

MODIFY COMMAND

Objectives

At the end of this chapter, you should be able to:

- Locate and invoke the Modify command
- Erase objects from the drawing.
- Move objects from a base point to a second point of a displacement.
- Rotate objects about a basepoint.
- Enlarge or reduce objects with scale.
- Make mirror images of selected objects.
- Make rectangular and polar arrays of existing objects
- Stretch selected object.
- Trim away parts of objects at cutting edges.
- Extend objects to selected boundary edges.
- Create a fillet between two objects.
- Create a Chamfer between two objects.
- Create parallel copies of objects with offset.

INTRODUCTION


Draw commands are used to create new objects. Modify commands or edit commands are used to change existing objects or to use existing objects to create new and similar objects. The commands listed below are covered in this unit:

Erase
Move
Rotate
Trim
Scale
Mirror
Array
Stretch
Extend
Fillet
Chamfer
Offset
Divide
Measure
Change
Pedit

ERASE

The *erase* command deletes the objects you select from the drawing. Any of the object selection methods can be used to highlight the object to erase. The only other required action is press *Enter* to cause the erase to take effect.

Methods for invoking the ERASE command include:

| | | |
|----------------|---|---|
| Toolbar | : |  |
| Pull-down Menu | : | Modify > Erase |
| Command | : | Erase |

An example of the erase command prompt is as follows:


Command: **Erase**
Select Objects: **PICK** (Use any object selection method.)
Select Objects: **PICK** (Continue to select desired objects)
Select objects: **Enter** (Confirm the object selection process and causes Erase to take effect.)
Command:

If objects are erased accidentally, *U* or *Undo* command can be used immediately following the mistake to undo one step, or *oops* can be used to bring back into the drawing whatever was erased the last time *erase* was used.

MOVE

Move allows you to relocate one or more objects from the existing position in the drawing to any other position you specify. After selecting the objects to move, you must specify the base point and second point of displacement.

Methods for invoking the MOVE command include:

| | | |
|----------------|---|---|
| Toolbar | : |  |
| Pull-down menu | : | Modify > Move |
| Command | : | Move |

The prompts for the MOVE command are as follows:

Command: **move**
Select objects: (Select objects you want to move)
Select objects: (Press ENTER)
Base point or displacement: (Select a point)

Second point of displacement: (Select a point)

Example

In the following figure, you want to put a corner of the rectangle at the center of the circle.

Command: **move**

Select objects: 1 found (select the rectangle)

Select objects: (Press ENTER)

Base point or displacement: end of (Select the corner of the rectangle)

Second point of displacement: center of (Select any point on the circle circumference)

Command:

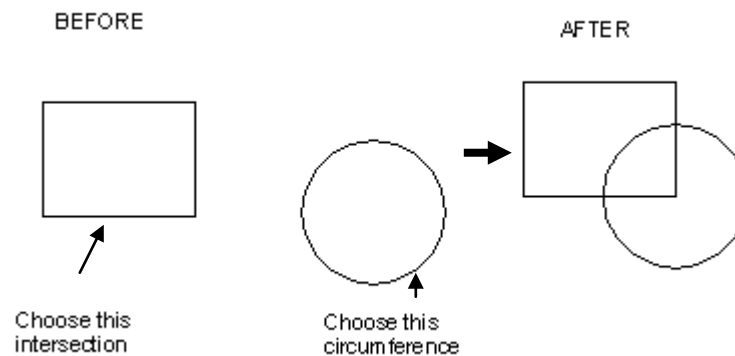



Figure 1 : Using MOVE Command to put the rectangle at the center of the circle

ROTATE

Selected object can be rotated to any position with this command. After selecting object to rotate, you select a "basepoint" (a point to rotate about) then specify an angle for rotation.

Methods for invoking the ROTATE command include:

| | | |
|----------------|---|---|
| Toolbar | : |  |
| Pull-down menu | : | Modify>rotate |
| Command | : | Rotate |

An example of the ROTATE command prompt is as follows:

Command: **rotate**

Select Object: (Specify the objects to rotate)

Base Point : end of (Specify the basepoint)

< Rotation angle>/Reference:

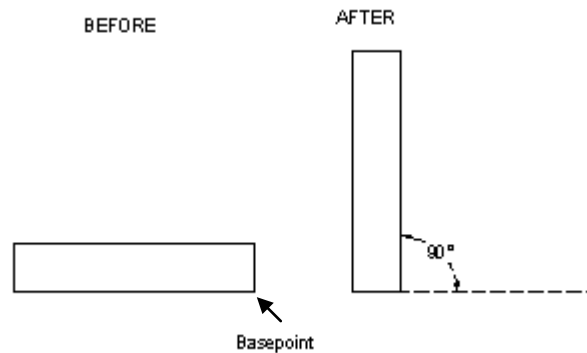


Figure 2: Using the ROTATE command to rotate the rectangle about 90° from basepoint


TRIM

The Trim command allows you to trim (shorten) the end of end object back to the intersection of another object (figure). The middle section of an object can also be trimmed between two intersection objects. There are two steps to this command;

1. PICK one or more "cutting edge" (existing object)
2. Then PICK the object or objects to trim (portion to remove)

The cutting edges are highlighted after selection. Cutting edges themselves can be trimmed if they intersect with other cutting edges, but lose their highlight when trimmed.

Methods for invoking the TRIM command include:

| | | |
|----------------|---|---|
| Toolbar | : |  |
| Pull-down menu | : | MODIFY > TRIM |
| Command | : | TRIM |

An example of the erase command prompt is as follows:

Command: **trim**

Select cutting edges: (Projmode = UCS, Edgemode = No Extend)

Select Objects: **PICK** (Select an object to use as a cutting edge)

Select Objects: **PICK**

Select Objects: **Enter**

< Select object to trim >/Project/Edge/Undo:PICK (select the end of and object to trim)

< Select object to trim >/Project/Edge/Undo: **PICK**

< Select object to trim >/Project/Edge/Undo:**Enter**

Command:

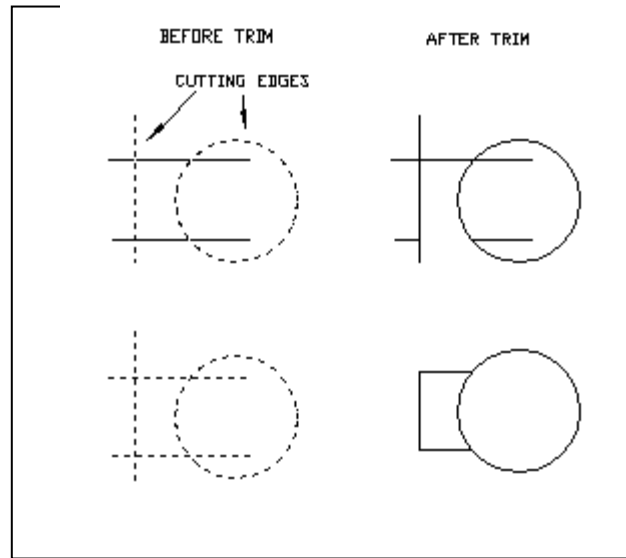



Figure 3: Using TRIM command to trim the objects.

SCALE

The scale command is used to increase or decrease the size of objects in a drawing. The scale command does not normally have any relation to plotting a drawing to scale.

Methods for invoking the SCALE command include:

| | | |
|----------------|---|---|
| Toolbar | : |  |
| Pull-down menu | : | MODIFY > SCALE |
| Command | : | SCALE |

An example of the scale command prompt is as follows:

```

Command : scale
Select Object : PICK or ( coordinates ) ( Select the object to scale )
Select Object : ENTER ( Indicates completion of the object selection )
Base point : PICK or ( coordinates ) ( Select the stationary point )
Scale factor <Reference>: PICK or ( value ) or ( coordinates ) ( Enter a value for the
scale factor or interactively scale the set of object )
Command :
```

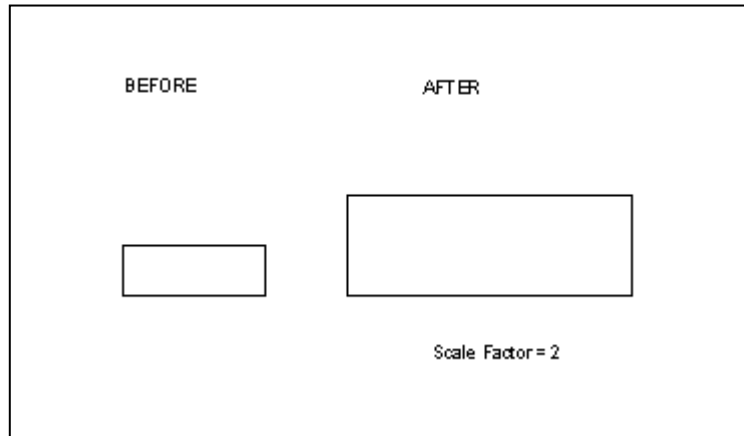



Figure 7 : Using SCALE Command to enlarge the rectangle with scale factor;2

MIRROR

This command creates a mirror image of selected existing objects. You can retain or delete the original objects ('old object'). After selecting objects, you create two points specifying a 'rubberband line', or 'mirror line', about which to mirror.

Methods for invoking the MIRROR command include:

| | | |
|----------------|---|--|
| Toolbar | : |  |
| Pull-down menu | : | MODIFY > MIRROR |
| Command | : | MIRROR |

An example of the scale command prompt is as follows:

```

Command : mirror
Select Object : PICK ( Select object or group of object to mirror )
Select Object : Enter ( Press Enter to indicate completion of object selection. )
First point of mirror line : PICK or (coordinates) (Draw first endpoint of line to
represent mirror axis by PICKing or entering coordinates)
Second point of mirror : PICK or (coordinates) (Draw second point of line by PICKing
or entering coordinates)
Delete old objects ? <N> Enter or Y ( Press Enter to yield both sets of objects or enter
Y to keep only the mirrored set.)
Command :
```

ARRAY

The array command creates either a Rectangular or Polar (circular) pattern of existing object that you select. The pattern could be created from a single or from a group of objects. Array copies a duplicate set of objects for each 'item' in the array.

Methods for invoking the ARRAY command include:

Toolbar :



Pull-down menu : **MODIFY > ARRAY**

Command : Array

Rectangular

This option creates an Array of the selection set in a pattern composed of rows and columns. The command syntax for a rectangular is given next:

Command : **Array**

Select Objects : **PICK** (Select object to be arrayed)

Select Objects : **Enter** (Indicates completion of object selection)

Rectangular or Polar array (<R>/P) : **R** (indicates rectangular)

Number of rows (---)<1> : (**value**) (enter value for number of rows)

Number of columns (III)<1> : (**value**) (enter value for number of columns)

Unit cell or distance between rows (---): (**value**) (enter a value for the distance from any of one object to the same point on an object in the adjacent

row.

Distance between columns (III) : (**value**) (enter a value for the distance from any point on one object to the same point on an object in the adjacent column.

Command :

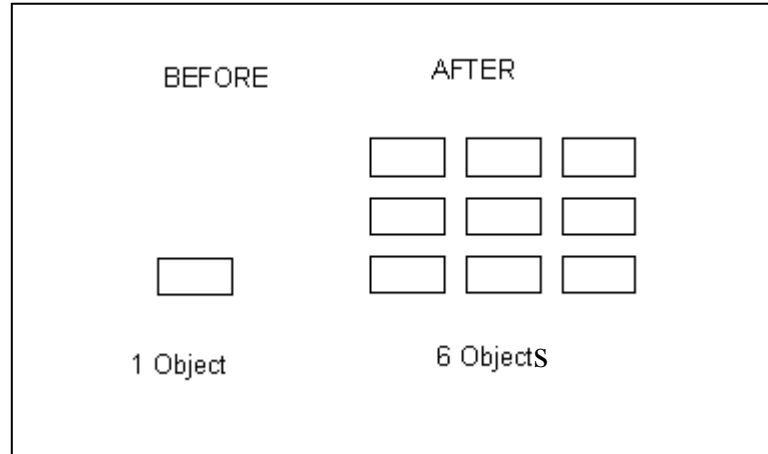


Figure 8: Using Array with Rectangular option

Polar

This option creates a circular pattern of the selection set with any number of copies or 'items'. The number of item specified includes the original selection set. You also specify the center of the array, angle to generate the array through and orientation of 'item'.

Command : **Array**

Select Object : **PICK** (select object to be arrayed)

Select Object : **Enter**

Rectangular or Polar array (<R>/P) : **P** (indicates Polar array)

Center point of array : **PICK** (select point from array to be generated around)

Number of items : (value) (enter value for number of copies including original selection set)

Angle to fill (+=ccw,-=cw) <360>: **Enter** or (value) (press enter for full circular array, enter value for less than 360 degree array; enter negative value for clockwise generation array)

Rotate object as they are copied? <Y> **Enter** or N (Press Enter for rotation of copies object about center, N for keeping objects in original orientation.)

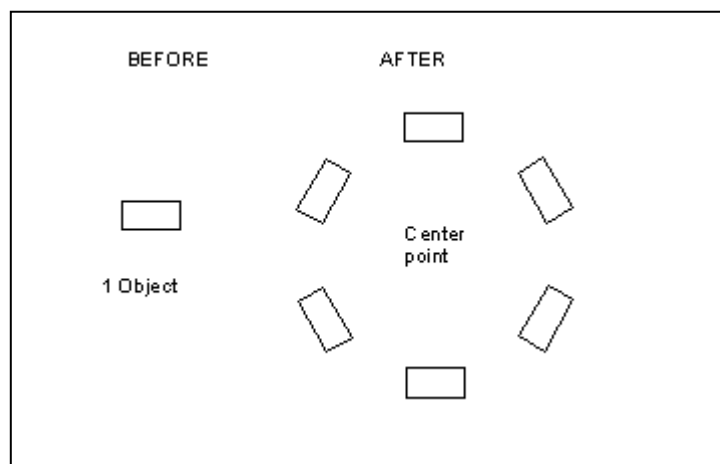



Figure 9 : Using ARRAY Command with polar option

STRETCH

Object can be made longer or shorter with STRETCH. When 'Stretched', Line and Plines become longer or shorter and Arc change radius to become longer or shorter. Circle do not stretch; rather, they move if the circle is selected within the Crossing Window.

Methods for invoking the STRETCH command include:

| | | |
|----------------|---|---|
| Toolbar | : |  |
| Pull-down menu | : | MODIFY > STRETCH |
| Command | : | STRETCH |

An example of the stretch command prompt is as follows:

Command : **STRETCH**

Select Object(s) : *to stretch by crossing-window or polygon*

Select Object :

First Corner : **PICK**

Other Corner : **PICK**

Select Object : **Enter**

Base point or displacement : **PICK** or (**coordinates**) (Select a point to use as the point to stretch from.)

Second point of displacement : **PICK** or (**coordinates**) (Select a point to use as the point to stretch to)

Command :

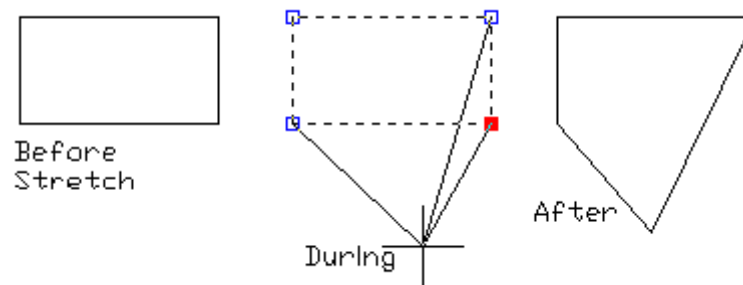



Figure 10 : Stretching the drawing

EXTEND

Extend can be thought of as the opposite of *Trim*. Objects such as *Lines*, *Arc*, and *Pline* can be extended until intersecting another object called a 'boundary edge' as in *Figure 5.18*. The command first requires selection of existing object to serve as 'boundary edge(s)', which become highlighted, then the objects to extend are selected. Objects extend until, and only if, they eventually intersects 'boundary edge'. An *Extended* object acquires a new endpoint at the boundary edge intersection.

Methods for invoking the EXTEND command include:

| | | |
|----------------|---|---|
| Toolbar | : |  |
| Pull-down menu | : | MODIFY > EXTEND |
| Command | : | Extend |

Command : **extend**

Select boundary edges : (Projmode = UCS, Edgemode = No extend)

Select Objects : **PICK**

Select Objects : **PICK**

Select Objects : **Enter**

<Select object to extend>/Project/Edge/Undo: **PICK** (Select object to extend)

<Select object to extend>/Project/Edge/Undo: **PICK**

<Select object to extend>/Project/Edge/Undo: **Enter**

Command :

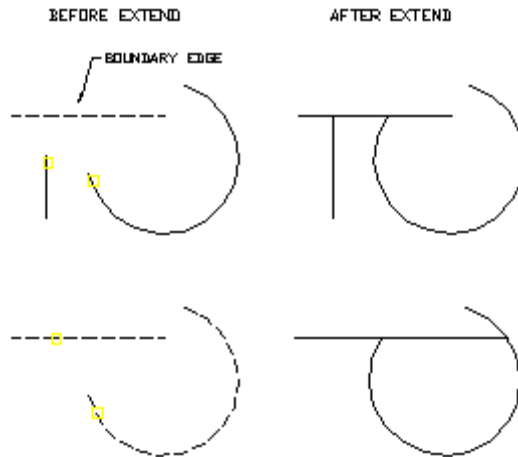


Figure 18

Edge mode/Project mode

The edge mode and Project mode switches operate identically to their function with the Trim command. Use Edge mode with the Extend option if you want a boundary edge object to be imaginarily extended.

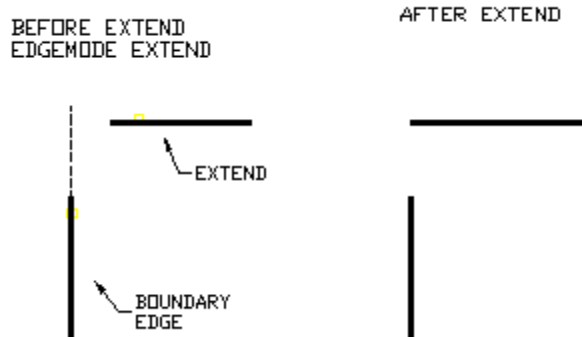


Figure 19

FILLET

The fillet command automatically rounds a sharp corner (intersection of two *Lines*, *Arcs*, *Circles*, or *Pline* vertices) with a radius. You only specify the radius and select the objects to be filleted. The objects to fillet do not have to completely intersect but can overlap. You can specify whether or not the object are automatically extended or trimmed as necessary (Figure 5.20)

Methods for invoking the FILLET command include:

Toolbar : 
 Pull-down menu : **MODIFY > FILLET**

Command : Fillet

The fillet command is used first to input the desired radius (if other than the default 0.500 value) and a second time to select the object to fillet.

Command: fillet
 (TRIM Mode) Current fillet radius = 0.5000
 Polyline/Radius/Trim/<Select first object>: r (Indicates the radius option)
 Enter fillet radius < 0.5000>: (value) or PICK (Enter a value for the desired fillet radius or select two points to interactively specify the radius)
 Command:

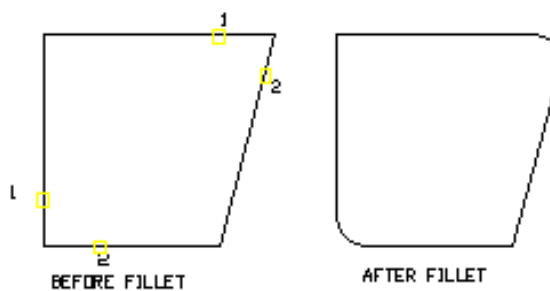


Figure 20

CHAMFER

Chamfer is a manufacturing process used to replace a sharp corner with an angled surface. In AutoCAD, *Chamfer* is commonly used to change the intersection of two *Lines* or *Plines* by adding an angled line. The *Chamfer* command is similar to fillet, but rather than rounding with a radius or “fillet”, an angled line is automatically drawn at the distances (from the existing corner) that you specify.

Methods for invoking the CHAMFER command include:

| | | |
|----------------|---|---|
| Toolbar | : |  |
| Pull-down menu | : | MODIFY > CHAMFER |
| Command | : | Chamfer |

Chamfer can be created by two methods: *Distance* (specify two distances) or *Angle* (Specify a distance and an angle).

Distance option → is used to specify the two values applied to create the chamfer. The value indicate the distances from the corner (intersection of the two line) to each chamfer endpoint (*Figure 21*). Use the chamfer command once to specify distances and again to draw the chamfer.

Command: **chamfer**

(TRIM Mode) Current chamfer Dist1 = 0.0000, Dist2 = 0.0000

Polyline/Distance/Angle/Trim/Method/<Select first line>:**d** (Indicates the distance option)

Enter first chamfer distance<0.0000>: (value) or *PICK*

Enter second chamfer distance <value of first distance>: *Enter* or *PICK*

Command:

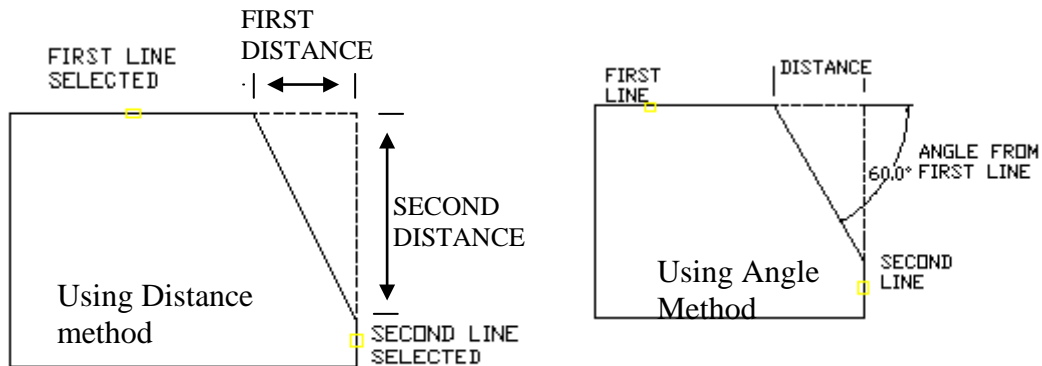



Figure 21

OFFSET

Offset creates a parallel copy of selected object. Selected object can be *Lines*, *Arcs*, *Circles*, *Plines* or other objects. Two options are available with offset; *offset a specified distance* and *offset through a specified point*.

Methods for invoking the OFFSET command include:

- Toolbar : 
- Pull-down menu : **MODIFY > OFFSET**
- Command : Offset

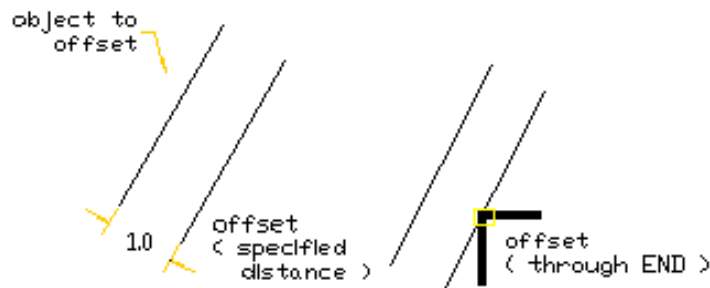


Figure 22

DIVIDE

DIVIDE marks off a specified number of equal lengths on a selected object by placing point objects or blocks along the length or perimeter of the object. Valid objects that can be divided include arcs, circles, ellipses and elliptical arcs, polylines, and splines.

Methods for invoking the DIVIDE command include:

Pull-down menu : **DRAW > POINT>DIVIDE**
Command : **Divide**

- 1 From the Draw menu, choose Point Divide.
- 2 Select a line, arc, spline, circle, ellipse, or polyline.
- 3 Enter the number of intervals you want to represent.

AutoCAD places a point at each interval on the object.

The point objects that are added to the object can be used for subsequent construction by allowing you to OSNAP to equally spaced intervals (*Nodes*). After using the **Divide** Command, the **point** objects may not be visible unless the point style is changed with the **Point Style...** dialog box (**Format** pull-down menu). A **Regen** must be invoked before the new **Point** style will be displayed. *Figure 23* below shows Points displayed at the object.

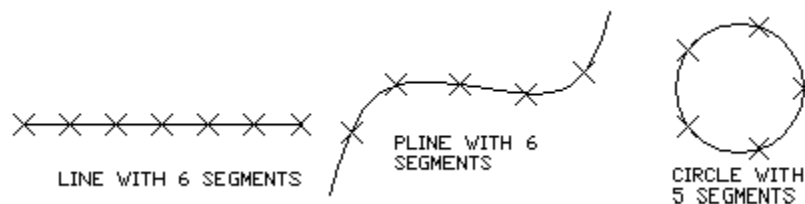


Figure 23

MEASURE

The Measure command will take an entity such as a line or arc and the measurement along it depends on the length of the segment. It similar with divide command, accomplishes this by placing a point entity at a specified distance given in the measure command.

Methods for invoking the MEASURE command include:

Pull-down menu : **DRAW > POINT>MEASURE**
Command : **Measure**

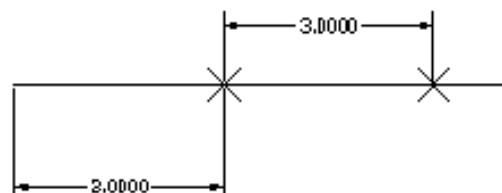


Figure 24 Using Measure to divide and measure the line with 3.0 inches length.

CHANGE

Using the change command allows the characteristics of an entity to be modified. The Change command allows changing three options: *Points*, *Properties* or *Text*

Methods for invoking the CHANGE command :

Command : **Change**

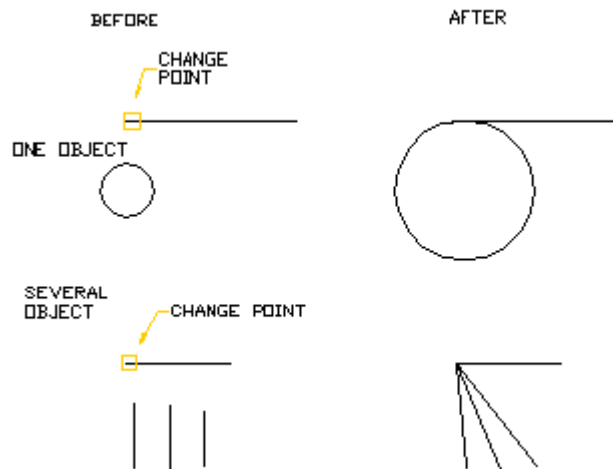


Figure 23 : Using Change Command with changing point method

This point allows changing the endpoint of an object or endpoints of several objects to one position:

Command: Change

Select Objects :**PICK**

Select Object :**Enter**

Properties/<Change point>:**PICK** (Select a point to establish as new endpoint of all objects)

PEDIT

This command provides numerous options for editing polylines (Plines). The list of options below emphasizes the great flexibility possible with polylines. The first step after invoking Pedit is to select Pline to edit.

Methods for invoking the PEDIT command include:

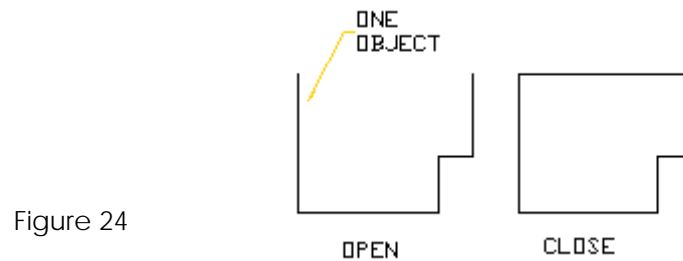
| | | |
|----------------|---|---|
| Toolbar | : |  |
| Pull-down menu | : | MODIFY > Object > Polyline |
| Command | : | Pedit. |

An example of the Pedit command prompt is as follows:

Command : **Pedit**
 Select Polyline : **PICK** (select the polyline for subsequent editing)
 Close or Open/ Join/Width/Edit Vertex/Fit/Spline/Decurve/Ltype
 gen/Undo/eXit<X>: (**option**) (Select the desired option from the screen
 menu or enter the capitalized letter for the desired option.

Close

Close connect the last segment with the first segment of an existing 'open' Pline, resulting in a 'closed' Pline. (Figure 5.24). A closed Pline is one continuous object having no specific start or endpoint, as opposed to one closed by PICKing points. A closed Pline reacts differently to the Spline option and to some commands such as Fillet, Pline option.



Open

Open removes the closing segment if the Close option was used previously. (Figure 24)

Join

This option join, or connect, any Pline, Lines, or Arc that have exact matching endpoint and add them to the selected Pline (Figure 25) Previously closed Plines cannot be joined.

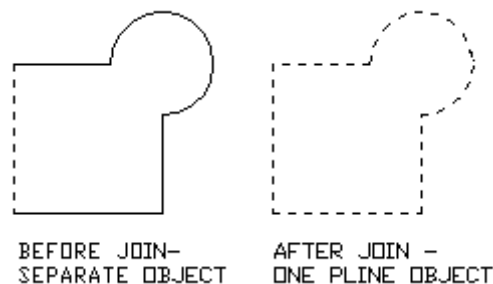


Figure 25

Width

Width allows specification of a uniform width for Pline segment. (*Figure 26*). Non-uniform width can be specified with the Edit Vertex option.

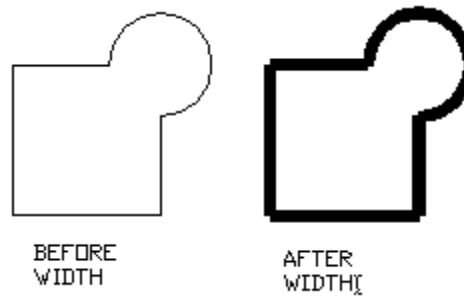


Figure 26

Another methods that available under **Pedit** Command are *Edit vertex, fit, Spline,Decurve, Ltype gen, Undo* and *Exit*. Please practice that methods