BASIC DRAWING COMMANDS FOR AUTOCAD

Measuring Commands

GRID: Displays a grid of dots at a desired spacing on the screen.
Command: GRID (enter)
On/Off/Tick spacing(x)/Aspect: (enter value) (enter)

SNAP: Specifies a "round off" interval so that points entered with the mouse can be locked into alignment with the grid spacing.
Command: SNAP (enter)
On/Off/Value/Aspect/Rotate/Style: (enter value) (enter)

Basic Draw Commands

CIRCLE: Draws circles of any size.
Command: Circle (enter)
3P/2P/TTR/<center point>: (pick a center point)
Diameter or <Radius>: (Pick a point on the circle)

LINE: Draws straight lines between two points
Command: LINE (enter)
From Point: (pick a point using the mouse)
To Point: (Pick a point using the mouse)
To Point: (Press return to end the command)

ARC: Draws an arc (any part of a circle or curve) through three known points.
Command: ARC (enter)
Center/ < Start point > : (pick the first point on the arc)
Center/End/ < Second point > : **C**
Center: (pick the arc's center point)
Angle/Length of chord/ <End point > : (pick the arc endpoint)

Display Commands

LIMITS: Sets the size of the drawing paper. For size "A" drawing paper the limits should be set for 10.5 x 8.
Command: LIMITS (enter)
On/Off/Lower left corner <0.0000> (enter)
Upper right corner: 10.5,8 (enter)

ZOOM: Enlarges or reduces the display of a drawing.
Command: ZOOM (enter)
All/Center/Dynamic/Extents/Left/Previous/Vmax/Window/<Scale(x/XP)>:
(pick a point to define one corner of a rectangular viewing window then pick a point to
define the second point to define the opposite diagonal corner of the viewing window)

Note: To return the picture to its original viewing size enter **ALL** and press the enter key when prompted instead of defining a window.

PAN: Allows you to move your view point around the drawing without changing the magnification factor.
Command: PAN (enter)

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Editing Commands

CHANGE: Alters properties of selected objects
Command: CHANGE (enter)
Select objects or window or Last (select objects to be changed)
Properties/<Change point>: (type P)
Change what property (Color/Elev/LAyer/LType/Thickness)? (type Layer)
New Layer: (enter new layer name and press enter)

ERASE: Erases entities from the drawing.
Command: ERASE (enter)
Select objects or Window or Last: (Select objects to be erased and press enter when
finished)

EXTEND: Lengthens a line to end precisely at a **boundary edge.**
Command: Extend (enter)
Select boundary edge(s)...
Select Objects (pick the line which represents the boundary edge which lines will be
extended   to)
(press enter when finished selecting cutting edges)
<Select object to extend>/Undo: (pick the line(s) that need to be extended

TRIM: Trims a line to end precisely at a **cutting edge**.
Command: Trim (enter)
Select cutting edge(s)...
Select Objects (pick the line which represents the cutting edge of line in which objects will
be   trimmed to)
(press enter when finished selecting cutting edges)
<Select object to trim>/Undo: (pick the line(s) that need to be trimmed)

GRIPS
You can edit selected objects by manipulating grips that appear at defining points on the object. Grips is not a command. To activate grips simply pick the object. Small squares will appear at various entity-specific positions. By selecting an end grip you can stretch the entity to change its size. By selecting the center grip you can move the entity to a new location. To remove grips press CTL-C twice. You can perform the following using grips: Copy, Multiple Copy, Stretch, Move, Rotate, Scale, and Mirror.

Creating Layers

LAYER: Creates named drawing layers and assigns color and linetype properties to those layers.
Command: LAYER (enter)
A Layer & Linetype Properties dialog box will be displayed. To add a new layer, pick the New button. A new layer listing appears, using a default name of Layer1. the layer name can be changed by highlighting the layer name. Colors and Linetypes can be assigned to each new layer by picking the color box to assign a color and picking the linetype box to assign a line type.

Standard AutoCAD colors

1 = Red	2 = Yellow	3 = Green	4 = Cyan
5 = Blue	6 = Magenta	7 = White	

Standard AutoCAD linetypes
Hidden2 = hidden lines
Center2 = center lines
Phantom2 = phantom or cutting-plane lines

BASIC DRAWING COMMANDS FOR AUTOCAD

Construction Commands

ARRAY: Makes multiple copies of selected objects in a rectangular or circular pattern
Command: ARRAY (enter)
Select objects or Window or Last: (select object to array)
Rectangular or Polar array (R/P) <current>: (P)
Center point of array: (pick the point around which to form the array)
Angle to fill (+=CCW, -=Cw) <360>: (enter)

COPY: Draws a copy of selected objects.
Command: COPY (enter)
Select objects or Window or Last: (select objects to be copied)
Base point or displacement: (pick a point on the object to be use as a reference point)
Second point of displacement: (pick a point which represents the new location of the
copied

object)

MIRROR: Makes mirror images of existing objects.
Command: MIRROR (enter)
Select objects or Window or Last: (select objects to be mirrored)
First point of mirror line: (pick a point on top of the mirror line)
Second point: (pick a point on the bottom of the mirror line)
Delete old objects? <N> y or n (enter)

MOVE: Moves designated entities to another location.
Command: MOVE (enter)
Select objects or Window or Last: (select objects to move)
Base point or displacement: (pick a point on the object to be use as a reference point)
Second point of displacement: (pick a point which represents the new location of the
object)

OFFSET: Constructs an entity parallel to another entity at a specified distance. Offset can be used
with   lines, circles, arcs, and polylines.
Command: OFFSET (enter)
Offset distance or Through <last>: (enter a distance value)
Select object to offset: (select object to offset)
Side to offset: (Pick any point on the side of the object you wish to offset)

FILLET: Changes any corner to a rounded corner.
Command: FILLET
Polyline/Radius/Angle/Trim/Method/ <Select first line > : (pick the first line)
Select second line: (pick the second line)

CHAMFER: Changes any corner to an angled corner.
Command: CHAMFER
Polyline/Distance/Angle/Trim/Method/ < Select first line > : (pick the first line)
Select second line: (pick the second line)

OSNAP
Instantly locates exact points relative to existing objects (points).
Object Snap Modes: Endpoint, Midpoint, Center, Quadrant, Intersection, Insertion, Perpendicular, Tangent,
Nearest, Node, and None.

BASIC DRAWING COMMANDS FOR AUTOCAD

Placing lettering on a drawing

TEXT: Draws text characters of any size.
Command: TEXT (enter)
Justify/Style/<Start point>: (pick a starting point or enter a justification letter)
Height (0) (enter the height of the lettering)
Rotation Angle (0) (enter)
Text: (enter the desired lettering) (enter)

Summary of Options

<Start Point>	Left-Justifies text along its baseline
Justify	Justifies text according to the alignment options
Style	Enters a new text style
Null reply	Enters a new line of text below the previous text. (space or Enter key will give a Null reply)

Text Alignment Options

Alignment	Abbreviation	Orientation
Aligned	A	Aligns text between two points. Text height will adjust automatically
Fit	F	Fits text between two points. Text will not change
Centered	C	Centers text at the baseline of a point
Middle	M	Centers text horizontally and vertically the baseline of a specified point
Right	R	Right Justify text at the baseline of a specified point

Alignment	Abbreviation	Orientation
Top Left	TL	Left Justifies text at the top of text
Top Center	TC	Centers text at the top of text
Top Right	TR	Right justifies text at the top to text
Middle Left	ML	Left justifies text at the middle of text
Middle Center	MC	Centers text both horizontally and at the middle of the text
Middle Right	MR	Right justifies text at the middle of text
Bottom Left	BL	Left justifies text at the bottom of text
Bottom Center	BC	Centers text a the bottom of text
Bottom Right	BR	Right justifies text at the bottom of text

height specified at vertically

The SPELL command will check the spelling of a group of text.

Crosshatching a drawing

BHATCH: Allows the user to crosshatch areas of a section view.
Command: BHATCH (enter)
The Boundary Hatch dialogue Box will be displayed. Select the Hatch Options box.
The Hatch Options box will be displayed. Select the Patterns box.
The Choose Hatch Pattern box will be displayed. Select the desired hatch pattern.
The Hatch Options box will be displayed again. You can select a scale and rotation angle for the crosshatch pattern. Select the OK box when finished.

The Boundary Hatch dialogue box will be displayed again. Select the Pick Points box. When prompted select the internal point of the are to be crosshatched. Press the enter key when finished.

The Boundary Hatch dialog box will be displayed again. Select the Apply box to add the crosshatching to the drawing.