



Politechnika Wrocławska

FUNDAMENTALS OF ENGINEERING DRAWING

AutoCAD

Lab1

Dr inż. Andrzej Iwańczuk



Basic information about the course

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- Students have an access to a **free** version of AutoCAD, which must be downloaded and installed on home computers from:

<https://www.autodesk.com/support/account/education/onboarding/students-guide>

- Students make drawings during classes and possibly finish them at home
- Drawings should be sent within a week to the following address: andrzej.iwanczuk@pwr.edu.pl
- **The Test** will be held during the last AutoCAD class (10.06): simple drawing in 1.5h
- Students can use the tutorial or the internet at any time

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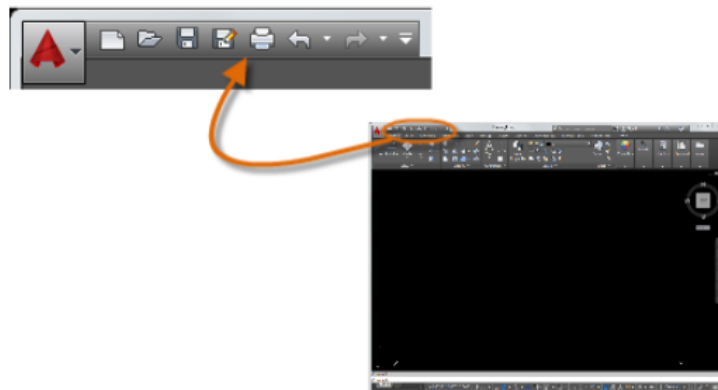
Basics

Review the basic AutoCAD controls.

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.





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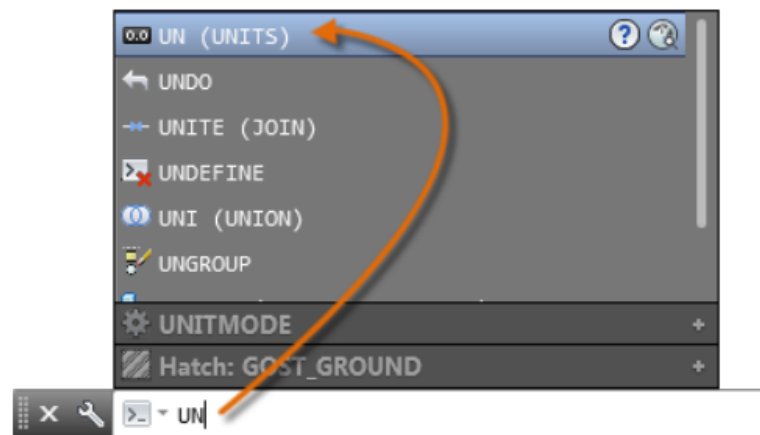
The Command Window

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.



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, an autocomplete menu appears. When several options are available, such as in the example below, make your choice by clicking the correct option or using the arrow keys and then pressing Enter or the Spacebar to confirm your selection.



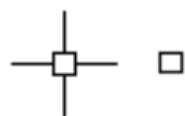


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The Mouse

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



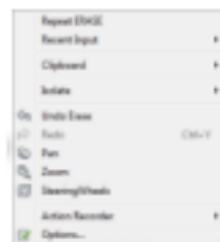
select objects

specify locations



pan and zoom

shortcut menus



Here's a Tip:

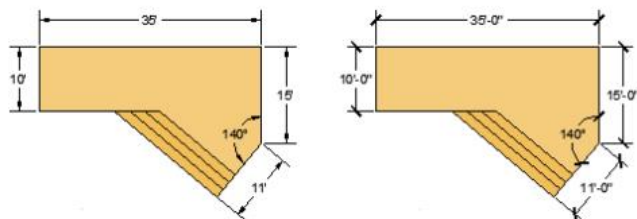
When looking for a command or option, try right-clicking. Depending on where your cursor is located, different menus will display relevant commands and options.

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New Drawings

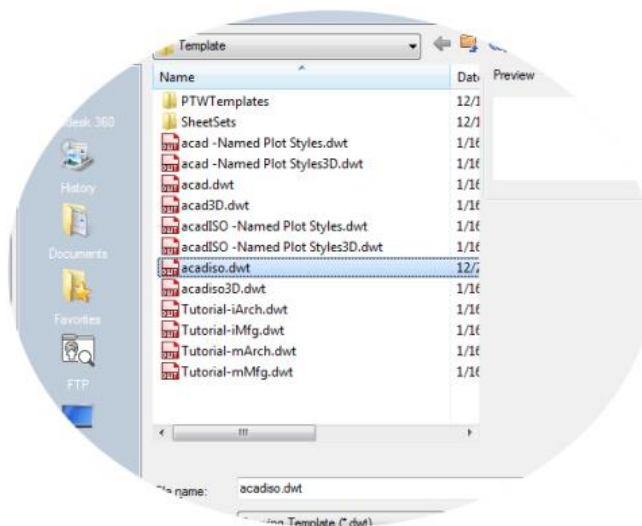
You can easily conform to industry or company standards by specifying settings for text, dimensions, linetypes, and several other features. For example, this backyard deck design displays two different dimension styles.



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



- 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 examples of the architectural or mechanical design templates using both imperial (i) and metric (m) measurements. You might want to experiment with them.

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

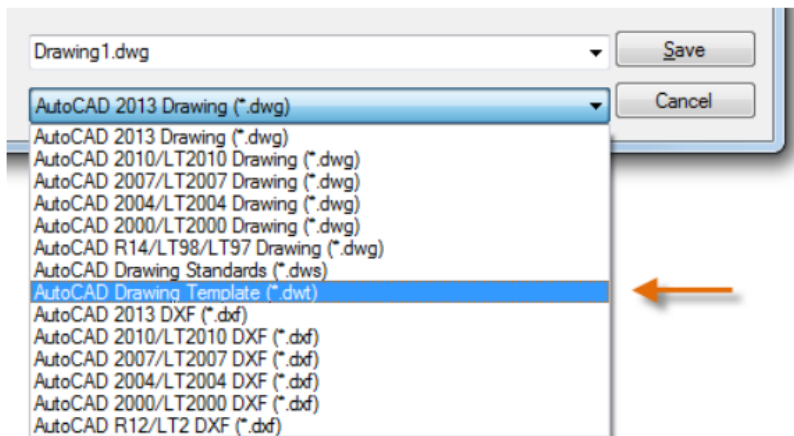


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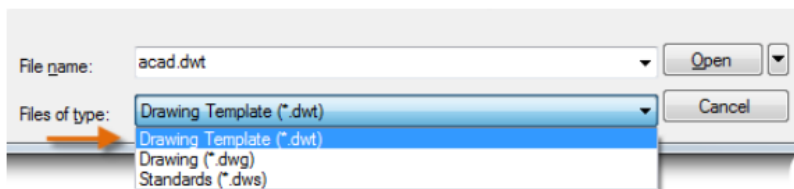
Create Your Own Drawing Template File

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.



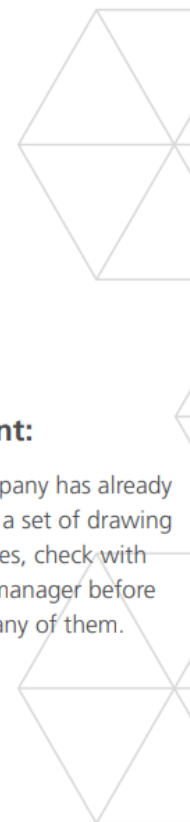
If you work independently, you can develop your drawing template files to suit your working preferences, adding settings for additional features as you become familiar with them.

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.



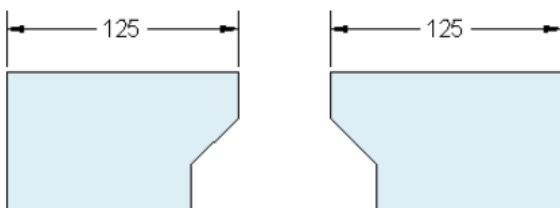


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Units

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.

Here's a Tip:

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



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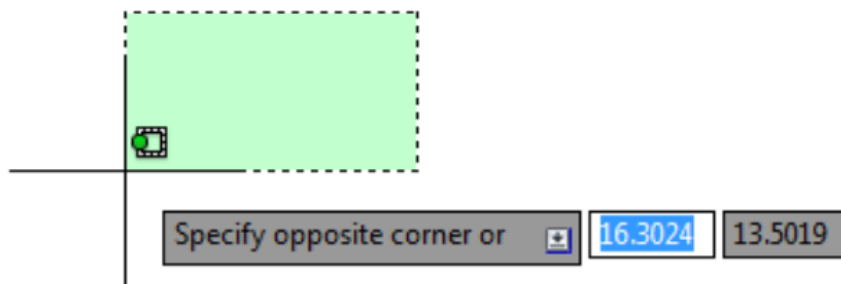
Model Scale

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** for information about the command in progress, 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:



Here's a Tip:

Press Esc to cancel this preselection operation.

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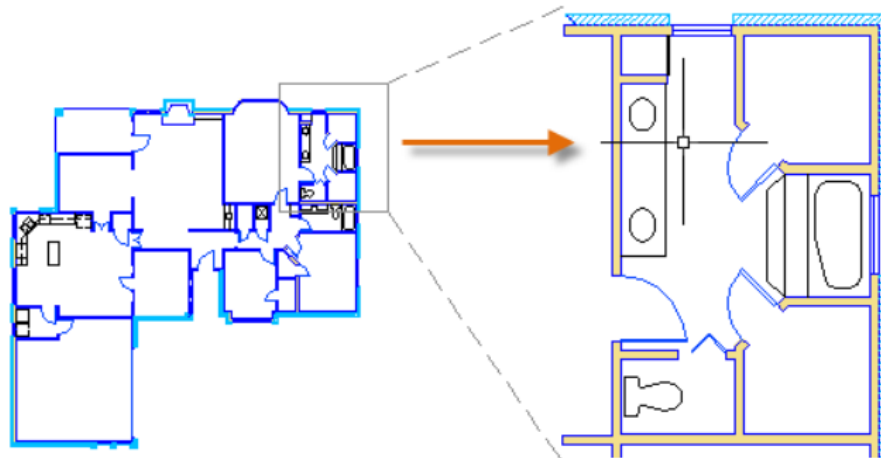
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Viewing

Zoom in on a drawing to better control the order of overlapping objects.

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

- Zoom in or out by rolling the wheel.
- Pan a view in any direction by holding the wheel down while moving your mouse.
- Zoom in on a specific area for greater detail holding your mouse over the area and clicking the wheel twice.



Here's a 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.

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Geometry

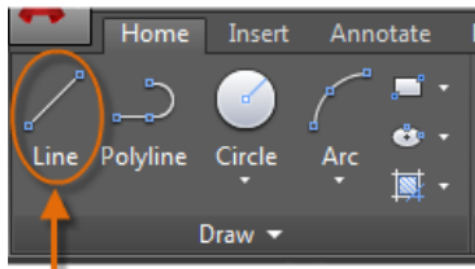
Create basic geometric objects such as lines, circles, and hatched areas.

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

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

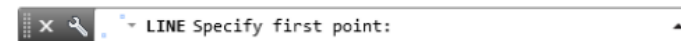
Lines

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

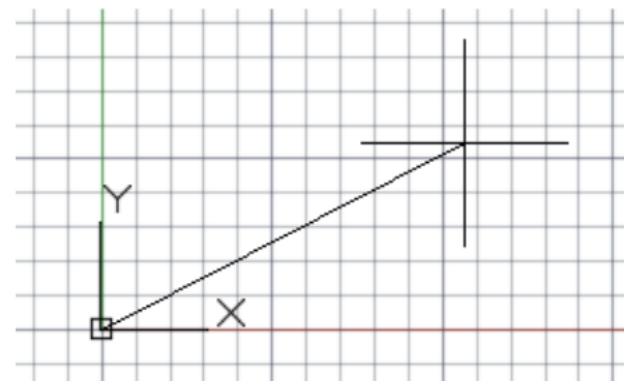


Alternatively, you can type **LINE** or just **L** in the Command window, and then press Enter or the Spacebar.

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.



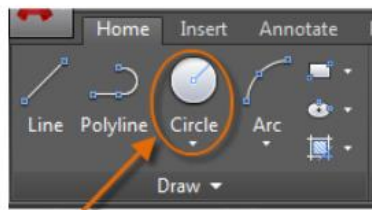
After you specify the next point, the **LINE** command automatically repeats itself, and it keeps prompting you for additional points. Press Enter or the Spacebar to end the sequence.

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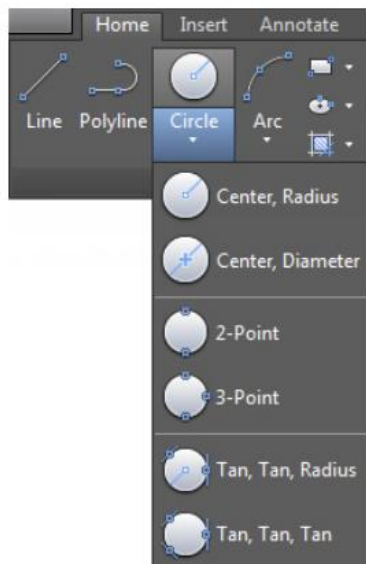
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Circles

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



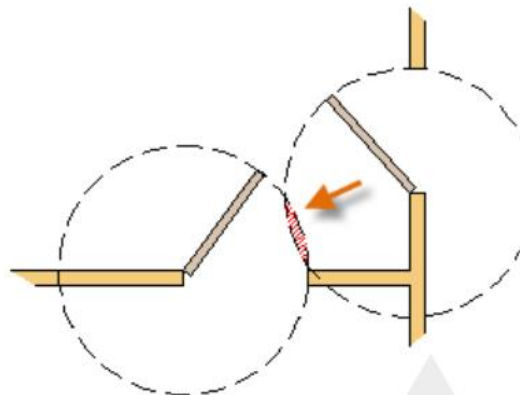
The other circle options are available from the drop-down:



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.

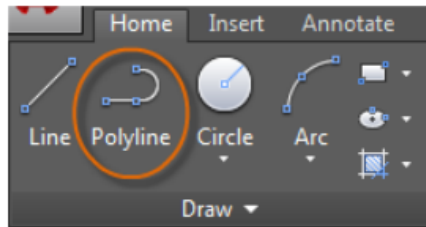


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Polylines and Rectangles

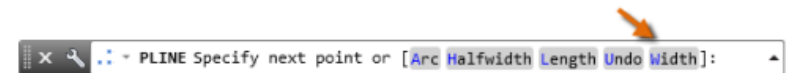
A polyline is a connected sequence of line or arc segments that is created as a single object.



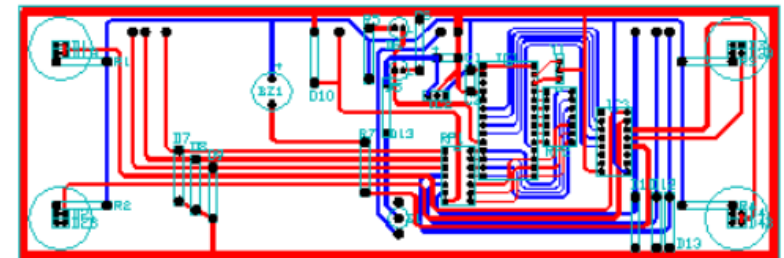
Use the **PLINE** command to create open or closed polylines for:

- Geometry that requires 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.



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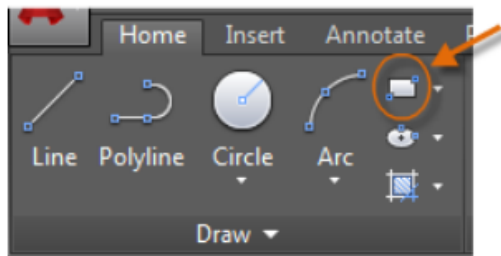
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Polylines and Rectangles (continued)

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.

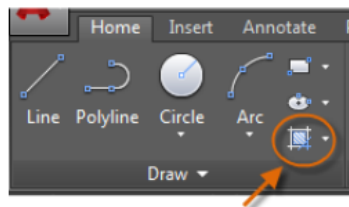


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Hatches and Fills

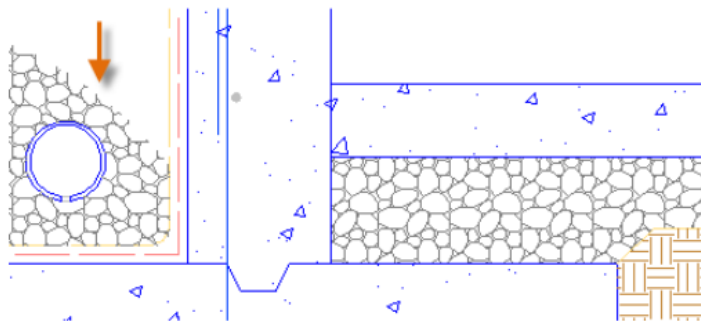
In AutoCAD, a hatch is a single, compound object that covers a specified area with a pattern of lines, dots, shapes, a solid fill color, or a gradient fill.



When you start the **HATCH** command, the ribbon temporarily displays the Hatch Creation tab. On this tab, you can choose from over 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 must specify the scale factor for the hatch in order 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:



Here's a Tip:

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

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Precision

Ensure the precision required for your models.

There are several precision features available, including:

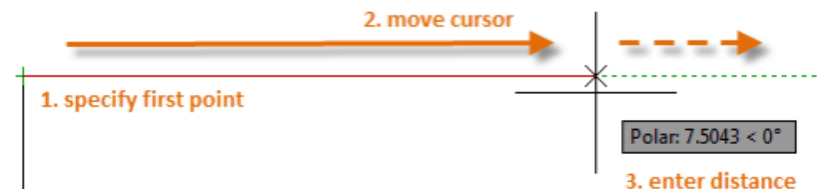
- **Polar tracking:** Snap to the closest preset angle and specify a distance along that angle.
- **Locking angles:** Lock to a single, specified angle and specify a distance along that angle.
- **Object snaps:** Snap to precise locations on existing objects, such as an endpoint of a polyline, the midpoint of a line, or the center point of a circle.
- **Grid snaps:** Snap to increments on 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).

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Locking Angles

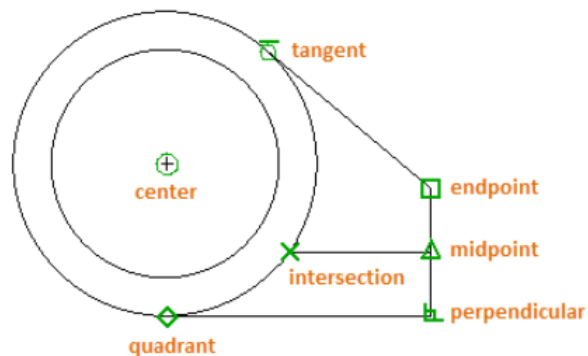
If you need to draw a line at a specified angle, you can lock the angle for the next point. For example, if the second point of a line needs to be created at a 45 degree angle, you would enter '<45' in the Command window.



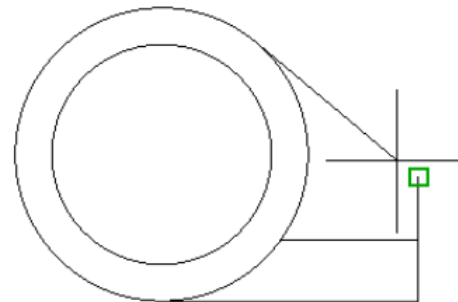
After you move your cursor in the desired direction along the 45-degree angle, you can enter the length of the line.

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.

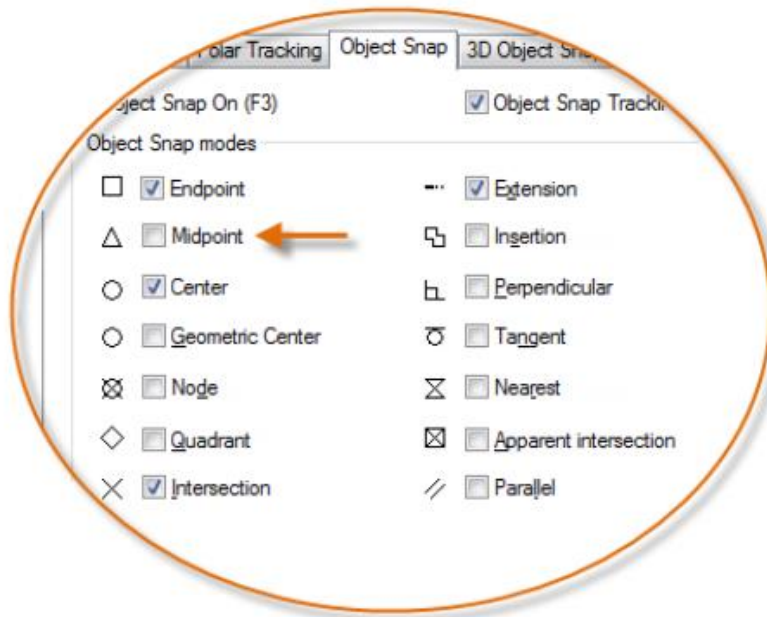


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Set Default Object Snaps

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

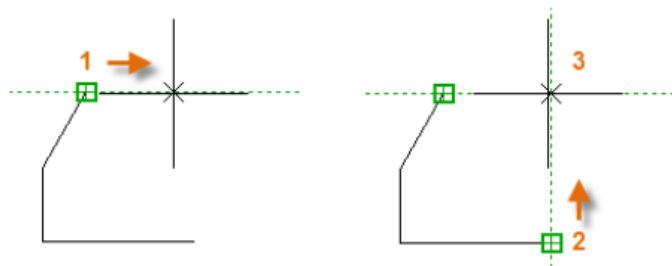
- ➔ At any prompt for a point, you can specify a single object snap that overrides all other object snap settings. Hold down Shift, right-click in the drawing area, and choose an object snap from the Object Snap menu. Then move the cursor to select a location on an object.
- ➔ Make sure that you zoom 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.

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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.



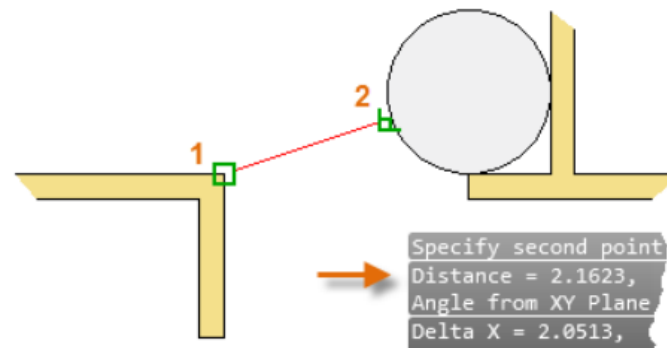
You can now finish creating the line, circle, or other object that you were creating from that location.

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.



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To be continued next week:

- Layers
- Properties
- Linetypes
- Lineweights
- Modyfing
- Offset
- Trim and Extend
- Mirror
- Stretch
- Dimensions