King Fahd University of Petroleum & Minerals CIVIL ENGINEERING DEPARTMENT

ENGINEERING GRAPHICS WITH AUTOCAD CE101



LABORATORY MANUAL

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PREFACE

This laboratory manual is designed to help students to use AutoCAD software for engineering graphics.

The objectives of this manual are focused on how to use AutoCAD Program as a drawing tool to draw objects and to develop basic concepts of engineering graphics, which are essential for every student enrolled in engineering. It is also an excellent method of learning the basics of AutoCAD software for more general purpose.

The class-work and homework assignments in this manual are arranged to match the

lecture sessions and are saved in the laboratory server.

Every command is explained, and examples of its use are shown during the lecture.

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1. INTRODUCTION TO AUTOCAD

1.1 Introduction:

In this section you are going to learn how to deal with the different AutoCAD

Windows and their contents.

When you start AutoCAD, three windows will be displayed on your screen:

- a. AutoCAD window: AutoCAD Window consists of the following components:
- The standard menu bars
- The floating toolbars
- The command line
- The drawing area
- The status bar

Draw toolbar, Modify toolbar and Object Snap toolbar can be added to your AutoCAD window by selecting the command View from the standard menu bar and choose any toolbar from the toolbar dialog box.

- b. Today Window: The use of Today window will help you:
- Start a new drawing
- Open a drawing
- Find a drawing
- Save a drawing
- Load AutoCAD symbol libraries

And you will also learn how to set up tools such as grid and snap to make drawing accurate and easier.

c. Active Assistance Window: This window will display information about any

AutoCAD command.

1.2 Starting AutoCAD

Opening a Drawing.

- a. From the pull-down menu, select the command Open.
- b. Choose the drive where the drawing is saved.
- c. Select the drawing you want open from the listed drawing names.

Starting a new drawing.

- a. From the pull-down menu, select the command new.
- b. Specify the new name of the drawing.
- c. Click the OK box.

Prototype Drawing.

- a. From the pull-down menu, select the command Open.
- b. From the template dialog box, select the file you want to use.
- c. You May choose to erase the contents of the file or modify it.

Any drawing can be used as a template.

Closing a drawing

You can close a drawing without exiting AutoCAD program, or you can close a drawing and exiting AutoCAD program.

- a. From the file menu, select Exit command, this will close the program.
- b. From the file menu, select New or Open command, this will allow you to continue using the program.

Saving a drawing.

- a. When you type at the command line, or select from the file menu the save command AutoCAD will display a save dialog box which will allow you to save your drawing under the current name or choose another name.
- b. If the drawing is already named, you can select the Qsave command from the standard toolbar menu.

While you are working on your drawing, remember to save your work every 15 to 20 minutes. This way will save you time and frustrations in case something goes wrong with your computer.

2. DRAWING WITH AUTOCAD

2.1 Drawing lines with AutoCAD

AutoCAD Entry Commands:

You can draw lines by using anyone of the five coordinate entry methods below:

- Absolute Rectangular Coordinates; (100, 0)
- Relative Rectangular Coordinates; @ (100, 0)
- Absolute Polar Coordinates; 100< 90°
- Relative Polar Coordinates; @ 100< -90°
- Direct Distance Entry; 100

Steps to begin drawing a line:

- a- Type L at the command prompt and press the Enter key, or
- b- Click le Line icon tool from the Draw toolbar.

In any case, AutoCAD will prompt you with a message asking you to specify the first point to begin your line.

- You can type the coordinates (x, y) of a point, and press enter to start the line,

(Cartesian coordinate entry).

3- After you have selected the first point AutoCAD will ask you to specify the next point:

Enter @ 45, 0 to extend the line 45 units to the right from the last point, (Relative

Cartesian coordinate entry); then

- Enter @55< 30° to extend the line 55 units 30° upward from the last point, (relative polar coordinate entry).

- Move the cursor horizontally to the right, and then enter 100 at the command prompt to extend the line 100 units in the horizontal direction, (direct distance entry).

4- Press Enter to terminate the line command.

Drawing Hidden lines

- From the object properties toolbar, choose the line type command, and then select Hidden
- Proceed as above.



Plate 1

2.2 Drawing circles with AutoCAD

You can draw circles by specifying one of the options cited below:

- The center and the radius or the diameter of the circle.
- 2 points (2P option). The two points are located on the circle.
- 3 points (3P option). The three points are located on the circle.
- 2 tangents and a radius, (TTR option).
- 3tangents, (TTT option).

Steps to begin drawing a Circle:

- 1- Type C at the command prompt and press the Enter key, or
- 2- Click le Circle icon tool from the Draw toolbar.

In any case, AutoCAD will prompt you with a message asking you to select one of the options cited above.

A- Center Radius or Diameter Option:

- a- You can type the coordinates (x, y) of a center point, or
- b- Select a point on the screen, using the mouse.
- c- After you have selected the center point AutoCAD will ask you to specify the radius

of the circle.

d- Press Enter to terminate the command.

B- 2P (two points) Option:

AutoCAD will ask you to specify 2 points (first end point of the diameter and then the second end point of the diameter).

C- 3P (three points) Option:

AutoCAD will ask you to specify 3 points (first point on the circle, second point on the circle and third point on the circle).

D- Tangent, Tangent, Radius Option:

AutoCAD will ask you to specify the first point on an object for the first tangent, the second point on an object for the second tangent of the circle.

E- Tangent, Tangent, Tangent Option:

AutoCAD will ask you to specify 3 points.

2.3 Drawing Arcs with AutoCAD

You can draw arcs by specifying one of the following options:

- Drawing an Arc by specifying 3 points on the screen.
- Drawing an Arc by specifying, the starting point, the center of the arc and the ending point.
- Drawing an Arc by specifying the Starting point, the Center of the arc, and the Angle.
- Drawing an Arc by specifying the Starting point the Center of the arc, and the Length of the arc.
- Drawing an Arc by specifying the Starting point, the End point, and the Direction of the arc.
- Drawing an Arc by using the option continue to join two lines by an arc. (See example)

3. TEXT WITH AUTOCAD

Text can be created and added to your drawing through the keyboard.

Selecting the text font and height:

 Select the text style command from the Format Pull-Down menu, and then make all the necessary changes on the text style dialog box, such as: Text height, text font ect.

Creating single and multiple line text:

- Select the text command from the draw pull-down menu, then select Single line text or Multiple line, or
- Type Dtext or DT at the command line, or
- Select the text icon A from the Draw toolbar.



4. ORTHOGRAPHIC PROJECTION

Definition:

Orthographic projection is a drawing representation using the front, the top, and the left or the right side views of an object.

This drawing representation is designed to describe the object through its principle views, the relation between them, the size and the location dimensions of each one and how they are related compared to the pictorial view. The emphasis is placed mainly on the visualization of the object.

One of the most efficient methods of creating orthographic views with AutoCAD is through the use of AutoCAD Drawing and Editing Commands, such as: Line, Circle,

Erase, Copy, Offset, Mirror, Block, ect.

The understanding of orthographic projection is essential for the topics that follow such as: Auxiliary views, Sectioning and Surface Intersections.



Plate 3



Plate 4



Plate 5



Plate 6



Plate 7



Plate 8



plate 9



Plate 10

5. GEOMETRIC CONSTRUCTION WITH AUTOCAD

Geometric construction is based on plane geometry, including; points, lines, circles, arcs and polygons.

AutoCAD provides powerful commands for creating complex geometric figures.

AutoCAD commands used in this section are: Circle command with the TTR option

(Tangent, Tangent and Radius); Fillet command, Trim command Offset command and

Polygon command. Every command is explained and examples are shown during the

lecture session.

The techniques and applications explained are used throughout the class work and homework assignments



Plate 11



6. DIMENSIONING WITH AUTOCAD

Dimensioning is the process of describing an object by its size.

The dimension values indicate the measurement of the dimension of the object. These

dimensions are extracted automatically from the drawing view.

Here are some selected AutoCAD dimensioning commands that you will use in this course:

- Linear: It creates distance between two points in the XY plane.
- Aligned: The dimension line is parallel to the distance to be dimensioned.
- Baseline: All dimensions are measured from the same baseline.
- Continue: All dimensions are placed end to end.
- Radius: It measures the radius of a circle or an arc.
- Diameter: It measures the diameter of a circle.
- Angular: It measures the angle between two intersecting lines.
- Leader: Creates leader line with annotation
- **Center Mark:** Indicates the center of circles and arcs. It can be changed to center lines.

After you select the type of dimension you want to place on your drawing, AutoCAD will ask you to select two points between the distances you want to measure, then the dimension will appear automatically on the object.

7. SECTIONAL VIEWS WITH AUTOCAD

There are three type of sections used in engineering graphics:

- a. Full Section.
- b. Offset Section.
- c. Half Section.

Each section is distinguished through its cutting plane line

The use of sectional views in engineering graphics is to make orthographic views less complicated to visualize. Cutting the view to reveal the interior will eliminate all hidden lines which will make the view easier to understand and to draw.

AutoCAD has several hatch patterns stored as a built-in library. The setting for

patterns, scale for pattern, angle for pattern and object selection can be done through

the hatching dialog box.

Procedure:

- Select Hatch command from the Draw pull-menu,
- Select the desired pattern,
- Set the scale pattern,
- Set the angle pattern,
- Select the area to hatch,
- Press enter to terminate the Hatch command.



Plate 13



Plate 14



Plate 15







Plate17



Plate 18

8. AUXILIARY VIEWS WITH AUTOCAD

Objects with inclined surfaces required more than two views to convey all information regarding the shape of the object. Orthographic views of the object will not show the true shape of the inclined surfaces. Auxiliary views are therefore drawn on an auxiliary plane.

Auxiliary plane are always parallel to the inclined surfaces.

Main AutoCAD commands used in drawing auxiliary views are:

- Line
- Point
- Offset
- Trim
- Distance
- Divide



Plate 19



Plate 20



Plate 21



Plate 22



Plate 23

9. INTERSECTION OF SURFACE WITH AUTOCAD

Surface intersection is a study of solid penetrations. When two planes intersect each others, there is a line common to both planes. This line is called Line of Intersection. There two types of line of intersection.

- Line of intersection between lateral side objects; this line is drawn as a straight line.
- Line of intersection between curved side objects; this line is drawn as a curved line.

Finding lines of intersection between two solids using AutoCAD software requires the use of the following commands:

Line: From Pull-down menu Draw, or Draw toolbar.

Point: Set from Pull-down menu Format and Point Style.

Copy: Set from the Pull-down menu Modify, or from the Modify toolbar.

Trim: Set from the Pull-down menu Modify, or from the Modify toolbar.

Offset: Set from the Pull-down menu Modify, or from the Modify toolbar.

Distance: Set from the Pull-down menu Tools, and then Inquiry.

Divide: Set from the Pull-down menu Draw, and then Point.

Pline: Set from the Pull-down menu Draw, or from the Draw toolbar.

Pedit: Set from the Pull-down menu Modify, then object, then Polyline or from the

Modify II toolbar.

Measure: Set from the Pull-down menu Draw, and then Point.



Plate 24







Plate 26







Plate 28



Plate 29



Plate 30



Plate 31

10. PICTORIAL DRAWING WITH AUTOCAD

Unlike the orthographic projection which is a two or three views drawing of an object,

The pictorial representation is a single view drawing projected in a single plane of

projection by one of the following methods:

- Isometric projection,
- Oblique projection, or
- Perspective projection.

Isometric Drawing with AutoCAD

AutoCAD provides the following commands to help you draw an isometric drawing:

- Isometric snap grid,
- Three isoplanes; Left, Top and Right. You can switch from one isoplane to another

by the function key F5 or Control + E.

Procedures:

- A. Lines.
- a. Select the SNAP command.
- b. Select STYLE from the snap prompt.
- c. Standard/Isometric: Type I and press Enter, this will display an isometric grid on the screen.
- d. Turn the grid on by pressing F7.
- e. Use line command to draw a cube.

B. Isometric circles.

- a. Select Ellipse command from the Draw menu.
- b. Select Isocircle.
- c. Enter centre of the circle.
- d. Enter the Radius of the circle.
- e. Use F5 to change from one isoplane to another.

Once you learn how to get into the isometric mode an toggle from one isoplane to another, isometric drawings become considerably easier with AutoCAD. Most the commands you have learned previously can be used for isometric.

Isometric drawing is the first step to 3D drawing.





Plate 32



Plate 33







11. FREEHAND SKETCHING

In this section students will use the skill and all information they have learned on the

previous topics to create freehand drawings such as: pictorial and multiview drawings.

Type of sketching:

- Pictorial sketching, (Isometric, Oblique and Perspective).
- Multiview sketching, (Front, Top and Side views).

Equipment for sketching:

- Pencil, (HB, H)
- Eraser (white)
- Paper

12. 3D DRAWINGS

Introduction:

After the isometric drawing, now it is time to switch to the 3D drawing.

In this section student will learn the fundamental concepts of producing a 3D drawings.

It shows how to create 3D drawing coordinates, (User Coordinate Systems), and how to

change viewpoints to see the object in 3D.

AutoCAD uses two types of coordinates:

- World Coordinate Systems (WCS).
- User Coordinate Systems (UCS).

World Coordinate Systems (WCS) represent the XY plane, use as AutoCAD default for all 2D drawings.

User Coordinate Systems (UCS) are the user defined coordinates; they help the user to create his own working plane.

How to change the viewpoints:

- Select View from the pull-down menu.
- Select 3D Viewpoint
- Select SW Isometric and watch the new orientation of your drawing.

How to create a UCS:

- Select the command View from the pull-down menu.
- Highlight toolbar, and then select UCS.
- Select one option from the followings: Origin/Zaxis/3point/Object/View/X/Y/Z.

AutoCAD provides three methods of creating a 3D drawing:

- a. Wireframe modeling.
- b. Surface modeling.
- c. Solid modeling.

Wireframe Modeling

Commands:

- Vpoint
- Сору

Example: Creating a 3D Wireframe rectangular prism.

- a. Draw a 100x50 rectangle in the center of your screen
- b. Use **3D** Views from the View pull-down menu, then select **SE** (South East), to view your rectangle taking a new orientation.
- c. Type Co for copy at your keyboard; then select the rectangle, then type (0,0,30). AutoCAD will create a copy of the rectangle 30 mm above the original.
- d. Use the line Command and the Object Snap to connect the upper and the lower corners.

Surface Modeling

Procedure:

- a. From the pull-down menu Draw, highlight Surfaces, then select 3D face.
- b. AutoCAD will ask you to pick the first point, then the second point, then the third and fourth points to complete the 3D surface.
- c. Use the copy command to copy the surface in the z-direction.

Solid Modeling

Solid modeling requires combining or subtracting primitive solid objects such as boxes, cylinders, wedges, cones, to create the final 3D solid model. AutoCAD provides two toolbars: **Solids** and **Solid editing** toolbars to create a solid model.



Plate 36







Plate38



APPENDICES

APPENDIX A



TYPES OF LINES IN ENGINEERING GRAPHICS

- 1) Dimension line: dark, thin and continuous.
- 2) Extension line: dark, thin and continuous.
- (3) Visible line: dark, thick and continuous.
- (4) Hidden line: Dark, medium and Dashed.
- 5) Center line: Dark, thin long dash and small dash.
- (6) Section line: Dark, Thin and Continuous.
- 7) Cutting Plane line: dark, thick long dash and small dash.

APPENDIX B

TYPES OF HOLES USED IN ENGINEERING GRAPHICS













Deep





APPENDIX C

AUTOCAD DRAWING AND EDITING COMMANDS

- COORD- Function Key F6 will allow you to turn coordinates ON or OFF.
- **GRID-** A matrix of dots that help you find your way around the Screen. **F7** key will turn the GRID ON or OFF.
- SNAP- Will help you move on the screen to a predetermined point.F9 Key turns the SNAP ON or OFF.
- **ORTHO-** Will help you draw perpendicular lines. F8 key turns the Orthogonal ON or OFF.
- UNITS- Will allow you to select the units of measurement.
- LAYERS- Will allow you to separate entities on your drawing.
- LINE- L. Will allow you to draw lines.
- POLYLINE- PL. Will allow you to draw a series of segments as one line.
- **PEDIT-** Will allow you to edit a polyline.
- OFFSET- Will allow you to create parallel lines.
- ARCS- A. Will allow you to draw arcs.
- CIRCLES- C. Will allow you to draw circles.
- **COPY- CO**. Will allow to copy objects on the screen from one position to another.
- **ARRAY- R**. Rectangular array, it will allow you to create a matrix of objects.
- **ARRAY- P.** Polar array, it will allow you to create a circular array.

- **ROTATE- RO**. Will allow you to rotate objects around a base point to any angle.
- MIRROR- Will allow you to mirror objects around a 90° axes.
- **MOVE-** Will allow you to move objects within the screen.
- BREAK- Will allow you to break a line into two entities.
- **EXTEND-** Will allow you to extend a line to another graphic entity.
- **TRIM-** Will allow you to shorten a line or a group of lines to their intersections.
- HATCH- Will allow you to place section lines using different patterns.
- FILLET- Will allow you to convert sharp corners to round corners.
- CHAMFER- Will allow you to cut sharp corners.
- **ERASE** E. Will allow you to delete any graphic entity from the screen

D. AUTOCAD FUNCTION KEYS

Keys	Functions
F1.	Help.
F2.	Switches from Graphics mode to Text mode and vise versa.
F3.	Turns Object Snap ON/OFF.
F6.	Turns Coordinate read out ON/OFF.
F7.	Turns Grid display ON/OFF.
F8.	Turns Perpendicularity ON/OFF.
F9.	Turns Snap ON/OFF.

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E. DRAWING ASSIGNMENTS