

CADD Lab-II

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I-USAGE OF CAD PACKAGES

INTRODUCTION

Preparation of 2d &3d drawing, quantity and cost estimate, testing and certifying various building materials by appropriate authority, ect are important parts of the industry before starting any work. Any information can be transferred to the manufacturing people more easily through drawing and writings. Than us in any other means. In order days drawings and documents were prepared by hand by using only simple equipments like T-Square, mini drafter ect, which would take very long time to finish a work, and the accuracy , neatness and clarity poor as well, the delay in preparing the required paper works owed to delay in starting the work also. So the man began to think about preparing drawing and documents with less effort and more clarity and neatness, and it lead to the invention of computer aided Drafting and documentation. The computer aided works packages can be leaned and use easily by anyone, even without much experience in computer. This is because of the fact that programs involving these works are essentially menu driven, so that the computer if self will prompt what to do next after each and every operation. Today the use of computer for civil works has become very popular in the field of manufacturing industry because_of its miraculous accuracy and speed.

ADVANTAGES OF COMPUTER AIDED WORKS:-

1. Quality of drawing high and dimension, symbols, texts ect, are independent of skill of person drawing them.
2. Calculation involving quantity and lost are done automatically by the computer if respective equations are fed, saving a huge amount of time.
3. Copying and modifying a work, from already prepared work is very simple.
4. Data bases are created so that use by another person is possible very easily.
5. Library is created for any work, because of the storage of them in memory of the computer.
6. Use of layers is possible in which different parts of drawing can be enlarged and kept as separate sheets of drawing to view all parts in detail.
7. Editing and modification are possible in the work with less effort and time if required.
8. High accuracy in drawing dimensions which cannot be achieved in hand drawing.
9. Use of three dimensional drawing is also possible in CAD to see the interior details as well.
10. Taking print outs of the done works in paper are very easy.

IMPORTANT COMMANDS IN CAD

Command is any predetermined query given to the computer, which will be remembered and executed by it.

The usually used commands in CAD are listed below:

1. LINE: - To draw line with definite start and end point.
2. OFFSET: - To draw a parallel object specified distance to any previously drawn object.

3. TRIM: - To cut & remove any drawn line at any required point.
4. EXTEND: - To lengthen any drawn line to specified point.
5. ERASE: - To completely remove any selected item from the drawing.
6. COPY: - To get a similar object from a drawn object.
7. ARRAY:- To get a specified number of similar objects from a drawn object in a rectangular or circular pattern
8. FILLET: - To join two different lines with an arc of specified radius.
9. CHAMFER: - To join two different lines with a third line at specified points.
10. RECTANGLE: - To draw a rectangle with specified length and width.
11. CIRCLE: - To draw a circle with specified radius.
12. POLYGON:- To draw a polygon of any given number of sides.
13. ARC: - To draw an arc of given radius and angle of rotation.
14. ELLIPSE: - To draw an ellipse of given major and minor axes.
15. ROTATE: - To rotate a draw figure to any desired angle with respect to a specified point.
16. MOVE: - To Change the position of any figure with respect to specified point.
17. DIMENSION: - To mark the dimension of the drawn figure.
18. TEXT: - To write the details of the drawing using text of any chosen style.
19. DONUT: - To draw a filled circle or pipe section of specified inner and outer diameter.
20. HATCH: - To hatch a specified are in drawing with a selection pattern so that, the area is high light.
21. POLYLINE: - To draw a sequence of lines connected one after another.
22. SCALE: - To change the size drawn figure with specified multiple or fraction.
23. MIRROR: - To get the mirror image of a drawn figure with respect to a specified line of symmetry.
24. LAYER: - To show different parts of a drawing in different sheets so that details can be seen more clearly.
25. AREA: - To get the area and perimeter of an enclosed area and by simply clicking the corners of the figure in succession up to the first clicked point.

3D MODELING

Most of the popular and large CAD system has the ability to the model in 3dimension. The spatial image of the object is drawn in a pictorial projection using X-Y-Z coordinate geometry of it's stored in memory. It can be recalled or redrawn in 3D pictorial projection or in any orthographic projection .Representing the image of the object in number of a 2D views. A wire frame representation is a 3Dline drawing of an object showing only the edges without any side ,surface in between the image of the object wire frame , as the name indicates , as the appearance of the frame constructed by thin wire representing the edges, projected line and curves.

3D AUTO CAD COMMANDS

In Auto CAD , various solid modeling commands provide an easy method to generate three dimensional models. Apart from crating basic solid such as box, con, cylinder, torus, wedge, and sphere etc . You can additionally create solid from 2D object by extruding and revolving them more complex solids can be created by adding and subtracting volumes.

BOX

The box commands enable you to create a 3D cube or cuboids.

Command: Specify first corner of cube or center and Specify other corner or [Cube/Length]: Specify height or [2Point

CYLINDER

Command: cylinder

Specify center point for base of cylinder or(elliptical): <0,0, and0>

Specify center point for base of cylinder or(diameter):

Specify height of cylinder or (center of other end)

SPHERE

This command enables you to create a 3D solid sphere with its central axis parallel to the Z axis of the current UCS

Command: Specify center of shape and Specify radius or [Diameter] of sphere

TORUS

This command enable to create a 3D donut shaped solid.

Command: Torus

Specify centre of to torus and Specify radius of tours or [diameter]

UCS

While working in auto cad by default you are placed in world co-ordinate system .UCS or user coordinate system is used as a tool for creating and editing 3D drawing. It can also be described as customized or user defined co-ordinate system to crate 3D object. using command you can change the orientation in which 2D section are drawing and the direction they in which the objects are extruded.

UNION

This command enable you to combine two or more solids.

Command: union(uni) and select object

SUBTRACT

This command enables you to subtract a solid from another solid object.

Command: Subtract

INTERSECT

This command enables you to create a composite solid that contain volume to two or more overlapping solid objects.

SOLID MODELING

A surface model is made up to surfaces and it is more realistic model than a wire frame model for presentations. But since it is lucking the information about the mass, it is not a true presentation of the actual object in the real world . A solid model on the other hand is fully fledged presentation., which consist of on opaque clothing of an you can also perform mass property analysis on it. It is mathematically equalant representation of its physical counterpart.

The following method may be used to create solid model in AutoCAD.

EXTRUDE

This command enables you to draw solid by extruding 2D object along a specified path. You can extrude closed object such as circle, ellipse, polygon and closed poly line etc..

Not that intersection or unclosed cross section poly line that that self cross does not extrude.

Command: Extrude

Specify object :(select the object to extrude)

Specify height or extrusion or (path) : enter value of height enter pat the prompt area if the path option chosen
select an object that describes the path)

Specify angle of taper for extrusion <0>: (enter value for tapering the extrude object if required)

REVOLVE:

Close object such as circle, ellipse, polygon and splines can be revolved about an axis to generate a solid.

Command: Revolve

Select object: (pick the object of revolve)

Specify start point for the axis of revolution or define axis by [object, x-axis/y-axis]

Enter a value to specify start point and end point of the axis: (type X or Y axis to specify x-axis or y-axis or type to select an object as the axis of revolution)

SLICE

This command enable you to a new solid or a set of solid by slicing an existing solid with a plane and removing a selected side. You may keep one or both side of the sliced solid.

Command: slice

Specify first point of slicing plane [object/ z-axis/view/xy/yz/zx/3 point] < 3 point >:

Specify a point or enter an option if 3 options is used specify other two points to define the slicing plane.

Specify a point on the desired side of the plane

(Enter or select an option)