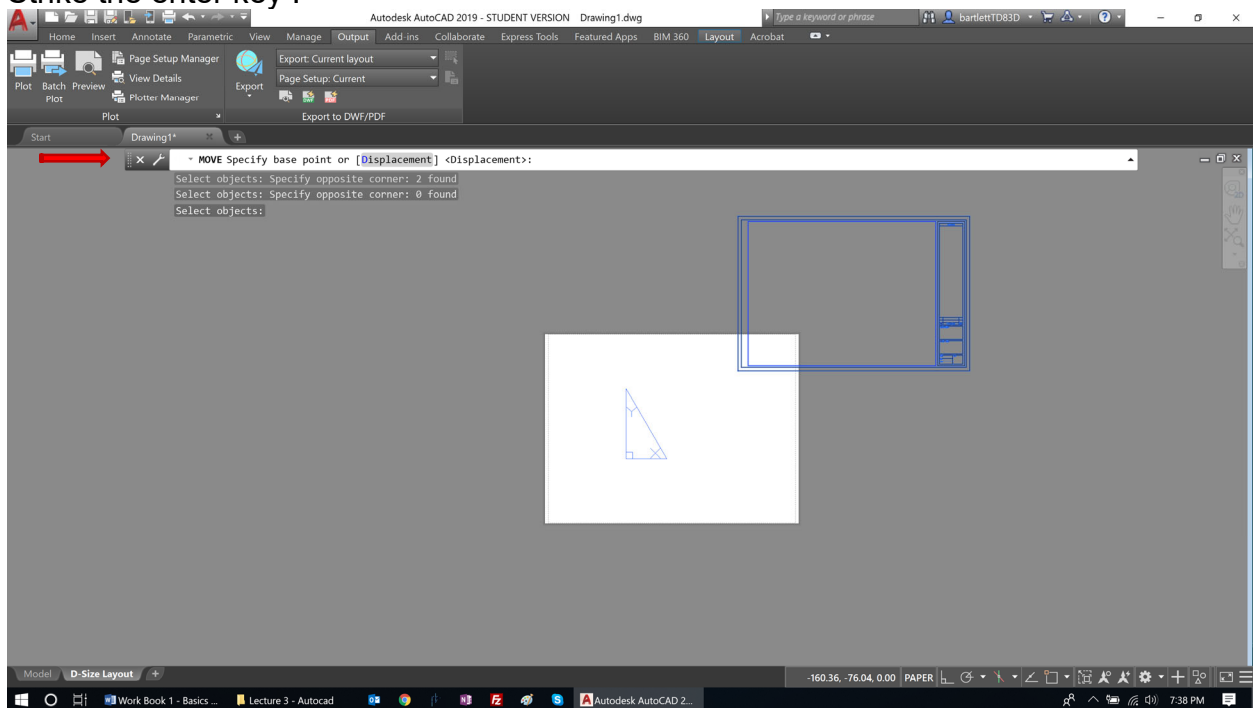


Select the base point in the lower left-hand corner of the drawing. Push the cross hairs to the right and the drawing box will increase in size.

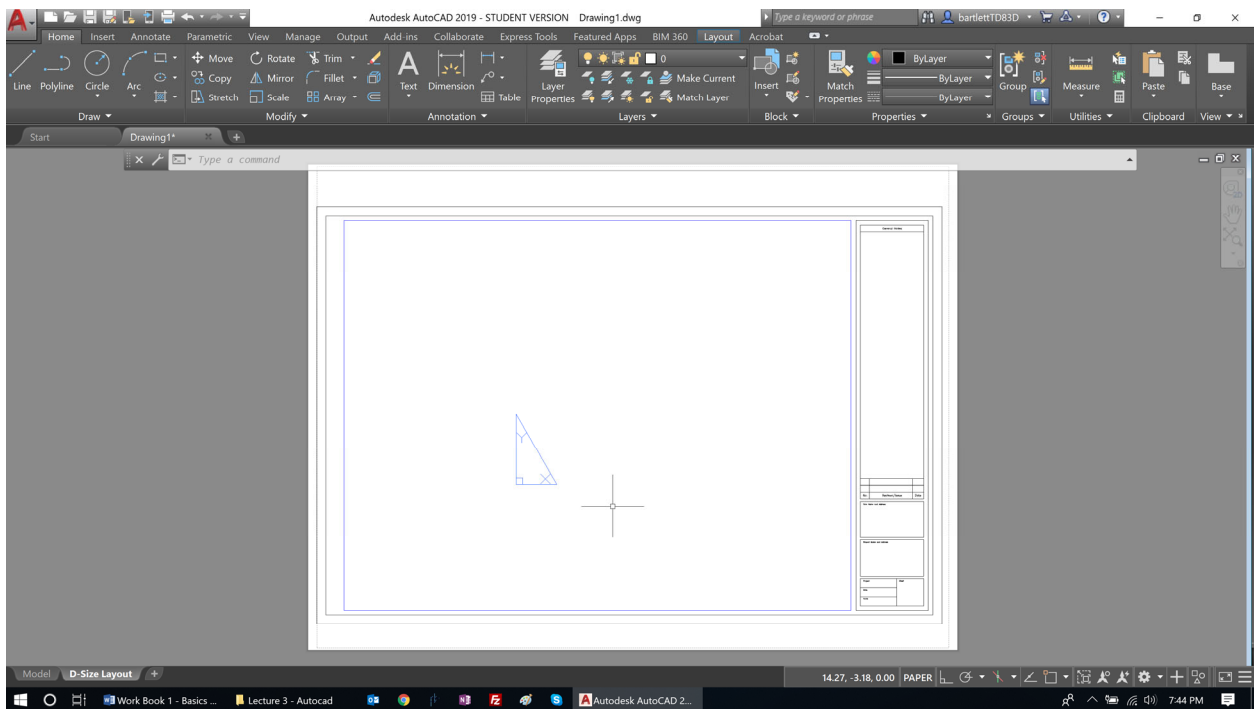
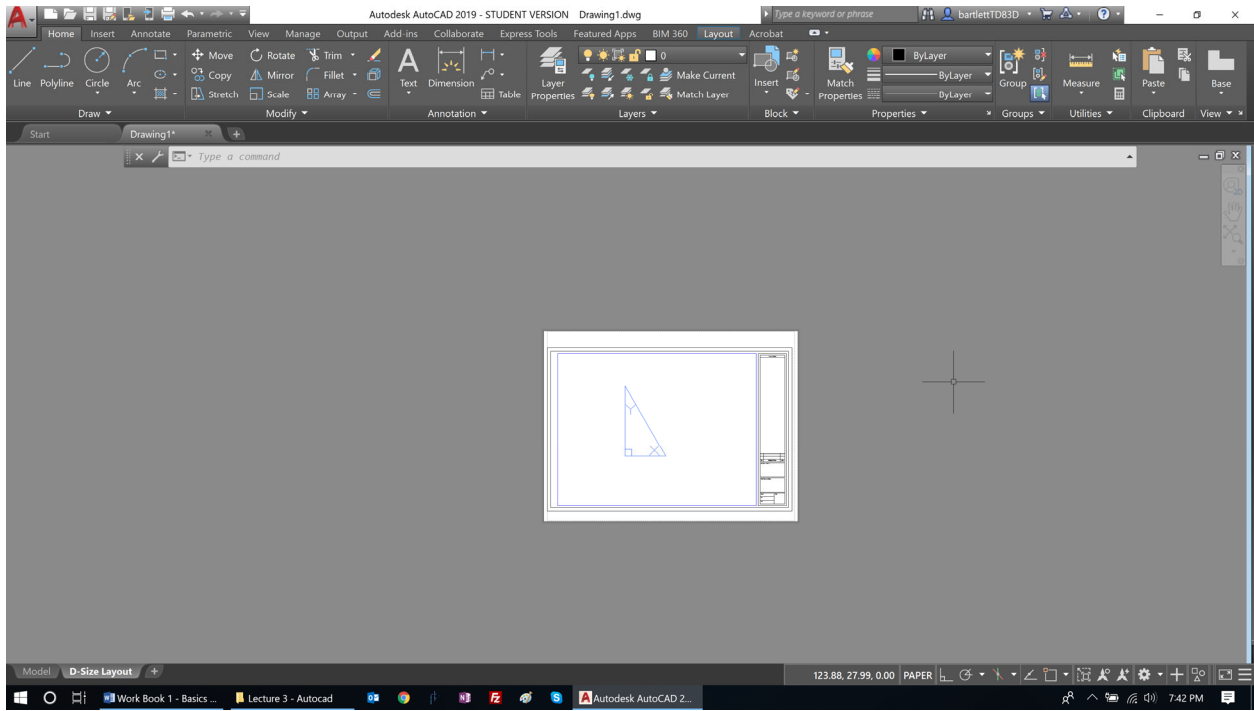




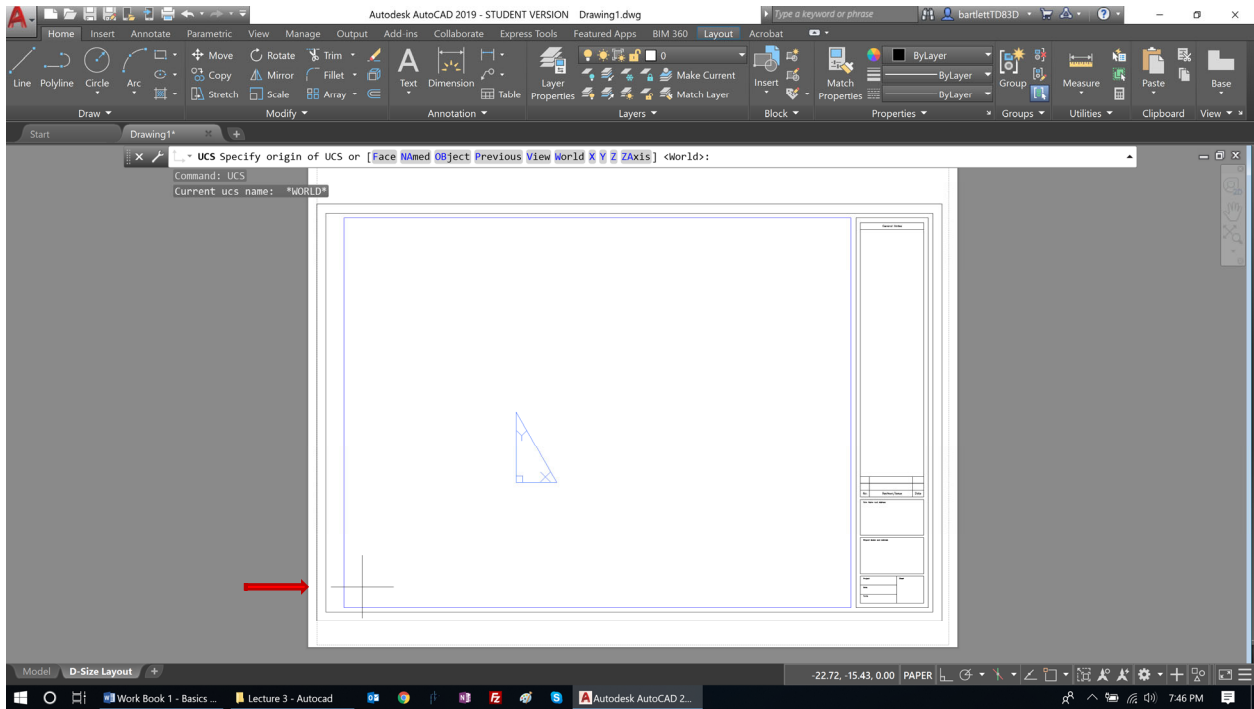
Strike the enter key .



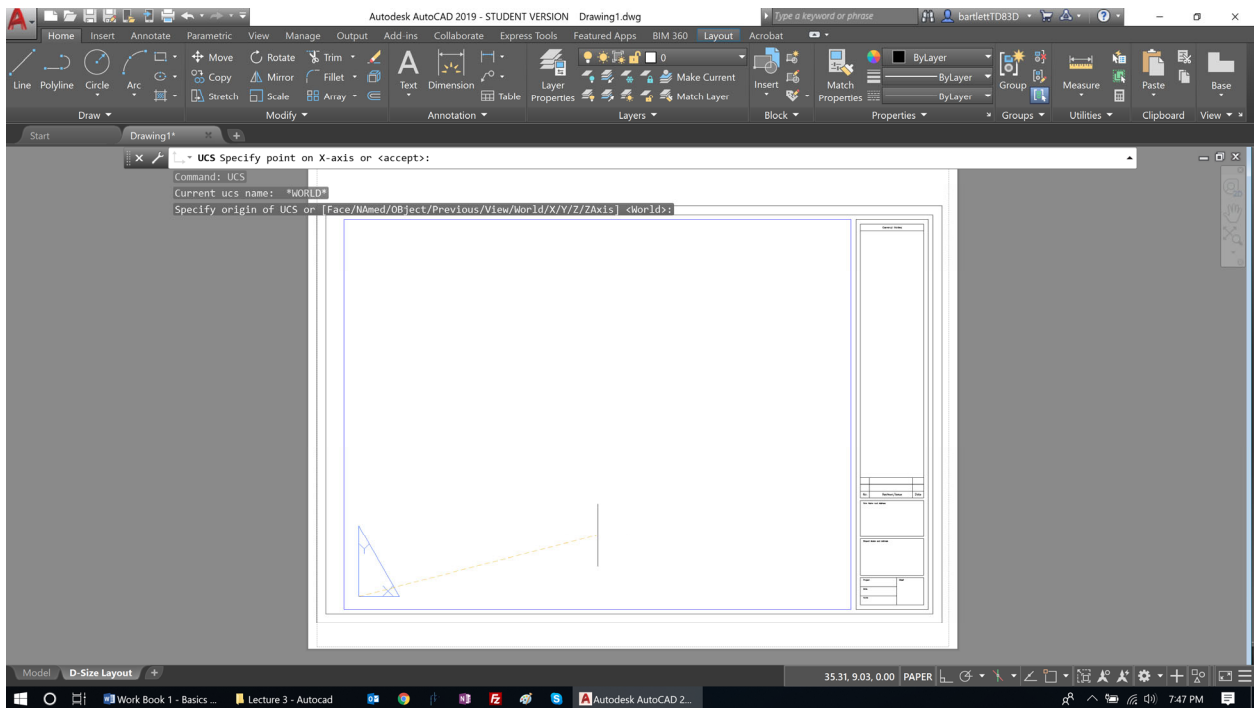
Select the second point at the lower corner of the drawing and drag the title block back onto the paper.

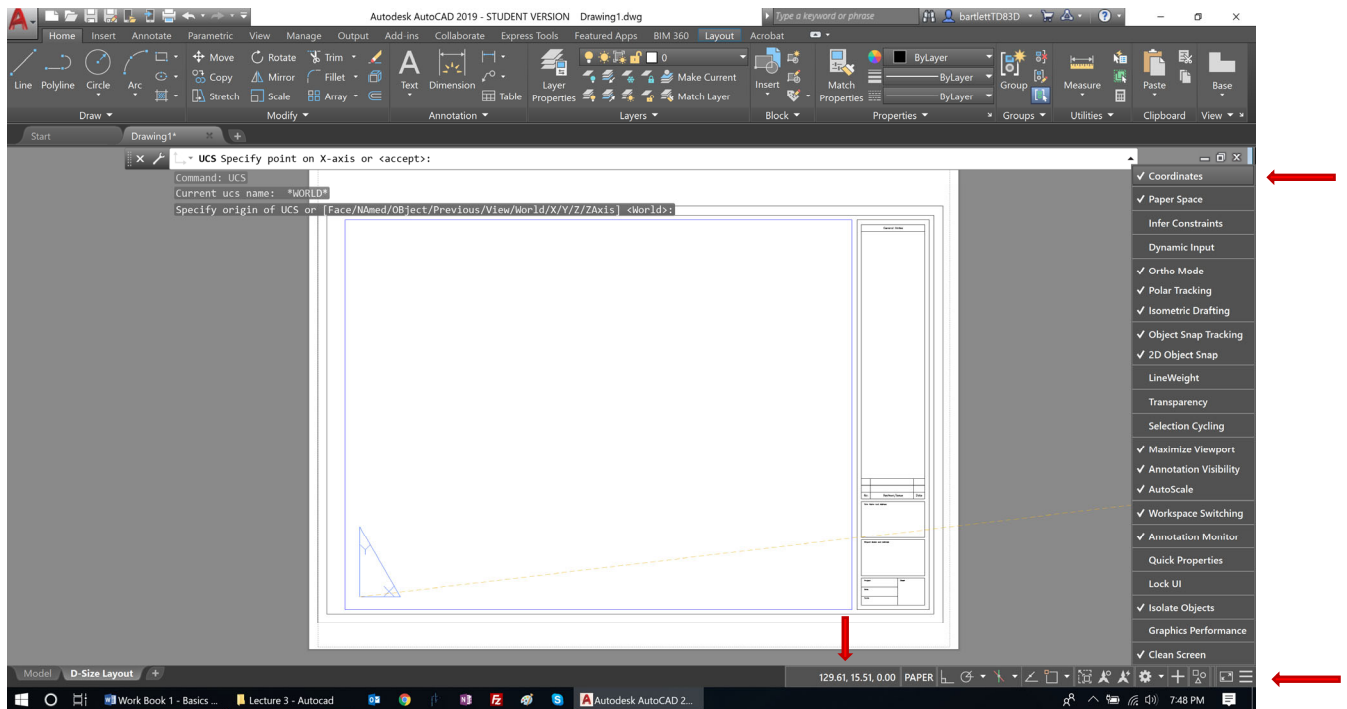


Move the UCS back to the lower left corner of the drawing by typing **UCS** in the **command** line.



Click on the lower left corner of the drawing to move the coordinate system to that location and strike the enter key.





Turn on the coordinate system by selecting the pop of menu at the bottom of the screen, then select coordinates in the pop up menu. The coordinates will then be displayed.