

AutoCAD - COMPUTER AIDED DRAFTING

by

Dr Mukul Shukla

Associate Professor, Mechanical Engg. Department, MNNIT, Allahabad, UP

Email – mukulshukla@mnnit.ac.in

(In association with Er. Saurav Kumar and Mr Ajeet Kumar)

Introduction:- AutoCAD (Version 2014/15) (from Autodesk Inc., USA) is a general purpose, easy to use, Computer Aided Drafting software, from which a variety of 2D drawings and 3D models can be made.

1. **Basics:-** From the **View** menu put **ON** the **Standard, Draw, Modify, Object snap** and **Dimension** toolbars. (Optionally you may put ON the **View, Zoom, Solids** and **Shade** toolbars also.) Help on any of the Commands can be taken anytime by pressing the **F1** key.

a) Units:- After opening AutoCAD, start with setting of the system of units (e.g. mm, inches etc.) alongwith the desired precision (1 or 2 or 3) of the dimensions. (*Default units = ??*)

b) Limits:- Changes the default limits (= ??) and sets the limits of the drawing that is to be made, by planning *a priori*. The origin is usually set to the Lower Left corner of the screen and the Upper Right corner is to be specified. (Avoid taking a square sized drawing area because the computer monitor resolution is rectangular, e.g. 640 X 480)

c) Zoom all:- Command – **zoom** (or **z**) then **all** (or **a**) (for implementing above changed drawing limits)

d) F1 key:- Online Help

F3 key:- Object snap mode On/Off

F6 key:- For Coordinates On/Off

F7 key:- For Grid On/Off


F8 key:- For Ortho mode On/Off

F9 key:- For Snap mode On/Off


e) Undo:- (Shortcut – u) Reverses the most recent action.


f) Up / Down arrow keys – Repeat immediate previous commands (without retyping the command)


2. Draw tools: They can be selected either from the Draw pull-down menu in the menu bar or from the Toolbar. Some of the important drawing commands are:-


LINE:-  Specify a point or press ENTER to continue from the last drawn line or arc.









CONSTRUCTION LINE:-  Specify a point or enter an option [Hor /Ver /Ang /Bisect /Offset]


MULTILINE:-  Specify a point or enter an option [Justification /Scale /Style].


POLYLINE:-  Specify a first point or enter an option [Arc/Close/Half width/Length/Undo/Width]

POLYGON:-  Enter number of sides or press ENTER
Specify center of polygon or [Edge]

RECTANGLE:-  Specify first corner point or Enter an option or specify a point [Chamfer /Elevation /Fillet /Thickness /Width].


Command	Icon	Shortcut key
Line		l
Polyline		Pline
Polygon		Polygon
Rectangle		Rectang
Arc		a
Circle		c
Ellipse		EL
Hatch		h

ARC:-  Creates an arc using three points. (*Explore other options*)

CIRCLE:-  Creates a circle using a specified radius. (*Explore other options*)


ELLIPSE:-  Creates an ellipse using a specified center point.


POINT:- Creates a point object.


HATCH:-  (Bhatch) Fills / sections an enclosed area or selected objects with a chosen hatch pattern.


3.Object Snap tools:- These are used for proper alignment and positioning of a drawing entity with respect to another drawing entity.


Temporary Tracking Point-  Creates a temporary point used by Osnaps.


Snap from-  Locates a point offset from a reference point within a command.


Snap to Endpoint-  Snap to the closest endpoint of an object.


Snap to Midpoint-  Snaps to the midpoint of an object.


Snap to Intersection-  Snaps to the intersection of two objects.

Snap to apparent intersect -  Snaps to the apparent intersection of two objects.










Snap to Extension-  Snaps to the phantom extension of an arc or line.

Snap to Center -  Snaps to the center of an arc, circle, ellipse, or elliptical arc.

Snap to Quadrant -  Snaps to a quadrant point of an arc, circle, ellipse, or elliptical arc.

Snap to Tangent-  Snaps to a point tangent to an object.

Snap to Perpendicular- Snaps to a point perpendicular to an object.

Command	Icon	Shortcut
Snap from		FROM
Snap to Endpoint		ENDP
Snap to Midpoint		MID
Snap to Intersection		INT
Snap to Apparent intersect		APPINT
Snap to Extension		EXT
Snap to Center		CEN
Snap to Quadrant		QUA
Snap to Tangent		TAN
Snap to Perpendicular	???	PER

4. Modify tools:- These are tools for the modification or editing of drawn objects.

ERASE:- Removes objects from a drawing. Select the objects to erase, right-click in the drawing area, and choose Erase.

COPY:- Duplicates objects. Select the objects to copy, right-click in the drawing area, and choose Copy Selection.

MIRROR:- Creates a mirror image copy of objects. Use an object selection method and press ENTER to finish.

OFFSET:- Creates concentric circles, parallel lines, and parallel curves. Specify a distance, enter *t*, or press ENTER

ARRAY:- Creates multiple copies of objects in a pattern. You can create rectangular or polar arrays by choosing the appropriate option. (*Explore its applications*)

MOVE:- Displaces objects a specified distance in a specified direction. Select the objects to move, right-click in the drawing area, and choose Move












ROTATE:- Rotates objects about a base point. Select the objects to rotate, right-click in the drawing area, and choose Rotate.

SCALE:- Enlarges or reduces objects proportionally in the X, Y, and Z directions. Select the objects to scale, right-click in the drawing area, and choose Scale

TRIM:- Trims objects at a cutting edge defined by other objects. Select one or more objects and press ENTER, or press ENTER to select all objects.

CHAMFER:- Bevels the edges of objects. Select first line or [Polyline /Distance /Angle /Trim /Method]

FILLET:- (Shortcut – f) Rounds and fillets the edges of objects. Select first object or [Polyline /Radius /Trim]







Command	Icon	Shortcut
Erase		e
Copy		cp
Mirror		mi
Offset		o
Array		ar
Move		m
Rotate		ro
Scale		sc
Trim		tr
Chamfer		cha
Fillet		f

5. Dimensioning Tools:- (Shortcut – d) These tools are used for dimensioning a drawing.

LINEAR DIMENSION:- Specify a point or enter an option [Mtext /Text /Angle /Horizontal /Vertical /Rotated]

ALIGNED DIMENSION:- Specify a point for manual extension lines, or press ENTER for automatic extension lines.

RADIUS DIMENSION:- Specify a point or enter an option [Mtext/Text/Angle].

Command	Icon	Shortcut
Linear dimension		Dimlinear
Aligned dimension		Dimaligned
Radius dimension		Dimradius
Diameter dimension		Dimdiameter
Angular dimension		Dimangular
Quick dimension		Qdim

DIAMETER DIMENSION:- Specify a point or enter an option [Mtext/Text/Angle].

ANGULAR DIMENSION:- Select an arc, circle, or line, or press ENTER to create the angular dimension by specifying three points.

QUICK DIMENSION:- Select the objects you want to dimension and press ENTER an option [Continuous/Staggered/Baseline/Ordinate/Radius/Diameter/datumPoint/Edit].

6. View tools:- (Shortcut – v) These tools help in visualizing a drawing from different views for better interpretation e.g. **Zoom** (Shortcut – z) to increase or decrease the drawn object's size.

REGENERATE:- Command - **Regen** to regenerate a drawing after making any modifications to it.

ZOOM REALTIME:- With no objects selected, right-click in the drawing area and choose Zoom to zoom in real time.

ZOOM WINDOW:- Zooms local portions of a drawing selected within a rectangular window.

ZOOM IN:- Increase the size of the drawing.

ZOOM OUT:- Decreases the size of the drawing.

ZOOM ALL:- (Self study)

ZOOM EXTENTS:- (Self study)

Command	Icon
Zoom Window	
Zoom IN	
Zoom OUT	
Zoom All	

PAN:- (Shortcut – p) With no objects selected, right-click in the drawing area and choose pan.

PAN REALTIME:-

3d ORBIT:- (Self study)

7. 3D Solid Modelling tools:- These are advanced features of AutoCAD which enable 3-d object modeling using a Building-block approach. The following solids / primitives are commonly used:

BOX:- Specify corner of box or press ENTER for corner of box, or enter c for center.

SPHERE:- Specify radius of sphere or [Diameter] and Specify a distance or enter d.

CYLINDER:- Specify center point for base of cylinder or [Elliptical] or press ENTER.

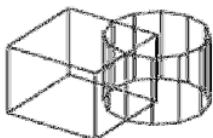
CONE:- Specify center point for base of cone or [Elliptical] or press ENTER.

EXTRUDE:- Select objects and Specify height of extrusion or [Path].

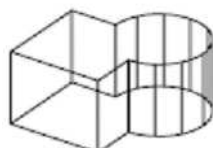
REVOLVE:- Specify start point for axis of revolution or define axis by [Object/X (axis)/Y (axis)]:
Specify a point or enter an option.

By a combination of **union**, **difference (subtract)** and **intersect** operations on a combination of the above solids, any complicated 3d object can be created.

Union: Combines (or adds) two or more 3D solids, surfaces, or 2D regions into a single, composite 3D solid, surface, or region.



Solids before union

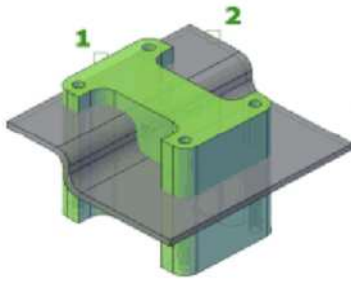


Solid after union

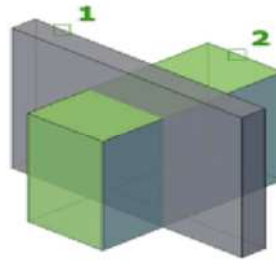
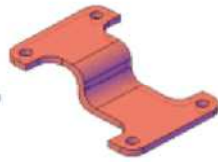
Intersect: Creates a 3D solid, surface, or 2D region from overlapping solids, surfaces, or regions.

Subtract: Creates a new object by subtracting one overlapping region or 3D solid from another.

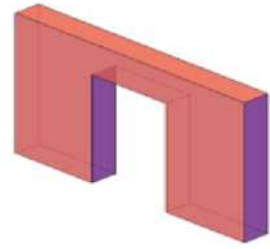
Command	Icon	Shortcut Key
Union		UNI
Intersect		INT
Subtract		SU
Presspull		PRESS
Loft		LOF



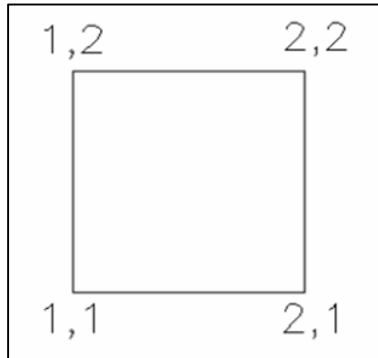
Output of intersection of two solids (1 and 2)



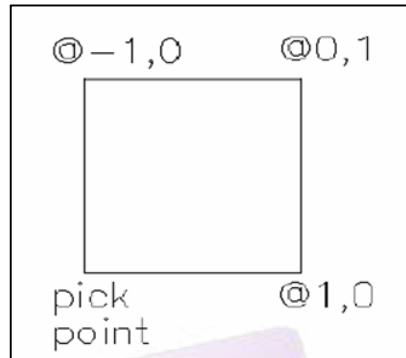
Output of subtraction of two solids (1 and 2)



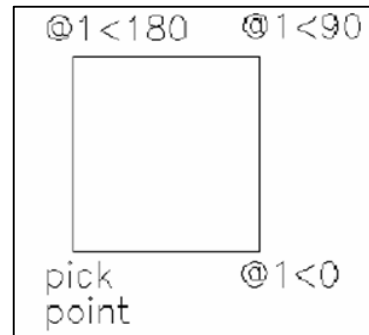
Coordinate System:



Absolute Coordinates: Input: x, y coordinates wrto origin.



Polar Coordinates from pick point. Input: ?? and ??



Relative Coordinates from pick point. Input: distance and angle

Text Command:

Text Style: You can specify the current text style to determine the appearance of all new text. A text style includes the font, size, oblique angle, orientation, and other text characteristics.

Editing Text: Edits single-line text, dimension text, attribute definitions, and feature control frames.

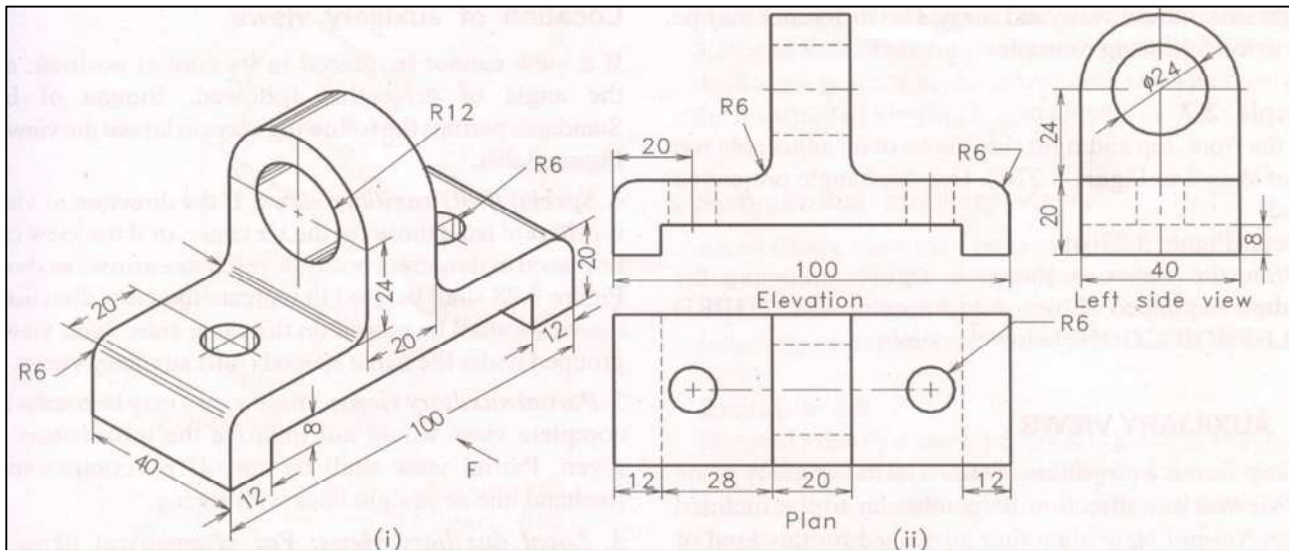
Mtext Editor

Command	Icon	Shortcut Key
Text		TEXT
Multiline Text		MTEXT
Text Style		STYLE
Editing Text		DDEDIT



Supplementary Reading: The concept and usage of **Layers** (e.g. to group objects in a drawing by function and to enforce standards for color, linetype, lineweight and other properties. **Command: LAYER** (or **LA**)), **Block** (shortcut – b), **Spline**, **Viewport**, **UCS**, **Isometric views**, **Hide**, **Render**, **Shade**, **dxf file**, **Export file** (formats) and **Surfaces** in AutoCAD?

Example 1: Draw front, top and left side views of the object shown below.



Front view;

Step-1: Pick line command>Pick 1st point> Pick 2nd point

Step-2: Pick Fillet command, or Enter "F">Enter "R">Radius value Enter>1st line>2nd line

Step-3: Pick offset command, or Enter "O">distance value enter>select line>give direction

Step-4: Mirror command, select object >pick mirror command>pick mirror point>give direction

Top view-:

Step-1: Pick Rectangle (X=100,Y=40)Enter

Step-2: Select Rectangle>click explode

Step-3: Offset, Enter "O" >offset distance value enter>select line>give direction

Step-: Pick circle, or Enter "C">pick circle point>give radius value enter

Left Side view-:

Step-1: Pick line command>pick 1st point>pick 2nd point

Step-2: Select object>pick mirror command>pick mirror point>give direction

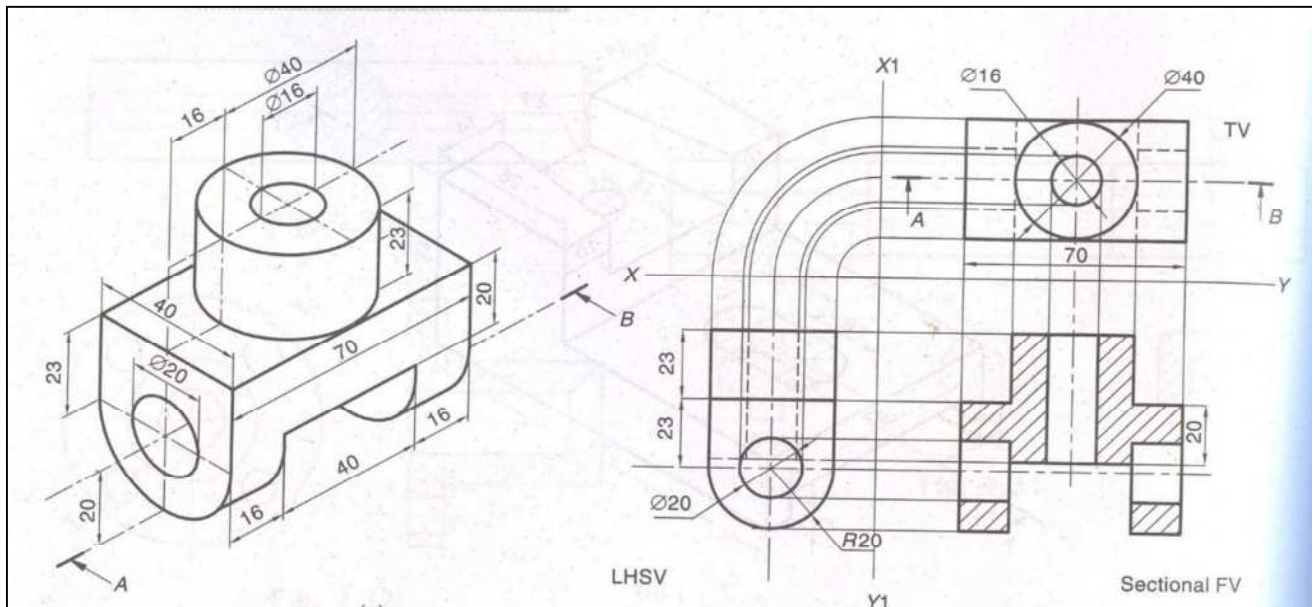
Step-3: offset, Enter "O" >offset distance value enter>select line>give direction

Step-4: Pick circle or Enter "C">pick circle centre point and give radius value enter

Step-4: Trim command, "TR" double enter>click remove line

Step-5: Dimension >pick 1st point & pick 2nd point>give direction

Example 2: Draw all the views as shown of the object given below.



Sectional Front view:-

Step-1: pick line command>pick 1st point & pick 2nd point

Step-2: centre line, select line>by layer>choose centre line

Step-3: offset command, Enter "O" >offset distance value enter>select line>give direction

Step-4: Mirror command, select object >pick mirror command>pick mirror point>give direction

Step-5: boundary command>Enter "BO">ok> pick internal point>Enter

Step-6: Pick hatch command>Enter "H" >choose pattern> add pick point>pick internal Point>Enter>ok

Step-7: Dimension >pick 1st point & pick 2nd point>give direction

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Example 3: Draw front view (right half in section) and top view of the object shown below:

