



KEMENTERIAN PENGAJIAN TINGGI



Introduction

utoCad

To

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INTRODUCTION TO AUTOCAD

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Alhamdulillah, thanks be to Allah SWT because with His bounty and permission we were able to complete the AutoCad 2019 eBOOK successfully. Thank you to our family and friends at Tuanku Sultanah Bahiyah Electrical Department Polytechnic for being the support team to complete this e-book. Finally, we would also like to thank the "PTSB E-Book Team" and the technical team who have guided us in completing this e-book. Hopefully with this eBOOK you can gain a little knowledge and experience in computer aided Electrical Drawing assisted drawing for excellence. Alhamdulillah, thanks be to Allah SWT because with His bounty and permission we were able to complete the AutoCad 2019 eBOOK successfully. Thank you to our family and friends at Tuanku Sultanah Bahiyah Electrical Department Polytechnic for being the support team to complete this e-book. Finally, we would also like to thank the "PTSB E-Book Team" and the technical team who have guided us in completing this e-book. Hopefully with this eBOOK you can gain a little knowledge and experience in computer aided Electrical Drawing assisted drawing for excellence.



ABSTRACT

Computer Aided Electrical Drawing (AutoCAD) The purpose of this e-book is to provide the knowledge and experience needed by students in drawing objects using AutoCAD software. Students can learn in brief the use of icons found in AutoCAD 2019. There are 3 topics that need to be given attention to give a clear picture of Autocad, are Introduction, Basic Drawing Tool and Display Control. In this e-book will provide valuable information with support diagrams to help students more understanding in conducting their practical

laboratories. Finally, it is hoped that this book can benefit all who want to deepen the knowledge of Computer Aided Design.



Chapter 1: Introduction To AutoCAD 1.1 Introduction 1.2 Advantages of autocad over conventional technical drawing 1.3 Concept of Computer Aided Design (CAD) 1.4 Hardware Computer in Autocad Software 1.5 Autocad 2019 icon Menu

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CHAPTER 1

INTRODUCTION TO AUTOCAD

1.1 Introduction

Computer-aided design (CAD) is computers aided drafting and design software from Autodest Inc. It to aid in the creation, modification, analysis, or optimization of a design.

CAD software is used :

- To increase the productivity of the designer,
- Improve the quality of design,
- Improve communications through documentation,
- To create a database for manufacturing.

1.2 Advantages of autocad over conventional technical drawing

- Improves the productivity, efficiency, and accuracy of drawings
- Enables drawings to be created, checked, and edited quickly
- Improves the quality of products
- Retains uniformity in reproduction
- Minimizes storage space



1.3 Concept of Computer Aided Design (CAD)

Computer-aided design (CAD) is the use of computers (or workstations) to aid in the creation, modification, analysis, or optimization of a design. CAD software is used :Designs made through CAD software are helpful in protecting products and inventions when used in patent applications. CAD output is often in the form of electronic files for print, machining, or other manufacturing operations. The term CADD (for computer aided design and drafting) is also used.

1.4 Hardware Computer in Autocad Software

- Computer/Laptop
- Keyboard



Printer/Plotter



1.5 Autocad 2019 icon Menu





1.6 Create A New Drawing

1. Click Start Drawing - Click New



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A new drawing is created for you based on the drawing template selected.

<Grid off



2. Set the LIMITS in AutoCAD

There are two methods of doing this.

1. Type Limits at the command prompt. OR

2. Click FORMAT LIMITS from the Pull-down menu (Autocad Classic).



The sample of paper size

Example

For A2 size

Command: limits Reset Model space limits: Specify lower left corner or [ON/OFF] <0.0,0.0>: 0.0 Specify upper right corner <0.0,0.0>: 594,420

For A3 size Command: limits Reset Model space limits: Specify lower left corner or [ON/OFF] <0.0,0.0>: 0.0 Specify upper right corner <0.0,0.0>: 420,297

For A4 size Command: limits Reset Model space limits: Specify lower left corner or [ON/OFF] <0.0,0.0>: 0.0 Specify upper right corner <0.0,0.0>: 297,210



1.7 Saving A Drawing

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Saving in AutoCAD File Format

- 1.On [File] menu, click [Export] [Export to DXF File] / [Export to DWG File].
- 2.[Export] dialog box will be displayed.
- 3. Select a saving location.
- 4. Enter a file name in [File Name] box.
- 5. Click [Save] button.
- 6. The drawing created in RootPro CAD will be saved as an AutoCAD drawing file





CHAPTER 2 AUTOCAD BASIC DRAWING

2.1 Basic Drawing Tools

In AutoCAD before drawing, we need familiar with the user interface of AutoCAD and the basic tools. In this chapter, we will learn how to make simple drawings using the basic draw and modify tools available in AutoCAD. Using these tools, you will be able to make and modify the simplest of drawings in AutoCAD.

2.1.1 Line

This is the most basic of the draw tools and one of the most frequently used ones in the drawing workflow. Line are creates straight line in drawing

area. With Line, can build object. Have 2 way use line in AutoCAd

i. Line Command in ribbon panel



Figure 2.1 : Line

ii. Line Command in Command line



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Figure 2.2 : Line Command

2.1.2 Arc

An Arc is a segment of a quarter circle. It combinations of center, endpoint, startpoint, radius, angle, chord length, and direction values.



Figure 2.3 : Arc

Select the Arc tool from the Draw panel in the Home tab. You can also use its command: **ARC**.



Figure 2.4 : Type of Arc



i. Point



ii. Start, Center, End

It creates an arc using the start point, center point and endpoint.



Figure 2.6 : Start, Center, End

iii. Start, Center, Angle

It creates an arc using the start point, center point and included angle.





iv. Start, Center, Length

It creates an arc using the start point, center point and length of chord.



v. Start, End, Angle

It creates an arc using start point, end point and included angle.



vi. Start, End, Direction

It creates an arc using the start point, end point, and starting direction.



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vii. Start, End, Radius

It creates an arc using the start point, center point and length of chord.



viii. Center, Start, End

It creates an arc using start point, end point and included angle.



ix. Center, Start, Angle

It creates an arc using the start point, end point, and starting direction.





x. Center, Start, Length

It creates an arc using the center, start point and length of chord.



2.1.3 Circle

Select the Circle command from the Draw panel in the Home tab, or you can also use its command, C. In order for students to draw an circle, they can use the Pull-Down Menu, Icon or Command Line. There are 6 methods to draw an circle from the Pull-Down Menu.





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a. Center, Radius and Center, Diameter

i. Center and Radius



Figure 2.16 : Center and Radius

ii. Center and Diameter



iii. 2P (Two Points)

Creates a circle based on two endpoints of the diameter.



With the 2 Point (2P) method, you can click or enter two points that will establish a diameter for the circle. you can specify various combinations of center, radius, diameter, points on the circumference, and points on other objects.

To draw the circle that sits at the top of the left rectangle, you'll need to specify the two diameter points so that the diameter of the circle matches the upper edge of the rectangle. Align the cursor with the top-left corner so that you see a green square appear, then click to specify the first point

Figure 2.18 : Circle Two Point



iv. 3P (Three Points)



With the 3 Point (3P) method, click or enter three points to create a circle that will touch all three points. we need to specify the first point as (1). we need to specify the third point like the second point by entering a length value and angle value (2). Or directly specifying the third point as (3)

To draw the circle that sits at the top of the left rectangle, you'll need to specify the two diameter points so that the diameter of the circle matches the upper edge of the rectangle. Align the cursor with the top-left corner so that you see a green square appear, then click to specify the first point

Figure 2.19 : Circle Three Point

v. Tan, Tan, Radius



2 circles or 2 lines are sufficient for this type of circle. select 2 point on 2 tangents of a line or circle it will specify the perimeter of a circle and then give the radius of the circle

It means this type of circle can be draw by Tangent, Tangent and Radius

Figure 2.20 : Tan, Tan, Radius

vi. Tan, Tan, Tan

Creates a circle tangent to three objects.



Figure 2.21 : Tan, Tan, Tan



To create circles, you can specify various combinations of center, radius, diameter, points on the circumference, and points on other objects. The default method is to specify the center and the radius. Three other ways to draw a circle are shown below.



Figure 2.22 : Three Ways Draw Circle

2.1.4 Ellipse

An ellipse is defined as a curve that surrounds two focal points. The shape of an ellipse is determined by two axes determined by length and width. We can also create an ellipse by defining two points on one axis and an end point on the other axis.



Figure 2.23 : Ellipse



Creates an ellipse or an elliptical arc.

The first two points of the ellipse determine the location and length of the first axis. The third point determines the distance between the center of the ellipse and the end point of the second axis.

i. Center

Center method, the first axis is determined by the center point and endpoint, while the second axis is determined by the length. We can either determine the point or distance value to create such ellipse.





Figure 2.24 : Center



TOTURIAL 2.1 : CENTER

Toturial 1 : Center

 Click on the Ellipse icon on the robbon panal and choose Center from the pull down of the Ellipse – choose <Center>



- 2. Specify the center of the Ellipse on the viewport
- 3. Specify endpoint of the first axis, as shown in the below



- 4. Press Enter. Specify the endpoint or distance value of the second axis.
- 5. Press Enter. The Ellipse will be created, as shoe below Vertical and herazotal





ii. Axis, End

The Ellipse is formed by defining the three points similar to above. The location and length of the first axis is specified by the points 1 and 2, while the distance between the center and the endpoint of the second axis is specified by the point 3.



Figure 2.25 : Axis, End

iii. Elliptical Arc

The Elliptical Arc command is used to create an elliptical arc.



Figure 2.26 : Elliptical Arc

The location and length of the first axis is specified by the points 1 and 2, while the distance between the center and the endpoint of the second axis is specified by the point 3.

The fourth and fifth specified points are the start and end angles.

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TOTURIAL 2.2: ELLIPTICAL ARC

Toturial 2 : Elliptical Arc

 Click on the Ellipse icom on the ribbon panel and chocse Elliptical Arc from the pull doen list, as shown in the below:



- 2. Specify the endpoint or the center of the elliptical arc.
- 3. Specify endpoint of the axis, as below :



4. Specify endpoint of other axis, as below :



5. Specify the start angle and the end angle.



6. Ellipticad are created.





Example ellipse show two different ellipses created by specifying axis and distance. The third point specifies only a distance and does not necessarily designate the axis endpoint.



first axis as major axis



najor axis axis Figure 2.27 : Different Ellipse

2.1.5 Polygon

Polygons are closed geometries made with three or more sides. The smallest polygon is a triangle and the largest polygon is a circle, which is made up of an infinite number of sides. In AutoCAD, you can make a polygon with a minimum of 3 and a maximum of 1 024 sides

1,024 sides.



Figure 2.28 : Polygon

Polygon is basically divided into 2:

i. Incribed in circle

A polygon is a shape drawn inside a circle. The order used in making a polygon with vertices or angles touching a circle is Inscribed in circle or shortcut command for inscribing is I





Figure 2.29 : Incribed in circle

- Specifies the radius of a circle on which all vertices of the polygon lie.
- Specifying the radius with your pointing device determines the rotation and size of the polygon.
- Specifying the radius with a value draws the bottom edge of the polygon at the current snap rotation angle.

ii. Circumscribed about circle

Use circumscribed polygon when you want to specify the distance between the center of the

polygon and midpoint of each side.



Figure 2.30 : Circumscribed about circle

- Specifies the distance from the center to the midpoints of the edges of the polygon.
- Specifying the radius with your pointing device determines the rotation and size of the polygon.
- Specifying the radius with a value draws the bottom edge of the polygon at the current snap rotation angle.



2.1.6 Polyline

Creates a 2D polyline, a single object that is composed of line and arc segments.

Drawing Polyline is more complex than drawing Line. A single Polyline can be composed of a number of a straight line or arc-segment. Polyline can also be given line widths to make them appear solid.



Specify start point	Specify next point
Sets the starting point for the polyline.	If you specify a second point, you create straight segments.
A temporary plus-shaped marker displays at the first point.	If you enter a (for Arc), you create arc segments
Processed Enter starts a new polyling from the last	

endpoint specified in creating a polyline, line, or arc.

Table 2.1 : Specify start point

2.1.7 Donut

This command draws a solid donut shape. Drawing doughnut is a quick way to create filled-rings or solid filled-circles. Creates a filled circle or a wide ring.



Figure 2.32 : Donut



Donut consists of two arc polylines that are joined end-to-end to create a circular shape. The width of the polylines is determined by the specified inside and outside diameters. If you specify an inside diameter of 0, the donut is a filled circle.



Figure 2.33 : Diameter Donut

2.2 Drawing Aids

2.2.1 Snap

Snap mode restricts the movement of the crosshair to intervals that are defined. When snap mode is on, the cursor seems to adhere or snap, to an invisible rectangular grid. Snap is useful for specifying precise points with the arrow keys or the pointing device. Restricts cursor movement to specified intervals. The following prompts are displayed. **Snap Spacing** - Activates Snap mode with the value you specify.

On - Activates Snap mode using the current settings of the snap grid.



Figure 2.34 : Snap

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Off - Turns off Snap mode but retains the current settings



Figure 2.35 : Invoking OSNAP



Figure 2.36 : Example of OSNAP in Drawing

2.2.2 Properties Palette

The properties palette is an essential tool. For properties palette can open with the PROPERTIES command (enter PR in the Command window) or



Figure 2.37 : Palette





Figure 2.38 : Properties Palette

The properties palette display a list of all important property setting like

- Color
- Layer

2.3 Edit Commands

AutoCAD drawings are rarely completed simply by drawing lines, circles etc. Most likely you will need to Modify these basic drawing objects in some way in order to create the image you need. AutoCAD provides a whole range of modify tools such as Move, Copy, Rotate and Mirror. As you can see, the command names are easily understandable. Perform editing operations such as erase, move, and trim on the objects in a drawing. The most common of these tools are located on the Modify panel of the Home tab.



Figure 2.39 : Edit Commands



The erase command is used to remove or delete objects from the drawing, as shown in the below image.



Figure 2.40 : Erase

No 1 and 2 are the numbers of the selected objects. The selected objects can also be removed with the help of the 'Delete' button on the keyboard



2.3.2 The Move and Copy Command



The move command in AutoCAD is used to move objects at a predetermined distance and direction. There are 4 steps to move an object at a certain distance in a specified direction. We can use Move Command from the Modify panel or use command M in command line.



Select objects Select the objects.

Specify the base point

Specify the starting point to move, which acts as a base point of an object.

Specify the point to place a moving object

Specify a point to put the object being moved. We can either specify it using a cursor or can enter the displacement value in the X, Y, and Z-direction. For example, 3, 4. The object will be moved 3 Units in the X-direction and 4 Units in the Y-direction from its current position.

Displacement

It signifies the relative distance and direction of the object from its last position. It determines how far the object is placed from its original position

Figure 2.41 : Move and Copy Command

The copy command is used to copy the objects.



Figure 2.42 : Copy Object



2.3.3 Rotate C Rotate

The rotate command changes the orientation of existing entities by rotating them about a specified base point. The rotation angle may be given as a numeric value or by dragging the entity.



Figure 2.43 : Rotate

2.3.4 Break



Allow partial erasure of line, arc, circle, 2D, 3D polylines and ellipse. Circle and arc are broken in a counter clockwise direction.



Figure 2.44 : Break



2.3.5 Trim

The Trim command in AutoCAD is used to remove objects/lines which is not desired in another object meets the edge of another object. It is also used to remove extra lines or extra parts of objects. We can also do the trim using different selection ethods.



Figure 2.45 : Trim



2.3.7 Mirror

The Scale command is used to increase or decrease the size of an object by a specified factor. If we

specify a value of 0 to 1, the size of the object will be reduced according to that factor. If we want to increase the size of an object, we need to specify its value greater than 1.



Figure 2.46 : Scale

Mirror

The mirror command in AutoCAD is used to make a copy (mirror copy) of the selected object. we can

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also discard the actual cbjek after copying the original object. An object copied by a mirror equal to half of its object can be mirrored across the center line to create the other half of the object.

It is shown in the picture below:



Figure 2.47 : Mirror

Note: By default, when you mirror a text object, the direction of the text is not changed. Set the MIRRTEXT system variable to 1 if you do want the text to be reversed.

2.3.8 Array



The array command creates either a Rectangular or Polar (circular) pattern existing object that you select. The pattern could be created from a single or a group of objects. Array duplicate set of objects for each item in the array. There are three types of arrays, which are listed below



 Rectangular Array - Distributes object copies into any combination of rows, columns and levels.



Figure 2.48 : Rectangular Array

 Polar Array - Evenly distributes object copies in a circular pattern around a center point or axis rotition

Polar Array

The Polar Array distributes a copy of the object along a circular path around the midpoint. We need to define the center to distribute the array. We can also change the number of items, angles, rows, and levels of the polar array.





Figure 2.49 : Polar Array

• Path Array - distributes the copies of the object along its path. We need to select the corresponding path to distribute the array.



Figure 2.50 : Path Array



2.3.9 Fillet and Chamfer



The fillet command is used to create rounded edges between the adjacent lines and faces in 2D and 3D. It is also used to connect two tangent arcs in 2D.



Figure 2.51 : Fillet and Chamfer





The chamfer command in AutoCAD is used to create slanted edges.We are required to select the two

adjacent lines to create chamfer between them



Figure 2.52 : Chamfer



2.3.10 Offset



The offset command in AutoCAD is used to create parallel lines, concentric circles, and parallel curves. It can offset any object through a point or at a specified distance. We can create as many parallel lines and curves with the help of the offset command. The offset objects can also be modified further according to the requirements. Here, modification of offset object means that we can apply trim, extend, and other methods on it.



Figure 2.53 : Offset



2.3.11 Polyline Edit (P edit)



You can choose from several useful options when you want to modify a polyline. The PEDIT command (enter PE in the Command window) is located on the drop-down list of the Modify panel. With this command, you can

- Ø Width (constant or varying)
- Ø Consist of arcs and lines.
- Ø Edited and joined together.
- ØExploded into individual segments

Command	Keystroke	lcon	Location	Result
Polyline	Pline / PL	Polyline	Home > Draw > Polyline	Creates a polyline of arcs and/or lines.
Polyline Edit	Pedit / PE	2	Home >Modify > Polyline Edit	Edits polyline objects

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CHAPTER 3 MEASURE AND DIMENSION

3.1 MEASURE

i. Measures the distance, radius, angle, area, and volume of selected objects or sequence of points.



Figure 3.1 : Measure Icon

ii. The MEASURE command performs many of the

same calculations as the following commands:

- a. Distance
- b. Area
- c. Radius
- d. Angle



Figure 3.2 : Measure



3.1.1 Distance

Measures the distance between specified points along with the X, Y, and Z component distances and the angle relative to the UCS.



Figure 3.3 : Distance

 Multiple points - Displays a running total of the distance between successive points. If you enter the Arc, Length, or Undo options, additional options similar to those for creating a polyline are displayed.

3.1.2 Area

Measures the area and perimeter of an object or defined area.



Figure 3.4 : Area

Specify corner points. Calculates the area and perimeter defined by specified points. If you enter the Arc, Length, or Undo options, additional options similar to those for creating a polyline are displayed.



- Add area -Turns on Add mode and keeps a running total of area as you define areas.
- Subtract area Subtracts a specified area from the total area.

3.1.3 Diameter and Radius

Measures the radius and diameter of a specified arc, circle, or polyline arc.



Figure 3.5 : Radius

3.1.4 Angle

Measures the angle



Figure 3.6 : Angle



3.2 DIMENSIONING

It is used to display the dimensions of drawings or models in AutoCAD. We can modify the dimensions in different drawing units according to the requirements.

The dimension area on the ribbon panel will look like the below image:



Figure 3.7 : Dimension



Figure 3.8 : Part of Dimension

3.1.4 Example 3



Figure 3.9 : Example



3.2.1 Linear

Linear Dimension scan create horizontal, vertical, aligned and dadial domensions eith DIM command. The type of dimension depends on the object that select and the direction that drag the dimension line.



Figure 3.10 : Linear

- Select the first point and second point of a line to be measured
- The dimension of the corresponding line will be displayed

3.2.2 Aligned



The aligned dimension will give us the exact distance between the two selected points or endpoints. It is commonly used to measure the slanted lines. The dimensions will be parallel to the created slanted line.



Figure 3.11 : Aligned



3.2.3 Angular



Angular dimensions measure the angle between two selected geometric objects or three points. From left to right, the example shows angular dimensions created using a vertex and two points, an arc, and two lines. Measures the angle between selected geometric objects or 3 points.



Figure 3.12 : Angular





Creates a diameter dimension for a circle or an arc. Measures the diameter of a selected circle or arc, and displays the dimension text with a diameter symbol in front of it. You can use grips to easily reposition the resulting diameter dimension.



Figure 3.13 : Diameter



3.2.5 Radius



Creates a radius dimension for a circle or an arc. Measures the radius of a selected circle or arc and displays the dimension text with a radius symbol in front of it. You can use grips to reposition the resulting radius dimension easily.



Figure 3.14 : Radius



TOTURIAL 3.1: AUTOCAD 2D



Step 1

Use the **CIRCLE** command to create the following circles. They all have the same center. Have 4 circle have do.











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Step 4 Use tan, tan, radius to create circle between 2 circle.







Step 6

Select the circle/line in the polar array and trim such to have the image

Step 7

Create these 2 circles and do a polar array of 3 items as show on the imege below







Step 8 Use the FILLET command to fillet the inner edges as image below .



Step 9 Do the all dimension for all part as same like a image





Exercise: AUTOCAD 2D

PROCEDURE

- 1. By using all commands, draw the figure 2 by using metric unit.
- 2. Using layer command, create layer object and layer dimension.
- 3. Insert all dimension
- 4. Save your work in your computer with name : NO MATRIC
- 5.







PROCEDURE

- 1.
- By using all commands, draw the figure 3 by using metric unit. Using wblock command create each part with the name as legend 1. 2.
- З. Save your work in your computer with name PRACTICAL WORK 3











11





*	Transformer
К	NPN Transister
κ	PNP Transister
₩	Zener Diode
₩	Diode
rhr	Gnd
	Block Resistor
-~~~-	Resistor
ŧ	Polarity Cap. 1
Ť	Polarity Cap. 2
÷	No Polarity Cap

Legend 1



REFERENCES

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