Chapter 16
More Draw Commands
AutoCAD 2D Tutorial

Polygon 16.1

1. **Choose** Draw, Polygon.
   
   or

2. **Click** the Polygon icon.
   
   or

3. **Type** Polygon at the command prompt.
   
   Command: **POLYGON**

4. **Type** The number of sides for the polygon
   
   (3-1024)

   Number of sides <default>: **number**

5. **Pick** The center of the polygon. Edge/<Center of polygon>:
   
   pick

   or

6. **Type** **E** to define the polygon by two edges.

7. **Type** **I** or **C** to place the polygon inside or outside of an
   
   imaginary circle.

   Inscribed in circle/Circumscribed about circle (I/C):

   Polygon Inscribed in an imaginary circle

   ![Polygon Inscribed in an imaginary circle](image1)

   Polygon circumscribed around an imaginary circle

   ![Polygon circumscribed around an imaginary circle](image2)

   Polygon drawn with Edge

   ![Polygon drawn with Edge](image3)
Rectangle 16.2

1. **Choose** Draw, Rectangle.
   
   or

2. **Click** the Rectangle icon.
   
   or

3. **Type** Rectang at the command prompt Command:
   
   RECTANG Chamfer/Elevation/Fillet/Thickness/Width/
   
   <First corner>:

4. **Pick** first corner.

5. **Pick** other corner or type coordinates (i.e. @4,2).
The SPLINE command creates a particular type of spline known as a nonuniform rational B-spline (NURBS) curve. A NURBS curve produces a smooth curve between control points.

1. Choose Draw, Spline.
   or
2. Click the Spline icon.
   or
3. Type SPLINE at the command prompt
   Command: SPLINE
4. Pick A start point for the spline
   Object / <Enter first point>: (pick point)
5. Pick Points until you are done drawing splines
   Enter point: (pick points)
6. Press Enter or close to complete the spline
7. Pick Starting tangent point for the spline
   Enter start tangent (pick point)
8. Pick Ending tangent point for the spline
   Enter end tangent: (pick point)
Spline options:

- **Object**: Converts 2D or 3D spline-fit polylines to equivalent Splines
- **Points**: Points that define the spline
- **Close**: Closes a spline.
- **Fit Tolerance**: Allows you to set a tolerance value that creates a smooth spline.

**TIP**: Refer to AutoCAD online help topic for more information on spline options.

**Editing Splines**

1. Choose **Modify, Object, Spline**.

**TIP**: Drawings containing splines use less memory and disk space than those containing spline-fit polylines of similar shape.
Covert PLINE to Spline 16.4

1. **Draw**  a PLINE.
2. **Type**  PEDIT to edit the polyline as a spline.
3. **Choose** Draw, Spline.
4. **Type**  Object at the command prompt.
5. **Click**  once on the polyline to turn it into a spline.

**TIP:** Use the LIST command to determine if an object is a PLINE or SPLINE.
Donut 16.5

Donuts are filled rings or solid-filled circles that actually are closed polylines with width.

1. **Choose** Draw, Donut.
   or

2. **Type** Donut at the command prompt.
   Command: **DONUT**

3. **Type** A value for the inside diameter.
   Inside diameter <last>: .5

4. **Type** A value for the outside diameter.
   Outside diameter <last>: 1

5. **Pick** A point for the center of the donut.
   Center of doughnut: (point)
**Ellipse 16.6**

Creates an ellipse or an elliptical arc.

1. **Choose** Draw, Ellipse.
   
   or
2. **Choose** the Ellipse or Partial Ellipse icon.
   
   or
3. **Type** ELLIPSE at the command prompt
   
   Command: **ELLIPSE**

4. **Type** One of the following options: Arc/Center/Isocircle /<Axis endpoint 1>:

**Ellipse options:**

**Axis endpoint 1**

Defines the first axis by two specified endpoints. The angle of the first axis determines the angle of the ellipse. The first axis can define either the major or the minor axis of the ellipse.

**Axis endpoint 2:**

<Other axis distance> / Rotation: Specify a point or enter a distance

**Arc**

Creates an elliptical arc. The angle of the first axis determines the angle of the elliptical arc. The first axis can define either the major or the minor axis of the elliptical arc.

**Center**

Creates the ellipse by a specified center point.

**Isocircle**

Creates an isometric circle in the current isometric drawing plane.

**Rotation**

The major axis is now treated as the diameter of a circle that will be rotated a specified amount around the axis. You enter an angle between 0 and 89.4 degrees.
**AutoCAD 2D Tutorial**

**ELLIPSE,**
Axis, Eccentricity (Axis Endpoint, Axis Endpoint, Other Axis Distance)

![Ellipse with axis and eccentricity](image)

**ELLIPSE,**
Center, Axis, Axis

![Ellipse with center and axis](image)

**ELLIPSE,**
Axis Endpoint, Axis Endpoint, Rotation=60

![Ellipse with endpoint and rotation](image)
Multilines 167

MLINE Command

Creates multiple parallel lines.

1. **Choose** Draw, Multiline.
   or
2. **Type** MLINE at the command prompt.
   Command: MLINE
3. **Pick** A point to start the multiline.
   Justification/Scale/STyle/<From point>: pick point
4. **Pick** A second point to continue the multiline.
   <To point>: pick point
5. **Pick** The next point to continue drawing multilines.
   Undo/<To point>: pick point
6. **Press** ENTER to end the multiline
   Close/Undo/<To point>: press enter or
7. **Type** C to close the multiline back to the first point.
   Close/Undo/<To point>: c
Multiline Justifications

Top Justification

Bottom Justification

Zero Justification
Multiline Styles

1. **Choose** Format, Multiline Style...
   or
2. **Type** MLSTYLE at the command prompt.

   Command: **MLSTYLE**

3. **Rename** The existing style called STANDARD to your newstyle.
4. **Choose** Element Properties to change the appearance of the multilines.
5. **Choose** ADD to create the new multiline.
AutoCAD 2D Tutorial

Editing Multilines

1. Choose Modify, Multiline...
   or
2. Type MLEDIT at the command prompt
   Command: MLEDIT
3. Choose From one of the mledit options:
Construction Line and Ray Command 16.8

Creates an infinite line.

1. Choose Draw, ConstructionLine

   or

2. Choose the XLINE icon.

   or

3. Type XLINE at the command prompt.

   Command: XLINE

   Specify a point or [Hor/Ver/Ang/Bisect/Offset]:

Construction Line Options

<table>
<thead>
<tr>
<th>Option</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>HOR</td>
<td>Creates a horizontal line passing through a specified point.</td>
</tr>
<tr>
<td>VER</td>
<td>Creates a vertical line passing through a specified point.</td>
</tr>
<tr>
<td>ANG</td>
<td>Creates a line at a specified angle.</td>
</tr>
<tr>
<td>BISECT</td>
<td>Creates a line that passes through the selected angle vertex and bisects the angle between the first and second line.</td>
</tr>
<tr>
<td>OFFSET</td>
<td>Creates a line parallel to another object.</td>
</tr>
</tbody>
</table>
Ray

Creates an infinite line in one direction.

1. **Choose**  
   Draw, RAY
   or

2. **Type**  
   RAY at the command prompt.
   Command: **RAY**
   Specify a point: *(pick through point)*