


Drawing Lines, Polygons and Rectangles

The **line** is the basic object in AutoCAD. You can create a variety of lines: single lines, multiple line segments with and without arcs, multiple parallel lines, and freehand sketch lines. In general, you draw lines by specifying coordinate points, properties such as linetype or color, and measurements such as angles. The default linetype is CONTINUOUS, an unbroken line, but various linetypes are available that use dots and dashes.

A **line** can be one segment or a series of connected segments, but each segment is a separate line object. Use lines if you want to edit individual segments. If you need to draw a series of line segments as a single object, use a polyline.

To draw a line


- From the Draw menu, choose Line. 
- Specify the start point.
- Specify the endpoint.
- Specify the endpoints of the next segments.
- Press ENTER to complete the line.

You can create **construction lines** that extend to infinity in one or both directions. The term construction line usually refers to xlines, which extend to infinity in both directions. Construction lines that extend to infinity in only one direction are called **rays**.

You can use construction lines as references for creating other objects. For example, you can use construction lines to find the center of a triangle, prepare multiple views of the same item, or create temporary intersections that you can use for object snaps.


Construction lines do not change the extents of the drawing; therefore, their infinite dimensions have no effect on zooming or viewpoints. You can move, rotate, and copy construction lines just as you move, rotate, and copy other objects. You may want to create construction lines on a construction line layer that can later be frozen or turned off, so that the construction lines are not plotted.

To create a construction line by specifying two points

- From the Draw menu, choose Construction Line. 
- Specify a point to define the root of the construction line.
- Specify a second point through which the construction line should pass.
- Continue to specify construction lines as needed.
- Press ENTER to end the command.

A **polyline** is a connected sequence of line or arc segments created as a single object. Use polylines if you want to edit all segments at once, although you can also edit them singly. You can set the width of individual segments, make segments taper, and close the polyline. When you draw arc segments, the first point of the arc is the endpoint of the previous segment. You can specify the angle, center point, direction, or radius of the arc. You can also complete the arc by specifying a second point and an endpoint.


To draw a polyline with straight segments

- From the Draw menu, choose Polyline. 
- Specify the first point of the polyline.
- Specify the endpoint of each polyline segment.
- Enter c (Close) to close the polyline, or press ENTER to end the command.

Polygons are closed polylines with between 3 and 1,024 equal-length sides. You draw a polygon by inscribing it in, or circumscribing it about, an imaginary circle or by specifying the endpoints of one of the edges of the polygon. Because polygons always have equal-length sides, they provide a simple way to draw squares and equilateral triangles.


Use inscribed polygons when you want to specify the distance between the center of the polygon and each vertex. This distance is the radius of the circle within which the polygon is inscribed. In this example, you draw an inscribed square, the default polygon.

To draw an inscribed square

- From the Draw menu, choose Polygon. 
- Enter 4 to specify four sides for the polygon.
- Specify the center point for the polygon.
- Enter i (Inscribed in Circle).
- Specify the radius .


Use circumscribed polygons when you want to specify the distance between the center of the polygon and the midpoint of each side. This distance is the radius of the circle the polygon circumscribes.

To draw a circumscribed hexagon

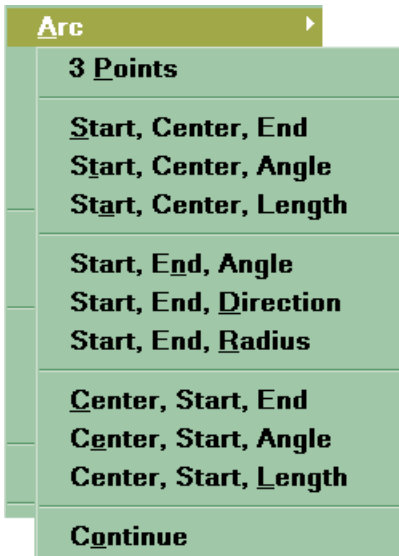
- From the Draw menu, choose Polygon. 
- Enter 6 for the number of sides.
- Specify the center of the polygon.
- Enter c (Circumscribed about Circle).
- Specify the radius length .

A **rectangle** is a rectangular polyline created using two picks.

To draw a rectangle

- From the Draw menu, choose Rectangle. 
- Specify the first corner of the rectangle.
- Specify the opposite corner of the rectangle.

Drawing Arcs




You can create **arcs** in many ways.

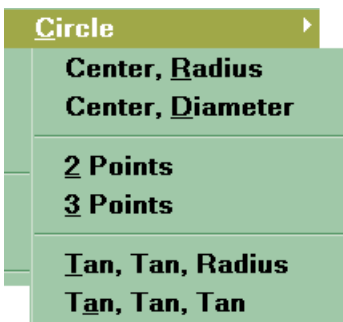
The default method is to specify three points—a start point, a second point on the arc, and an endpoint. You can also specify the included angle, radius, direction, and chord length of arcs. The chord of an arc is a straight line between the endpoints.

By default, AutoCAD draws arcs counterclockwise.

To draw an arc by specifying three points

- From the Draw menu, choose Arc Start, Center, End. 
- Specify the start point by entering endp and selecting the line. The arc snaps to the endpoint of the line.
- Specify the second point by entering cen and selecting the existing arc to define the center of the arc.
- Specify the endpoint of the arc.


Drawing Circles

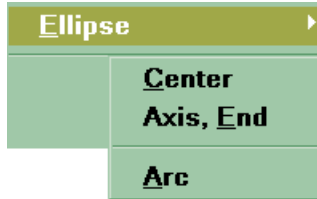


You can create **circles** in several ways.

The default method is to specify the center and radius. You can also specify the center and diameter or define the diameter alone with two points. You can define the circle's circumference with three points. You can also create the circle tangent to three existing objects or create it tangent to two objects and specify a radius. In the following illustrations, the darker circles are the ones being drawn.

To draw a circle by specifying a center point and radius


- From the Draw menu, choose Circle Center, Radius. 
- Specify the center point.
- Specify the radius.

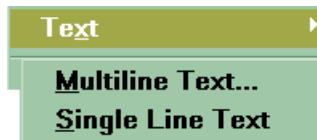
Drawing Ellipses

You can create full ellipses and elliptical arcs, both of which are exact mathematical representations of ellipses.

The default method of drawing an ellipse is to specify the endpoints of the first axis and the distance, which is half the length of the second axis. The longer axis of an ellipse is called the major axis, and the shorter one is the minor axis. The order in which you define the axes does not matter.

To draw a true ellipse using endpoints and distance

- From the Draw menu, choose Ellipse Axis, End. 
- Specify the first endpoint of the first axis.
- Specify the second endpoint of the first axis.
- Drag the pointing device away from the midpoint of the first axis and click to specify the distance.

Text


The default is **Multiline text**. Multiline text consists of any number of text lines or paragraphs that fit within a width you specify. Unlike single-line text, multiline text includes as part of the same mtext object all text lines or paragraphs created in a multiline text editing session. You can move, rotate, erase, copy, mirror, stretch, or scale mtext objects

Using **Single Line Text** you can create one or more lines of text and end each line when you press ENTER. Each text line is a separate object that you can relocate, reformat, or otherwise modify.

Text conveys important information in your drawing. You use text for title blocks, to label parts of the drawing, to give specifications, or to make annotations.

AutoCAD® provides various ways to create text. For short, simple entries, use line text. For longer entries with internal formatting, use multiline text. Although all entered text uses the current text style, which establishes the default font and format settings, you can customize the text appearance.

To create multiline text

- From the Draw menu, choose Text Multiline Text. 
- At the Specify First Corner prompt, use your pointing device to specify the corner.
or
Enter coordinate values on the command line.
- At the next prompt, define the text width by using your pointing device to specify the opposite corner of the boundary box.
or
Enter a width value on the command line.

If you choose to enter options on the command line, AutoCAD continues to display prompts on the command line until you specify the opposite corner of the boundary box.
The Multiline Text Editor is displayed after you specify the second corner of the boundary box.
Arrows inside the boundary box indicate the flow of the entered text based on the current justification setting.

- In the Multiline Text Editor, enter the text.

Text that extends beyond the boundary box width wraps to the next line.

- To convert text to uppercase as you enter it, double-click AutoCAPS.
- Choose OK.

To create Single Line Text

- From the Draw menu, choose Text Single Line Text.
- The system will display:
Current text style: "TEXT" Text height: 0.1250
Specify start point of text or [Justify/Style]:
This tells you that you will be placing .125" text and that the current style is "Text".
- Specify a different justification or style and the insertion point for the first character. Press ENTER to locate the new text immediately below the last text object you created, if any.

If the text height is set to 0 in the current text style, you are prompted to specify the height of the text.
Set the text height by dragging the pointing device until the distance between the cursor and the insertion point indicates the text height you want.

or

On the command line, enter a value in drawing units.

- Set a text rotation angle by dragging the pointing device until the angle between the cursor and the insertion point represents the text rotation angle you want.

or

On the command line, enter the X,Y coordinate.

- Enter the text. Press ENTER to end one line of text and begin another.

The TEXT command displays the text in the drawing as you type. Each line of text is a separate object. If you select another point in the drawing while TEXT is active, the cursor moves to that point, and you can continue entering text from there.

- Press ENTER on a blank line to end text creation.