Minimal AutoCAD® 2011

Instructor: Thomas Mason, P.E.

2020

PDH Online | PDH Center

5272 Meadow Estates Drive
Fairfax, VA 22030-6658
Phone: 703-988-0088
www.PDHonline.com

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AUTOCAD DRAWING COMMANDS
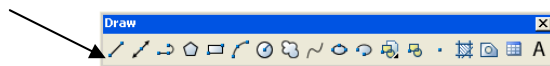
LINE

POLYLINE

CIRCLE

ELLIPSE

ARC

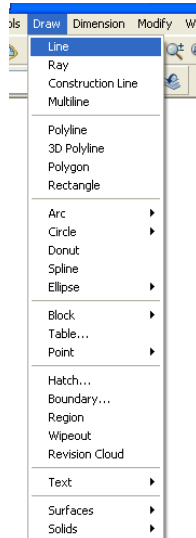POLYGON

RECTANGLE

SPLINE

AUTOCAD DRAWING COMMANDS

AutoCAD Line Command: The line command can allow you to create a single line or a series of lines.

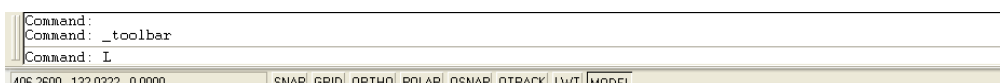To activate the Line command use one of the following options:

1- Draw Toolbar.

2- Draw from the pull-down menu bar.

3- Key board: Type **L** for line on the command line and press **Enter**.
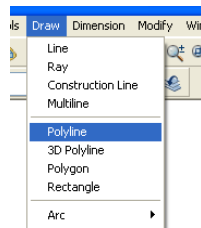

AutoCAD Polyline Command: A polyline is connected sequence of lines created as a single line.

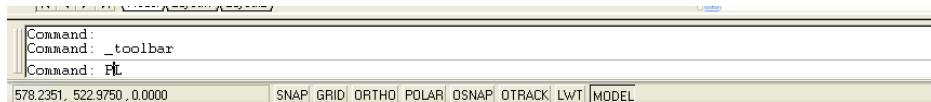To activate the PLine command use one of the following options:

1- Draw Toolbar.

2- Draw from the pull-down menu bar.

1

3- Key Board: Type Pl at the command line and then press **Enter**.


```
Command :
Command : _toolbar
Command : PL
578.2351, 522.9750 , 0.0000        SNAP GRID ORTHO POLAR OSNAP OTRACK LWT MODEL
```

AutoCAD Circle Command: The circle command will allow you to create circle with the following options:
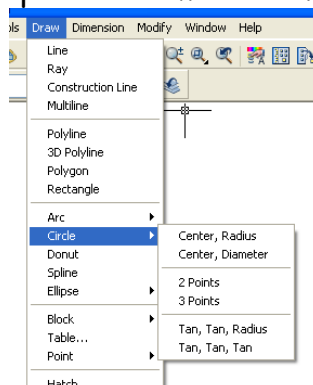
- Center Radius

- Center Diameter

- Two Points

- Three Points

- Tangent, Tangent, radius (TTR)

- Tangent, Tangent, Tangent.

To activate the Circle Command use one of the following options:

1- Draw Tool bar.

2- Draw from the pull-down menu bar.

3- Key Board: Type **C** on the command line and then press **Enter**.

2

Center, Radius Option

- Click Circle on the Draw Toolbar.

- Specify the center point.

- Specify the radius.

Center, Diameter Option

- Click Circle on the Draw Toolbar.

- Specify the center point.

- Specify the diameter.

2P Option

- Click Circle on the Draw Toolbar.

- At the specify the center point for the circle or[3P/2P/TTR].

- Click 2P

- Specify the first point of the circle's diameter.

- Specify the second point of the circle's diameter.

3P Option

- Click Circle on the Draw Toolbar.

- At the specify the center point for the circle or[3P/2P/TTR].

- Click 3P

- Specify the first point of the circle.

- Specify the second point of the circle.

- Specify the third point of the circle.

TTR Option

- Click Circle on the Draw Toolbar.

- At the specify the center point for the circle or[3P/2P/TTR].

- Click 3P

- Specify a point on drawing for the first tangent point of the circle.

- Specify a point on drawing for the second tangent point of the circle.

- Specify the radius of the circle.

AutoCAD Ellipse Command
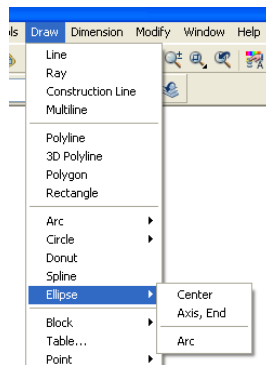
The ellipse command has the following options:

- Endpoint Axis

- Center

- Arc

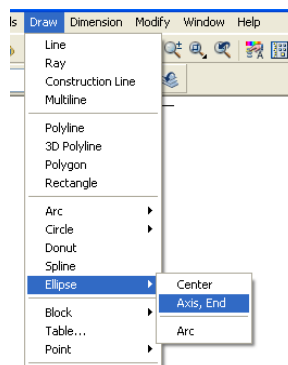To activate the ellipse command use one the following options:

1- Draw Toolbar.

2- Draw from the pull-down menu bar.

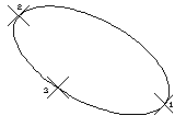3- Key Board: Type Ellipse at the command line and then press Enter.

Drawing an ellipse with a **Distance Option:**
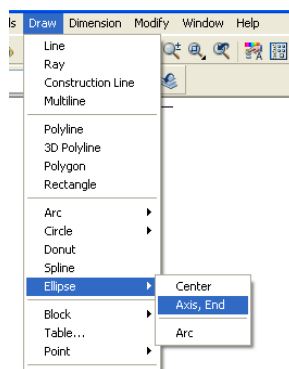
- From the Draw Menu, select Ellipse.

- Specify the first endpoint of the first axis. (1)

- Specify the second endpoint of the first axis. (2)

- At the specify the distance to other axis or [rotation] prompt, drag the cursor away from the previous point and click to specify a distance. (3). See figure bellow.
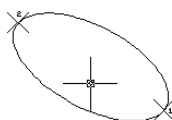

Drawing an ellipse with **Rotation Option**
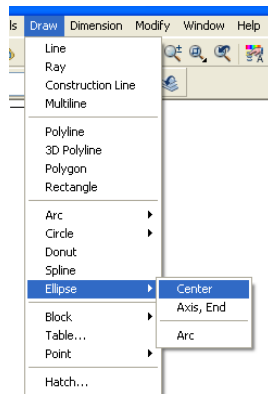
- From the Draw Menu, select ellipse.

- Specify the first endpoint of the first axis. (1)

- Specify the second endpoint of the first axis. (2)

- At the specify the distance to other axis or [rotation] prompt, click Rotation or enter R.

- Move the cursor around the center of the ellipse and click, or specify a value for the angle of rotation. See figure bellow.
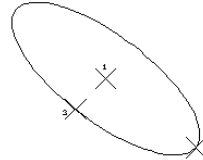
Drawing an ellipse with a **Center Option**

  - From the Draw pull-down menu, select ellipse, then click Center.
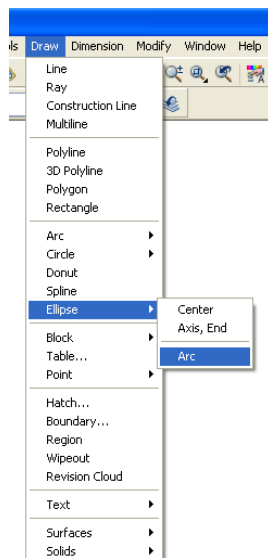

  - Specify the center point. (1)

  - Specify endpoint of axis. (2)

  - At the specify the distance to other axis or [rotation] prompt, click Rotation or enter R.

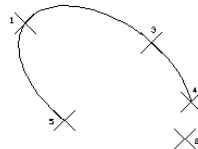 - Specify a distance or locate point. (3) See figure bellow.

Drawing an ellipse with **Arc Option**

  - From the Draw pull-down menu, select ellipse, then click Arc.

- Specify first endpoint on the elliptical arc. (1)

- Specify second endpoint on the elliptical arc. (2)

- Specify distance point. (3)

- Specify the start angle point. (4)

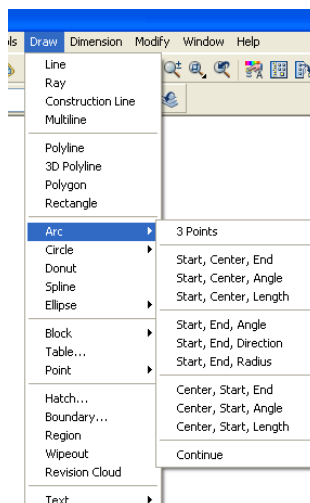- Specify the end angle point. (5), See figure below.

AutoCAD Arc Command

Arc command allows you to create arcs in several ways. Most arcs are drawn in the counter-clockwise direction from the start point to the end point.

To activate the arc command use one the following options:
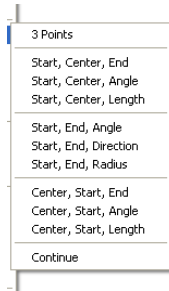
1- Draw Tool bar, on the toolbar select Arc
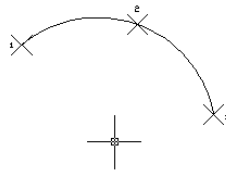

2- Draw from the pull-down menu bar.

3- Type **A** for Arc on the command line then press **Enter**.

List of Arc Options available on the drop-down menu.

3 Points
Start, Center, End
Start, Center, Angle
Start, Center, Length
Start, End, Angle
Start, End, Direction
Start, End, Radius
Center, Start, End
Center, Start, Angle
Center, Start, Length
Continue

Drawing Arcs with **3Points** Option: This type of arc can be drawn in both directions, counter-clockwise and clockwise.

1- Specify the first starting point. (1)

2- Specify the second point. (2)

3- Specify the end point of the arc. (3). See figure below.

Drawing Arcs with **Start, Center, End** Option: This type of arc can be drawn in counter-clockwise direction.

1- Specify the start point. (P1)

2- Specify the center point. (P2)

3- Specify the end point of the arc. (P3). See figure below.

Note: You can use the same procedure to experiment the other options of AutoCAD ARC command.
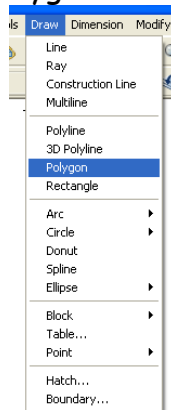
AutoCAD Polygon Command
The polygon command can allow you to create polygons of different shapes and sizes.

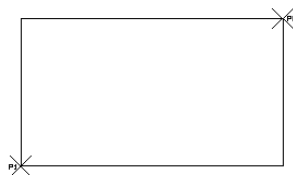To activate the Polygon command use one the following options:

1- Draw Tool bar

2- Draw from the pull-down menu: Enter the number of sides required for the polygon.

3- Key Board, type Polygon> enter the number of side> Specify the center of the polygon>Select an option from Inscribed in a circle or Circumscribed about a circle.

4- Specify the Radius of the circle, enter a value and press Enter.

AutoCAD Rectangle Command

The rectangle command will allow you to create rectangles of different sizes by specifying two points P1 and P2 diagonally opposite as shown in the figure below.
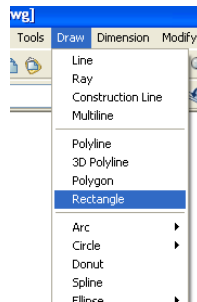

To activate the Rectangle command use one the following options:
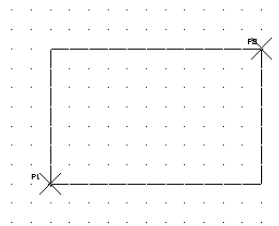
1- Draw Toolbar.

2- Draw from the pull-down menu.

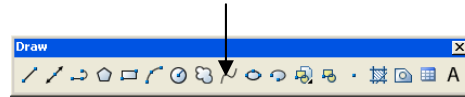3- From the Command line: Type **Rec** for rectangle .

Example:
- At the line command type: REC then press enter.
- At the Prompt: Specify first corner point
  or[Chamfer/Elevation/Fillet/Thickness/Width]:
   Pick P1, see figure below.
- At the prompt: Specify second corner point or [Dimensions]:
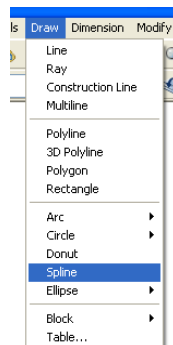   Pick P2, see figure below.

AutoCAD Spline Command: Creates a nonuniform rational B-spline (NURBS) curve.

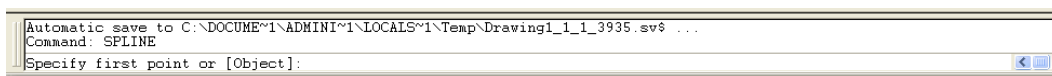To activate the Spline command use one of the following options:

- Draw Toolbar

- Draw from the pull-down menu.

- Type Spline at the command line:

You create splines by specifying points. You can close the spline so that the start and endpoints coincide to form a closed shape. Tolerance describes how closely the spline fits the set of points you specify. The lower the tolerance, the more closely the spline fits the points. At zero tolerance, the spline passes through the points.