



NAMAL UNIVERSITY, MIANWALI

Department of Electrical Engineering

EE-254– Engineering Drawing

Lab – 4

Use of Basic drawing tools of AutoCAD 2016

Student Name	Institute ID	Marks

Date: _____

Instructor: Engr. Rizwan Shabbir

Document History

Rev.	Date	Comment	Author
1.0	02-2022	Initial Draft	MB
1.1	04-2023	Revision	RS

Course Learning Outcomes

CLO-3: Reproduce 2-D and 3-D sketches using AutoCAD by applying engineering drawing principles.

CLO-4: Present Auto CAD designs effectively through design documentation and reports.

Equipment

- Software
 - AutoCAD 2016 Educational Version

Instructions

The following instructions are to be followed while performing the labs

- The manual must be thoroughly read before starting the lab.
- The theoretical concepts related to the lab and experiments must be revised.
- All attempt shall be made to complete the lab during the lab session.
- Any attempt to plagiarize from any source will be reported to the disciplinary committee for further action, so keep the work original.

Objectives

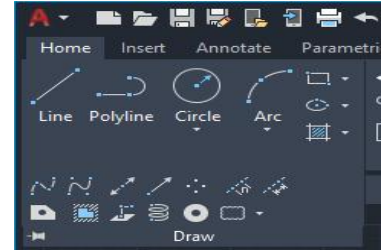
- To get hands on experience of AutoCAD software.

Background Information

You will create these drawings using the basic drawing tools. These tools include **Line**, **Circle**, **Polyline**, and **Rectangle** and so on and they are available in the Draw panel of the ribbon, as shown below. You can also activate these tools by typing them in the command line.

Drawing Lines

You can draw a line by specifying its start point and end point using the **Line** tools. However, there are various methods to specify start and end of a line. These methods are explained in the following examples.



Using the Absolute Coordinate System

In this example, you will create lines by specifying points in the absolute coordinate system. In this system, you specify the points with respect to the origin (0, 0). A point will be specified by entering its X and Y coordinates separated by a comma, as shown in below.

- Click **Zoom** > **Zoom All** on the Navigation Bar; the entire area in the graphics window will be displayed.
- Turn OFF the **Dynamic Input** icon.
- To draw a line, click **Home** > **Draw** > **Line** on the ribbon, or enter **LINE** or **L** in the command line.
- Type 10, 30 and press ENTER.
- Type 50, 30 and press ENTER.
- Type 50,60 and press ENTER.
- Type 10,60 and press ENTER. Select the Close option from the command line.

Using Relative Coordinate system

In the relative coordinate system, you define the location of a point with respect to the previous point. For this purpose, the symbol, '@' is used before the point coordinates. This symbol means that the coordinate values are defined in relation with the previous point.

- Type-in **Z** in the command line to activate the **ZOOM** command. Click the **All** option in the command line. This displays the entire area in the graphics window.
- Turn OFF the **Grid** icon on the status bar.
- Type 100, 100 and press ENTER. This defines the first point of the line.
- Type @40, 0 and press ENTER.
- Type @0, 40 and press ENTER.
- Type @60, 0 and press ENTER.
- Type @0, 80 and press ENTER.
- Type @100, 0 and press ENTER.
- Type @50, 50 and press ENTER.
- Type @0, 30 and press ENTER.
- Type @-30, 0 and press ENTER.
- Type @-50, 50 and press ENTER.

- Type @-30, 0 and press ENTER.
- Type @0, -50 and press ENTER.
- Type @-50, -50 and press ENTER.
- Type @-90, 0 and press ENTER.
- Type @0, -50 and press ENTER.
- Type @30, -30 and press ENTER.
- Type @-30, 0 and press ENTER.
- Type @0, -70 and press ENTER.

Using Polar Coordinate system

In the polar coordinate system, you define the location of a point by entering two values: distance from the previous point and angle from the zero degrees. You enter the distance value along with the @ symbol and angle value with the < symbol.

- Type L and press ENTER
- Type @100<0 and press ENTER.
- Type @70<45 and press ENTER.
- Type @30<90 and press ENTER.
- Type @70<135 and press ENTER.
- Type @50<180 and press ENTER.
- Type @50<270 and press ENTER.
- Type @40<315 and press ENTER.
- Type @80<180 and press ENTER.
- Type @50<270 and press ENTER.

Using Direct Distance Entry

In the direct distance entry method, you draw a line by entering its distance and angle values. You use the Dynamic Input mode in this method.

Click **Zoom > Zoom All** on the **Navigation Bar**. Activate the **Dynamic Input** icon on the Status Bar. Define the first point of the line by typing 50,50 and pressing ENTER.

- Move the pointer horizontally toward right and type-in 150 in the length box. Press the TAB key and type 0 as angle. Next, press ENTER.
- Move the pointer vertically upwards and type-in 100 as length. Press the TAB key and type 90 as angle. Next, press ENTER.
- Move the pointer horizontally toward left and type 50. Press the TAB key and type 180 as angle. Next, press ENTER.
- Move the pointer vertically downwards and type 20. Press the TAB key and type 90 as angle. Next, press ENTER.
- Move the pointer horizontally toward left and type 50. Press the TAB key and type 180 as angle. Next, press ENTER.
- Move the pointer vertically downwards and type 40. Press the TAB key and type 90 as angle. Next, press ENTER.
- Move the pointer horizontally toward left and type 50. Press the TAB key and type 180 as angle. Next, press ENTER.
- Click the Close option in the command line.

Erasing, Undoing and Redoing

- Draw any sketch using the Line tool.
- Click Home > Modify > Erase on the ribbon or Enter ERASE or E in the command line.
- Select the lines shown below and press ENTER. This erases the lines.
- Click the Undo button on the Quick Access Toolbar. This restores the lines.
- Click the Redo button on the Quick Access Toolbar. This erases the lines again.

Drawing Circles

The tools in the **Circle** drop-down on the **Draw** panel can be used to draw circles. You can also type in the **CIRCLE** command in the command line and create circles. Various methods are used to create circles. These methods are explained in the following.

Circle (Center, Radius)

By using this method, a circle can be created by specifying its center and radius value.

- Click Home> Draw> Circle> Center, Radius on the ribbon.
 - Select an arbitrary point in the drawing area to specify the center point.
 - Enter any positive valued number as the radius and press ENTER.
- OR**
- In command line, type 'Circle' and press ENTER.
 - Then specify its center point and press ENTER.
 - And finally, enter any positive valued number as the radius and press ENTER.

Circle (Center, Diameter)

By using this method, a circle can be created by specifying its center and diameter value.

- Click Home> Draw> Circle> Center, Diameter on the ribbon.
 - Select an arbitrary point in the drawing area to specify the center point.
 - Enter any positive valued number as the diameter and press ENTER.
- OR**
- In command line, type 'Circle' and press ENTER.
 - Specify its center point and press ENTER.
 - Then type 'D', to switch from radius to diameter.
 - And finally, enter any positive valued number as the diameter and press ENTER.

Drawing Arcs

An arc is a portion of a circle. The total angle of an arc will always be less than 360 degrees. AutoCAD provides you with 11 ways to draw an arc. You can draw arcs in different ways by using the tools available in the Arcs drop-down of the Draw panel. The usage of these tools will depend on your requirement.

Arc (3-Point)

Draws an arc using three specified points on the arc's circumference.

- The first point is the start point of an Arc.
- Specify the second point, is a point on the circumference of the arc.
- Specify the final point on the arc

Arc (Start, Center, End)

- The first point is the start point of an Arc.
- Specify the center point, instead of 2nd point as above.
- Specify the end point on the arc

Arc (Start, Center, Angle)

Draws an arc counterclockwise from the start point (1) using a center point (2) with a specified included angle. If the angle is negative, a clockwise arc is drawn.

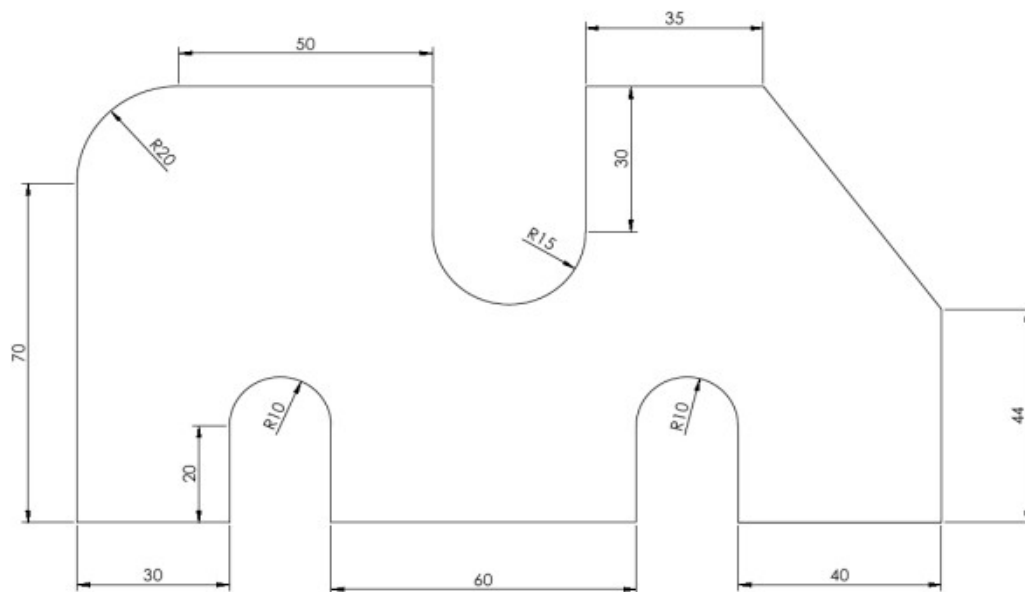
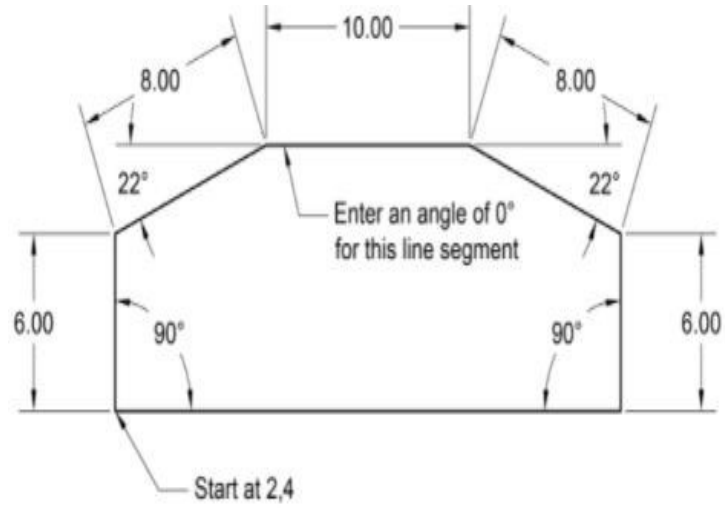
Drawing Polylines

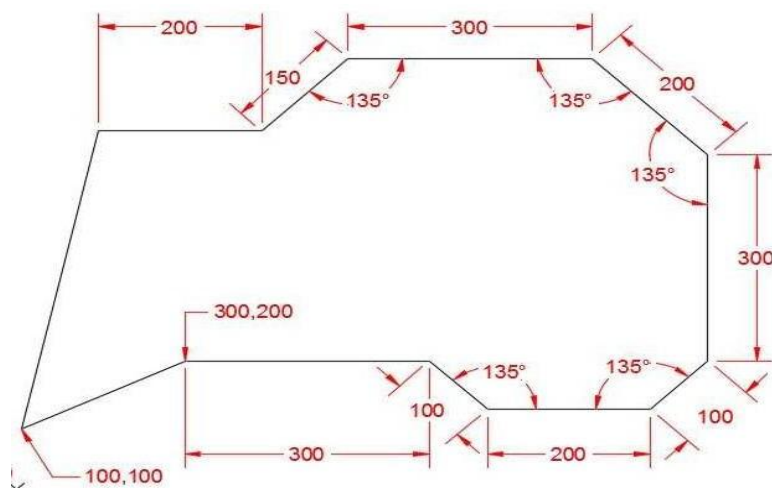
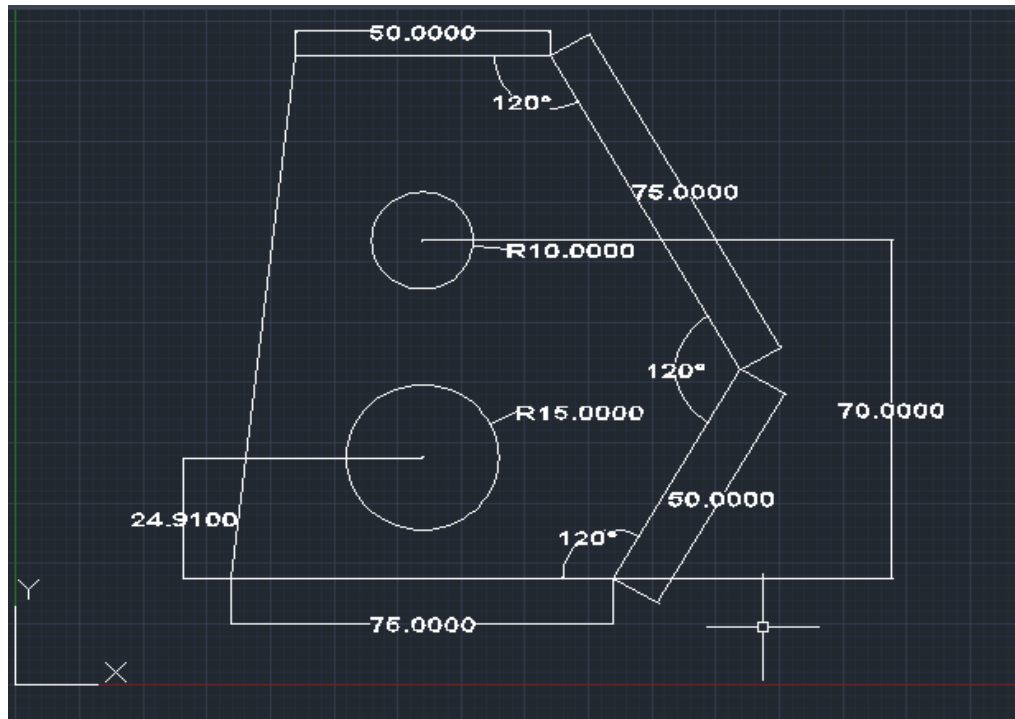
A Polyline is a single object that consists of line segments and arcs. It is more versatile than a line as you can assign a width to it.

- We can understand Polyline in AutoCAD as a line with a number of connected lines in one shape without any break.
- You can find the **Polyline** command in the **Draw** menu of the home tab, or you can press **PL** from the keyboard then press the enter button of the keyboard.
- Now, take the first point from where you want to start this polyline, then make your desired shape with a line in continuation of this polyline command.
- Draw the same pattern with the **line** as we draw with the **polyline** command. Now when you select both of them with a mouse cursor, you can see polyline will select in the single selection and show you different connected line segments, but if we see the pattern of line command, it will select one segment of the line pattern. So, polyline gives you the number of the connected line segments in one shape of the structure.
- Now draw a polyline, then press the **A** button of the keyboard for the Arc option of this command, then press the enter button of the keyboard.
- Now click again and again on a new point as the endpoint of the next arc, and you can make a number of the arc with it.
- Leaving this arc option of polyline command, press **L** from the keyboard, then press enter button of the keyboard for back into line form of polyline command.

You can analyze its parameters more and more by practicing on it for making a good drawing with the help of the polyline command of the auto cad.

Tasks:





Method of Evaluation: Lab Report and in-lab marking by instructors

Measured Learning Outcomes

CLO-3: Reproduce 2-D and 3-D sketches using AutoCAD by applying engineering drawing principles.

CLO-4: Present Auto CAD designs effectively through design documentation and reports.

	Excellent (10)	Good (9 to 7)	Satisfactory (6 to 4)	Unsatisfactory (3 to 1)	Poor (0)	Marks Obtained
Report (CLO4)	Required document filled-in neatly with meaningful answers to all questions, proper grammar and punctuations with proper conclusion drawn	Required document filled-in neatly with meaningful answers to most questions and proper conclusions drawn with some grammar mistakes	Some correct/meaningful answers and conclusions with some irrelevant ones. Some parts of the document not neat or some grammar mistakes.	Answers not understandable/ not relevant to questions. Conclusions not based on results. Illegible writing with no proper grammar/punctuation	Report/Hand out Not submitted	
Task Completion (CLO3)	All Tasks were completed successfully in the time of the lab	Most of the tasks were completed in the given time of the lab	Some of the tasks were completed in the given lab time	Very few tasks were performed and completed in given lab time	Lab tasks weren't performed at all	
Total						