AutoCad
Basic Tutorial
Launching AutoCad

1. Start
2. 3D and CAD
3. AutoCad
Typing Commands

Typing a Command

All AutoCAD commands can be typed in at the command line. Many commands also have one or two letter aliases that can also be typed as shortcuts to the commands.

1. Type the desired command at the command prompt.
   Command: **LINE**

   or

2. Type the command's alias. Command: **L**

3. Press **ENTER/Space** to end.

4. Type an option at the command prompt.

   **TIP:** Many AutoCAD commands require you to press ENTER to complete the command. You know you are no longer in an AutoCAD command when you see a blank command line.

Reissuing the Last Command

The last used AutoCAD command can be re-entered by one of the following three methods of ENTER. The ENTER key on the keyboard will always act as ENTER, the SPACEBAR and RIGHT MOUSE will act as enter most of the time (exceptions include placing TEXT).

1. Press the **ENTER** key on the keyboard

   or

2. Press the **Space bar** on the keyboard.

   or

3. Click the **right** mouse button.
Pointing Device (Mouse)

AutoCAD uses either a mouse or digitizing tablet to select objects in a drawing.

**Left Mouse Button**

Used to pick or select objects

1. Click the left mouse button to select an object area in the drawing.
2. Press **ESC** twice to deselect an object (or to cancel a command).

**Right Mouse Button**

Used to enter a command, repeat last command, or access shortcut menus.

1. Click the right mouse button.

**TIPS:**

- **SHIFT** + the right mouse button brings up the object snap menus.
- Various screen locations for the mouse brings up different menus.
5.2 PAN

Shifts the location of a view.

1. **Choose** View, Pan.
   or

2. **Click** the Pan icon.
   or

3. **Type** PAN from the command prompt.

Command: **PAN** or **P**

**TIPS:**

- While in the PAN command, click with the right mouse button to see the following menu.

- Panning can also be done by using the window scroll bars
5.1 ZOOM

Increases or decreases the apparent size of objects in the current viewport

1. **Choose** View, Zoom.
   or
2. **Click** a Zoom icon.
   or
3. **Type** ZOOM at the command prompt.
   Command: **Zoom** or **Z**
4. **Type** One of the following zoom options:

   The following are basic zoom options:
   - **All** Places entire drawing (all visible layers) on display at once. Forces a regeneration.
   - **Extents** Displays current drawing content as large as possible.
   - **Previous** Restores previous view.
   - **Window** Designates rectangular area to be drawn as large as possible.
   - **Number** Magnification relative to ZOOM All display
   - **Number X** Magnification relative to current display (1X)
   - **Center** Specifies center point and new display height.
   - **Dynamic** Permits you to pan a box representing the viewing screen around the entire generated portion of the drawing and enlarge or shrink it.

**TIPS:**

- While in the ZOOM command, click with the right mouse button to see the menu to the right.
2.2 Creating a New Drawing

NEW Command

Creates a new drawing file.


   or

2. Press CTRL + N

   or

3. Click the New icon.

   or

4. Type NEW at the Command prompt.

   Command: **NEW**

5. Choose One of the options for creating a new drawing.

6. Click The OK button.

7. Save the drawing as another name.

**TIP:**

New drawings can also be created from Template Files.
1.11 Undo and Redo

Reverses the last action.

1. Choose **Edit, Undo**.
   
   or

2. Click the Undo icon.
   
   or

3. Press **CTRL + Z**.

4. Type U at the command prompt to undo the last command.
   
   **Command:** U

---

Redo

Reverses the effects of a single UNDO or U command.

1. Choose **Edit, Redo**.
   
   or

2. Click the Redo icon.
   
   or

3. Type REDO at the command prompt to redo the last undo command.
   
   **Command:** REDO

**TIPS:**

- UNDO has no effect on some commands and system variables, including those that open, close, or save a window or a drawing, display information, change the graphics display, regenerate the drawing, or export the drawing in a different format.

- REDO must immediately follow the U or UNDO command.
1.12 Function Keys and Accelerator Keys

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<td>CTRL+X</td>
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<tr>
<td>CTRL+Y</td>
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<td>CTRL+Z</td>
<td>Reverses last action</td>
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<tr>
<td>CTRL+I</td>
<td>Cancels current command</td>
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<tr>
<td>CTRL+A</td>
<td>Cancels current command</td>
</tr>
<tr>
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<td>Moves to the next layout tab to the left of the current tab</td>
</tr>
<tr>
<td>CTRL+PAGE DOWN</td>
<td>Moves to the next layout tab to the right of the current tab</td>
</tr>
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</table>
2.1 Open Existing Drawings

1. **Choose** File, OPEN.
   - or
2. **Press** CTRL + O.
   - or
3. **Click** the OPEN icon.
   - or
4. **Type** OPEN at the command prompt.
   - Command: **OPEN**
5. **Press** ENTER
6. **Double Click** the desired directory to find the drawing to open.
7. **Click** the drawing name to open.
8. **Click** The OK button.

-TIP: Preview shows a bitmap image of the drawing selected. This image is the view that was last saved in the drawing. It will not show a preview of drawings saved before R13 AutoCAD.
Quick Save

The QSAVE command is equivalent to clicking Save on the File menu.

If the drawing is named, AutoCAD saves the drawing using the file format specified on the Open and Save tab of the Options dialog box and does not request a file name. If the drawing is unnamed, AutoCAD displays the Save Drawing As dialog box (see SAVEAS) and saves the drawing with the file name and format you specify.

1. Press CTRL + S.
   or
2. Click the Save icon.
   or
3. Type QSAVE at the command prompt,
   Command:QSAVE

TIPS:

Drawings can be saved as different versions of AutoCAD (e.g. R13, R14, R 2000, etc.)

AutoSave settings under Tools, Options…
Useful Commands
7.1 Running Object Snaps

An object snap mode specifies a snap point at an exact location on an object. OSNAP specifies running object snap modes, which remain active until you turn them off.

1. **Choose** Tools, Drafting Settings...
   or
2. **Type** DDOSNAP at the command prompt
   Command: **DDOSNAP**
   or
3. **Click** OSNAP on the Status Bar.
4. **Right Click** the Object Snap TAB.
5. **Choose** an object snap to turn ON/OFF from the dialog box.

![Drafting Settings dialog box](image)

- **Snap and Grid**
- **Polar Tracking**
- **Object Snap**
- **Dynamic Input**
- **Quick Properties**

**Object Snap modes**
- Endpoint
- Insertion
- Midpoint
- Perpendicular
- Center
- Tangent
- Node
- Nearest
- Quadrant
- Apparent intersection
- Intersection
- Parallel
- Extension

To track from an Osnap point, pause over the point while in a command. A tracking vector appears when you move the cursor. To stop tracking, pause over the point again.
UNITS Command 8.5

1. Choose Format, Units...
or
2. Type DDUNITS at the command prompt.
   Command: DDUNITS or UN
3. Choose a units and angle setting.
4. Choose a precision setting.

![Drawing Units dialog box](image)

- Length
  - Type: Decimal, Architectural, Decimal, Engineering, Fractional, Scientific

- Angle
  - Type: Decimal Degrees
  - Precision: 0

- Insertion Scale
  - Units to scale inserted content: Inches

- Sample Output
  - 1.5000, 2.0039, 0.0000
  - 3.0000 x 45.0000

- Lighting
  - Units for specifying the intensity of lighting: International
3.1 Line Command

Creates single straight line segments

1. **Choose** Draw, Line.  
   **or**

2. **Click** the Line icon.  
   **or**

3. **Type** LINE from the command prompt  
   Command: LINE or L

4. **Press** ENTER

5. **Pick** From point: (point)

6. **Pick** Specify next point or [Close/Undo]: (point)

7. **Pick** Specify next point or [Close/Undo]: (point)

8. **Press** ENTER to end line sequence  
   **or**

9. **Type** U to undo the last segment  
   To point: U (undo)  
   **or**

10. **Type** C to create a closed polygon  
    To point: C (close)

**TIPS:**

- You can continue the previous line or arc by responding to the From point: prompt with a space or ENTER.
- Choose the right mouse button for the line pop-up menu to appear while in the line command.
Pline Command 15.1

A polyline is a connected sequence of line segments created as a single object. You can create straight line segments, arc segments, or a combination of the two.

1. **Choose** Draw, Polyline.
   
   or

2. **Pick** the Pline icon.

3. **Type** PLINE at the command prompt
   
   Command : PLINE or PL

4. **Pick** A point on the drawing to start the polyline
   
   From point: (select)

5. **Type** One of the following options
   
   Arc/Close/Halfwidth/Length/Undo/Width/<endpoint of line>:

   or

6. **Pick** A point to continue drawing
   
   Arc/Close/Halfwidth/Length/Undo/Width/<endpoint of line>: (pick point)

*Polyline as one segment*
3.4 Orthogonal Lines

Controls lines from being drawn at various angles to straight lines. When the snap grid is rotated, ortho mode rotates accordingly.

1. **Press** Function Key F8.
   or
2. **Double Click** ORTHO from the Status Bar.
   or
3. **Press** CTRL + L.
Rectangle 16.2

1. **Choose** Draw, Rectangle.
   
   or

2. **Click** the Rectangle icon.
   
   or

3. **Type** Rectang at the command prompt Command:
   
   RECTANG Chamfer/Elevation/Fillet/Thickness/Width/
   
   <First corner>:

4. **Pick** first corner.

5. **Pick** other corner or type coordinates (i.e. @4,2).
3.6 Circles

Circle Command

1. Choose Draw, Circle.
   or

2. Click the Circle icon.
   or

3. Type CIRCLE at the command prompt.
   Command: CIRCLE

4. Type One of the following options:
   3P/2P/TTR/<<center point>>:
   or

5. Pick A center point.

6. Type A radius or diameter.
   or

7. Pick A radius or diameter
   Diameter/<<radius>>:

TIPS:
- To create circles that are the same size, press ENTER when asked for the circle radius.
- When selecting a circle with a pickbox, be sure to select the circumference of the circle.
3.7 Arc Command

1. **Choose** Draw, Arc.
   or

2. **Click** the Arc icon.
   or

3. **Type** ARC at the command prompt
   Command: **ARC**

4. **Draw** One of the arcs.

**TIPS:**
- Except for 3 point arcs, arcs are drawn in a COUNTERCLOCKWISE direction.
- While in the arc command, press the right mouse button to select the following options for arcs:

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<tr>
<td>start, center, end</td>
</tr>
<tr>
<td>Start, end, radius</td>
</tr>
<tr>
<td>Start, center, included angle</td>
</tr>
<tr>
<td>Start, end, direction</td>
</tr>
</tbody>
</table>
The SPLINE command creates a particular type of spline known as a nonuniform rational B-spline (NURBS) curve. A NURBS curve produces a smooth curve between control points.

1. Choose Draw, Spline.
   or
2. Click the Spline icon.
   or
3. Type SPLINE at the command prompt
   Command: SPLINE
4. Pick A start point for the spline
   Object / <Enter first point>: (pick point)
5. Pick Points until you are done drawing splines
   Enter point: (pick points)
6. Press Enter or close to complete the spline
7. Pick Starting tangent point for the spline
   Enter start tangent (pick point)
8. Pick Ending tangent point for the spline
   Enter end tangent: (pick point)
Editing
AutoCAD 2D Tutorial

Editing Polylines 15.2

1. Choose *Modify, Polyline.*
   or
2. Pick the Pedit icon from the Modify II toolbar.
3. Type PEDIT at the command prompt
   Command: PEDIT
4. Pick Pick a polyline to edit
   Select Polyline:(pick)
5. Type One of the following options: Close/Join/ Width/Edit vertex/FitCurve/Spline/Curve/
   Decurve/Undo/eXit

PEDIT options:

- **Close** Closes open polyline segments
- **Join** Connects polylines, lines, and arcs to existing polylines.
- **Width** Changes the width for all polyline segments.
- **Fit curve** Creates curved arc segments around pline vertices at the direction you specify.
- **Spline Curve** Creates a curve through control points on a polyline.
- **Decurve** Straightens curved segments.
- **Edit Vertex** Displays the following Edit Vertex Options:

*Polyline width change*  
*Splined Polyline*
AutoCAD 2D Tutorial

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)

Circle before move

Circle after move

TIP:

To move an object a specified distance, type a distance at the second point of displacement prompt: @1<0
AutoCAD 2D Tutorial

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

Duplicate objects copied

Multiple objects copied

TIP:

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

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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

8. **Press** Enter to end the command.

*Offsetting objects by specifying a distance*
Explode Command 15.4

1. Choose *Modify, Explode.*
   or
2. Pick the Explode icon.
3. Type EXPLODE at the command prompt.
   Command: **EXPLODE**
   or
4. Pick The object to explode. Select objects: (pick)

![Polyline before explode](image)

![Polyline (line) after explode](image)
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

   **TIP:**

   - Use the object selection option FENCE to choose multiple objects
Stretch 13.2

1. Choose Modify, Stretch.
   or
2. Click the Stretch icon.
3. Type STRETCH at the command prompt.
   Command: STRETCH Select objects to stretch by window...
4. Type C to choose CROSSING window
   Select objects: C
5. Pick A first corner to stretch. First corner: (point)
6. Pick The opposite corner to window the objects to stretch.
   Other corner: (point)
7. Press ENTER to accept objects to stretch.
8. Pick A base point to stretch from Base point: (point)
9. **Pick**  A point to stretch to Newpoint: (point)

or

10. **Type**  A distance to stretch. Newpoint: @1<0

TIP:
The Stretch command must use a CROSSING window or a CROSSING POLYGON window.
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.
AutoCAD 2D Tutorial

Break 13.1

1. Choose Modify, Break. or
2. Click the Break icon.
   or
3. Type BREAK at the command prompt. Command: **BREAK**
4. Pick Object to break.
   Select object: (select one object)
5. Pick A second break point.
   Enter second point: (point)

   ![Diagram](image)

   or

6. Type **F** to choose a different break point
   Enter second point (or F for first point):(F)
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**
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*/Reference: (number)*
   
   *or*

7. **Pick** A rotation angle*/Reference: *(point)*
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)
AutoCAD 2D Tutorial

Text Command 11.1

Text

Creates a single-line text object

1. **Type** TEXT at the command prompt
   Command: **TEXT**
   or

2. **Pick** the Single Line Text icon from the Text Toolbar.

3. **Pick** A start point
   Justify/Style/<Start Point>: (point)
   or

4. **Type** J to change the justification or S to change the text style.

5. **Type** A text height
   Height <default>: (type value or pick two points)

6. **Type** A rotation angle
   Rotation angle <default>: (angle or point)

7. **Type** A text string
   Text: (type text string)

8. **Press** enter to exit the Text: prompt.

**DTEXT (Dynamic Text)**

Creates a single-line text object, showing the text dynamically on the screen as it is entered.

1. **Choose** Draw, Text, Single Line Text.
   or

2. **Type** DTEXT at the command prompt
   Command: **DTEXT**

3. **Follow** the steps 3-8 from above.
Layers
**AutoCAD 2D Tutorial**

**Introduction to Layers and Layer Dialog Box**

1. **Choose** Format, Layer.
   
   or

2. **Type** LAYER at the command prompt.
   Command: LAYER (or LA)

   or

3. **Pick** the layers icon from the Layer Control box on the object properties toolbar.

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**AutoCAD 2005**

**Layer Properties**
AutoCAD 2D Tutorial

Layer Options 12.2

? Lists layers, with states, colors and linetypes.
Make Creates a new layer and makes it current.
Set Sets current layer.
New Creates new layers.
ON Turns on specified layers.
OFF Turns off specified layers.
Color Assigns color to specified layers.
Ltype Assigns linetype to specified layers.
Freeze Completely ignores layers during regeneration.
Thaw Unfreezes specified layers Ltype.
Lock Makes a layer read only preventing entities from being edited but available visual reference and osnap functions.
Unlock Places a layer in read write mode and available for edits.
Plot Turns a Layer On for Plotting
No Plot Turns a Layer Off for Plotting
LWeight Controls the line weight for each layer

TIP:

Layers can be set using the command line prompts for layers. To use this, type –LAYER or -LA at the command prompt

1. Type Command: -LAYER or LA
2. Type One of the following layer options

/?/Make/Set/New/ON/OFF/Color/Ltype/Freeze/Thaw:
Layer Shortcuts 12.3

Changing the Layer of an Object

1. **Click** Once on the object to change.
2. **Select** the desired layer from the Layer Control Box dropdown.

AutoCAD will move the object to the new layer.
Colours + Line Weights
Color Command 12.6

1. **Choose** Format, Color.
   or

2. **Type** DDCOLOR at the command prompt.
   Command: **DDCOLOR or COL**
   or

3. **Choose** Color on the Object Properties toolbar and then select a color from the list or select Other to display the Select Color dialog box.

**TIP:**

These settings ignore the current layer settings for color.

**By Layer**

If you enter bylayer, new objects assume the color of the layer upon which they are drawn.

**By Block**

If you enter byblock, AutoCAD draws new objects in the default color (white or black, depending on your configuration) until they are grouped into a block. When the block is inserted in the drawing, the objects in the block inherit the current setting of the COLOR command.
Linetypes 12.7

Loading and Changing Linetypes

1. Choose Format, Linetype...
   or

2. Type DDLTYPE at the command prompt.
   Command: DDLTYPE or LT

3. Choose Load... to see a list of available linetypes.

4. Choose the desired linetype to assign.

5. Click OK.
AutoCAD 2D Tutorial

Lineweights 12.8

Loading and Changing Lineweights

1. Choose Format, Lineweight...

   or

2. Type LINEWEIGHT at the command prompt.

   Command: LINEWEIGHT or LWEIGHT

   or

4. Pick a lineweight to make current from the Object Properties menu.

TIPS:

- Lineweights can also be assigned to layers.

- The Display Lineweights feature can be turned on/off on the status bar to show or not show lineweights in the drawing, thus making regenerations faster.

- Lineweights are displayed using a pixel width in proportion to the real-world unit value at which they plot. If you are using a high-resolution monitor, you can adjust the lineweight display scale to better display different lineweight widths.
AutoCAD 2D Tutorial

Object Properties 12.9

1. **Choose** Modify, Properties.
   or
2. **Click** the Properties icon.
   or
3. **Type** DDCHPROP or DDMODIFY at the command prompt.
   Command: **DDCHPROP** (CH) or **DDMODIFY** (MO)
4. **Pick** Objects whose properties you want to change
   Pick a window for DDCHPROP, single object for DDMODIFY.
   Select objects:(select)
5. **Press** ENTER to accept objects.
   Select objects: (press enter)
6. **Choose** One of the following properties to change.
Other Useful Functions
BHATCH Command 17.1

1. **Choose**  
   Draw, Hatch...
   or

2. **Click**  
   the Hatchicon.
   or

3. **Type**  
   BHATCH at the command prompt

Command: **BHATCH**
Measuring Distances 8.2

1. **Choose** Tools, Inquiry, Distance.
   
or
2. **Click** the Distance icon from the Inquiry Toolbar.
   
or
3. **Type** DIST at the command prompt
   
Command: **DIST**
4. **Pick** The first point to measure from
   
First point: **pick point**
5. **Pick** The second point to measure to
   
Second point: **pick point**

Distance Between Circle Centers

**TIP:**
Be sure to use Object Snaps with the MEASURE command.
Linear Dimensions 26.1

1. Choose Dimension, Linear.
   
   or

2. Click the Linear Dimension command from the toolbar.
   
   or

3. Type DIM at the command prompt.
   Command: DIM
   Dim: HOR or VER
AutoCAD 2D Tutorial

Aligned Dimensions 26.2

1. Choose Dimension, Aligned.
   or
2. Click the Aligned Dimension command from the toolbar.
   or
3. Type DIM at the command prompt.
   Command: DIM
   Dim: ALIGNED
Radial Dimensions 26.3

1. Choose Dimension, Radius or Diameter.
   
   or

2. Click the Radial Dimensions command from the toolbar.

3. Type DIM at the command prompt.
   Command: DIM
   Dim: RADIUS or DIAMETER
AutoCAD 2D Tutorial

Calculating Areas 8.3

   or
2. Click the Area icon.
   or
3. Type AREA at the command prompt
   Command: AREA
4. Pick The first point for area calculation
   <First point>/Object/Add/Subtract: pick
5. Pick Next point: pick
6. Pick Next point: pick
7. Press ENTER when you are finished choosing points.

Area of Rectangle

Object Allows user to pick an object to calculate area (circle or polyline).
Add Adds separate areas for a total area calculation
Subtract Subtracts areas from each other.

TIPS:
Be sure to use Object Snaps with the MEASURE command
To subtract an area, you must first be in “add” mode to add the first area.
Plot Settings

1. Choose the Plot Settings tab.
2. Choose the appropriate paper size based on the chosen plotter.
3. Choose the paper units (inches or mm).
4. Choose the drawing orientation (Portrait, Landscape, Upside down).
5. Choose the plotting area.
6. Choose the plot scale.
7. Choose plot to center or specify an x or y offset.
8. Click OK.
1.13 On-Line Help

1. Choose Help, AutoCAD Help.
   or
2. Click the Help icon.
   or
3. Type HELP at the command prompt
   Command: HELP
   or
4. Press Function Key F1