Chapter 10
Edit Commands
Move Command 10.1

1. Choose Modify, Move.
   or

2. Click the Move icon.
   or

3. Type MOVE at the command prompt
   Command: MOVE or M

4. Pick Objects to move
   Select objects: (select)

5. Pick A point to move from
   Base point or displacement: (pick point)

6. Pick A point to move to
   Second point of displacement: (pick point)

**TIP:**
To move an object a specified distance, type a distance at the second point of displacement prompt: @1<0
Copy Command 10.2

1. **Choose** Modify, Copy.
   or

2. **Click** the Copy icon.
   or

3. **Type** COPY at the command prompt.
   
   Command: **COPY or CP**

4. **Pick** Objects to copy.
   
   Select objects: (select)

5. **Pick** A point to move from.
   
   Base point or displacement/Multiple: (pick point).

6. **Pick** A point to copy to.
   
   Second point of displacement: (pick point)
   or

7. **Type** A point to copy to.
   
   Second point of displacement: @ 1<0

**TIP:**

- To copy many objects in the same copy command, type M for Multiple at the "Base point or displacement/Multiple" option.
Previous Selection 10.3

Places selected objects in the Previous selection set

1. **Choose**  Modify, Move.
   
   or

2. **Click**  the Move icon.
   
   or

3. **Type**  MOVE at the command prompt.
   
   Command: MOVE or M

4. **Pick**  Objects to move.
   
   Select objects: (P)

**TIP:**

AutoCAD requires that objects be selected in order to be processed. The Select Objects prompt occurs after many commands, including the SELECT command itself.
Offset Command 10.4

Offset Distance

To offset a specified distance:

1. **Choose** Modify, Offset.
   or
2. **Choose** the Offset icon.
   or
3. **Type** OFFSET at the command prompt.
   Command: **OFFSET** or **O**
4. **Type** The distance to offset.
   Offset distance or <Through point>: *(number)*
5. **Pick** The object to offset.
   Select object to offset: *(select object)*
6. **Pick** A side to offset object to.
   Side to offset: *(pick side)*
7. **Pick** Another object to offset
   Select object to offset: *(pick side)*
   or
1. **Press** Enter to end the command.

*Offsetting objects by specifying a distance*
Offset Through Point

To offset through point:

1. **Type** 
   OFFSET at the command prompt
   
   Command: **OFFSET**

2. **Type** 
   T to specify a through point
   
   Offset distance or <Through point>: (T)

3. **Pick** 
   A point to offset through (HINT: use object snaps) Select object to offset: *(pick)*
   
   Through point: *(select object)*

*Offset through a point*
EXTEND 10.5

1. **Choose** Modify, Extend.
   
   or

2. **Click** the Extend icon.
   
   or

3. **Type** EXTEND at the command prompt
   
   Command: EXTEND
   
   Select boundary edge(s)...

4. **Pick** The BOUNDARY edge to extend to
   
   Select objects: (select)

5. **Press** ENTER to accept the boundary edge
   
   Select objects: (press enter)

6. **Pick** The objects to extend
   
   <Select object to extend> / Project / Edge / Undo: Select an object, enter an option, or press enter: (select)

7. **Press** ENTER when you are done choosing objects

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**TIP:**

- Use the object selection option FENCE to choose multiple objects
TRIM 10.6

The TRIM command allows you to trim objects in a drawing so they end precisely at a cutting edge defined by one or more other objects in the drawing.

1. **Choose**  Modify, Trim.
   
   or
   
2. **Click**  the Trim icon.
   
3. **Type**  TRIM at the command prompt
   
   Command: **TRIM**
   
   Select cutting edge(s)...

4. **Pick**  The CUTTING edge to extend to
   
   Select objects: (select)

5. **Press**  ENTER to accept the cutting edge
   
   Select objects: (press enter)

6. **Pick**  Objects to trim
   
   <Select object to trim> / Project / Edge / Undo:
   
   Select an object, enter an option, or press enter

7. **Press**  ENTER when you are done choosing objects
   
   Select object to trim/Undo: (press enter)

   **TIP:** Hold the SHIFT key to interactively extend instead of trim.
Edgemode

Controls how the TRIM and EXTEND commands determine cutting and boundary edges.

0 Uses the selected edge without an extension.
1 Extends the selected edge to its natural boundary.
MIRROR 10.7

1. **Choose** Modify, Mirror.
   or
2. **Click** the Mirror icon.
   or
3. **Type** MIRROR at the command prompt.
   Command: **MIRROR**
4. **Pick** Objects to mirror.
   Select objects: *(select)*
5. **Pick** First point of mirror line: *(point)*
6. **Pick** Second point: *(point)*
7. **Type** Yes to delete the original objects and No to keep them.
   Delete old objects? **Y** or **N**
**Mirrtext**

Mirror reflects (mirrors) text if 1, retains text direction if 0.

1. **Type** MIRRTTEXT at the command prompt.
   
   Command: **MIRRTTEXT**

2. **Type** 1 to reflect the text and 0 to retain the text.
   
   Current value <0> New value: **1** or **0**

![Diagram showing text before and after mirroring with MIRRTTEXT settings ON and OFF](image-url)
1. Choose Modify, Rotate.
   
   or

2. Click the Modify icon.
   
   or

3. Type ROTATE at the command prompt
   
   Command: ROTATE

4. Pick Objects to rotate:
   
   Select objects: (select)

5. Pick A pivot point to rotate around
   
   Base point: (point)

6. Type A rotation angle<Rotation angle>/Reference: (number)
   
   or

7. Pick A rotation angle<Rotation angle>/Reference: (point)
Reference Angle Rotation

A positive angle causes counterclockwise rotation, and a negative angle produces clockwise rotation. If you respond to the last prompt with r, you can specify the current rotation and the new rotation you want. AutoCAD prompts:

1. **Type** R for a rotation angle<Rotation angle>/Reference: (R)

2. **Choose** An existing rotation angle Rotation angle:
   (number or points)

3. **Choose** A new rotation angle New angle:
   (number or points)

**TIP:**

You can show AutoCAD the reference angle (by pointing to the two endpoints of a line to be rotated), and then specify the new angle. You can specify the new angle by pointing or by dragging the object.
SCALE 10.9

1. **Choose** Modify, Scale.
   
   or

2. **Click** the Scale icon.
   
   or

3. **Type** SCALE at the command prompt
   
   Command: `SCALE`
   
   Select objects: *(select objects)*

4. **Pick** A pivot point to scale about Base point: *(point)*

5. **Type** A rotation angle<Scale factor>/Reference: *(number)*
   
   or

6. **Pick** A scale factor<Scale factor>/Reference: *(point)*
   
   Scale factor/Reference: *(points)*
Scale by Specifying Length

You can show AutoCAD the reference length (by pointing to the two endpoints of a line to be scaled), and then specify the new length. You can specify the new length by pointing, or by dragging the object.

1. **Type** R to define a reference length
   Scale factor/Reference: (R)
2. **Choose** A reference scale factor
   Reference length : (number or points)
3. **Choose** A new scale factor
   New length: (number or points)