

LESSON 7

LEARNING OBJECTIVES

After completing this lesson, you will be able to:

1. Copy objects.
2. Create a Mirrored Image.
3. Add Rounded Corners.
4. Add Chamfered Corners.

Copy Multiple Copies

The **Copy** command creates a duplicate set of the objects selected. The **Copy** command is similar to the **Move** command. (Also refer to page 6-9 for an optional copy method using Drag and then Copy Here.)

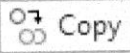
The steps required are:

1. Select the objects to be copied.
2. Select a basepoint.
3. Select a new location for the new copy.

The difference between Copy and Move commands:

The **Move** command **merely moves** the objects to a new location.
The **Copy** command **makes a copy**, and you select the location for the new copy.

1. Select the **Copy** command using one of the following:





Ribbon = Home Tab / Modify Panel /  **Copy**
or
Keyboard = CO <Enter>

The following prompt will appear on the Command Line:

    **COPY** Select objects:





2. Select the objects you want to copy and then press **<Enter>**.

The following prompt will appear on the Command Line:

    **COPY** Specify base point or [Displacement mode] <Displacement>:





3. Select a basepoint (**P1**).

The following prompt will appear on the Command Line:

    **COPY** Specify second point or [Array] <use first point as displacement>:





4. Select the new location (**P2**).

The following prompt will appear on the Command Line:

    **COPY** Specify second point or [Array Exit Undo] <Exit>:

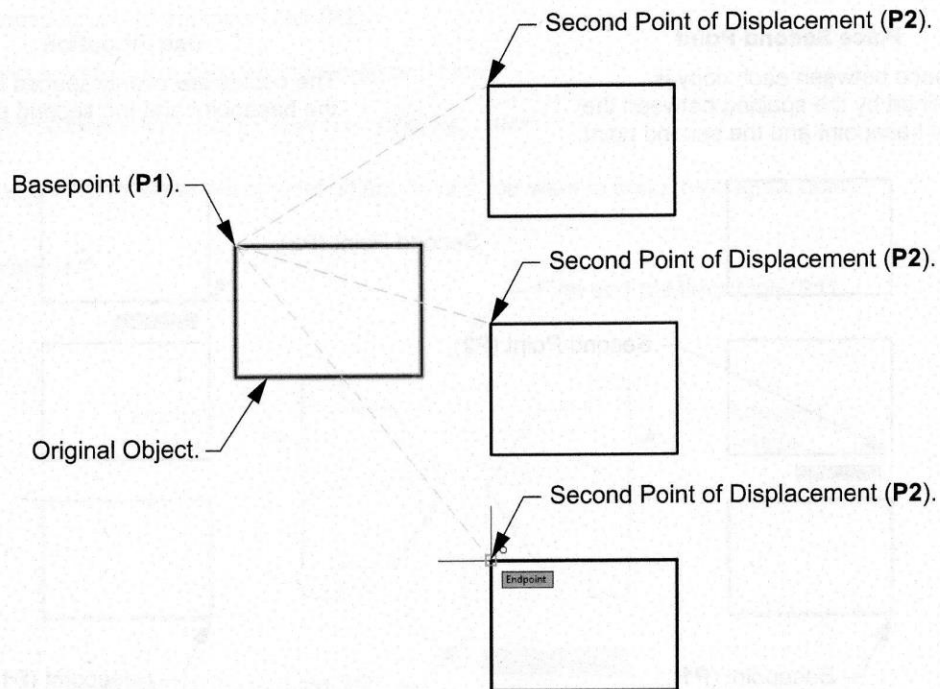
5. Select the new location (**P2**) for the next copy.

The following prompt will appear on the Command Line:

    **COPY** Specify second point or [Array Exit Undo] <Exit>:

6. Select the new location (**P2**) for the next copy, or press **<Enter>** to stop selecting more copies.

Refer to the example on the next page.






Note: The Copy command continues to make copies until you press **<Enter>** to exit.

Changing the “Mode”

You may change the “Mode” to **Single** if you prefer to have AutoCAD stop the Copy command automatically after a single copy.

After you have selected the object(s) to copy, the following prompt will appear on the Command Line:




   COPY Specify base point or [Displacement mOde] <Displacement>:

If you select the option **mOde**, you may select **Single** or **Multiple** copy mode.




Copy “Array” Option

The **Copy** command allows you to make an **Array** of copies.

After you have selected the basepoint (**P1**), the following prompt appears:




   COPY Specify second point or [Array] <use first point as displacement>:

If you select the option **Array**, the following prompt will appear on the Command Line:

   COPY Enter number of items to array:

1. Enter the number of items to Array and then press **<Enter>**.

The following prompt will appear on the Command Line:

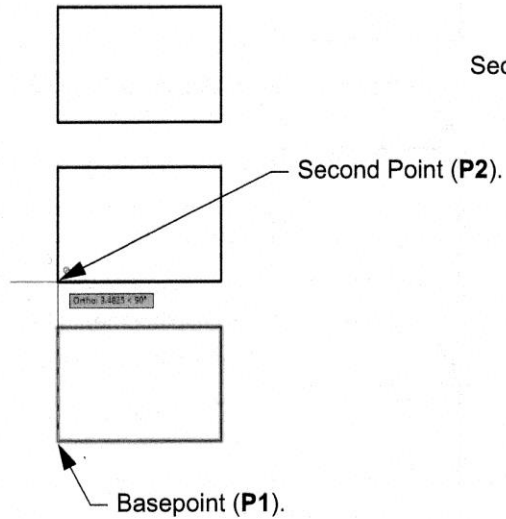
   COPY Specify second point or [Fit]:

2. Place the second point (**P2**) or select **F** and then press **<Enter>**.

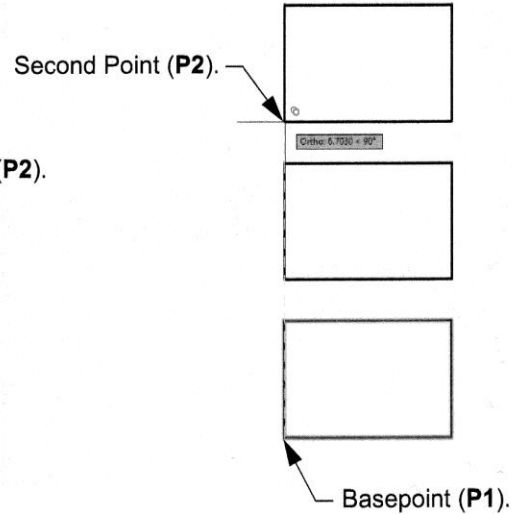
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Place Second Point

The space between each copy is determined by the spacing between the original basepoint and the second point.

**Use Fit option**

The copies are evenly spaced between the basepoint and the second point.




Note: The Array option within the Copy command is a quick method to create multiple copies. But AutoCAD has a more powerful and accurate Array command described in Lesson 13.

Mirror

The **Mirror** command allows you to make a mirrored image of any objects you select. You can use this command for creating right- or left-hand parts, or to draw half of a symmetrical object and mirror it to save drawing time.

1. Select the **Mirror** command using one of the following:




Ribbon = Home Tab / Modify Panel /  **Mirror**
 or
Keyboard = MI <Enter>

The following prompt will appear on the Command Line:

   MIRROR Select objects:




2. Select the objects to be mirrored and then press **<Enter>**.

The following prompt will appear on the Command Line:

   MIRROR Specify first point of mirror line:

3. Select the first end of the mirror line (**P1**).

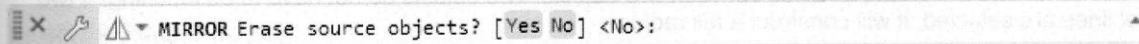
The following prompt will appear on the Command Line:

   MIRROR Specify second point of mirror line:

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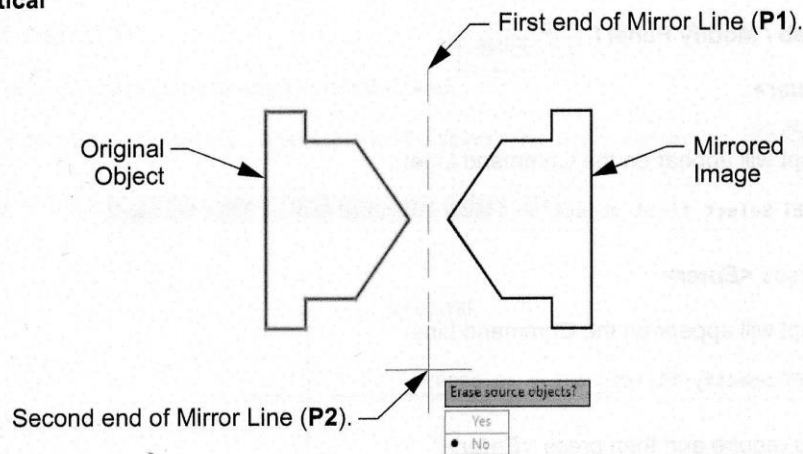
- Select the second end of the mirror line (**P2**).

The following prompt will appear on the Command Line:



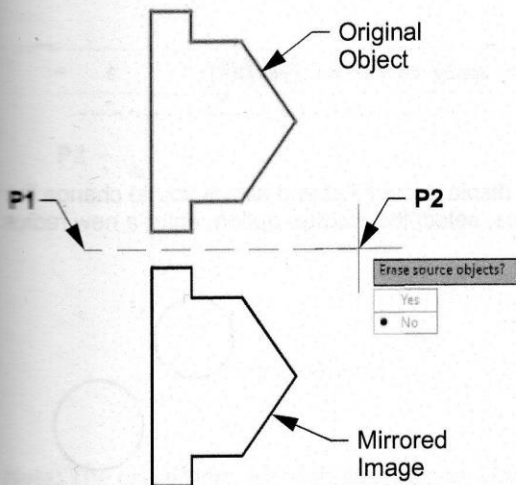
- Select **Y** if you want to erase the original object, or **N** if you want to keep the original object.

Mirror Line "Vertical"

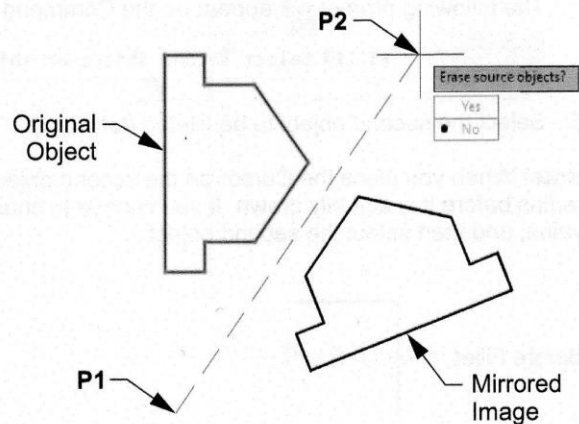


Note: The placement of the "Mirror Line" is important. You may make a mirrored copy **horizontally, vertically, or on an angle.**

Mirror Line "Horizontal"



Mirror Line "Angled"



How to control text when using the Mirror command

(Do the following **before** you use the Mirror command.)

- At the Command Line type: **mirrtext** and then press **<Enter>**.
- If you **want** the text to mirror (reverse reading), type: **1** and then press **<Enter>**.
If you **do not want** the text to mirror, type: **0** and then press **<Enter>**.

MIRRTEXT SETTING = 1

MIRRTEXT SETTING = 0

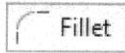
Fillet

The **Fillet** command will create a radius between two objects. The objects do not have to be touching. If two parallel lines are selected, it will construct a full radius.

Radius a corner

1. Select the **Fillet** command using one of the following:

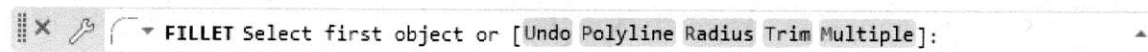
Ribbon = Home Tab / Modify Panel /



or

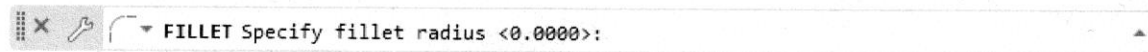
Keyboard = F <Enter>

The following prompt will appear on the Command Line:



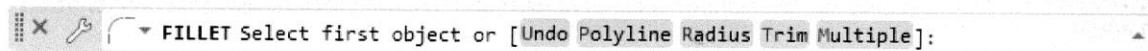
2. Type **R** and then press **<Enter>**.

The following prompt will appear on the Command Line:



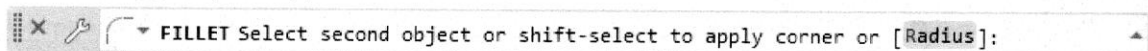
3. Type the radius you require and then press **<Enter>**.

The following prompt will appear on the Command Line:



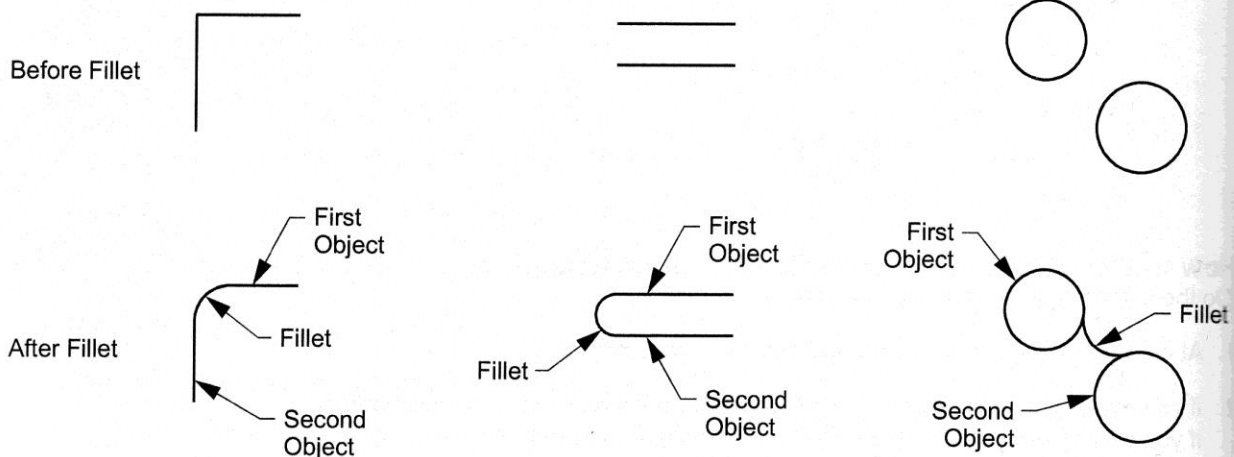
4. Select the first object to be filleted (left click).

The following prompt will appear on the Command Line:



5. Select the second object to be filleted (left click).

Note: When you place the Cursor on the second object, AutoCAD displays the Fillet and allows you to change the radius before it is actually drawn. If you choose to change the radius, select the **Radius** option, enter a new radius value, and then select the second object.



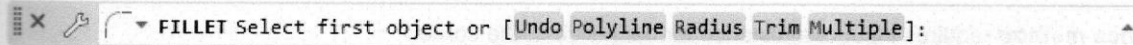
Continued on the next page...

The **Fillet** command may also be used to create a square corner.

Square corner

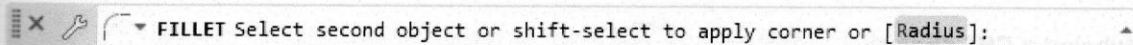
1. Select the **Fillet** command.

The following prompt will appear on the Command Line:

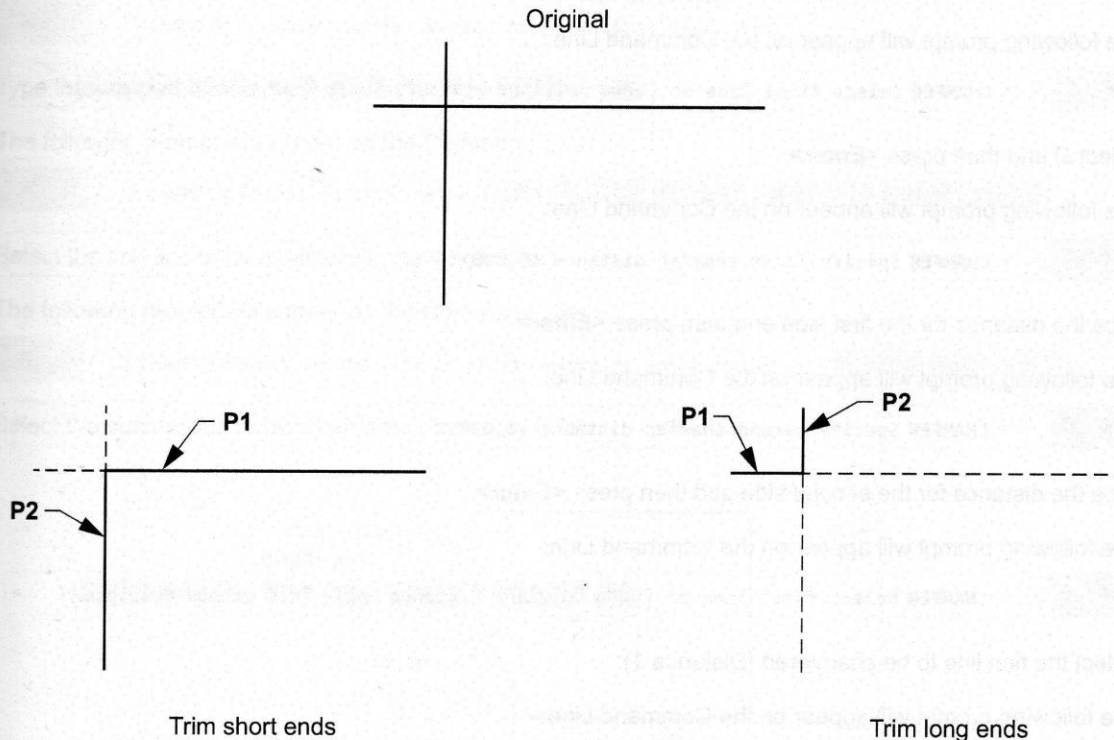


2. Select the first object (**P1**).

The following prompt will appear on the Command Line:



3. Hold down the **<Shift>** key while selecting the second object (**P2**).



Note: The corner trim direction depends on which end of the object you select. Select the ends that you want to keep.

Options:

Polyline: This option allows you to fillet all intersections of a polyline in one operation, such as all four corners of a rectangle.

Trim: This option controls whether the original lines are trimmed to the end of the arc or remain the original length. (Set to Trim or No trim.)

Multiple: Repeats the fillet command until you press **<Enter>** or the **<Esc>** key.

Chamfer

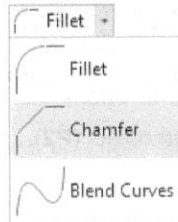
The **Chamfer** command allows you to create a chamfered corner on two lines. There are two methods: **Distance** and **Angle**.

Distance method

Distance method requires input of a distance for each side of the corner.

1. Select the **Chamfer** command using one of the following:

Ribbon = Home Tab / Modify Panel /
OR
Keyboard = CHA <Enter>



The following prompt will appear on the Command Line:

CHAMFER Select first line or [Undo Polyline Distance Angle Trim mEthod Multiple]:

2. Select **D** and then press **<Enter>**.

The following prompt will appear on the Command Line:

CHAMFER Specify first chamfer distance <0.000>:

3. Type the distance for the first side and then press **<Enter>**.

The following prompt will appear on the Command Line:

CHAMFER Specify second chamfer distance <0.000>:

4. Type the distance for the second side and then press **<Enter>**.

The following prompt will appear on the Command Line:

CHAMFER Select first line or [Undo Polyline Distance Angle Trim mEthod Multiple]:

5. Select the first line to be chamfered (Distance 1).

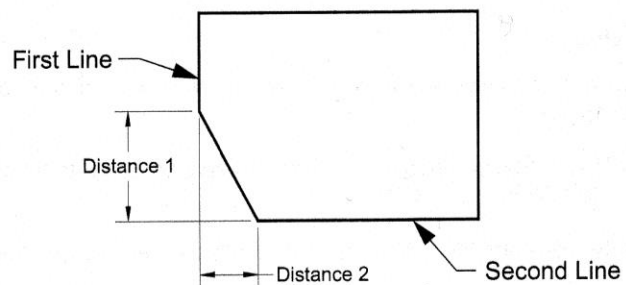
The following prompt will appear on the Command Line:

CHAMFER Select second line or shift-select to apply corner or [Distance Angle Method]:

6. Select the second line to be chamfered (Distance 2).

Note: When you place the cursor on the second line, AutoCAD displays the chamfer and allows you to change the distances before it is actually drawn.

If you choose to change the distance, select the Distance option, enter new distance values, and then select the second line.



Angle method

Angle method requires input for the length of the line and an angle.

1. Select the **Chamfer** command.

The following prompt will appear on the Command Line:

CHAMFER Select first line or [Undo Polyline Distance Angle Trim mEthod Multiple]:

2. Select **A** and then press **<Enter>**.

The following prompt will appear on the Command Line:

CHAMFER Specify chamfer length on the first line <0.000>:

3. Type the chamfer length and then press **<Enter>**.

The following prompt will appear on the Command Line:

CHAMFER Specify chamfer angle from the first line <0>:

4. Type the angle and then press **<Enter>**.

The following prompt will appear on the Command Line:

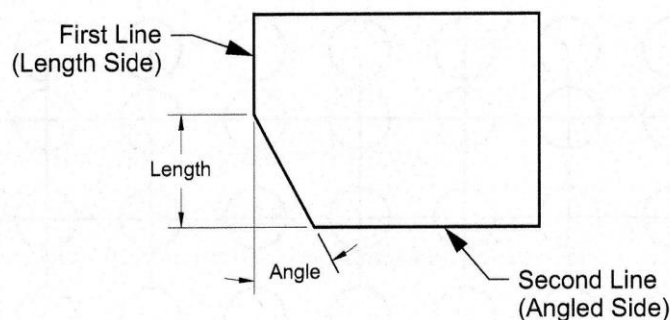
CHAMFER Select first line or [Undo Polyline Distance Angle Trim mEthod Multiple]:

5. Select the first line to be chamfered (the length side).

The following prompt will appear on the Command Line:

CHAMFER Select second line or shift-select to apply corner or [Distance Angle Method]:

6. Select the second line to be chamfered (the angle side).



Options:

Polyline: This option allows you to chamfer all intersections of a polyline in one operation, such as all four corners of a rectangle.

Trim: This option controls whether the original lines are trimmed or remain after the corners are chamfered. (Set to Trim or No trim.)

mEthod: Allows you to switch between **Distance** and **Angle** method. The distance or angle must have been set previously.

Multiple: Repeats the Chamfer command until you press **<Enter>** or the **<Esc>** key.