Agnihotri Engineering & GATE Classes

Scripting success stories

Introduction

Computer-Aided Design (CAD), also known as computer-aided design and drafting (CADD) , is the use of computer technology for the process of design and design-documentation. Computer Aided Drafting describes the process of drafting with a computer. CADD software, or environments, provides the user with input-tools for the purpose of streamlining design processes; drafting, documentation, and manufacturing processes. CADD output is often in the form of electronic files for print or machining operations. The development of CADD-based software is in direct correlation with the processes it seeks to economize; industry-based software (construction, manufacturing, etc.) typically uses vector-based (linear) environments whereas graphic-based software utilizes raster-based (pixelated) environments.

CAD definition : CAD extends for Computer Aided Drafting. CAD is a process of making drawing of an object on the screen of computer .

There are various types of drawing required in the field of engineering ,science & technology.CAD is one of them . overview of CAD with different streams.

Civil Engineering - In the field of civil engineering , plan & layout of building are prepared .

Mechanical Engineering – In the field of mechanical engineering ,drawing of machine component , machine design & layout are prepared .

Electrical Engineering - In the field of electrical engineering layout of power distribution system is prepared .

In all other field of engineering use of computer is made for drawing and drafting .

In modern CAD system interactive computer graphics (ICG) is used. ICG system is combination of hardware ,software & human designer.The ICG system is a main component of CAD . The other main component is a human designer. The system magnifies the power of designer.

Traditional Drawing -

In Traditional drawing practice, the designer prepares rough free hand sketches of individual components. On the basis designer's line work and strength these sketches can be modified.This is followed by a labourius & time consuming drawing work on the drawing board or drawing sheets.

Reasons for implementation of CAD system (Advantages)

1)To increase the productivity & accuracy of designer.
2)To improve the quality of design.
3)To create the Data-base for manufacturing.
4)To improve the efficiency in the design.
5)To save our time by shortening preparation and construction time.
6) It improves the quality of drawings produced as it offers better drawing visualization through colours.
7)It causes low wastage as does not require paper or drawing sheets to construct drawings.
8)It reduces man power requirement.
9)Printing can be done to any scale.

Application Of CAD

There are various process where we use CAD.

Automated Drafting :- It involves the creation of Hard Copy of engineering drawing directly from the CAD data base.

Geometical Modelling :- It is concerned with the mathematical description of the geometry of an object.

Limitation Of CAD

1)A 32 bits word computer is necessary because large amount of computer memory is required .
2)The size of software package is very large.
3)Skills & judgement are required by designer whille making a drawing.
4)Huge investment is need to be made.

Some Important CAD softwares

1)COREL DRAW   2)MICROSFT-OFFICE   3)PAGE MAKER   4)MICRO STATION   5)PAINT
6)PHTO-PHINISH   7)COREL CAD   8)AUTO-CAD

AUTO CAD

AUTO CAD is one of the 2D & 3D drafting & modelling software commonly used in engineering practice.
It is a powerful drafting tool that has a flexible features for the creation of drawing , editing & plotting.
The Machine drawing , isometric view & assembly drawing is possible in AUTO CAD.
AUTO CAD was first introduced by AUTO DESK (USA) in 1982.

Application Of AUTO-CAD

1)Preparation of 2D & 3D machine component drawing.
2)Preparation of graph using mathematical data
3)Preparation of building drawing.
4)Company logo and presentation.
5)preparation of work flow chart , map & layout etc.

AUTO CAD COMMANDS

To prepare the correct dimensional engineering drawing the co-ordinate system followed in Auto-Cad is used.
There are 4 co-ordinate system in AUTO CAD
i)Absolute Co-odinate System   ii)Polar Co-ordinate system iii)Relative Co-ordinate System
iv)Direct Distance Co-ordinate System
We generally use Polar Co-ordinate System in AUTO CAD,in polar co-ordinate system points are located to draw a line by defining the distance of points from the current position & the angle made to that line.

BASIC DRAWING COMMANDS FOR AUTOCAD

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)
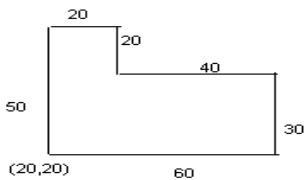
Specify first point	(20,20)
Specify Next point (Undo)	@60<0
Specify Next point (Undo)	@30<90
Specify Next point (Undo)	@40<180
Specify Next point (Undo)	@20<90
Specify Next point (Undo)	@20<180
Specify Next point (Undo)	@50<270
Specify Next point (Undo/Close)	press Enter to complete the drawing

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)

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)

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)

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.