Chapter 2: Fundamentals I

Construct Geometric Figures

AutoCAD provides various methods for constructing geometric figures:
- Drawing Lines
- Drawing Rectangles
- Drawing Wide Lines

Close Option

The close option closes a sequence of lines to form a polygon, joins the last and first point. AutoCAD performs two steps when you select the close option: closes the polygon, and terminates the LINE command.

Continue Option

The continue option sets the start of the line to the end of the most recently drawn line or arc.
Undo option

The undo option erases the most recent line segment and continues from the end of the previous line segment.

Command: (ENTER)
Specify first point: (specify Point 1)
Specify next point or [Undo]: (specify Point 2)
Specify next point or [Undo]: (specify Point 3)
Specify next point or [Undo]: (specify Point 4)
Specify next point or [Undo]: (specify Point 5)
Specify next point or [Undo]: (ENTER)
Specify next point or [Undo]: (specify Revised Point 4)
Specify next point or [Undo]: (specify Revised Point 5)
Specify next point or [Undo]: (specify Point 6)
Specify next point or [Undo]: (ENTER)

Drawing Rectangles

Rectangle boxes are drawn in AutoCAD with the RECTANLGE command. Invoke the RECTANGLE command using one of the following input formats.

<table>
<thead>
<tr>
<th>Command</th>
<th>Prompt</th>
</tr>
</thead>
<tbody>
<tr>
<td>Line</td>
<td>Command: prompt</td>
</tr>
<tr>
<td></td>
<td>Use Toolbox</td>
</tr>
<tr>
<td></td>
<td>Choose the Rectangle</td>
</tr>
<tr>
<td></td>
<td>command</td>
</tr>
<tr>
<td></td>
<td>Use Menu</td>
</tr>
<tr>
<td></td>
<td>Choose Rectangle</td>
</tr>
<tr>
<td></td>
<td>Command: prompt</td>
</tr>
<tr>
<td></td>
<td>Use Toolbox</td>
</tr>
<tr>
<td></td>
<td>Choose the Rectangle</td>
</tr>
<tr>
<td></td>
<td>Use Menu</td>
</tr>
<tr>
<td></td>
<td>Choose Draw Menu</td>
</tr>
<tr>
<td></td>
<td>Choose the Rectangle</td>
</tr>
<tr>
<td></td>
<td>Use Toolbar</td>
</tr>
<tr>
<td></td>
<td>Choose the Rectangle</td>
</tr>
<tr>
<td></td>
<td>tool</td>
</tr>
</tbody>
</table>

Drawing Wide Lines

Wide lines are drawn in AutoCAD with the TRACE command. Invoke the TRACE command using the following input format.

<table>
<thead>
<tr>
<th>Command</th>
<th>Prompt</th>
</tr>
</thead>
<tbody>
<tr>
<td>TRACE</td>
<td>Command: prompt</td>
</tr>
<tr>
<td></td>
<td>Use Toolbox</td>
</tr>
<tr>
<td></td>
<td>Choose the Trace</td>
</tr>
<tr>
<td></td>
<td>command</td>
</tr>
<tr>
<td></td>
<td>Use Menu</td>
</tr>
<tr>
<td></td>
<td>Choose Trace</td>
</tr>
<tr>
<td></td>
<td>Command: prompt</td>
</tr>
<tr>
<td></td>
<td>Use Toolbox</td>
</tr>
<tr>
<td></td>
<td>Choose the Trace</td>
</tr>
<tr>
<td></td>
<td>Use Menu</td>
</tr>
<tr>
<td></td>
<td>Choose Draw Menu</td>
</tr>
<tr>
<td></td>
<td>Choose the Trace</td>
</tr>
<tr>
<td></td>
<td>Use Toolbar</td>
</tr>
<tr>
<td></td>
<td>Choose the Trace</td>
</tr>
<tr>
<td></td>
<td>tool</td>
</tr>
</tbody>
</table>

Coordinate Systems

AutoCAD provides two coordinate systems to locate a point: World Coordinate System (WCS) and User Coordinate System (UCS).

The WCS (fixed) is designed in accordance with the conventions of the Cartesian coordinate system; horizontal distances increase in the positive X direction, toward the right, and vertical distances increase in the positive Y direction, upward. Distances perpendicular to the XY plane that you are viewing increase toward you in the positive Z direction. WCS is fixed but can be at any angle, side or rotation.

The UCS (movable) is the user defined version of the coordinate system and created with the UCS command. We will discuss this in more detail in Chapter 15.

Coordinate Point Entry

When a command prompts you for a point, you can use the pointing device to specify a point, or you can enter a coordinate value on the command line. You can enter coordinates either:

- Cartesian coordinate system
- Polar coordinate system

Cartesian coordinate system

In Cartesian coordinate system a point's distance (in units) and its direction (+ or -) are indicated along the X, Y, and Z axes. You can enter absolute coordinates based on the origin (0,0), or relative coordinates based on the last point specified. For example:

- 3,4 --- indicates 3 units along x axis and 4 units along y axis measured from origin (0,0), as absolute Coordinates.
- @3,4 --- indicates 3 units along x axis and 4 units along y axis measured from previous point as relative Coordinates.
Polar coordinate system

In Polar coordinate system a point is indicated in reference to a distance and an angle. By default the angle is measured in counter-clockwise direction from positive x axis. You can enter absolute coordinates based on the origin (0,0), or relative coordinates based on the last point specified.

For example:

3<45 --- indicates 3 units measured from origin as distance and 45 degrees measured counter-clockwise direction from x axis.

@3<45 --- indicates 3 units measured from previous point and 45 degrees measured counter-clockwise direction from x axis.

Coordinate Display

AutoCAD displays the current cursor location as a coordinates on the status bar. There are three types of coordinate display:

- **Dynamic display** updates the $X,Y$ coordinate location as you move the cursor.
- **Distance and angle display** updates the relative distance ($distance<angle$) as you move the cursor. This option is available only when you draw lines or other objects that prompt for more than one point.
- **Static display** updates the $X,Y$ coordinate location only when you specify a point.

You can switch between the types of display by pressing F6 key.

Constructing Geometric Figures

AutoCAD provides various methods for constructing geometric figures

- **Drawing Circles**
- **Drawing Arcs**

Drawing Circles

The CIRCLE command offers five different options for drawing circles:

- **Center-Radius**
- **Center-Diameter**
- **3 Points**
- **2 Points**
- **Tangent, Tangent, Radius (ttr)**

Center-Radius

Draws a circle based on a center point and a radius. Invoke the CIRCLE command command:

```
Command: prompt
Choose Circle > Center, Radius
```

```
Command: circle
(ENTER)
Specify center point for circle or [3P/2P/Ttr (tan tan radius)]: 2,2
(ENTER) Command: prompt
Choose Circle > Center, Radius
```

```
Command: circle
(ENTER)
Specify radius of circle or [Diameter]: 1
(ENTER)
```

Center-Diameter

Draws a circle based on a center point and a diameter. Invoke the CIRCLE Center-Diameter option command:

```
Command: prompt
Choose Circle > Circle, Diameter
```

```
Command: circle
(ENTER)
Specify center point for circle or [3P/2P/Ttr (tan tan radius)]: 2,2
(ENTER) Command: prompt
Choose Circle > Circle, Diameter
```

```
Command: circle
(ENTER)
Specify radius of circle or [Diameter]: 2
(ENTER)
```

3 Points
Draws a circle based on three points on the circumference. Invoke the CIRCLE 3 Points option command:

```
Invoke CIRCLE 3 Points command:
Command: circle
(ENTER)
Specify center point for circle or [3P/2P/Ttr (tan tan radius)]: 3P
(ENTER)
Specify first point on circle: 2,1
(enter coordinates)
Specify second point on circle: 3,2
(enter coordinates)
Specify third point on circle: 2,3
(enter coordinates)
```

2 Points
Draws a circle based on two end points of the diameter. Invoke the CIRCLE 2 Points option command:

```
Invoke CIRCLE 2 Points command:
Command: circle
(ENTER)
Specify center point for circle or [3P/2P/Ttr (tan tan radius)]: 2P
(ENTER)
Specify first end point of circle’s diameter: 1,2
(enter coordinates)
Specify second end point of circle’s diameter: 3,2
(enter coordinates)
```

Tangent, Tangent, Radius (Ttr)
Draws a circle tangent to two objects (either lines, arcs, or circles) with a specified radius. Invoke the CIRCLE Ttr option command:

```
Invoke CIRCLE Ttr command:
Command: circle
(ENTER)
Specify center point for circle or [3P/2P/Ttr (tan tan radius)]: ttr
(ENTER)
Specify point on object for first tangent of circle:
(enter coordinates)
Specify point on object for second tangent of circle:
(enter coordinates)
Specify radius of circle: 1
(enter value)
```

Drawing Arcs
The ARC command offers eleven different options for drawing arcs:

- Three-point (3 points)
- Start, center, end (S,C,E)
- Start, center included angle (S,C,A)
- Start, center, length of chord (S,C,L)
- Start, end direction (S,E,D)
- Start, end radius (S,E,R)
- Center, start, end (C,S,E)
- Center, start, included angle (C,S,A)
- Center, start, length of chord (C,S,L)
- Continuation from line or arc (LinCont or ArcCont)

Three-point (3 points)
This option draws an arc using three specified points on the arc’s circumference. Invoke the ARC 3 points option command:

```
Invoke ARC 3 points command:
Command: arc
(ENTER)
Specify start point of arc: 1,2
(enter coordinates)
Specify second point of arc: 1,2
(enter coordinates)
Specify end point of arc: 3,2
(enter coordinates)
```

Start, center, end (S,C,E)
This option draws an arc using three specified points – start point, center point, and end point. Invoke the ARC S.C.E option command:

```
Invoke ARC S.C.E command:
Command: arc
(ENTER)
Specify start point of arc or [Center]: 1,2
(enter coordinates)
Specify second point of arc: 1,2
(enter coordinates)
Specify end point of arc: 3,2
(enter coordinates)
```
Start, center, included angle (S.C.A)

This option draws an arc using three specified points – start point, center point, and included angle. Invoke the ARC S.C.A option command:

Command: arc (ENTER)
Choose Arc > Start, Center, Angle

Specify start point of arc or [Center]: 1,2 (specify start point of arc)
Specify second point of arc or [Center]: a (ENTER)
Specify end point of arc or [Angle/Radius]: 270 (specify included angle)

Start, center, length of chord (S.C.L)

This option draws an arc using three specified points – start point, center point and length of chord. A positive value for the length of chord will cause AutoCAD to draw the minor arc; a negative value will result in the major arc. Invoke the ARC S.C.L option command:

Command: arc (ENTER)
Choose Arc > Start, Center, Length

Specify start point of arc or [Center]: 1,2 (specify start point of arc)
Specify second point of arc or [Center]: a (ENTER)
Specify center point of arc: 2,2 (specify center point of arc)
Specify end point of arc or [Angle/Radius]: 1.414 (specify length of chord)

Start, end, included angle (S.E.A)

This option draws an arc using three specified points – start point, end point, and included angle. If you specify a positive angle for the included angle, an arc is drawn counterclockwise; for a negative angle the arc is drawn clockwise. Invoke the ARC S.E.A option command:

Command: arc (ENTER)
Specify start point of arc or [Center]: 2,3 (specify start point of arc)
Specify second point of arc or [Center]: e (ENTER)
Specify end point of arc: 1,2 (specify end point of arc)
Specify center point of arc or [Angle/Radius]: a (ENTER)
Specify included angle: 90 (specify included angle)

Start, end, direction (S.E.D)

This method allows you to draw an arc between selected points by specifying a direction in which the arc will start from the selected start point. Invoke the ARC S.E.D option command:

Command: arc (ENTER)
Specify start point of arc or [Center]: 3,2 (specify start point of arc)
Specify second point of arc or [Center]: e (ENTER)
Specify end point of arc: 2,3 (specify end point of arc)
Specify center point of arc or [Angle/Radius]: d (ENTER)
Specify direction from start point: 90 (specify direction from start point)

Start, end, radius (S.E.R)

This method allows you to specify a radius after selecting the two endpoints of the arc. A positive value for the radius causes AutoCAD to draw the minor arc; a negative value results in the major arc. Invoke the ARC S.E.R option command:

Command: arc (ENTER)
Specify start point of arc or [Center]: 2,3 (specify start point of arc)
Specify second point of arc or [Center]: a (ENTER)
Specify end point of arc or [Center]: 1,2 (specify end point of arc)
Specify radius of arc: 1 (specify radius of arc)

Center, start, end (C.S.E)

This method is similar to the Start, Center, End (S.C.E) method, except in this option the beginning point is the center point of the arc rather than the start point.

Center, start, included angle (C.S.A)

This method is similar to the Start, Center, Angle (S.C.A) method, except in this option the beginning point is the center point of the arc rather than the start point.

Center, start, length of chord (C.S.L)

This method is similar to the Start, Center, Length (S.C.L) method, except that the beginning point is the center point of the arc rather than the start point.
Continuation from line or arc (LinCont or ArcCont)

You can draw an arc using this method by pressing the ENTER key as a response to the first prompt of the ARC command. After pressing the ENTER key, the only other input is to select or specify the endpoint of the arc you wish to draw. AutoCAD uses the endpoint of the previous line or arc (whichever was drawn last) as the start point of the new arc. Invoke the ARC Continue option command:

```
arc (ENTER)
```

Specify start point of arc or [Center]:

Specify end point of arc:

Object Selection

The group of objects selected for modification is called the selection set. There are several different ways of selecting the objects for modification and some of the methods include:

- Window Option
- Crossing Option
- Previous Option
- Last Option

Window Option

The Window option for selecting objects allows you to select all the objects contained completely inside a rectangular area. Establish the rectangular area by placing a point at the desired location when the SELECT “Select Objects:” prompt of the modify command is displayed, and then move the cursor toward the right of the first point when AutoCAD prompts for the opposite corner.

Crossing Option

The Crossing option for selecting objects allows you to select all the objects contained completely inside the rectangular area as well as the objects that are overlapping the rectangle. Establish the crossing area by placing a point at the desired location when the SELECT “Select Objects:” prompt of the modify command is displayed, and then move the cursor toward the left of the first point when AutoCAD prompts for the opposite corner.

Previous Option

AutoCAD remembers the most recent selection set and allows you to reselect it with the Previous option. For example, if you moved several objects and now wish to copy them elsewhere, you can invoke the COPY command and respond to the SELECT “Select objects:” prompt by entering P to select the same group objects again.

Last Option

The Last option is an easy way to select the most recently created object currently visible. Invoke the appropriate modify command and enter L for last option at the SELECT “Select Objects:” prompt.
Modify Objects

AutoCAD not only allows you to draw objects easily, but also allows you to modify the objects you have drawn. Of the many modifying commands available, the ERASE command will probably be the one used most often.

- Erasing Objects
- Getting it back

Erasing Objects

To erase objects from a drawing, invoke the ERASE command:

Command: "erase" (ENTER)

Choose Erase
Modify Menu
Choose Erase

Command: "erase" (ENTER)
Select objects: (select objects to be erased and press ENTER or SPACEBAR)

Getting it back

The OOPS command restores objects that have been unintentionally erased. Whenever the ERASE command is used, the last group of objects erased is stored in memory. The OOPS command will restore the objects; it can be used at any time. It only restores the objects erased by the most recent erase command. Invoke the OOPS command:

Command: prompt "oops" (ENTER)

Chapter End Questions

Next Class

1st CAD exercises and homework #1 are assigned today; they must be completed by 5:00 pm January 26 (usually due in one week, but this one is an exception). Don’t forget to have your assignments signed off by the TAs before turning them in! Also other CAD exercises must be signed off and place all of them (signed-off ones and completed ones, if not complete at the time of sign-off) in a folder (we call it “portfolio”).

Practice commands discussed in today’s lecture.