# AUTOCAD

## Civil Engineering Drawing & Graphics
## (BSc Civil Engineering 2010)

# *Editing Commands*

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# Editing Command

**Move, Copy, and Stretch:**

**Base points and displacements:**

The MOVE, COPY, and STRETCH commands all require that you specify how far and in what direction you want the objects moved, copied, or stretched.

**MOVE COMMAND:**

The procedure to use move command is stated as:

**1. Press Esc to make sure that no command is active and no objects are selected.**

**2. Click the Move button on the Modify toolbar.**

**3. Select one or more objects.**

**4. Press Enter when you're finished selecting objects.**

**5. Specify a base point by clicking a point or typing coordinates. (**Specify a base point somewhere on or near the object(s) that you're moving. You can use an object snap mode to choose a point exactly on one of the objects.)

# Editing Command

**MOVE COMMAND:**
**6.	Specify the second point by clicking a point or typing coordinates (**These are common precision techniques for specifying the second point:
•use an object snap mode to pick a second point exactly on another object in the drawing.
•Type a relative or polar coordinate. For example, if you type **@6,2, AutoCAD moves the objects 6 units to the right and 2 units up. If you type @3<45, AutoCAD moves the objects 3 units at an angle of 45 degrees.)**

# Editing Command

## COPY COMMAND:

• The COPY command works almost identically to the MOVE command, except that AutoCAD leaves the selected objects in place and moves new copies of them to the new location. The COPY command creates multiple copies by default. If you want only one copy, press Enter after placing it in the drawing.

• The COPY command includes an Undo option with which you can roll back multiple copies within a single COPY operation.

• **Copy between drawings:** You can't copy objects from one drawing to another with the COPY command. Instead, you use the COPYCLIP command, together with its companion command, PASTECLIP.

# Editing Command

## STRETCH COMMAND:

STRETCH can make lines longer or shorter, depending on your crossing selection box and displacement vector. In other words, the STRETCH command really combines stretching and compressing.
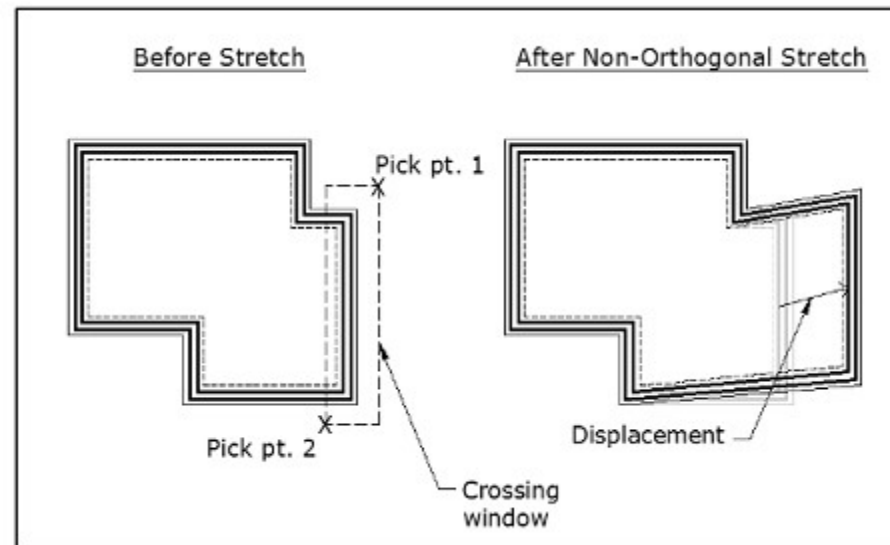


FIGURE: Strange stretching phenomena

# Editing Command

**ROTATE COMMAND:**

The ROTATE command swings one or more objects around a point that you specify. Follow these steps to use the ROTATE command:

**1. Press Esc to make sure that no command is active and no objects are selected.**

**2. Click the Rotate button on the Modify toolbar.**

**3. Select one or more objects and then press Enter to end object selection.**

**4. Specify a base point by clicking a point or typing coordinates.**

**5. Specify a rotation angle by typing an angle measurement and pressing Enter, or just press Enter to accept the default value shown in angle brackets. (**After you specify the rotation angle by typing or picking, AutoCAD rotates the objects into their new position. The ROTATE command's copy option makes a rotated copy while leaving the source object in place).

# Editing Command

**OFFSET:**

Offset command is use to create concentric circles, curve and parallel lines.

**ARRAY COMMAND:**

The ARRAY command is like a supercharged COPY: You use it to create a rectangular grid of objects at regular X and Y spacings or a polar wheel of objects at a regular angular spacing. For example, you can use rectangular arrays to populate an auditorium with chairs or a polar array to draw bicycle spokes.

The following is the procedure to use array command

**1. Press Esc to make sure that no command is active and no objects are selected.**

**2. Click the Array button on the Modify toolbar.**

# Editing Command

**ARRAY COMMAND:**
**3. Click the Select Objects button and then select one or more objects. Press Enter to end object selection and return to the Array dialog box.**
**4. Make sure that the Rectangular Array radio button is selected.**
**5. Fill in the five text boxes: Rows, Columns, Row Offset, Column Offset, and Angle Of Array**
**6 . Click the Preview button.**
**7. Click the Accept button if you're satisfied with the array or the**
**Modify button if you want to change the array parameters.**

# Editing Command

**SLICING & SPLICING COMMAND:**

The commands in this section — TRIM (TR), EXTEND (EX), BREAK (BR), FILLET (F), CHAMFER (CHA), and JOIN (J) — are useful for shortening and lengthening objects, for breaking them in two, and for putting them back together again.

**Trim and Extend:**

TRIM and EXTEND are the twin commands for making lines, polylines, and arcs shorter and longer. The two commands and their prompts are almost identical, so the following steps cover both. We show the prompts for the TRIM command; the EXTEND prompts are similar:

**1. Click the Trim or Extend button on the Modify toolbar.**

**2. Press Enter to select all drawing objects to act as the knife for trimming objects or the wall to which objects will be extended, or select individual objects by picking them. Press Enter to end object selection.**

**3. Select a single object to trim or extend. Choose the portion of the object that you want AutoCAD to trim away or the end of the object that's closer to the extend-to boundary.**

# Editing Command

## TRIM & EXTEND COMMAND:

TRIM and EXTEND normally allow you to select only one object at a time for trimming or extending. An exception is that you can type **F and press Enter to use the Fence object selection mode. Fence is useful for trimming or extending a large group of objects in one fell swoop.**
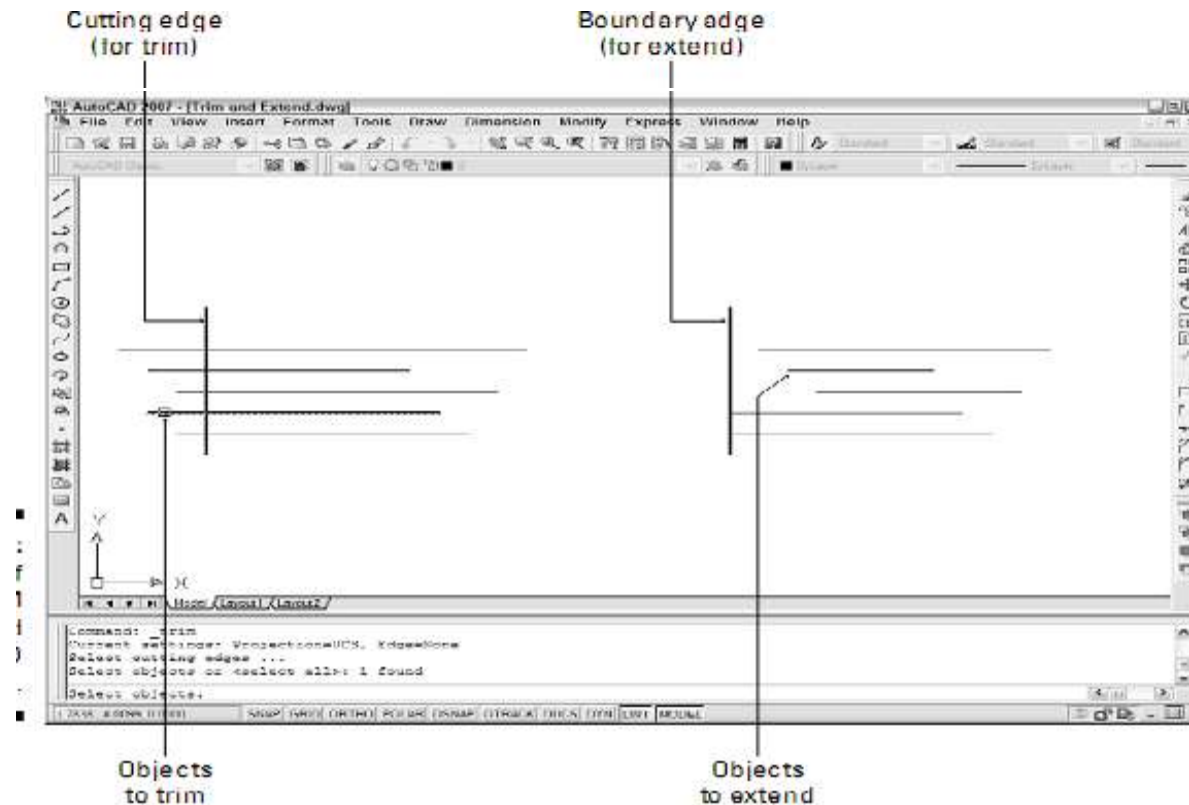
# Editing Command

## CHAMFER & FILLET:

Whereas TRIM, EXTEND, and BREAK alter one object at a time, the FILLET and CHAMFER commands require a pair of objects. FILLET creates a curved corner between two lines, whereas CHAMFER creates an angled, straight corner. (In case you wonder
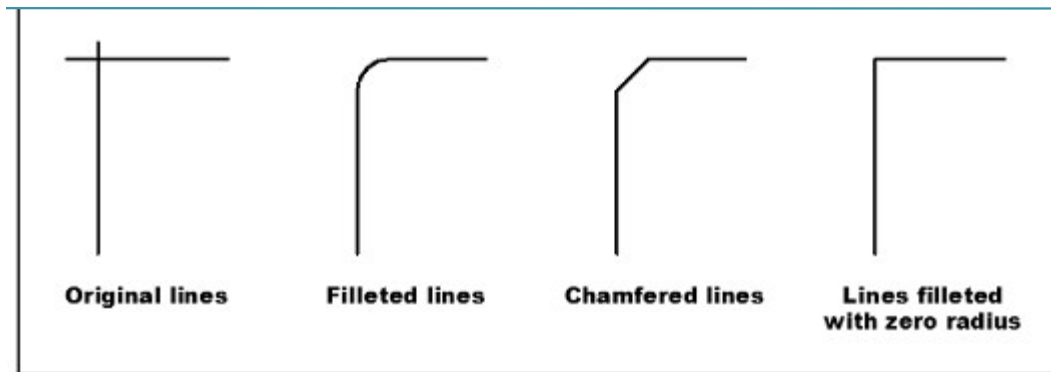


FIGURE: Cleaning up those corners with fillet & chamfer

# Editing Command

**<u>Chamfer & fillet:</u>**

The following steps describe how to use the FILLET command. The
CHAMFER command works similarly except that, instead of specifying a fillet
radius, you specify either two chamfer distances or a chamfer length and
angle.

**1. Click the Fillet button on the Modify toolbar.**
**2. Type R and press Enter to set the fillet radius.**
**3. Type a fillet radius and press Enter.**
**4. Select the first line of the pair that you want to fillet.**
**5. Select the second line of the pair that you want to fillet.**

**Note:**

You can fillet two lines and specify a radius of zero to make them meet at a
point. If you have lots of lines to fillet, whether with a
zero or nonzero radius, use the FILLET command's Multiple option to speed
the process.
Holding down the Shift key before picking the second line automatically gives
you a clean intersection, the same as if you'd explicitly set the fillet radius to 0.
The CHAMFER command has
the same Shift-select option.