



LEARN ABOUT AUTOCAD

An Introduction to AutoCAD for Beginners

Geometry

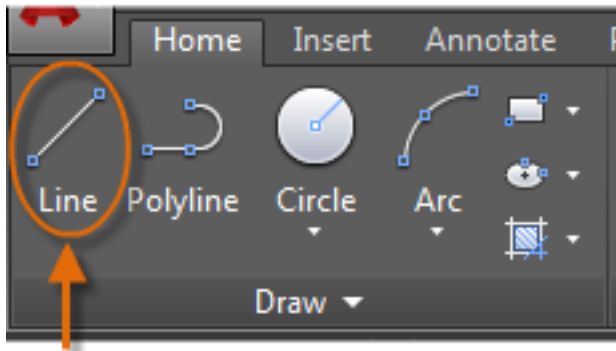
Create basic geometric objects such as lines, circles, and hatched areas.

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.

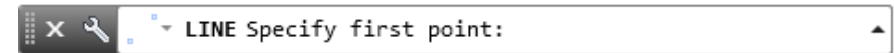
Lines

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

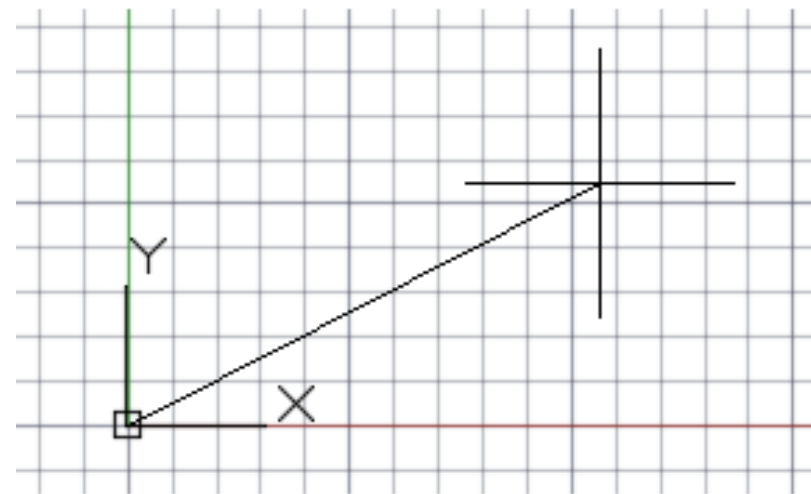


Alternatively, you can type **LINE** or just **L** in the Command window, and then press Enter or the Spacebar.

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



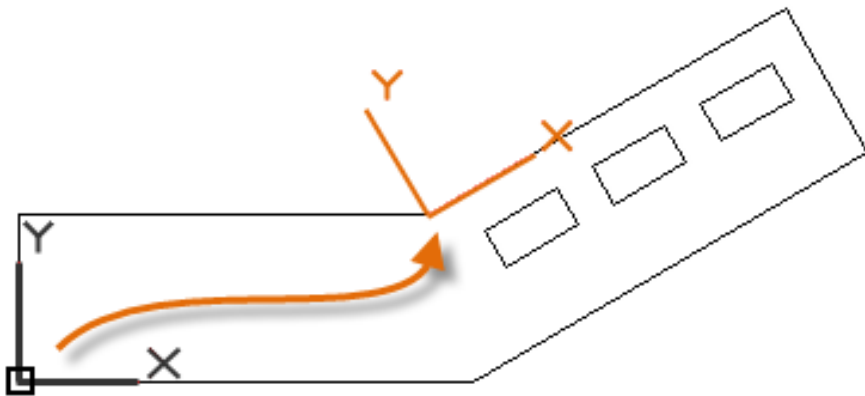
To specify the starting point for this line, you would type in the coordinates 0,0. It is a good idea to locate one corner of your model at 0,0, which is called the origin point. To locate additional points, you could specify additional X,Y coordinate locations in the drawing area, however more efficient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

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

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.

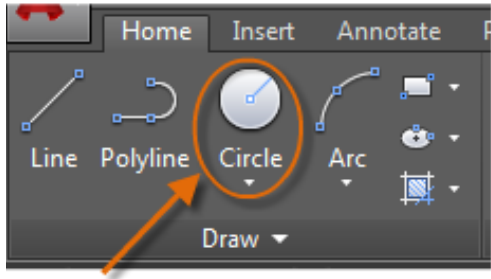
Lines as Construction Aids

Lines can serve as reference and construction geometry such as:

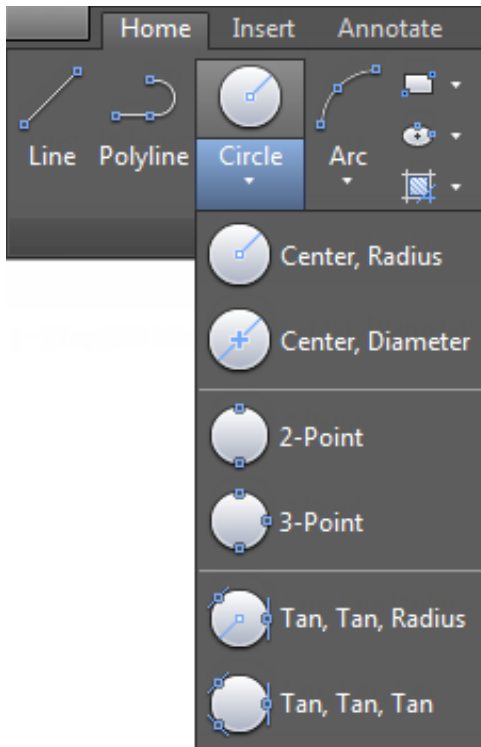
- Property line setbacks
- The mirror line of a symmetrical mechanical part
- Clearance lines to avoid interferences
- Traversal path lines

Circles

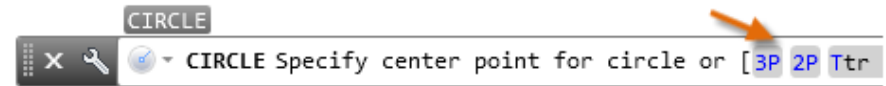
The default option of the **CIRCLE** command requires you to specify a center point and a radius.



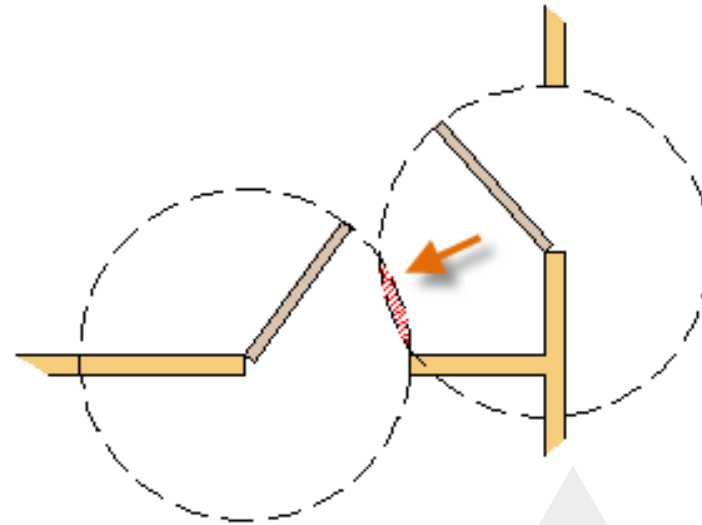
The other circle options are available from the drop-down:



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.

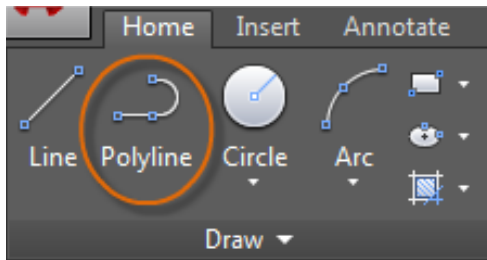


Circles can be useful as reference geometry. For example, you can see that the two doors in the illustration can interfere with each other.



Polylines and Rectangles

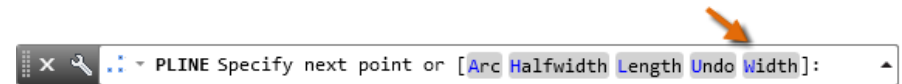
A polyline is a connected sequence of line or arc segments that is created as a single object.



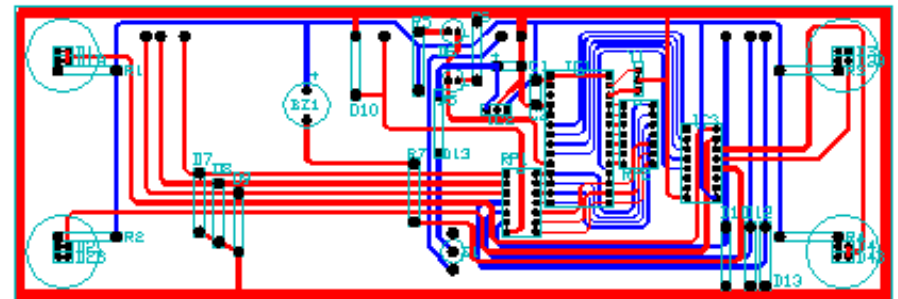
Use the **PLINE** command to create open or closed polylines for:

- Geometry that 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.

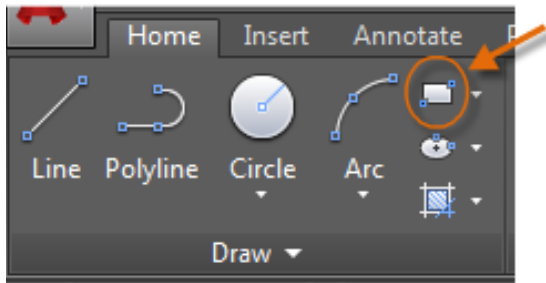


Polylines and Rectangles (continued)

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).

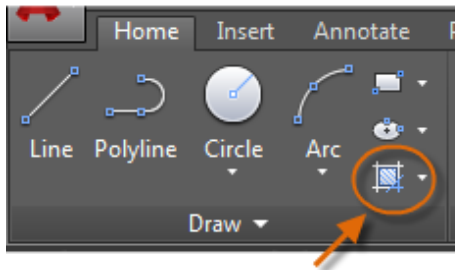


Simply click two diagonal points for the rectangle as illustrated. If you use this method, turn on grid snap (F9) for precision.



Hatches and Fills

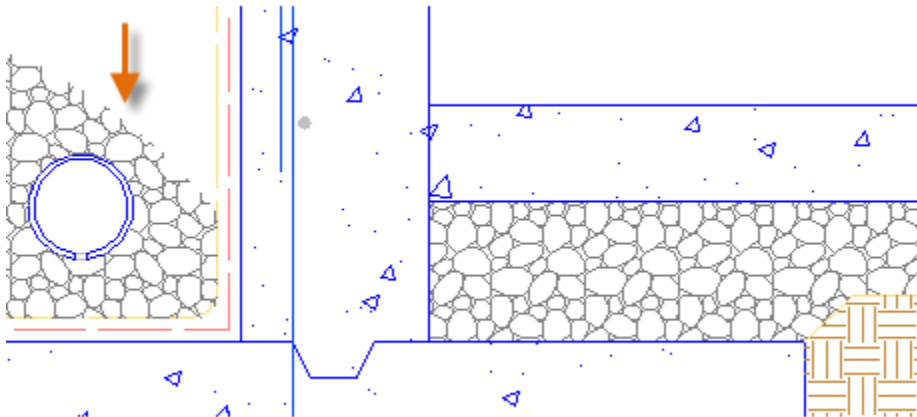
In AutoCAD, a hatch is a single, compound object that covers a specified area with a pattern of lines, dots, shapes, a solid fill color, or a gradient fill.



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:

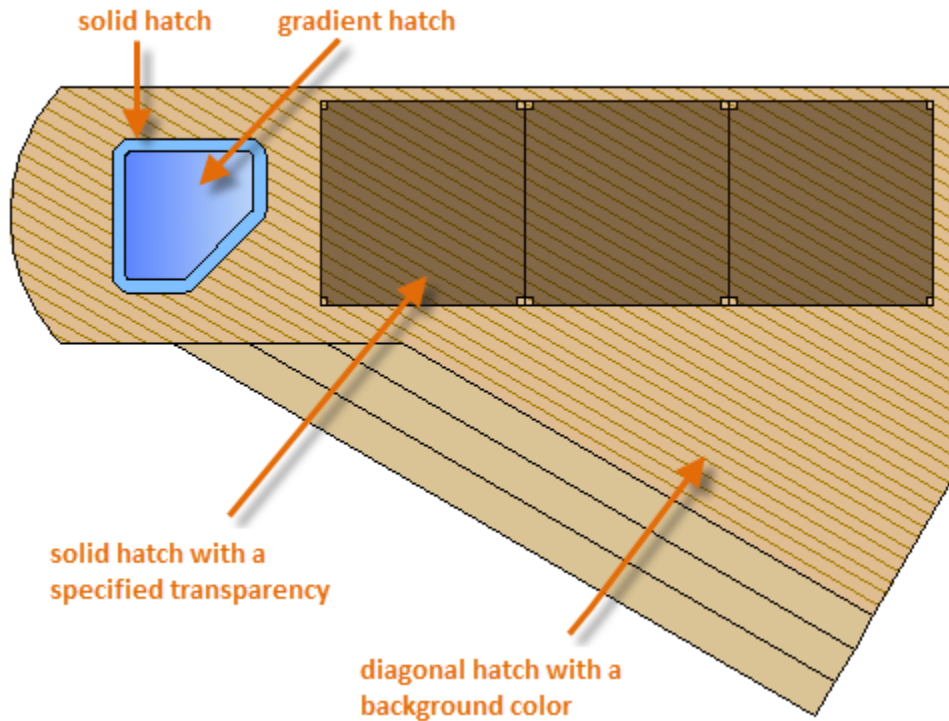


Here's a Tip:

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.

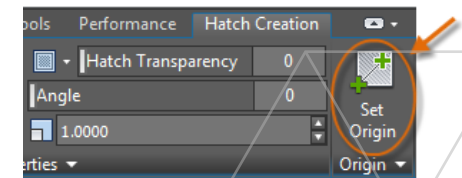
Hatches and Fills (continued)

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



Here's a 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 appear to indicate potential 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:

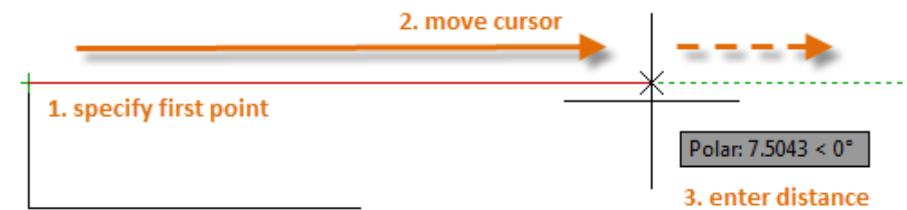
- ➔ **Polar tracking:** Snap to the closest preset angle and specify a distance along that angle.
- ➔ **Locking angles:** Lock to a single, specified angle and specify a distance along that angle.
- ➔ **Object snaps:** Snap to precise locations on existing objects, such as an endpoint of a polyline,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

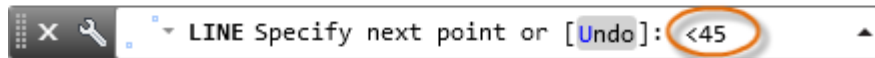
For example, after you specify the first point of the line below, move your cursor to the right, and then enter a distance in the Command window to specify a precise horizontal length for the line.



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

Locking Angles

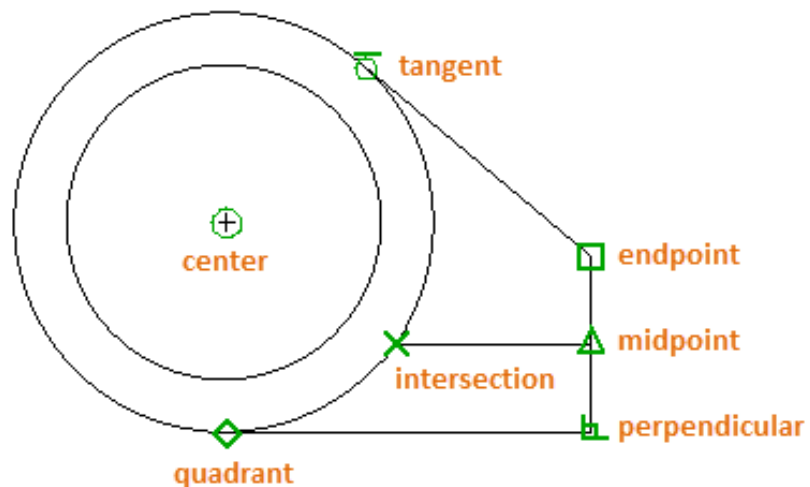
If you need to draw a line at a specified angle, you can lock the angle for the next point. For example, if the second point of a line needs to be created at a 45 degree angle, you would enter '<45' in the Command window.



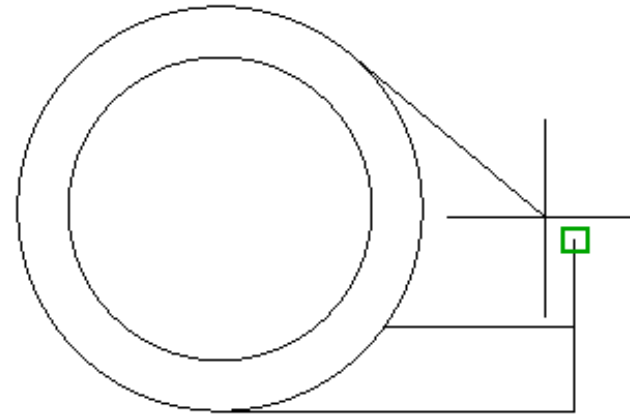
After you move your cursor in the desired direction along the 45-degree angle, you can enter the length of the line.

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.

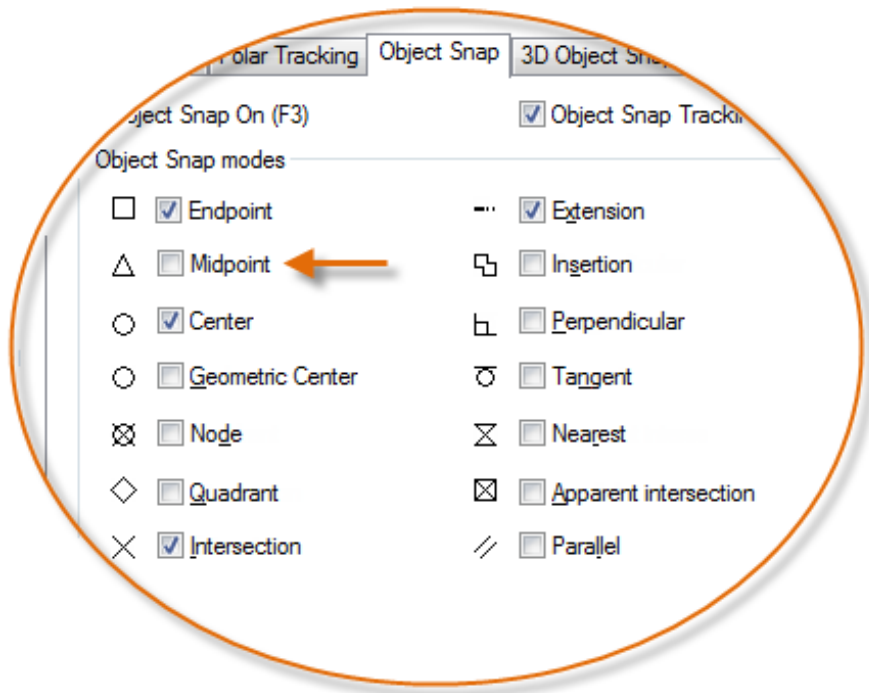


Object snaps become available during a command whenever AutoCAD prompts you to specify a point. For example, if you start a new line and move your cursor near the endpoint of an existing line, the cursor will automatically snap to it.



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.

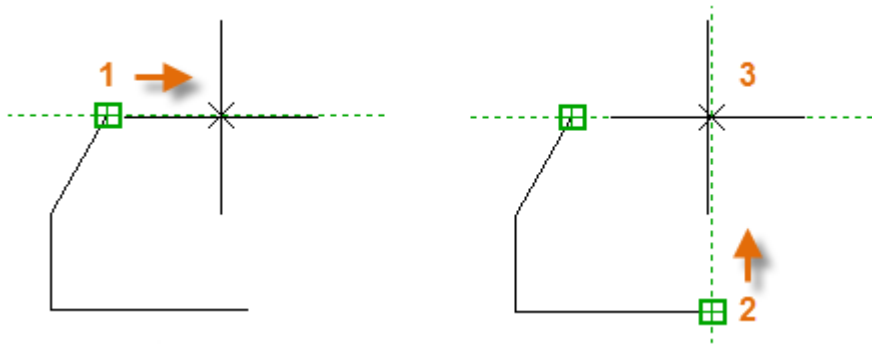


Recommendations

- ➔ At any prompt for a point, you can specify a single object snap that overrides all other object snap settings. 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.



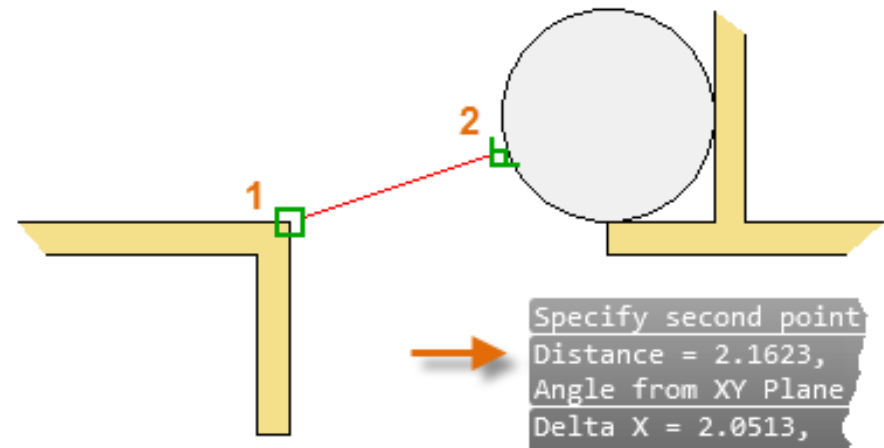
You can now finish creating the line, circle, or other object that you were creating from that location.

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.