Draw Commands

In AutoCAD, draw commands are instructions used to create and construct objects. These commands tell the software what actions to perform, such as drawing lines, circles, or polygons.

Accessing Commands

Frequently used commands are located in the toolbar, organized into panels and tabs. You can activate a command by:

- Typing its corresponding shortcut or command alias.
- Selecting the command icon in the toolbar.

Experienced users often prefer using shortcut commands because they:

- Facilitate easier communication with the application.
- Remain consistent across different versions of AutoCAD.

Line Command

The Line command is used to draw a simple line from one point to another. Each line segment is a separate object that can be moved or erased independently.

Activation

You can activate the Line command in two ways:

- **Using the Command Line**: Type L and press Enter.
- **Using the Line Icon**: Click the line icon in the draw panel bar.

Drawing a Line

1. Activate the Line command.
2. Specify the first point by clicking anywhere on the drawing area.
3. Specify the next point by clicking again.
4. Press Enter or Esc to end the command.

Creating a Line with Precise Distance

1. Activate the Line command.
2. Specify the starting point.
3. Enter the desired distance and press Enter.

Creating a Line at a Specific Angle

1. Click Home tab > Draw panel > Line.
2. Specify the start point.
3. Enter the left angle bracket (<) and Input the angle, for example <45.
4. Move the cursor to indicate the direction or specify the length of the line, for Example 5.
5. Press space bar or Enter.

Drawing Lines using Specific Coordinates

1. Click Home tab > Draw panel > Line.

2. Type the coordinate value for the first point by typing the X value, a comma, then the Y value, for example 2,5.

3. Press the Spacebar or Enter

4. Do one of the following:

   - If dynamic input is on: Type the pound sign (#) followed by the X-value, a comma, then the Y-value, for example #5,7.
   - If dynamic input is off: Type the X value, a comma, then the Y value, for example 5,7.

5. Press the Spacebar or Enter.

   Note: When dynamic input is on, relative coordinates are the default. When dynamic input is off, absolute coordinates are the default. Press F12 to turn dynamic input on or off.

Relative Coordinates

AutoCAD can measure coordinates from the last specified point by using the @ symbol before the coordinate.

> The @ symbol tells AutoCAD to measure from the current position. For example, @2,1 measures 2 units in the positive x direction and 1 unit in the positive y direction from the current location.

Command Sequence

1. Click Home tab > Draw panel > Line.

2. Specify the first point.

3. To specify the second point relative to the first point, do one of the following:

   - If dynamic input is on: Type the X-value, a comma, then the Y-value, forexample 4,7.
   - If dynamic input is off: Type the at sign (@) followed by the X-value, a comma,then the Y-value, for example @4,7.

4. Press the Spacebar or Enter.

Polar Coordinates

Polar coordinates are used to draw the next points at a specific angle. Specify a point, enter a distance, and an angle separated by a left angle bracket (<).

- Example: @3<135 (3 is the distance, <135 is the angle).
- Angles increase counterclockwise and decrease clockwise. Use negative values for clockwise direction (e.g., 1<315 is the same as 1<-45).

Polyline Command

The Polyline (Pline) command is similar to the Line command, but it creates a single object composed of multiple segments.

Activation

- Type `PL` and press Enter.
- Click the Polyline icon in the ribbon.

Command Sequence

1. Activate the Polyline command.
2. Specify the start point.
3. Specify subsequent points.
4. Press Enter or type `C` to close the polyline.

Rectangle Command

The **Rectangle command** draws a rectangle with vertical and horizontal sides. It is essentially a closed polyline drawn automatically.

Activation

- Type `REC` and press Enter.
- Click the Rectangle icon in the ribbon.

Command Sequence

1. Activate the Rectangle command.
2. Specify the first corner point.
3. Specify the opposite corner point.

Using Dimensions

1. Activate the Rectangle command.
2. Pick a point anywhere in your drawing area
3. Type `D` and press Enter to select the "Dimensions" option.
4. Enter the length and press Enter.
5. Enter the width and press Enter.

⬟ Polygon Command

The **Polygon command** creates an equilateral closed polyline with 3 to 1024 sides. You can create polygons either inscribed within a circle or circumscribed around a circle.

Command Sequence

1. Type `POL` and press Enter.
2. Enter the number of sides.
3. Specify the center of the polygon or choose the Edge option.
4. Choose either Inscribed in circle or Circumscribed about circle.
5. Specify the radius of the circle.

Circle Command

The **Circle command** is used to draw circles. There are several ways to define a circle:

- Center, Radius (default method)
- Center, Diameter
- 2-Point
- 3-Point
- Tan, Tan, Radius
- Tan, Tan, Tan

Center, Radius (Default)

1. Type `C` and press Enter.
2. Click anywhere in the drawing area to mark the center of the circle.
3. Enter the radius and press Enter.

Center, Diameter

1. Click the Circle icon in the ribbon.
2. Choose the "Center, Diameter" option.
3. Click anywhere in your drawing area to mark the center of the circle.
4. Enter the diameter and press Enter.

2-Point

1. Click the Circle icon in the ribbon.
2. Choose the "2-Point" option.
3. Click the first endpoint of the circle's diameter.
4. Click the second endpoint of the circle's diameter.

3-Point

1. Type `C` and press Enter.
2. Specify the "3P" option.
3. Pick the first point on the circle.
4. Pick the second point on the circle.
5. Pick the third point on the circle.

Tan, Tan, Radius

Creates a circle with a specified radius tangent to two objects.

1. Type `C` and press Enter.
2. Specify the "Ttr" option.
3. Pick the first tangent object.
4. Pick the second tangent object.
5. Enter the radius.

Tan, Tan, Tan

Creates a circle tangent to three objects.

1. Click the Circle icon in the ribbon.
2. Specify the "Tan, Tan, Tan" option.
3. Pick the first tangent object.
4. Pick the second tangent object.
5. Pick the third tangent object.

Arc Command

The **Arc command** draws an arc of a circle. The default method uses three pick points: start, second, and end points.

Activation

- Type A and press Enter.
- Click the Arc icon in the ribbon.

Command Sequence

1. Activate the Arc command.
2. Specify the start point.
3. Specify the second point.
4. Specify the end point.

Other Options

- **Start, Center, End**: Specifies the start point, center, and endpoint of the arc.
- **Start, Center, Angle**: Specifies the start point, center, and included angle.
- **Start, Center, Length**: Specifies the start point, center, and chord length.
- **Start, End, Angle**: Specifies the start point, endpoint, and included angle.
- **Start, End, Direction**: Specifies the start point, endpoint, and tangent direction at the start point.
- **Start, End, Radius**: Specifies the start point, endpoint, and radius.
- **Center, Start, End**: Specifies the center point, start point, and endpoint.
- **Center, Start, Angle**: Specifies the center point, start point, and included angle.
- **Center, Start, Length**: Specifies the center point, start point, and chord length.
- **Continue**: Creates an arc tangent to the last drawn line, arc, or polyline.

Ellipse Command

The **Ellipse command** creates an ellipse or an elliptical arc. The first two points determine the distance between the center of the ellipse and the endpoint of the second axis.

Activation

- Type EL and press Enter.
- Click the Ellipse icon in the ribbon.

Command Sequence

1. Type `EL` and press Enter.
2. Specify the first endpoint of the axis.
3. Specify the other endpoint of the axis.
4. Specify the distance to the other axis or select the Rotation option.