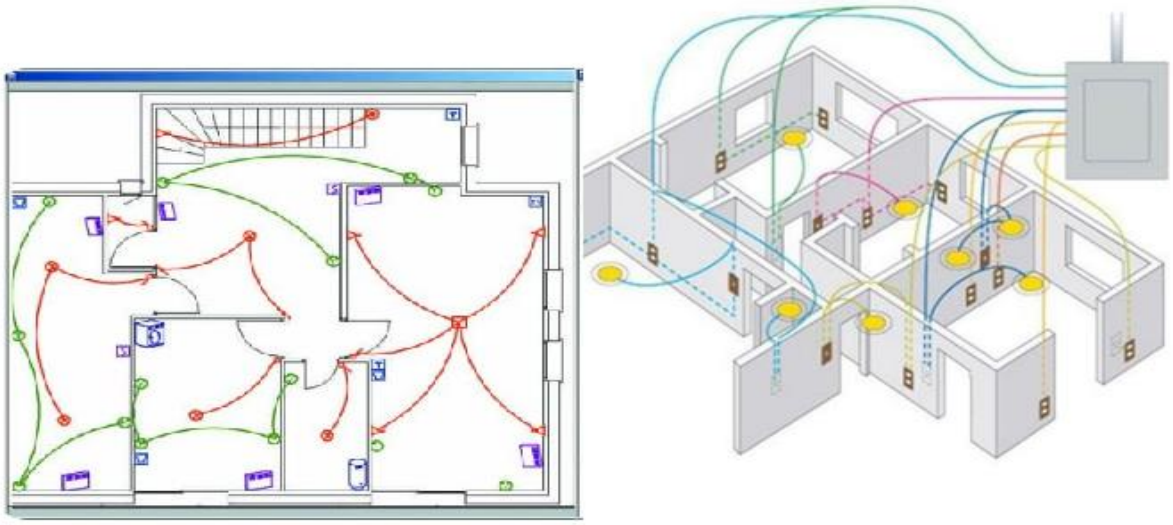
BUILDING ELECTRICAL INSTALLATION

LEVEL-III

Based on October 2023, Curriculum Version - II



Module Title: Computer Aid Design Systems

Module code: EIS BEI3 M1 1023

Nominal duration: 96 Hour

Prepared by: Ministry of Labor and Skill

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Contents

Acknowledgment

The Ministry of Labor and skill wishes to thank to MOLS experts and TVET trainers who contribute their time and professional experience to the development of this Training module for finishing construction work.

We would like also to express our appreciation to the regional labor and skill bureaus, TVT colleges for their cooperation and technical support of this training module development.

Acronym

CAD .....Computer aided design system

2D 2Dimenstional

3D 3Dimenstional

TVT..... Technical vocational training

MOLS .....Ministry of Labor and Skill

Auto CAD .....Automatic Computer Aid Design

UCS .....User Coordinate System

VP .....View port

Introduction to the Module

Computer aid design system helps to know the CAD system, Basic 2d drawing, Basic 3D Modeling and Layouts and Printing in finishing construction field. This module cover skill, knowledge and attitude required to Produce Basic Engineering drawing

This module is designed to meet the industry requirement under the finishing construction work occupational standard, particularly for the unit of competency: Computer Aid Design System

This module covers the units:

- Computer aid design system
- Basic 2D drawing and drafting.
- Basic 3D Modeling
- Layouts and Printing

Learning Objective of the Module

- Identify Computer aid design system
- Produce Basic 2d drawing and drafting
- Produce Basic 3D Modeling
- Layouts and Printing

Module Instruction

For effective use these modules trainees are expected to follow the following module instruction:

1. Read the information written in each unit
2. Accomplish the Self-checks at the end of each unit
3. Perform Operation Sheets which were provided at the end of units
4. Do the “LAP test” giver at the end of each unit and
5. Read the identified reference book for Examples and exercise

UNIT ONE: COMPUTER AID DESIGN SYSTEM

This unit is developed to provide you the necessary information regarding the following content coverage and topics:

- Introduction CAD system
  - The importance of CAD software
  - Types of CAD software and its applications
  - Auto CAD
- Booting up the auto CAD software
- Auto CAD Interface and tool

This unit will also assist you to attain the learning outcomes stated in the cover page. Specifically, upon completion of this learning guide, you will be able to:

- Define and importance of CAD system
- Understanding CAD software and its applications
- Explain function of Auto CAD
- Boot up the auto CAD software
- Describe Auto CAD Interface and tool

1.1. Computer Aid Design system

1.1.1. Introduction

CAD, or computer-aided design, is the use of computers to aid in the creation, modification, analysis, or optimization of a design. CAD software is used by a wide range of professionals, including architects, engineers, product designers, and artists.

CAD (Computer-Aided Design) systems are software tools used by engineers, architects, and designers to create, modify, analyze, and optimize designs. These systems provide a digital representation of physical objects or structures, allowing users to visualize and manipulate them before they are built.

CAD systems have revolutionized the design industry by replacing traditional manual drafting methods. They provide a more efficient and accurate way to create complex designs, reducing errors and saving time. CAD systems also enable designers to easily make changes and iterations to their designs, without having to start from scratch.

CAD software can be used to create both 2D drawings and 3D models. 2D drawings are typically used for construction or manufacturing documentation. 3D models are used for a variety of purposes, including product design, prototyping, and animation.

In addition, CAD systems integrate with other software tools and technologies, such as computer-aided manufacturing and computer-aided engineering. This allows for a seamless workflow from design to production, enabling the creation of prototypes and the simulation of real-world scenarios.

1.1.2. The importance of CAD software

CAD software is important because it offers a number of advantages over traditional manual design methods, including:

- **Increased accuracy and precision:** CAD software allows designers to create highly accurate and precise drawings and models. This is important for many industries, such as engineering and manufacturing, where even small errors can have serious consequences.

- **Improved productivity and efficiency:** CAD software can help designers to work more quickly and efficiently. This is because CAD software automates many of the tasks that would otherwise have to be done manually.
- **Enhanced communication and collaboration:** CAD software makes it easier for designers to communicate and collaborate with each other. This is because CAD software allows designers to share drawings and models with each other electronically.
- **Greater flexibility and ease of design changes:** CAD software makes it easy to make changes to designs. This is because CAD software is parametric, meaning that changes to one part of a design can be automatically reflected in other parts of the design.
- **Reduced costs:** CAD software can help to reduce costs by helping designers to create more efficient designs and by reducing the need for physical prototypes.

1.1.3. CAD software and its applications

The best CAD software for you will depend on your specific needs and requirements. However, here are a few of the most popular and well-regarded CAD software options available

1. **Autodesk Fusion 360:** Fusion 360 is a cloud-based CAD, CAM, and CAE tool for product development. It offers a comprehensive set of features for creating, modifying, and analyzing 2D and 3D designs. Fusion 360 is also notable for its ease of use and its affordable pricing.
2. **On shape**: On shape is another cloud-based CAD software program that is known for its ease of use and collaboration features. On shape is a good choice for teams of designers who need to be able to work on projects together from anywhere in the world.
3. **Autodesk AutoCAD:** AutoCAD is one of the most popular CAD software programs in the world. It is known for its powerful features and its wide range of applications. AutoCAD is a good choice for experienced CAD users who need a powerful and versatile software program.
4. **PTC Creo:** Creo is a powerful CAD software program that is used by many large manufacturing companies. It is known for its advanced design and simulation capabilities. Creo is a good choice for experienced CAD users who need a software program that can handle complex designs.

5. **SolidWorks:** SolidWorks is another popular CAD software program that is known for its ease of use and its wide range of applications. SolidWorks is a good choice for both experienced and novice CAD users.
6. **Solid Edge:** Solid Edge is a CAD software program that is known for its ease of use and its affordability. It is a good choice for novice CAD users or for small businesses that need a powerful CAD software program.
7. **SketchUp:** SketchUp is a popular 3D modeling software program that is known for its ease of use. It is a good choice for novice CAD users or for users who need to create 3D models for non-technical purposes.
8. **Blender:** Blender is a free and open-source 3D modeling software program that is known for its powerful features and its wide range of applications. Blender is a good choice for both experienced and novice CAD users.
9. **Revit:** Revit is a building information modeling (BIM) software program developed by Autodesk. It is used by architects, engineers, contractors, and other professionals to create and maintain digital representations of buildings and other structures. Revit software can be used to create 3D models, generate drawings and documentation, and track project progress.

When choosing a CAD software program, it is important to consider the following factors:

- Features
- Ease of use
- Cost
- Industry support

In our work Autodesk AutoCAD, the best ways.

1.1.4. Concept of Autodesk AutoCAD

AutoCAD is a commercial computer-aided design (CAD) and drafting software application developed and marketed by Autodesk. AutoCAD was first released in December 1982, running on microcomputers with internal graphics controllers. It was one of the first CAD programs to be widely adopted by personal computer users, and has remained the leading CAD software ever since.

AutoCAD can be used to create 2D and 3D drawings, and is used by a wide range of professionals, including architects, engineers, designers, and drafters. AutoCAD is also used by

students and hobbyists. AutoCAD is a powerful and versatile CAD software program that offers a wide range of features, including:

- **2D drafting:** AutoCAD provides a complete set of tools for creating and editing 2D drawings, including lines, arcs, circles, rectangles, polylines, and splines. It also includes tools for creating and managing layers, dimensions, and text.
- **3D modeling:** AutoCAD also includes a full set of tools for creating and editing 3D models. It supports a variety of 3D modeling techniques, including solid modeling, surface modeling, and mesh modeling.
- **Collaboration tools:** AutoCAD includes a variety of tools for collaboration, including the ability to share drawings with others, work on drawings with others simultaneously, and track changes to drawings.
- **Data management**: AutoCAD includes a variety of tools for data management, including the ability to create and manage libraries of drawings, symbols, and blocks.
- **Plotting:** AutoCAD includes a variety of tools for plotting drawings, including the ability to plot to paper, to plot to electronic files, and to plot to 3D printers.

AutoCAD is a powerful and versatile CAD software program that can be used to create a wide range of drawings and models.

The benefits of using AutoCAD

- **Accuracy and precision:** AutoCAD allows users to create highly accurate and precise drawings and models. This is important for many industries, such as engineering and manufacturing, where even small errors can have serious consequences.
- **Productivity and efficiency:** AutoCAD can help users to work more quickly and efficiently. This is because AutoCAD automates many of the tasks that would otherwise have to be done manually.
- **Enhanced communication and collaboration:** AutoCAD makes it easier for users to communicate and collaborate with each other. This is because AutoCAD allows users to share drawings and models with each other electronically.

- **Greater flexibility and ease of design changes:** AutoCAD makes it easy to make changes to designs. This is because AutoCAD is parametric, meaning that changes to one part of a design can be automatically reflected in other parts of the design.
- **Reduced costs:** AutoCAD can help to reduce costs by helping users to create more efficient designs and by reducing the need for physical prototypes.

1.2. Boot up the Auto CAD software

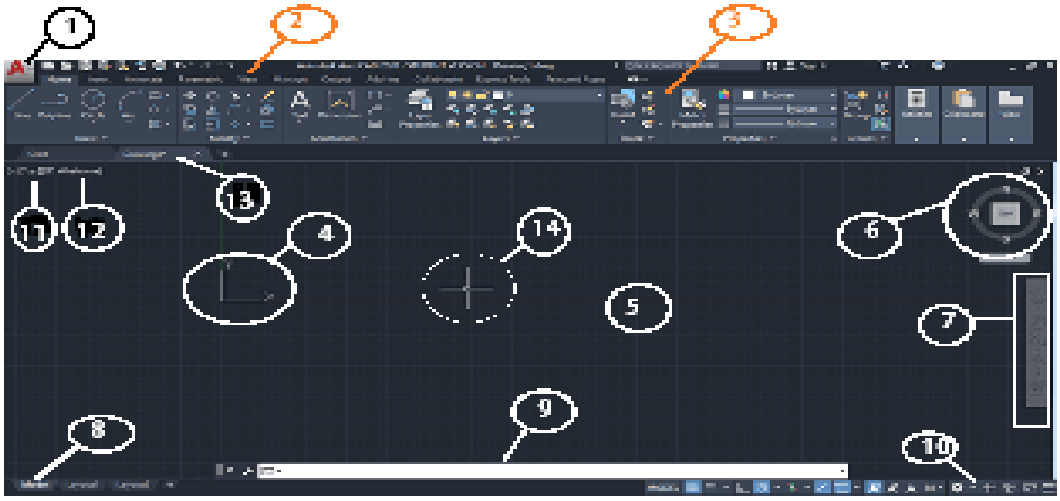
Booting up Auto CAD software typically involves the following steps:

1. **Launch the Auto CAD Software:** Locate the Auto CAD software icon on your computer's desktop or in the Start menu. Double-click on the icon to launch the software.
2. **Wait for the Software to Load:** Depending on your computer's speed and the complexity of the Auto CAD software, it may take a few seconds or minutes for the software to load. During this time, the software may display a splash screen or a loading indicator.
3. **Choose/Create a New Project or File:** Once the Auto CAD software is loaded, you will be prompted to either create a new project or open an existing file. If you are starting a new design, choose the option to create a new project. If you have an existing design file, choose the option to open the file.
4. **Set up Project Setting (if applicable):** If you are creating a new project, the Auto CAD software may prompt you to set up project settings such as units of measurement, drawing scale, and other project-specific parameters. Follow the prompts and enter the required information to set up the project.
5. **Familiarize yourself with the User Interface:** Once the project is set up or the file is opened, take a moment to familiarize yourself with the Auto CAD software's user interface. Explore the menus, toolbars, and panels to understand the available tools and features.
6. **Start designing or Modelling:** With the Auto CAD software booted up and the user interface ready, you can now start designing or modelling your project. Use the appropriate tools and commands to create, modify, and analyze your design. Refer to the software's documentation or help resources for guidance on specific tools and techniques.

It's important to note that the exact procedure for booting up an **auto** CAD software may vary depending on the specific software you are using. The steps mentioned above provide a general overview of the process. Always refer to the software's documentation or help resources for detailed instructions specific to the **auto** CAD software you are working with.

1.3. Auto CAD Interface and Tools

The AutoCAD interface is a graphical user interface (GUI) that provides users with a platform to create, edit, and manipulate digital drawings in a variety of industries such as architecture, engineering, and construction. AutoCAD, developed by Autodesk, is one of the most widely used computer-aided design (CAD) software applications. The AutoCAD interface is designed to be intuitive and user-friendly, with various elements that enable efficient navigation and access to tools and commands.



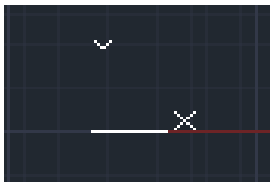
Figures.1.1. AutoCAD interface

The name of the numbered parts is listed below:

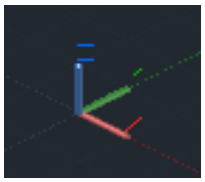
1. **Application menu/button:** It is present at the upper-left corner of the workspace. To close the application menu, we can click anywhere outside the application button or window. The button is shown in the image marked as number 1.

2. **Quick Access Toolbar:** The Quick Access Toolbar is located at the top of the application window and right of the application menu. It consists of the set of frequently used commands. We can add and remove the commands according to the requirements.
3. **Ribbon Panel:** It provides access to the dialog box related to that panel. If the ribbon panel disappears, then we need to enter the 'ribbon' on the command line to display the ribbon panel on the workspace.
4. **User Coordinate System (UCS):** The UCS is the active coordinate system that represents the XY plane in 2D and XYZ planes in 3D. It acts as the direction in the X, Y, and Z-axis for drawing and modeling. We can control the origin and orientation of the UCS to make drawings according to the specific points and coordinates. We can also work with drawing aids such as Grid and the ortho mode.

   The 2D and 3D UCS is shown in the given image:



Figures.1.2. 2d User Coordinate System

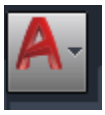


Figures.1.3. 3d User Coordinate System

5. **Model Space / Work Space / Drawing Window:** It is defined as the area to create 2D and 3D drawings, models, and objects. We can create using different commands according to the requirements.
6. **View Cube**: The View Cube is termed as the navigation tool that is displayed when we are working on a 2D or 3D model space. We can switch between the isometric view and the standard view of our drawings or model.
7. **Navigation Bar:** The Navigation Bar is used to access the navigation tools. It is a user interface element, where we can access both unified (common tools) and product-specific tools (unique product tools). The navigation bar is shown in the below image:

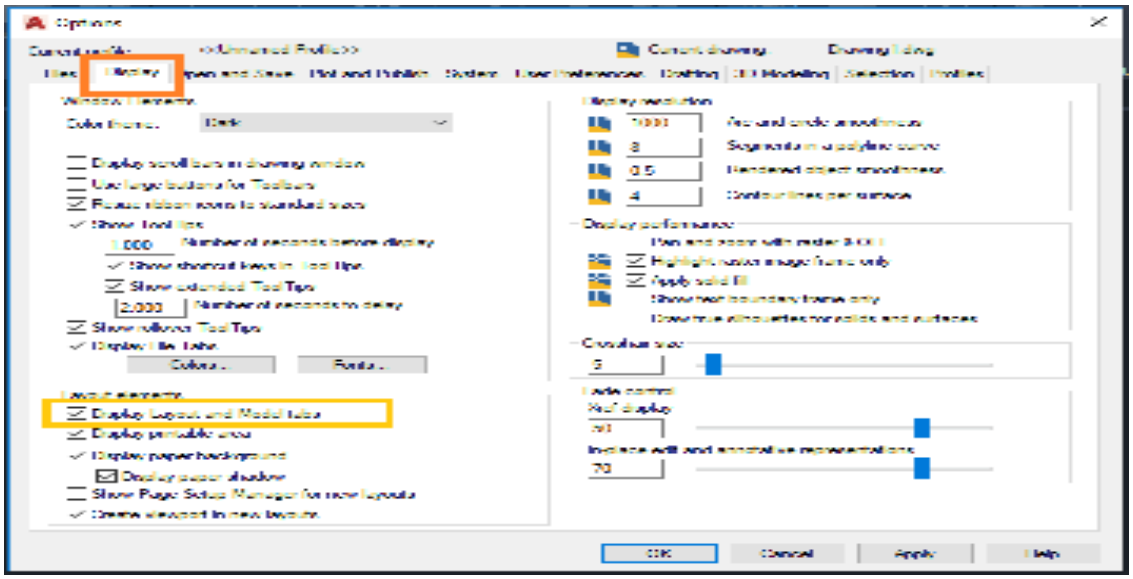
Figures.1.4. navigation bar

8. **Model Layout Tab:** A layout is defined as a 2D working environment for creating the drawing sheets. The model tab is the screen where we create the 2D and 3D drawings. If the model and layout tab is not visible, then follow the steps given below:
   a. We need to click on the AutoCAD option displayed on the top left corner of the screen, shown as



Figures.1.5. AutoCAD option

   b. Click on the 'Options' button at the bottom.
   c. Then on the 'display' option on the top, select the option 'Display Layout and Model tabs' and then click 'Ok' as shown below:

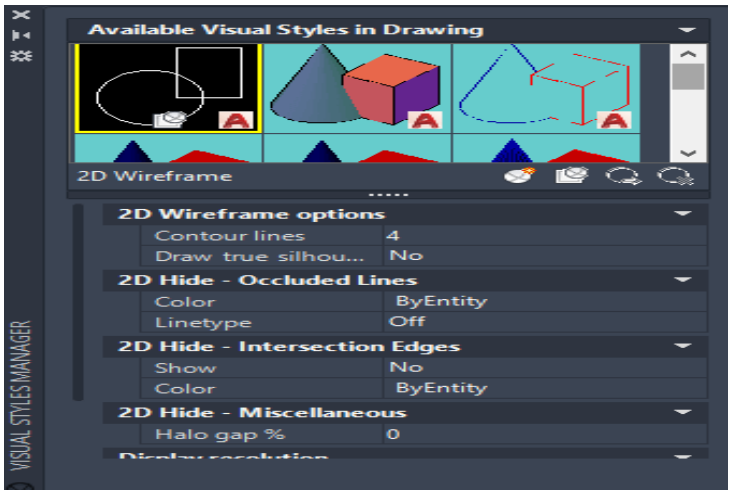


Figures.1.6. AutoCAD option display

9. **Command Line Window (shortcut key- ctrl+9)**: The command line window is used to write the commands. We need to press 'Enter' after typing any particular command. The AutoCAD also displays the steps after each command on the Command line. We need to select the option and press 'Enter' after each step.
10. **Status Bar:** The status bar displays the drawing tools that affect the drawing environment. It provides quick access to most of the commonly used drawing tools. It includes options such as ISODRAFT, ORTHOMODE, AUTOSNAP, OSNAP, etc.

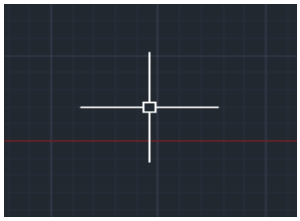
11. **View Control:** The View Control is displayed on the left corner of the viewport. It provides a suitable way to change views, visual settings, and styles. It includes options such as top, bottom, left, right, etc.
12. **Visual Style Control:** The Visual Style Control is the customized setting that controls the structure view of the models or 3D drawings created on the Viewport. The options include 2D Wireframe, Realistic, Shaded, Conceptual, Hidden, etc.

We can also enter the visual styles on the command line to display the options of the visual style control. After entering on the command line, the box will appear displaying clear information with figures for each option. It is shown in the below image:



Figures.1.7. Visual Style Control

13. **File Tab:** The file tab consists of the current drawing files opened on the screen. We can also type FILETAB command in the command line to turn on turn on the file tabs. To close the file tab, we can type FILETABCLOSE command in the command line window. We can click on the '+' sign to add a new tab on the File Tab.
14. **Mouse Cursor:** The cursor used to draw figures, etc. is known as the mouse cursor. The cursor in 2D and 3D is shown in the given image:



Figures.1.8. 2D Mouse Cursor

Figures.1.9. 3D Mouse Cursor

Self-check 1

I. Choose the best answer from the given alternatives

1. What does CAD stand for?
   a) Computer-Aided Design
   b) Computer-Aided Development
   c) Computer-Assisted Drafting
   d) Computer-Animated Design
2. Which of the following is NOT an importance of CAD software?
   a) Increased productivity and efficiency
   b) Improved accuracy and precision
   c) Enhanced collaboration and communication
   d) Reduced cost and time in the design process
   e) None
3. Which type of CAD software is primarily used for architectural design and drafting?
   a) 2D CAD software
   b) 3D CAD software
   c) Mechanical CAD software
   d) Electrical CAD software
4. Which CAD software is widely used in the engineering?
   a) AutoCAD
   b) SketchUp
   c) SolidWorks
   d) Fusion 360
5. Which CAD software is developed by Autodesk?
   a) AutoCAD
   b) CATIA
   c) Pro/ENGINEER
   d) Siemens NX

II. Give short & precis answer for the following questions

1. What is the importance of CAD software in the design process?
2. Name two types of CAD software and provide an example of their application.
3. Explain the process of booting up AutoCAD and briefly describe its interface and tools.

UNIT TWO: BASIC 2D DRAWING

This unit is developed to provide you the necessary information regarding the following content coverage and topics:

- Controlling the display in drawings
- Basic parameters AutoCAD
- basic 2d drawing
- Modifying and Editing Drawings
- Dimensioning

This unit will also assist you to attain the learning outcomes stated in the cover page. Specifically, upon completion of this learning guide, you will be able to:

- Controlling the display in drawings
- Using basic parameters AutoCAD
- Creating basic 2d drawing
- Modifying and Editing Drawings
- Applying dimensioning

2.1. Controlling the Display in Drawings

Controlling the display in drawings refers to various settings and commands that allow you to customize how your drawing is displayed in AutoCAD. Here is a brief explanation of each of the mentioned features:

I. Drawing Units Setup

To control the display of units in your drawings, you can set up the drawing units. This allows you to specify the unit type, precision, and format for your drawings. To set up the drawing units, follow these steps:

1. Go to the "Format" menu.
2. Select "Units" from the dropdown menu.
3. In the Units dialog box, choose the desired unit type, precision, and format.
4. Click OK to apply the changes.

1. Go to the "Format" menu: The AutoCAD Menu Bar contains multiple pulldown menus, where all of the AutoCAD commands can be accessed. Note that many of the menu items listed in the pull-down menus can also be accessed through the Quick Access toolbar and/or Ribbon panels.
2. Click on the Length Type option to display the different types of length units available. Confirm the Length Type is set to **Decimal.**
3. On your own, examine the other settings that are available.
4. Set the Precision to **two digits** after the decimal point as shown in the above figure.
5. Pick OK to exit the Drawing Units dialog box.

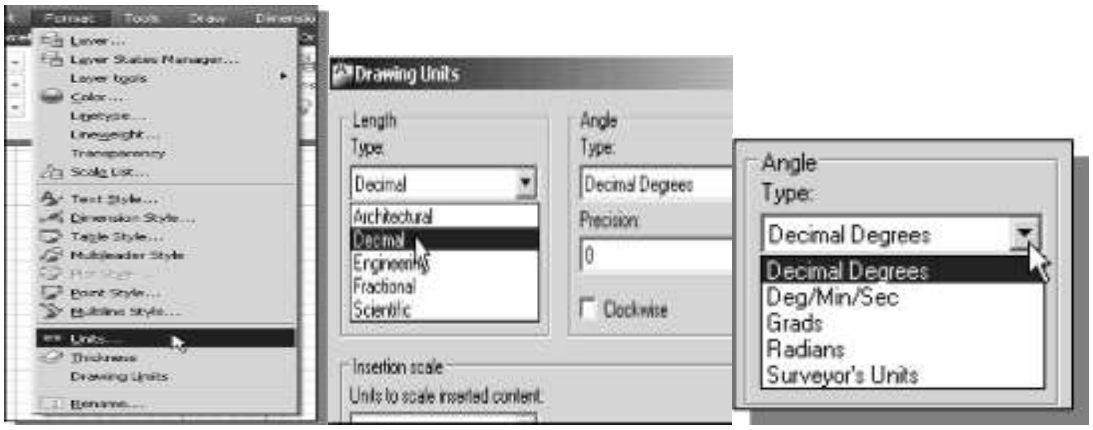


Figure2.1. Drawing Units

II. Zoom Extent

The Zoom Extent command allows you to quickly adjust the view of your drawing to fit all objects within the drawing area. To use this command, follow these steps:

1. Type "ZOOM" in the command line.
2. Select "Extents" from the options or press Enter.
3. AutoCAD will automatically adjust the view to fit all objects within the drawing area.

Inside the Menu Bar area select:

[View] → [Zoom] → [All]

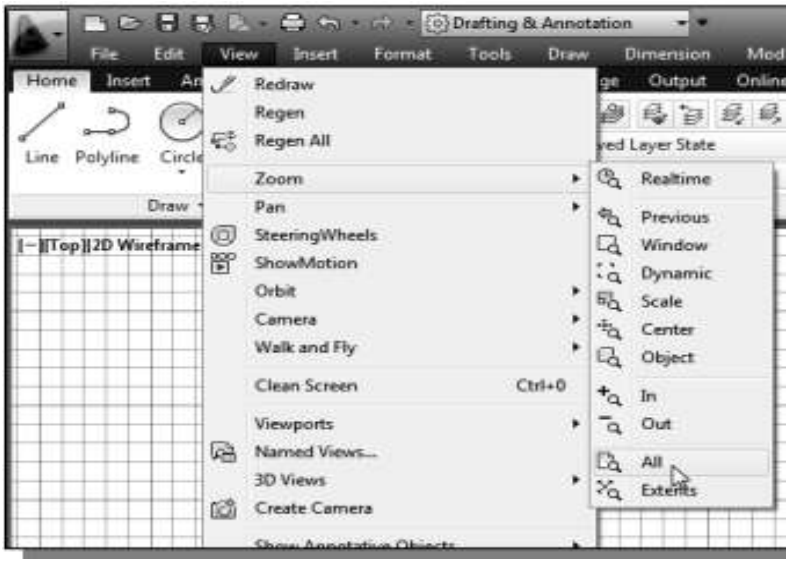


Figure:2.2. Drawing Zoom Extent

The Zoom All command will adjust the display so that all objects in the drawing are displayed to be as large as possible. If no objects are constructed, the Drawing Limits are used to adjust the current viewport.

III. Drawing LIMITS

The Drawing Limits command allows you to set the limits of the drawing area. This controls the area in which objects can be created and displayed. To set the drawing limits, follow these steps:

1. Type "LIMITS" in the command line.
2. Specify the desired limits by entering the X and Y coordinates.
3. Press Enter to apply the changes.

Note: The limits can also be adjusted by using the "Limits" option in the "Format" menu.In the Menu Bar select:

[Format] → [Drawing Limits]

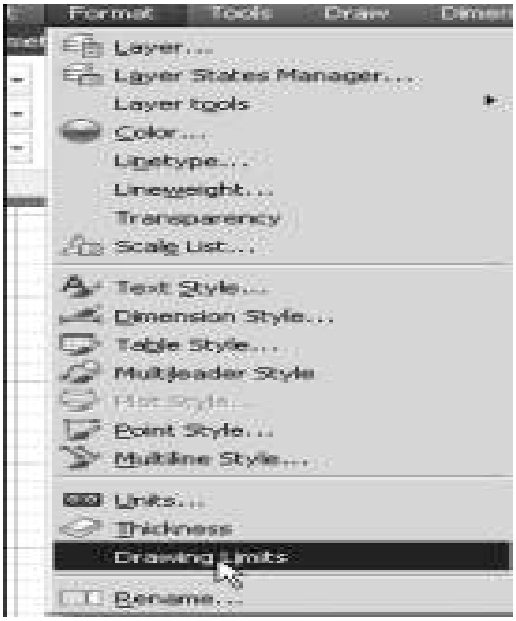


Figure:2.3. Drawing limits

a. In the command prompt area, the message "Reset Model Space Limits: Specify lower left corner or [On/Off] <0.00,0.00>:" is displayed. Press the ENTER key once to accept the default coordinates <0.00,0.00>

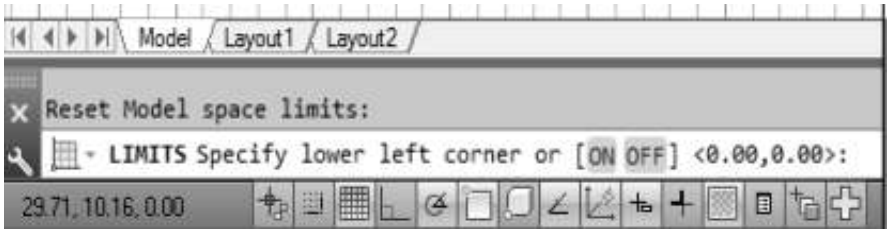


Figure:2.4. Drawing limits Specify lower left corner

b. In the command prompt area, the message "Specify upper right corner <12.00,9.00>:" is displayed. Press the ENTER key again to accept the default coordinates <12.00,9.00>.

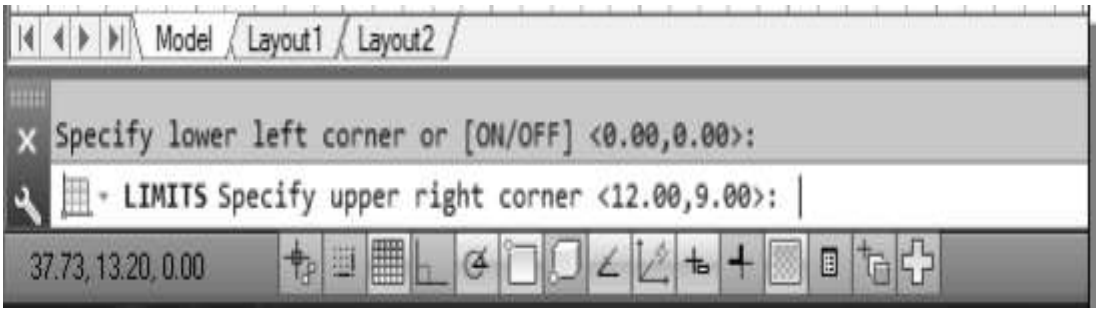


Figure:2.5. Drawing limits Specify upper right corner

IV. Status Bar

The Status Bar in AutoCAD is a horizontal bar located at the bottom of the application window. It provides important information and access to various settings and commands. Displays your current settings. These settings can be turned on and off by clicking on the word (Snap, Grid, Ortho, etc.) or by pressing the function keys, F1, F2, etc. See button descriptions below.

Figure:2.6. Status Bar

- **[SNAP]:** Increment Snap controls the movement of the cursor. If it is off, the cursor will move smoothly. If it is ON, the cursor will jump in an incremental movement. The increment spacing can be changed at any time using **Tools / Drafting Settings/ Snap and Grid**.
- **GRID** : The grid (dots) is merely a visual "drawing aid". The default spacing is 1 unit. You may change the grid spacing at any time using: **Tools / Drafting Settings /Snap and Grid.**
- **ORTHO**: When Ortho is ON, cursor movement is restricted to horizontal or vertical. When Ortho is OFF, the cursor moves freely.
- **POLAR:** POLAR TRACKING creates “Alignment Paths” at specified angles.
- **OSNAP:** RUNNING OBJECT SNAP Specific Object Snaps can be set to stay active until you turn them off.
- **OTRACK:** OBJECT SNAP TRACKING Creates “Alignment Paths” at precise positions using object snap locations.
- **LWT:** LINEWEIGHT. Displays the width assigned to each object.
- **MODEL:** Switches your drawing between paper space and model space.
- **Coordinate System:** The Status Bar shows the current coordinate system being used in the drawing, such as Cartesian or Polar coordinates. You can toggle between different coordinate systems by clicking on the coordinate system button on the Status Bar.

- Location: Lower left corner of the screen. The UCS icon indicates the location of the Origin. The UCS icon appearance can be changed using: **View / Display / Icon / Properties**.
- **ORIGIN:** The location where the X, Y and Z axes intersect. 0,0,0

Function Keys

- F1 Help Explanations of Commands.
- F2 Flip screen Toggles from Text Screen to Graphics Screen.
- F3 Osnap Toggles Osnap On and Off.
- F4 Tablet Toggles the Tablet On and Off.
- F5 Is plane Changes the is plane from Top to Right to Left.
- F6 Coordinate Display Changes the display from ON / Off /.
- F7 Grid Toggles the Grid On or Off.
- F8 Ortho Toggles Ortho On or Off.
- F9 Snap Toggles Increment Snap on or off.
- F10 Polar Toggles Polar Tracking On or Off.
- F11 Otrack Toggles Object Snap Tracking On and Off.

V. **GRID Display:** The GRID feature displays a grid of horizontal and vertical lines on the drawing area. It helps in aligning objects and maintaining consistent spacing. You can toggle the grid display on or off as per your requirement.

VI. **PAN Realtime:** The PAN command allows you to shift or move the view of your drawing in real-time without changing the zoom level. It is a convenient way to navigate and explore different areas of your drawing without altering the scale or perspective.

To use the PAN command in real-time, follow these steps:

1. Open your drawing in AutoCAD.
2. Activate the PAN command by either:
    a. Typing "PAN" in the command line and pressing Enter, or
    b. Clicking on the PAN button in the navigation toolbar.
3. Once the PAN command is active, your cursor will change to a hand symbol.
4. Click and hold the left mouse button to grab the drawing.

5. While holding the mouse button, move the cursor in the desired direction to shift the view of the drawing.
6. Release the mouse button to stop panning

2.2. Basic Parameters of Auto CAD

The basic parameters of AutoCAD related to layers, colours, and line types are essential for organizing and managing your drawings. They allow you to control the visibility, appearance, and plot settings of objects.

2.2.1. Layers

Layers in AutoCAD are used to organize and manage different elements of your drawing. They provide a way to group related objects together and control their visibility, properties, and organization. Some key points about layers:

- Layers are like transparent sheets that can contain various objects such as lines, dimensions, text, hatches, etc.
- Each layer can have a unique name to identify its purpose or content.
- You can assign objects to specific layers based on their properties, such as color, line type, line weight, and visibility.
- Layers can be turned on or off to control the visibility of objects on different layers.
- By controlling the properties of layers, you can easily make changes to specific objects or groups of objects in your drawing.

How to creating and modifying layer

a. **Create Layers:** Use the "Layer Properties Manager" or type "LA" and press Enter to open the Layer Properties Manager. Click on the "New Layer" button to create new layers and specify their names, colors, line types, and other properties.
b. **Set Current Layer:** To draw objects on a specific layer, select the desired layer in the Layer Properties Manager or use the "Layer" drop-down menu in the ribbon. The current layer determines where new objects are placed.

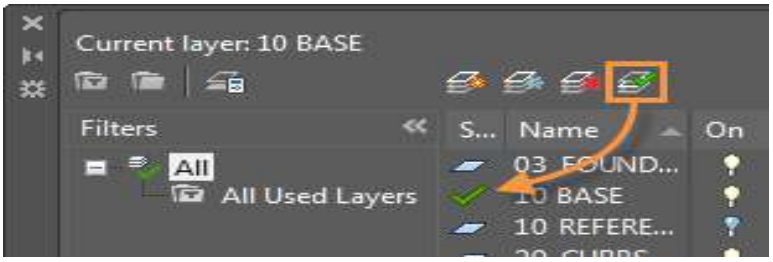


Figure:2.7. Set Current Layer

c. **Modify Layer Properties:** In the Layer Properties Manager, you can modify layer properties such as color, line type, line weight, and plot settings. You can also freeze, thaw, lock, or unlock layers to control object visibility and editing.

Figure:2.8. Modify Layer Properties

Example: The steps are listed below:

a) Type **LA** on the command line < press **Enter**.

b) A dialog box will appear.

c) Click on the New Layer icon on the top of the dialog box.

d) A new layer will appear, as shown below:

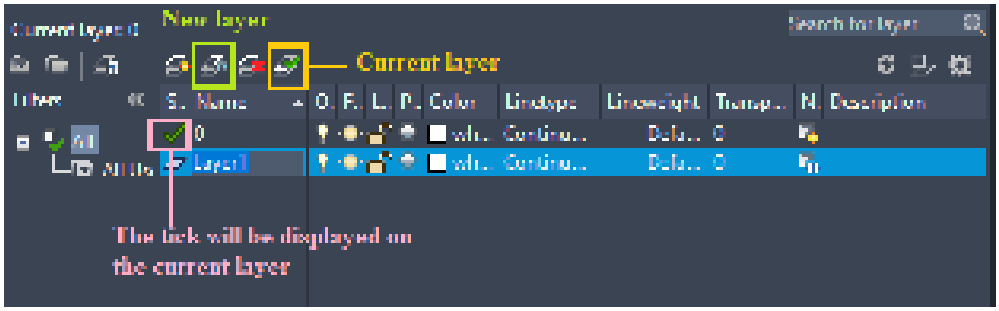


Figure:2.9. New Layer

e) Specify the name and color. We can also change Line type, Transparency, Layer FREEZE, etc. according to the requirements.

f) Let's create total 6 layers. It is shown below:

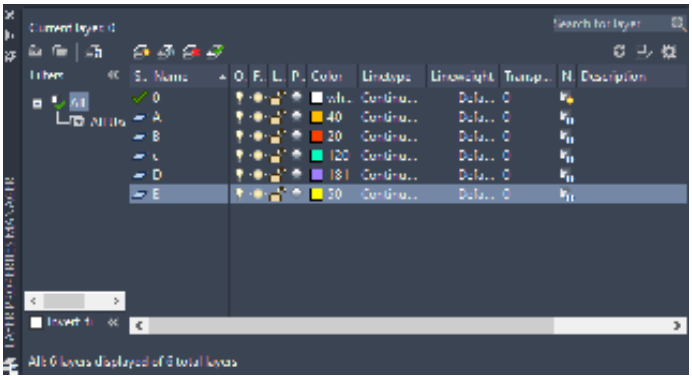


Figure:2.10. Created Layer

g) To create an object, specify the layer as the current layer.

h) The objects will be created based on the selected current layer.

i) To use all layers in the drawing, keep changing the current layer according to the object.

j) Let's draw an object with the use of all the above layers.

k) The object is shown below:

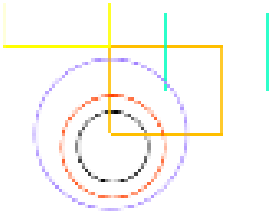


Figure:2.11. Layer type

The above object is just an example to explain the concept of layers.

In this example, the display is limited to the objects on the Floor Plan layer by turning off the other layers

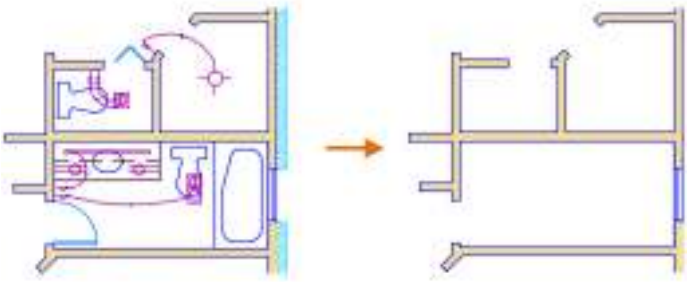


Figure:2.12. Floor Plan layer

2.1.1. Colors

Colors play an important role in AutoCAD drawings as they help differentiate and distinguish different entities.

What you need to know about colors in AutoCAD:

- AutoCAD supports a wide range of colors that can be applied to various objects in your drawing.
- Colors can be assigned to objects based on their properties, such as layers or individual objects.
- AutoCAD provides both indexed colors and true colors. Indexed colors are limited to a predefined color palette, while true colors allow you to specify any RGB color value.
- You can modify the color of objects using the properties palette, the layer properties manager, or by using the color commands.

Figure:2.13. Select Color HSL and RGB.

How to assign and modifying Colours?

a. Assign Object Colours: Select objects, right-click, and go to "Properties" or use the "Properties" panel. In the "Properties" window, you can change the color of selected objects by clicking on the colour swatch.

b. Modify Color Properties: In the "Layer Properties Manager," you can assign colors to layers. Objects on the same layer will inherit the layer color unless their individual color is explicitly set.

2.2.2. Line types

Line types define the appearance of lines in your AutoCAD drawing and help convey different types of information. Here are some details about line types:

- AutoCAD offers various pre-defined line types, such as continuous, dashed, dotted, centerline, hidden, and more.

- Each line type has a specific pattern of dashes, dots, or gaps that determine its appearance.

- Line types can be assigned to objects based on their properties, such as layers or individual objects.
- AutoCAD allows you to create custom line types using the "LINETYPE" command. You can define the pattern and scale of the custom line type.
- Line types can be modified and managed using the "LINETYPE" command and the "LINETYPES" dialog box.

Understanding and effectively using layers, colors, and line types in AutoCAD helps you organize your drawing, convey information clearly, and create visually appealing designs.

How to assign and modifying Line Types

a. **Assign Line Types:** Select objects, right-click, and go to "Properties" or use the "Properties" panel. In the "Properties" window, you can change the line type of selected objects by clicking on the line type dropdown.
b. **Modify Line Type Properties:** In the "Layer Properties Manager," you can assign line types to layers. Objects on the same layer will inherit the layer's line type unless their individual line type is explicitly set.

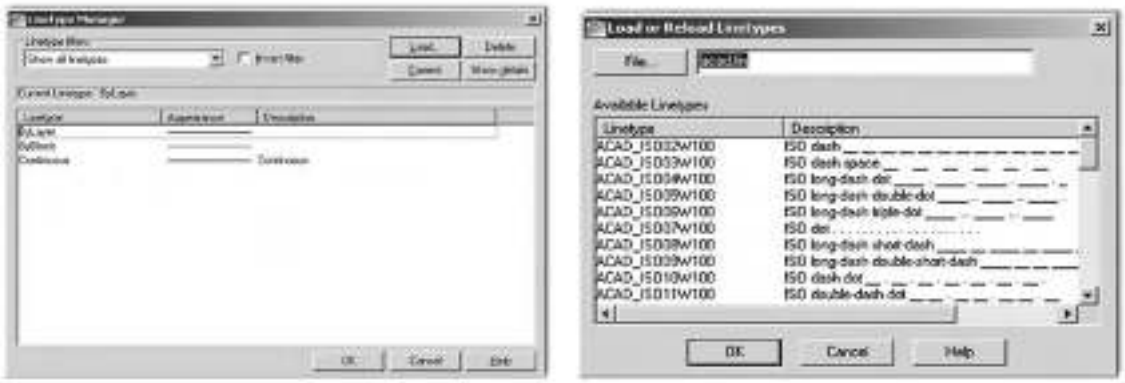


Figure:2.14. Line Type

Example: The steps to use the line type into the current drawing are listed below:

a) Type **LINETYPE** or **LT** on the command line or command prompt and press Enter.
b) Click on the **'Load...'** button on the dialog box.
c) Select the desired line type from the list. Here, we have selected the **Zigzag** line type.

d) Click on the chosen Line type and press **OK** button, as shown below:

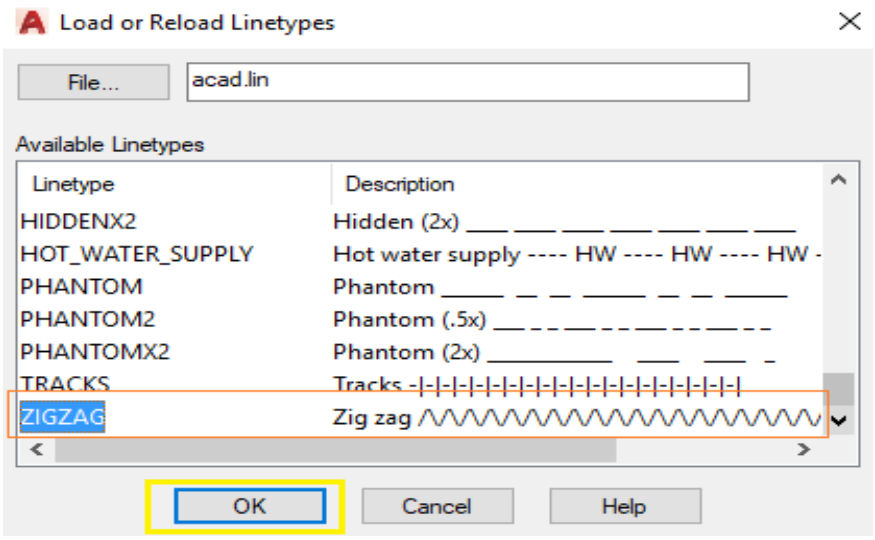


Figure:2.15. load Line Type

e) The Line type Manager will display the selected Zigzag Line type. It is shown below:

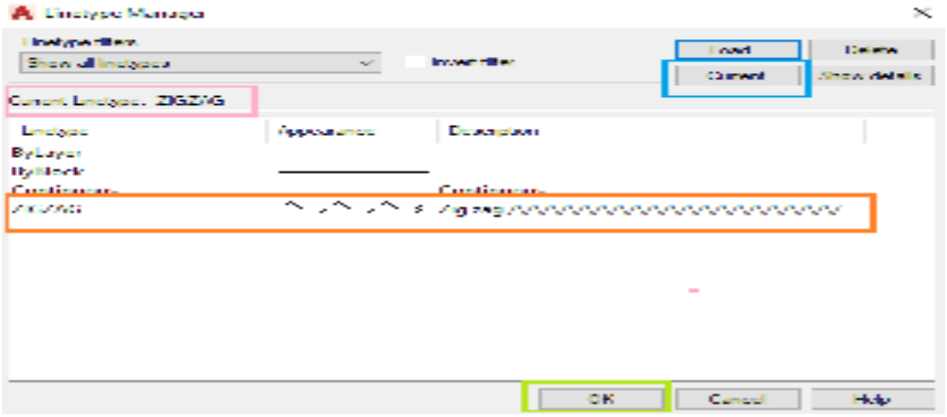


Figure:2.16. Zigzag Line Type

f) Click again on the Zigzag line type and press the **'Current'** button to reflect the line type into the current drawing.

g) Click on **OK** button. Now, let's create a square and two circles. It will look like the below image:

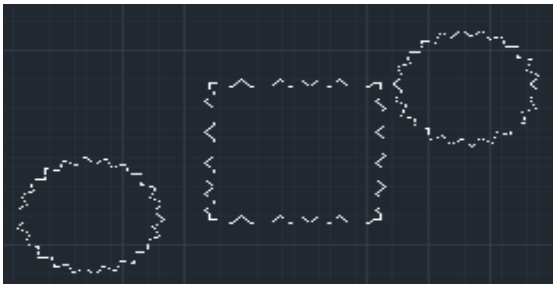


Figure:2.17. circle and rectangle by Zigzag Line Type

2.3. Basic 2D drawing

2.3.1. Introduction

Drawing and drafting are fundamental skills used in various fields such as architecture, engineering, design, and art. These skills involve creating accurate and detailed representations of objects, plans, or ideas on paper or digitally.

Drawing is the act of making marks on a surface to represent something visually. It can be done using various tools such as pencils, pens, charcoal, or digital software. Drawing can be used for artistic expression, creating illustrations, or simply for sketching ideas.

Drafting, on the other hand, is a more technical and precise form of drawing. It involves creating detailed plans, diagrams, or technical drawings that are used in fields like architecture and engineering. Drafting often requires the use of specialized tools and techniques to ensure accuracy and clarity.

Both drawing and drafting require a combination of observation, measurement, and creativity. They involve understanding perspective, proportions, shading, and other principles of visual representation.

Learning basic drawing and drafting skills can be beneficial for anyone interested in expressing ideas visually, communicating designs or plans, or simply exploring their creativity. These skills can be honed through practice, studying reference materials, and taking courses or workshops.

2.3.2. Drawing Tools

1. Line:

Lines are fundamental elements in 2D drawing and drafting. They are used to represent edges, outlines, or paths of objects. Key aspects of lines include:

- Different line types: Lines can be solid, dashed, dotted, or have custom patterns, serving specific purposes or indicating different features.
- Line weights: Lines can have different thicknesses or weights to distinguish between various elements or emphasize certain details.

How you can create these shapes:

- Type "Line" in the command line or click on the "Line" icon in the "Draw" toolbar.

- Specify the starting point of the line by clicking on the desired location in the drawing area.
- Specify the endpoint of the line by clicking on another location or typing coordinates.

Figure:2.18. Line drawing tool

2. **Circle:** Circles are closed curves consisting of all points equidistant from a common center. In 2D drawing and drafting, circles are used to represent cylindrical or rounded features. Key features of circles include:

- Center points and radius: Circles are defined by a center point and a radius, which determine their size and position.
- Diameter: The diameter is twice the radius and represents the distance across the circle passing through the center.

Example 1:

The steps to create a circle are given below:

1. Click on the **Circle** icon on the Ribbon Panel, as shown in the below image:

Figure:2.19. Circle drawing tool

OR Type **Circle** or C on the command line and press **Enter**.

2. Specify the center point of a circle on the workspace or viewport.
3. Specify the **radius** of the circle.

   Or To specify **diameter**, type **D** - press **Enter** - specify the diameter of the circle.
4. Press **Enter**.

Types of Circles

There are three types of circles, which are listed below:

1. 2-point circle
2. 3-point circle
3. Tangent circle

1. **2 Point circle:** We can create a circle by specifying the **two** endpoints. The distance between the two specified endpoints will be considered as the diameter of that circle.

Example: A circle between two lines. The steps to create a **two**-point circle are listed below:

a) Select the 2-Point circle option from the drop-down list of the circle

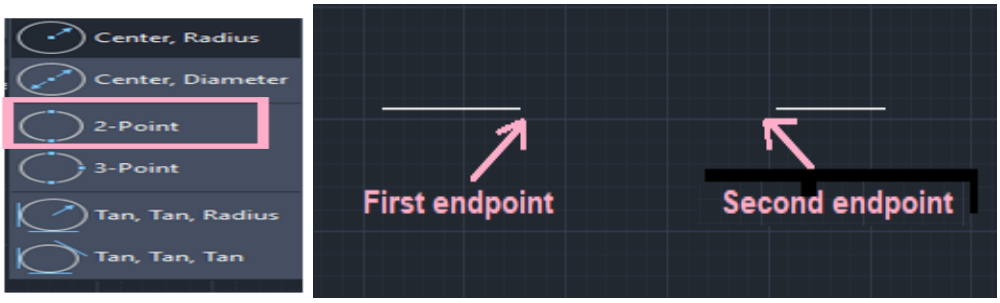


Figure:2.20. 2 Point Circle

Or Type **C** or **Circle** on the command line – Press **Enter**- type **2P** – press **Enter**.

b) Click on the first endpoint of a circle diameter. We can also type the value to specify the endpoint.

c) Click on the second endpoint of a circle diameter through the cursor

d) The circle will be created, as shown in the below image:

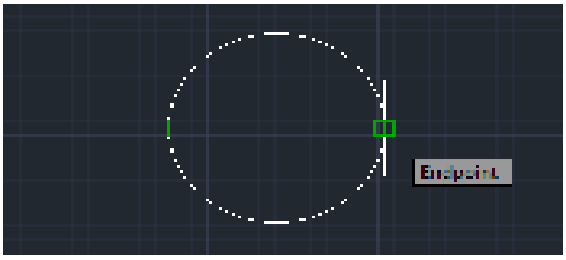


Figure:2.21. created Circle

2. **3 Point circle:** We can create a circle by specifying the **three** endpoints.

Example: A circle between three edges. The steps to create a **three**-point circle are listed below:

a) Create three edges.

b) Select the **3-Point** circle option from the drop-down list of the circle, as shown in the below image:

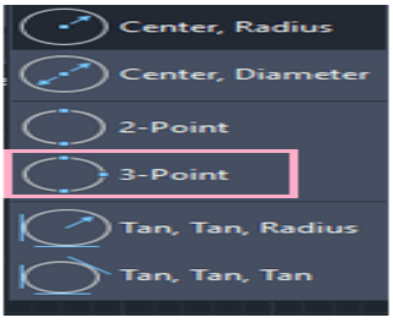


Figure:2.22. 3 Point Circle

Or Type **C** or **Circle** on the command line - Press **Enter**- type **3P** - press **Enter**.

c) Click on the **first** point.

d) Click on the **second** point.

e) Click on the third point. The three points are shown in the below image:

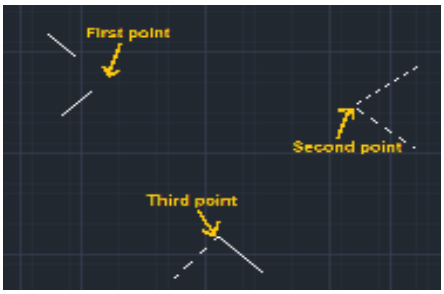


Figure:2.23. 3 Point Circle

f. The circle will be drawn, as shown in the below image:

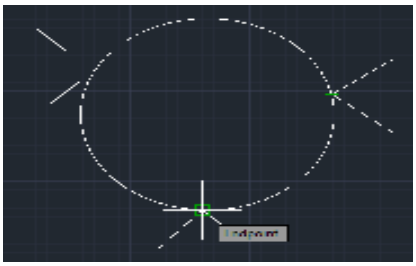


Figure:2.25. Created Circle

3. Tangent circle: The tangent circle command is used to draw circles on the tangent. There are two types of the tangent circle, that appear on the drop-down list of the circle icon on the ribbon panel, as shown in the below image:

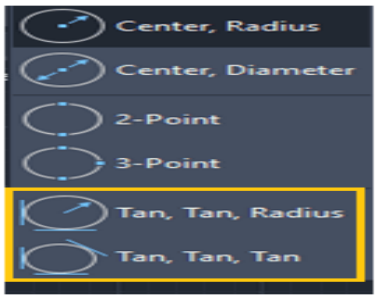


Figure:2.26. Tangent circle

Example 1: Tan, Tan, Radius. The steps to create a **Ttr** (Tangent tangent radius) circle are listed below:

a) Draw two lines in the viewport, as shown in the below image:

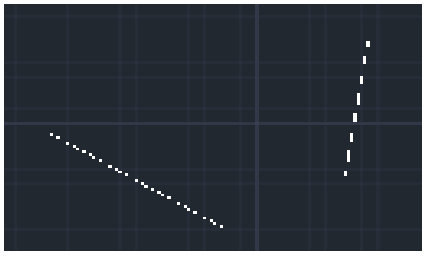


Figure:2.27. Draw two lines

b) Select the **Tan, Tan, Radius** circle option from the drop-down list of the circle on the ribbon panel.

   Or Type **C** or **Circle** on the command line - Press **Enter**- type **ttr** - press **Enter**.

c) Click on the first point on the first tangent.

d) Click on the second point on the other tangent.

e) Specify the radius of the circle. For example, **4**.

f) The circle will be drawn on the two tangents, as shown in the below image:

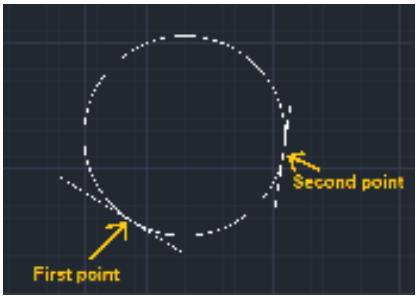


Figure:2.28. two lines created circle

The circle will be adjusted according to the specified radius.

Example 2: (Tan, Tan, Tan) The steps to create a (Tangent tangent tangent) circle are listed below:

a) Draw three lines on the viewport, as shown in the below image:

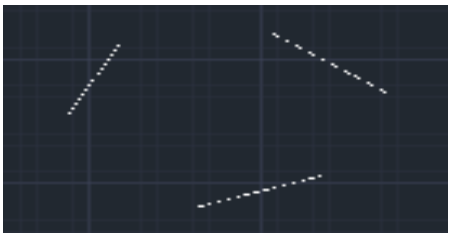


Figure:2.29. three lines (Tan, Tan, Tan)

b) Select the **Tan, Tan, Tan** circle option from the drop-down list of the circle.

c) Click on the **first** point on the first tangent.

d) Click on the **second** point on the second tangent.

e) Click on the **third** point on the third tangent.

f) The circle will be drawn on the three tangents, as shown in the below image:

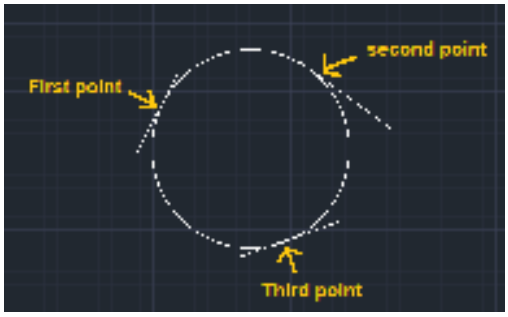


Figure:2.30. Created circle (Tan, Tan, Tan)

We can modify the tangent and points according to the requirements.

Example 3: (Tan, Tan, Radius) to create a circle inside a triangle. The steps to create a circle inside a triangle are listed below:

a) Draw a triangle in the viewport, as shown in the below image:

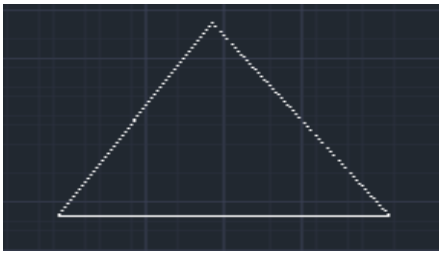


Figure:2.31. Create circle (Tan, Tan, Radius)

b) Select the **Tan, Tan, Radius** circle option from the drop-down list of the circle. Or Type **C** or **Circle** on the command line - Press **Enter**- type **ttr** - press **Enter**.

c) Click on the **first** point on one side of the triangle.

d) Click on the **second** and **third** points on the other side of the triangle.

e) The circle will be drawn inside the circle, as shown in the below image:

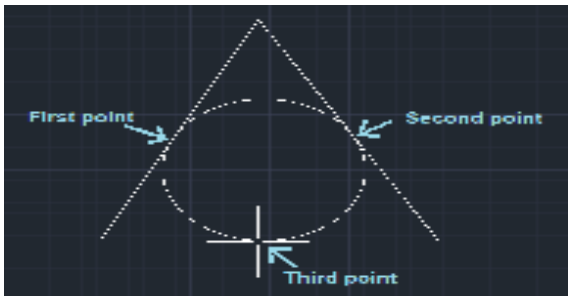


Figure:2.32. Created circle (Tan, Tan, Radius)

3. **Arc:** Arcs are curved segments of circles and are commonly used to represent rounded or curved features. Some important aspects of arcs include:
   - Center points and radius: Arcs are defined by a center point and a radius, which determine their size and position.
   - Start and end angles: Arcs have start and end angles that specify the extent of the curved segment.

Types of Arcs

The arrow below the arc consists of the drop-down list specifying the types of arcs. The list will look like the below image:

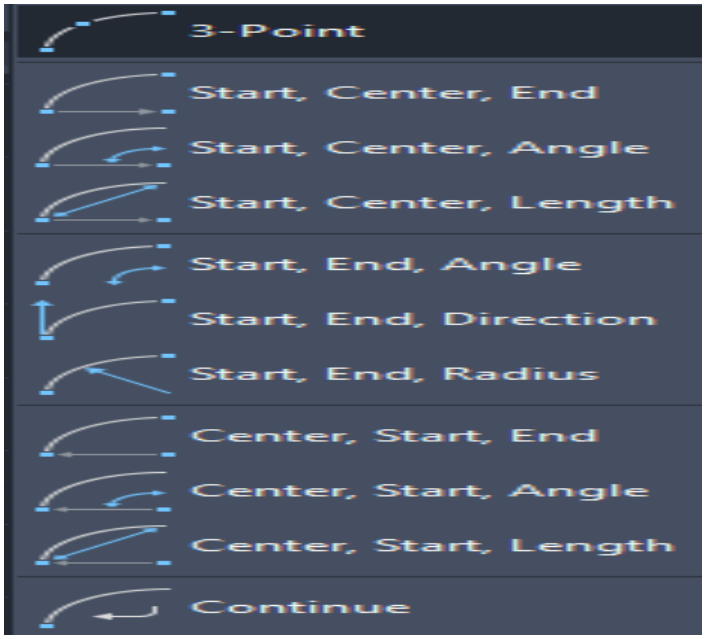


Figure:2.33. Types of Arcs

4. Rectangle:

How you can create these shapes:

- Type "Rectangle" in the command line or click on the "Rectangle" icon in the "Draw" toolbar.
- Specify the first corner of the rectangle by clicking on the desired location or typing coordinates.
- Specify the opposite corner of the rectangle by clicking on another location or typing coordinates.

Example 1: To draw rectangles randomly. The steps are given below:

1. Select the rectangle command from the ribbon panel. The rectangle icon will look like the below image:

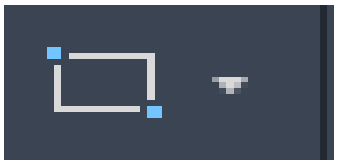


Figure:2.34. Rectangle command

Or Type Rec or Rectangle in the command line and press Enter.

2. Specify the first corner point on the viewport.
3. Specify the second corner point (diagonally opposite to the first point) on the viewport.

Example 2: To draw a rectangle by specifying the length and width.

The steps are given below:

a) Select the rectangle command from the ribbon panel. The rectangle icon will look like the below image:

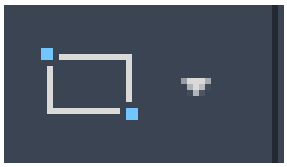


Figure:2.35. Rectangle command

Or Type Rec or Rectangle in the command line and press Enter.

b) Specify the first corner point on the viewport.

c) Specify the length and breadth of the rectangle in the form of @length, width. For example, @4,5.

Where, 4 is the length of the rectangle, while 5 is the width of the rectangle.

d) Press Enter.

5. Polyline

How you can create these shapes:

- Type "Polyline" in the command line or click on the "Polyline" icon in the "Draw" toolbar.
- Specify the vertices of the polyline by clicking on the desired locations.
- Press Enter to finish the polyline.

6. Polygons

Polygons are closed geometric shapes with straight sides. They can have any number of sides, but common examples are triangles, quadrilaterals (such as rectangles and squares), pentagons, and hexagons. Important aspects of polygons include:

- Number of sides: Polygons are defined by the number of sides they have, which determines their overall shape.
- Corner points: Polygons are defined by a series of corner points that connect to form the straight sides.

Types (Methods to use Polygon): There are two types of the polygon, which are given below:

a. Inscribe in Circle

The polygon formed will be drawn inside the circle. The command is best used when we want to create any polygon with its vertices or corners touching the circle. The shortcut command for inscribing is I.

b. Circumscribe about Circle

The polygon formed will be drawn outside the circle. The shortcut command for circumscribing is C.

Figure:2.36. Polygon (Circumscribe about Circle)

Example 1: The steps to create a pentagon (polygon with five sides) are listed below:

a. Select **Polygon** from the drop-down list of the **Rectangle** present on the ribbon panel, as shown in the below image:

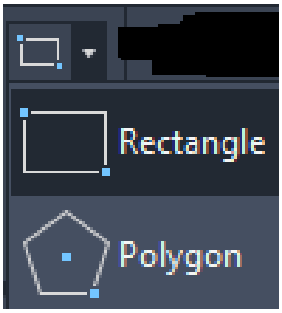


Figure:2.37. Polygon command

Or Type **POL** on the command line and press **Enter**.

b. Specify the number of sides of a polygon. Here, we are creating a pentagon, so we will enter **5**.
c. Press **Enter**.
d. Specify the center of polygon on the viewport or workspace.
e. Enter **I** or **C** for Inscribe or Circumscribe.
f. Press **Enter**.
g. Specify the radius of the polygon. For example, **4**.
h. Press **Enter**. The polygon will be drawn, as shown in the below image:

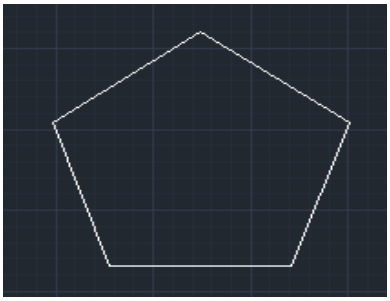


Figure:2.38. Polygon

Example 2: The steps to create a **heptagon** (polygon with seven sides) are given below:

a. Select **Polygon** from the drop-down list of the **Rectangle** present on the ribbon panel, as shown in the below image:

Figure:2.39. Polygon command

Or Type POL on the command line and press Enter.

b. Specify the number of sides of a polygon. Here, we are creating a **heptagon** (also called **Septagon**), so we will enter **7**.

c. Press **Enter**.

d. Specify the center of polygon on the viewport or workspace.

e. Enter **I** or **C** for Inscribe or Circumscribe.

f. Press **Enter**.

g. Specify the radius of the polygon. For example, **5**.

h. Press **Enter**. The polygon will be drawn, as shown in the below image:

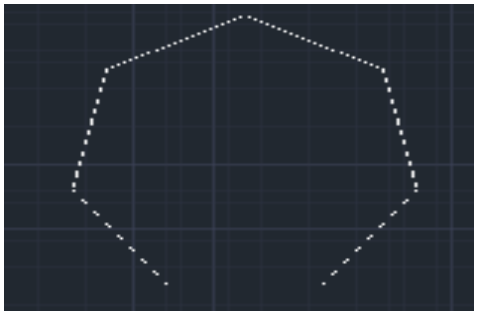


Figure:2.40. Polygon

7. Ellipses: Ellipses are oval-shaped curves defined by their two axes: the major axis (longest diameter) and the minor axis (shortest diameter). Some key points about ellipses include:

- Center points and axes: Ellipses are defined by a center point and the lengths of their major and minor axes.
- Eccentricity: The eccentricity of an ellipse determines how elongated or squashed it appears.

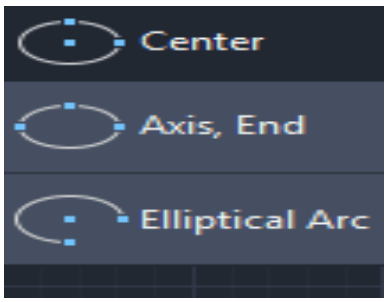


Figure:2.41. Ellipses command

Center: In this 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.

Example. The steps are listed below:

a) Click on the **Ellipse icon** on the ribbon panel and choose **Center** from the drop-down list of the Ellipse, as shown in the below image:

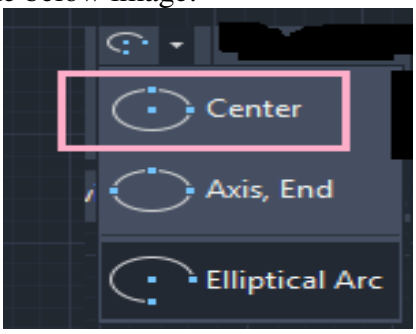


Figure:2.42. Ellipse icon

Or Type el or ellipse on the command line or command prompt and press Enter.
Type C or copy on the command line.
Press Enter.

b) Specify the center of the Ellipse on the viewport.

c) Specify endpoint of the first axis, as shown in the below image:

Figure:2.43. Specify the center of the Ellipse

We can either specify the point or the distance value. The distance will be calculated from the center to the endpoint of the first axis, such as 3.

We can also choose the vertical axis as the first axis, as shown in the below image:

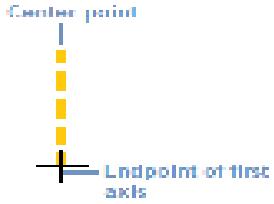


Figure:2.44. Vertical axis

d) Press **Enter**.

e) Specify the endpoint or distance value of the second axis. For example, 4.

f) Press **Enter**.

g) The Ellipse will be created, as shown in the below image:

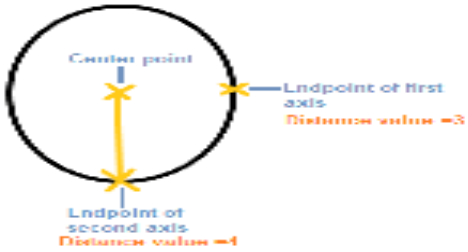


Figure:2.45. Created Ellipse

If the vertical axis was chosen as the first axis, the ellipse so formed is shown in the below image:

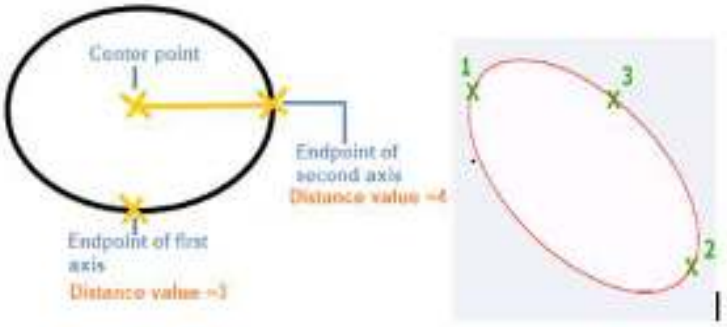


Figure:2.46. Created Ellipse

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.

8. **Hatching or Filling of Areas:** Hatching or filling is the process of adding patterns or shading to areas in a drawing to indicate different materials, surface textures, or to distinguish between objects. Hatching techniques involve lines, dots, or other patterns that are repeated within a defined area.

There are several hatch patterns available in AutoCAD. We can choose the desired pattern from the list of the patterns according to the requirements. The number of hatch lines represented in any pattern can be adjusted with the help of the **Hatch Pattern Scale**. The Hatch Pattern Scale signifies the spacing between the lines in a particular pattern.

Example 1: Consider the below image:

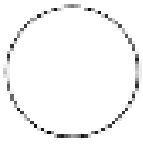


Figure:2.47. Hatching or filling of area

The steps to fill hatch pattern are listed below:

a) Select the **Hatch icon** from the ribbon panel, as shown in the below image:

Figure:2.48. Hatching tool

Or Type **H** on the command line or command prompt and press **Enter**.

b) Pick internal points of any object.

Or Select objects.

Here, we have picked an internal point inside a circle. We can pick any point inside a circle.

c) A hatch pattern will be created inside the circle, as shown below:

Figure:2.49. hatch pattern

d) To modify the lines, we have increased the Hatch Pattern Scale. The figure will now appear as:

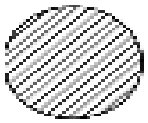


Figure:2.50. Modify the lines of hatch pattern

We can also select the desired patterns from the list of patterns, as shown below:

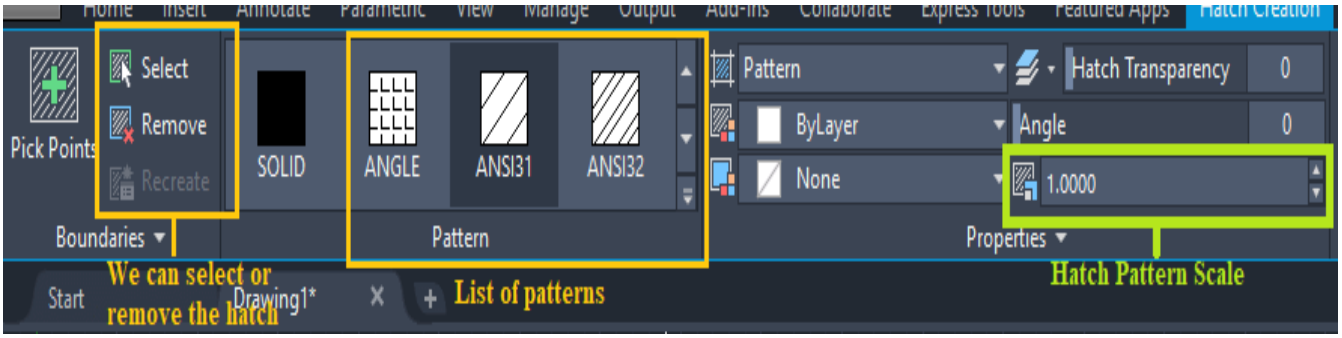


Figure:2.51. list of patterns hatch pattern

The greater the hatch pattern Scale, the higher will be the distance between the lines in the hatch pattern. When we click on the arrow near the **Hatch Icon**, a drop-down list appears, as shown in the below image:

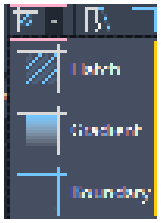


Figure:2.52. Hatch Icon

Gradient

Example 1

The steps are listed below:

a) Select the **Gradient Icon** from the drop-down list of the hatch.
b) Pick internal points of any object or select objects. Here, we have picked an internal point outside the circle.
c) A gradient will be created inside the rectangle, as shown below:

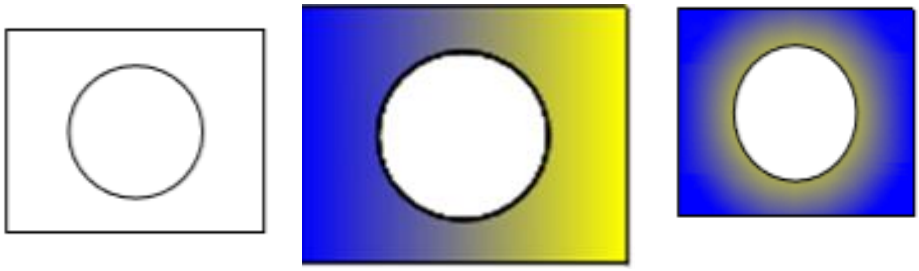


Figure:2.53. Created inside the rectangle Gradient surface

We can also select the desired patterns from the list of patterns, as shown below:

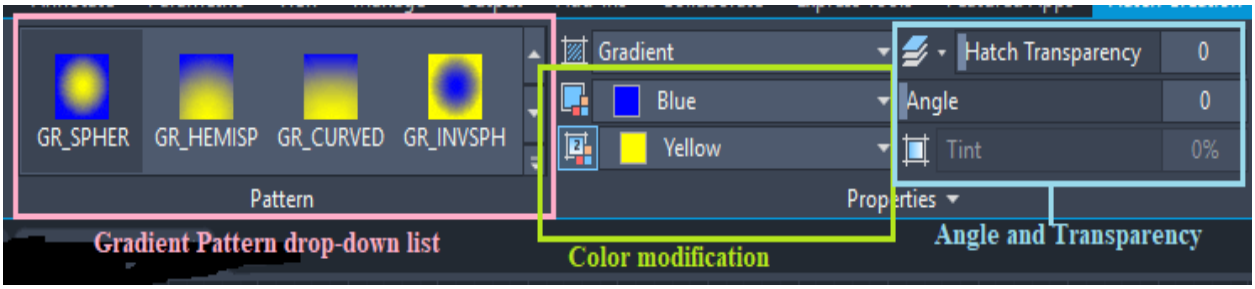


Figure:2.54. list of patterns and color modification

2.4. Modifying and Editing Drawings

When working with 2D drawings in drafting and design software like AutoCAD, there are numerous editing commands available to modify and manipulate objects within the drawing.

2.4.1. Editing commands

Editing commands refer to the specific actions or operations that can be performed on objects or elements within a digital design or drafting software, such as AutoCAD. These commands allow users to modify, manipulate, or transform the elements of a drawing to achieve the desired result.

Editing commands typically include actions like moving, copying, rotating, scaling, mirroring, offsetting, trimming, extending, filleting, chamfering, arraying, stretching, breaking, joining, and exploding objects. These commands provide users with the ability to make precise changes to their designs, adjust dimensions, create duplicates, modify shapes, connect or separate elements, and perform various other operations to refine and enhance the drawings.

Editing/modify tool and commands:

1. **Move (M):** Moves selected objects to a new location. You can specify a base point and then either enter specific coordinates or use object snaps to define the destination point.

The **move** command in AutoCAD is used to move objects at a specified distance and direction. To move the objects with precision, we can use object **snaps, grid snaps, coordinates,** etc.

There are four steps to move any object at a specific distance in a specified direction.

a) **Select objects**: We need to select the objects.
b) **Specify the base point:** Specify the starting point to move, which acts as a base point of an object.
c) **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.
d) **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.

Example 1: To move a circle. we will move a circle from a rectangle and will place it inside a triangle. The steps for such an example are listed below:

a) The figure is shown in the below image:

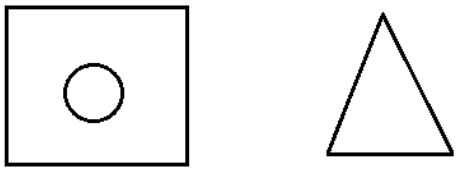


Figure:2.55. Move a circle

b) Select the move icon from the Ribbon panel, as shown in the below image

Figure:2.56. Move a tool

Or Type **M** or **Move** on the command line or command prompt and press **Enter**.

c) Select the object to move, through a small square cursor. Here, we will select the circle.

d) Press **Enter**.

e) Specify the base point or displacement.

Base Point

f) Specify the base point of an object.

g) Specify the second base point. It is the point where we want to move that object.

h) process is shown in the below image:

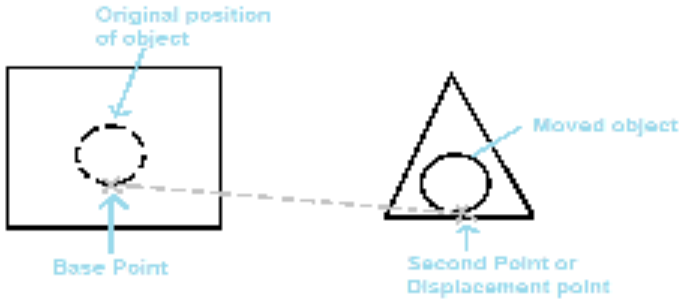


Figure:2.57. Moved circle

Or Displacement

i) Type D on the command line.

j) Specify the displacement value in the form of (X, Y, Z). For example, 2, 3. In this, the object will move 2 units in the X-axis and 3 units in the Y-axis from the original or current position.

2. **Copy (CO):** Creates copies of selected objects. Similar to the Move command, you specify a base point and then provide a destination point for the copied objects. The copy command is used to copy the objects. The concept is the same as Ctrl + C to copy and Ctrl + V to paste, which can also be used in AutoCAD. The objects are copied in a specified direction and at a specified distance.

The Copy icon on the ribbon panel looks like the below image:

Figure:2.58. Copy tool

The steps to copy the objects are listed below:

a. Select the copy icon from the ribbon panel.

Or Type co or copy in the command line or command prompt and press Enter.

b. Select the objects. To select it, click on the boundary of objects through a small square cursor.

c. After the selection is completed, press Enter.

d. Specify the displacement or base point.

Base Point

e. Specify the base point of the objects.

f. Specify the second base point. It is the point where we want to move that object.

The process is shown in the below image:

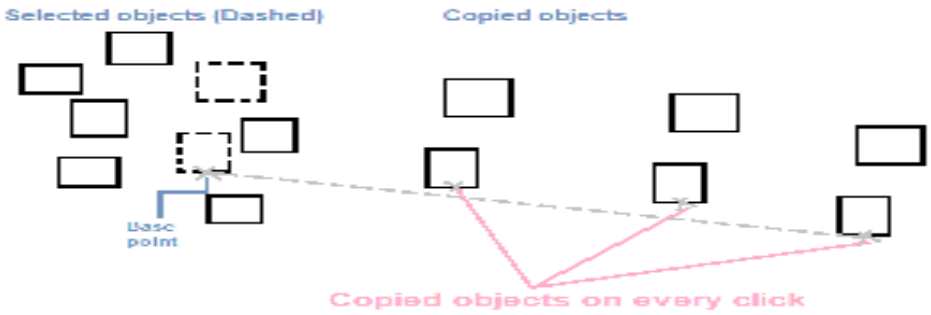


Figure:2.59. Copied object

g. To continue copying objects, repeat steps 5 and 6.

h. To Exit, press the **Esc** button.

Or

a) Specify the base point of the objects.

b) Type the displacement value, such as, 5. The distance 5 will be measured from the original object.

c) Press Enter.

d) To continue copying objects, repeat steps 6 and 7.

Or

Displacement

a) Type **D** on the command line.

b) Specify the value of displacement in the form of (X, Y, Z), for example, 2, 3. In this, the object will be moved 2 units in the X-axis and 3 units in the Y-axis from the original or current position.

c) Press **Enter**

3. **Rotate (RO):** Rotates selected objects around a specified base point by entering an angle of rotation or using reference points. The dragging of the cursor will rotate the object in the corresponding direction of the cursor.

Example: The steps for such an example are given below:

a) Select the **rotate icon** from the ribbon panel, as shown in the below image:

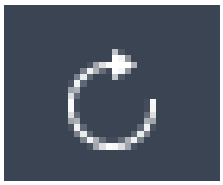


Figure:2.60. Copy tool

Or Type **ro** or **rotate** in the command line or command prompt and press **Enter**.

b) Select the object. Here, we will rotate the full object, so the whole object will be selected, as shown in the below image:

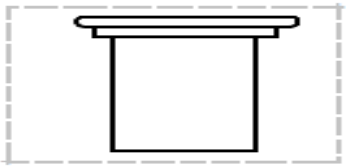


Figure:2.61. Select the object

c) Press **Enter**.

d) Specify the base point on the figure. Here, the specified base point is shown in the below image:

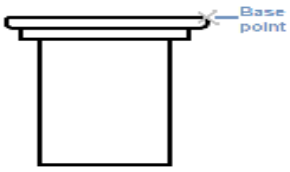


Figure:2.62. Specify the base point

We can specify any base point on the figure.

e) Drag the cursor **clockwise** to rotate the object, as shown in the below image:

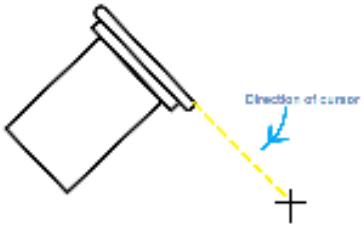


Figure:2.63. Drag the cursor clockwise

We can drag the cursor at the point according to the requirements.

f) Drag the cursor **counter-clockwise** (or **anti-clockwise**) to rotate the object, as shown in the below image:

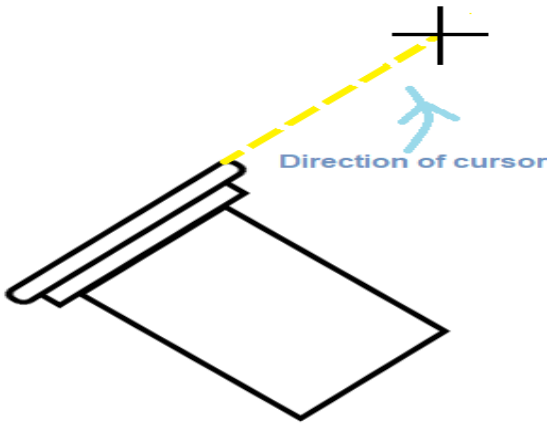


Figure:2.64. Drag the cursor counter-clockwise

4. **Scale (SC):** Changes the size of selected objects uniformly, either by specifying a scale factor or a reference length.
5. **Stretch (ST):** Resizes selected objects by stretching them in a specified direction or along a specified path. The **Stretch** command in AutoCAD is used to stretch the portion of the object partially enclosed by the polygon selection or window selection.

   The objects selected individually or completely enclosed by the window selection cannot be stretched. Those objects are rather moved such as ellipses, blocks, and circles.

Example 1: The steps for such an example are listed below:

a) Select **Stretch** icon from the ribbon panel, as shown below:

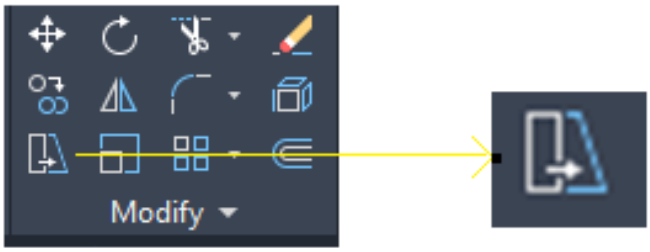


Figure:2.65. Stretch icon

Or Type **S** or **stretch** on the command line or command prompt and press **Enter**.

b) Select the object. We have selected the object with the help of box selection, as shown below:

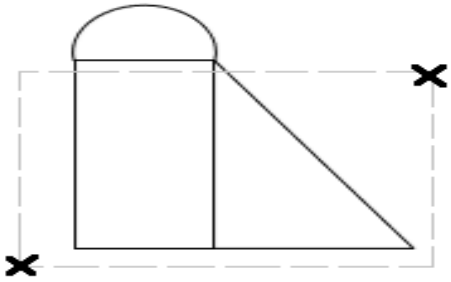


Figure:2.66. Select the object by Stretch

c) Press Enter.

d) Specify the base point or displacement value.

We can also specify the displacement value in coordinates (X, Y, Z format). Here, we have specified the base point, as shown below:

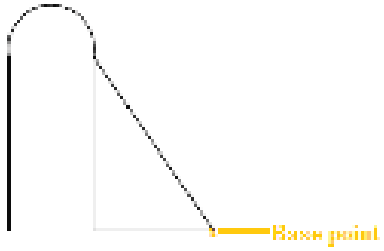


Figure:2.67. Specify the base point

e) Specify second point or the displacement value. If we specify the second point, as shown below:

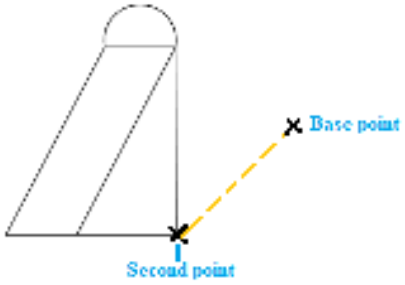


Figure:2.68. Specify second point

The base point was the original base point of the object and the second point is the stretched base point of that object. Let us specify the displacement value = 6, 7. The object will now look like the below image:

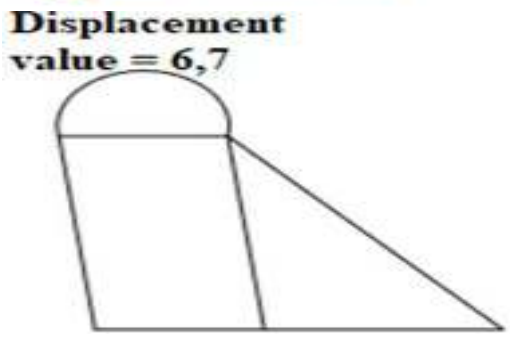


Figure:2.69. Stretched base point

Example 2: The steps for such an example are listed below:

a) Select **Stretch** icon from the ribbon panel

Or Type **S** or **stretch** on the command line or command prompt and press **Enter**.

b) Select the object. Here, we have selected a portion, as shown below:

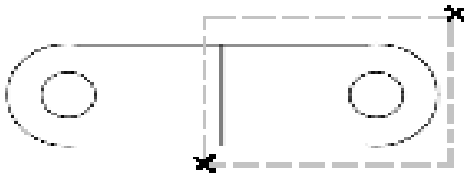


Figure:2.70. Select the object

c) Press **Enter**.

d) Specify the base point or displacement value.

Here, we have specified the **base point**, as shown below:

Figure:2.71. Specify the base point

e) Specify second point or the displacement value. We have specified the **second point**, as shown below:

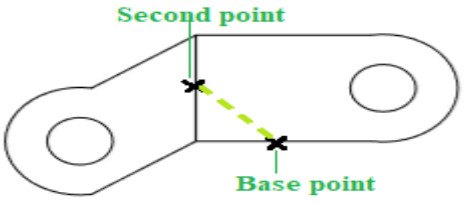


Figure:2.72. Specify second point

The **base point** was the original base point of the object and the second point is the **stretched base point** of that object. We can also stretch it by specifying the displacement value according to the requirements.

6. **Trim (TR):** Trims selected objects by removing portions that intersect with other objects or a specified cutting edge. The Trim command in AutoCAD is used to remove the objects, which meet the edges of other objects. It is used to remove extra lines or extra parts of an object. We can also perform trim using different selection methods. We are required to select the portion of the object to trim.

Or Type **Tr** or **trim** on the command line or command prompt and press **Enter**.

b. Press **Enter**

c. Select the line or curve, as shown below:

Here, we have selected three parts at once. We can choose objects according to our requirements.

d. Press **Enter**.

The selected portion will be removed. The object will now look like the below image:

Example: The steps to trim the portion of the above object are:

a. Select the **Trim** icon from the ribbon panel, as shown below:

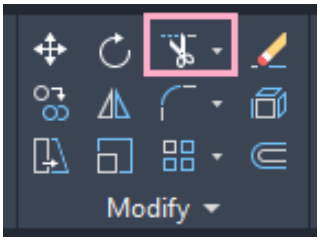


Figure:2.73. Trim icon

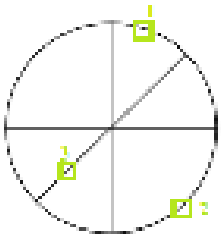


Figure:2.74. Select the line

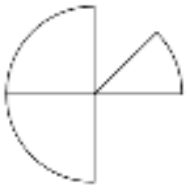


Figure:2.75. Removed lines

7. **Extend (EX):** Extends selected objects to meet other objects or a specified boundary. The **extend** command is used to extend the selected boundaries or edges. The objects are extended so that the edges of other objects can be converged.

Example: The steps to extend the boundaries or edges of the selected parts in the above figure are:

a. Select the **Extend** icon from the drop-down list of the Trim, as shown below:

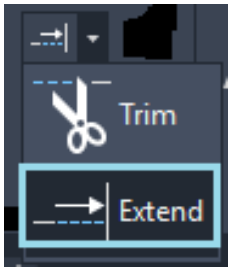


Figure:2.76. Extend icon

Or Type **EX** or **Extend** on the command line or command prompt and press **Enter**.

b. Press **Enter**
c. Select the part to extend, as shown below:

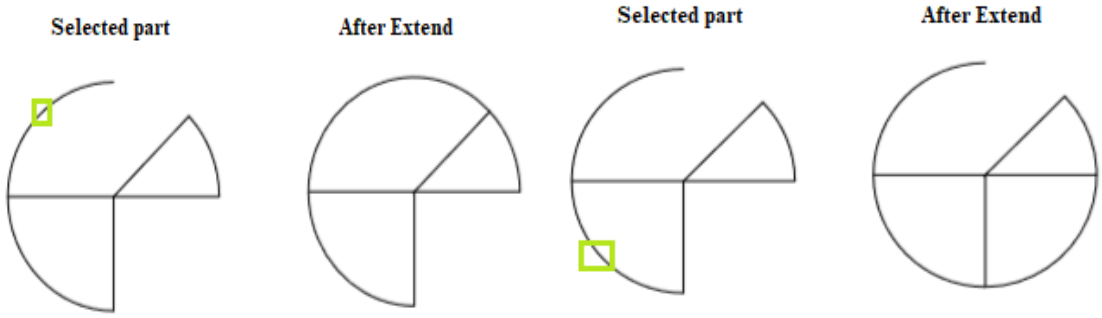


Figure:2.77. select the part and extend

Here, we have selected different parts for better understanding. We can select the objects according to our requirements.

d. Press **Enter**.

8. **Fillet (F):** Rounds the corners or intersections between two selected lines or objects by creating an arc of a specified radius. 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.

The steps to implement fillet are:

a) Select the **Fillet** command from the ribbon panel, as shown below:

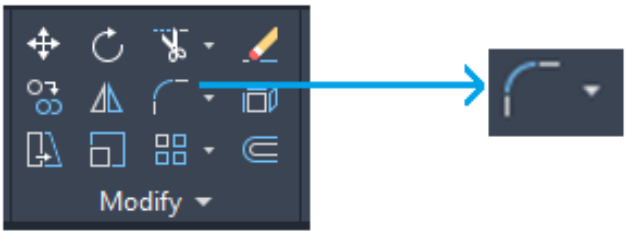


Figure:2.78. Fillet command

Or Type **F** on the command line or command prompt and press **Enter**.

b) Select the first object.
c) Type **R** or **Radius**.

d) Press **Enter**.

e) Specify the **radius** of the Fillet and press **Enter**.

f) Select the second object.

Example 1: The fillet process for the first figure is shown below:

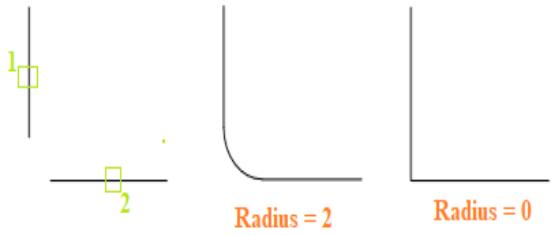


Figure:2.79. fillet processes

Example 2: The fillet process for the second figure is shown below:

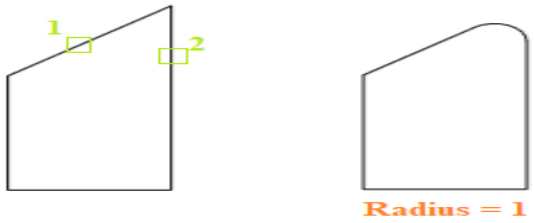


Figure:2.80. fillet processes

9. **Chamfer (CH):** Bevels the corners or intersections between two selected lines or objects by creating a straight-line segment between specified distances. 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 as shown in below image:

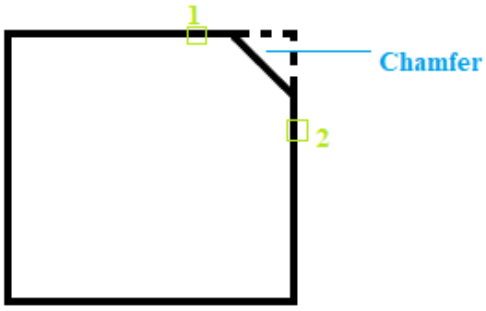


Figure:2.80. Select the two adjacent lines

Here, **1** and **2** are the selected lines. We can create the chamfer by specifying either the chamfer **distance** or **angle**.

Example 1: To create chamfer by specifying the distance. The steps are listed below:

a) Select **Chamfer icon** from the drop-down list of Fillet, as shown below:

Figure:2.81. Chamfer icon

Or Type **CHA** or **chamfer** on the command line or command prompt and press **Enter**.

b) Type **D** or **Distance** on the command line.

c) Press **Enter**.

d) Specify **first distance** value and press **Enter**.

e) Specify **second distance** value and press **Enter**.

Here, we have specified the first and second distance value as **2** and **3**.

The chamfer will be created. We can verify the first and second distance values, as shown below:

f) Click on the **first line** and the **second line**, as shown in the below image:

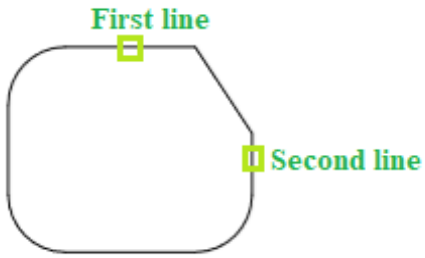


Figure:2.82. Select the first line and the second line

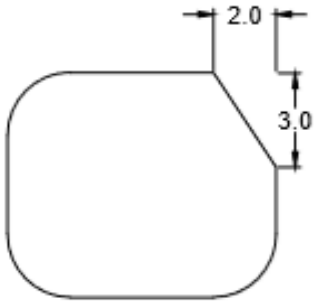


Figure:2.83. Created chamfer

We can either specify the endpoints of a chamfer on the lines.

10. **Offset (O):** Creates parallel copies of selected objects at a specified distance. The offset command in AutoCAD is used to create parallel lines, concentric circles, and parallel curves. We 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.

Example 1: Offset of a circle.

The steps are listed below:

a) Create a circle with any specified radius. For example, **3**.

b) Select the **Offset icon** on the ribbon panel.

Or Type **O** or **offset** on the command line or command prompt and press **Enter**.

c) Specify the value of offset distance. It is the distance value to create a concentric circle from the original circle. For example, **1**.

d) Press **Enter**.

e) Select the object to offset. We need to select the object with a small square cursor.

f) Move the cursor inside or outside to place the offset object, as shown in the below image:

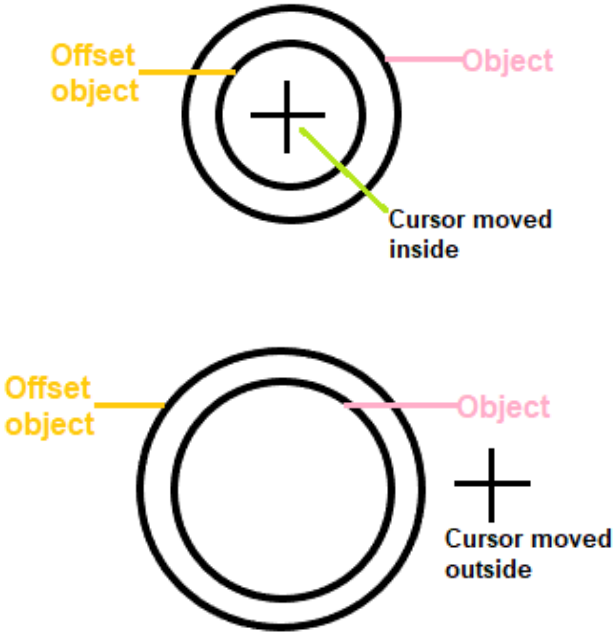


Figure:2.84. offset the object

g) Press **Esc** or **Enter** to exit from the offset command.

Example 2: The object to offset is shown in the below image:

The steps are listed below:

a) Create an object with two lines and an arc, as shown above.

b) Join the object. To offset the whole figure, we need to join the segments of that figure. It will be joined as a polyline. The steps to join the object are listed below:

- Type **J** or **join** on the command line.
- Select objects or multiple objects to join. Here, we will select the **two lines** and an **arc**.
- Press **Enter**.

c) Select the **Offset icon** from the ribbon panel.

Or Type **O** or **offset** on the command line or command prompt and press **Enter**.

d) Specify the value of offset distance. For example, **1**.

e) Press **Enter**.

f) Select the object to offset. We need to select the object with a small square cursor.

Move the cursor inside or outside to place the offset object, as shown in the below image:

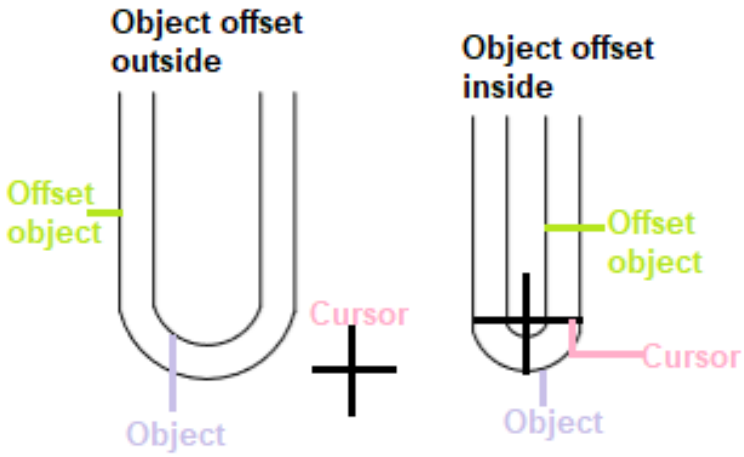


Figure:2.85. offset the object

g) Press **Esc** or **Enter** to exit from the offset command.

The distance between the object and the offset object is **1**, as mentioned in step **4**.

11. **Erase (E):** Removes selected objects from the drawing.
12. **Mirror (MI):** Reflects selected objects across a specified axis or line. The **mirror** command in AutoCAD is used to create a copy (mirror copy) of the selected object. We can also delete the source object after mirroring the object. The objects that represent the same as the half of their object can be mirrored across a center line to create the other half of the object. It is shown in the below image:

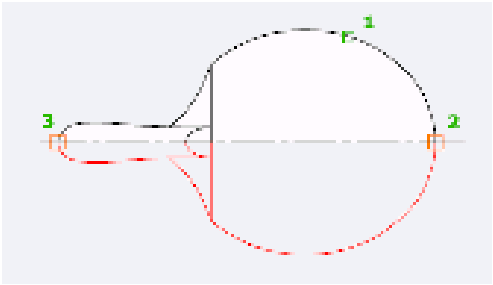


Figure:2.86. Mirror

Example 1: The dimensions of the below figure are given. We will create the upper half of the above figure. After that, we will use the Mirror command to create the lower half.

The steps are listed below:

a) The upper half is created, as shown in the below image:

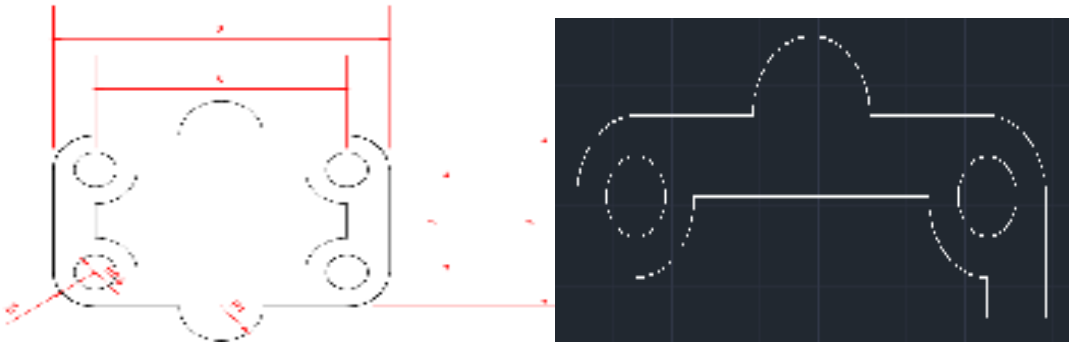


Figure:2.87. Mirror object and upper half

b) Select the **mirror** command from the ribbon panel, as shown below:

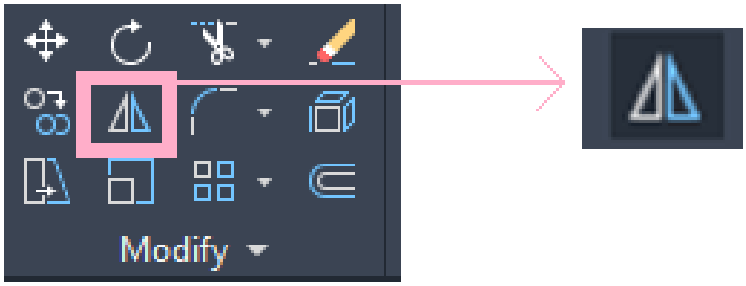


Figure:2.88. Mirror icon

Or Type **mi** or **mirror** on the command line or command prompt and press **Enter**.

c) Select the half part with the help of box selection, as shown in the below image:

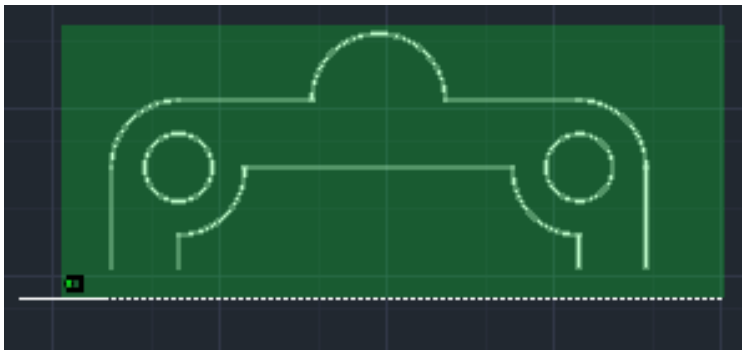


Figure:2.88(a). Select the half part

d) Press **Enter**.

e) Specify the first and second point of the mirror line, as shown in the below image:

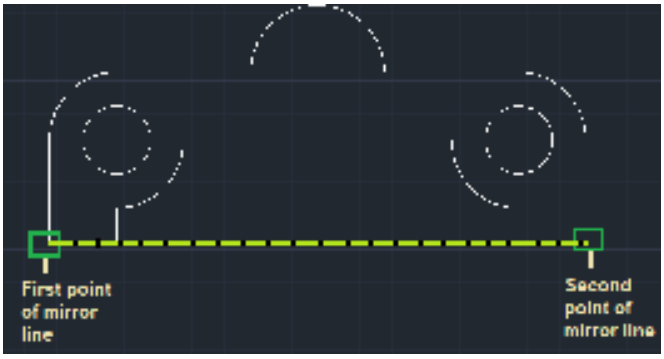


Figure:2.88(b). Specify the first and second point of the mirror line

Erase the source object

f) After the figure is formed, the command line will look like the below image:

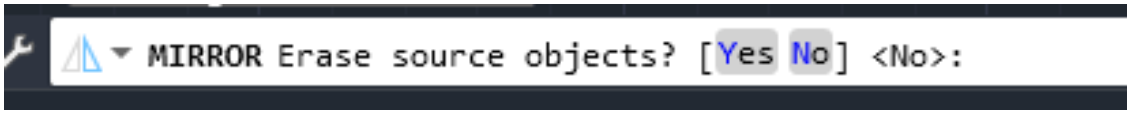


Figure:2.88(c). Erase source object

g) To erase source object, click on, Yes. If not, click on No.

h) The created figure is shown in the below image:

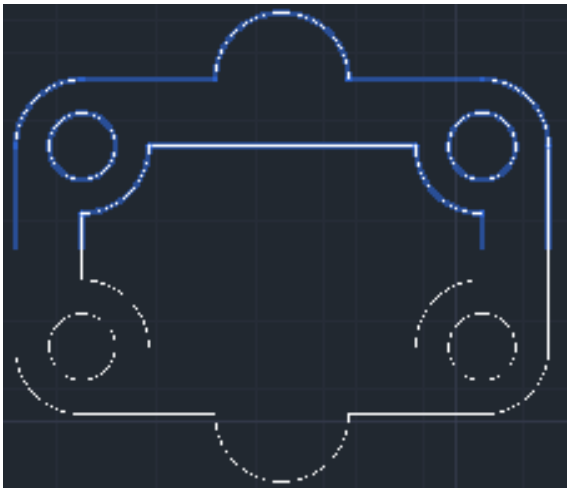


Figure:2.88(c). mirrored object

13. **Array (AR):** Creates multiple copies of selected objects in a specified pattern, such as rectangular, polar, or path arrays.

The array is the command used to draw multiple copies of an object in a particular pattern or order. We can also manage the distance between the objects in the array. The shortcut key for Array is **AR**. To implement an array, we can type **Array** or **AR** on the command line and then press **Enter**. We can also select the Array option from the Ribbon Panel. The Array icon will look like the given image:

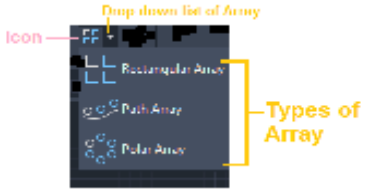


Figure:2.89. type of array

What is the difference between Copy and Array?

The copy command is used to copy the elements in an order defined by us. In contrast, the array is used to copy the objects with predefined angles and numbers in a fixed order (rectangular, polar, or path array).

There are three types of arrays, which are listed below:

- Rectangular Array
- Polar Array
- Path Array

1. **Rectangular Array:** The rectangular array is defined as the arrangement of the object into rows, columns, and levels that form a rectangle. In AutoCAD, we can create multiple copies of an object in the form of a rectangle using the Rectangular Array.

Example: The steps to create a rectangular array are given below:

a) Open the AutoCAD software.

b) Click on the **Circle** icon on the Ribbon Panel, as shown in the below image:

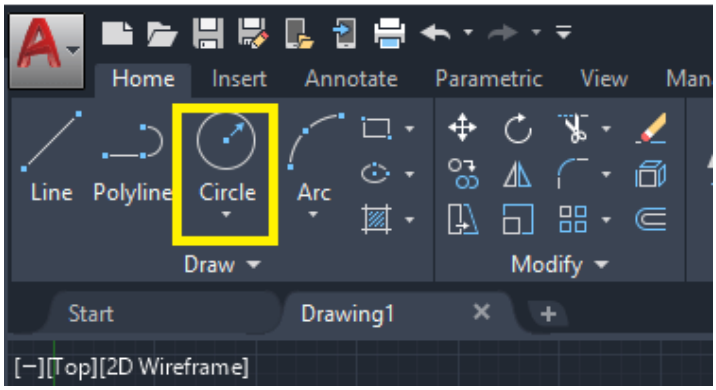


Figure:2.90. Circular icon

We can also write **Circle** or C on the command line.

c) Specify the center point on the workspace or drawing area to draw the circle.

d) Specify the diameter of the circle. Here, we have specified the value 1. We can provide any value of diameter according to the requirements.

e) Select the **Rectangular Array** from the ribbon panel.

f) Select the object with the small square cursor and Press **Enter**, as shown in the below image:

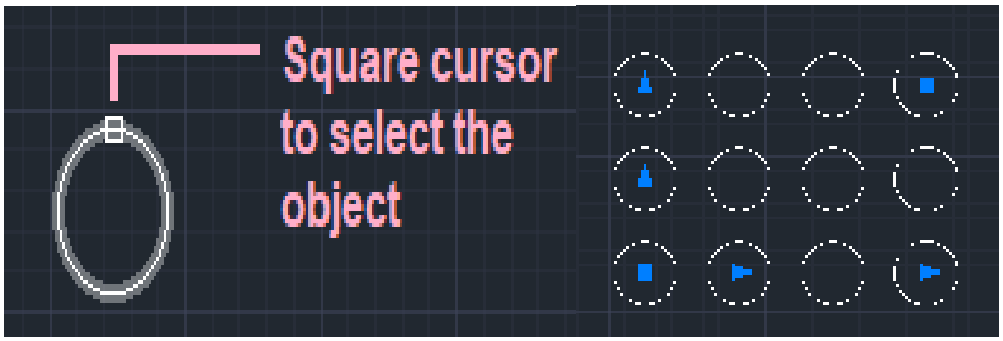


Figure:2.91. Select the object and default rectangular array

g) The default rectangular array will be created, as shown in the above image:

We can also type **AR** - select the object with the square cursor- select **Rectangular** - and then Press **Enter** on the command line.

h) Press the **Esc** button on the top left of the keyboard to exit the array command. If you want to modify the rectangular array further, without exit, follow the steps given below.

Elements Modification

To modify the above array, the steps are given below:

i) Go to the top row of the screen, which will look like the below image:

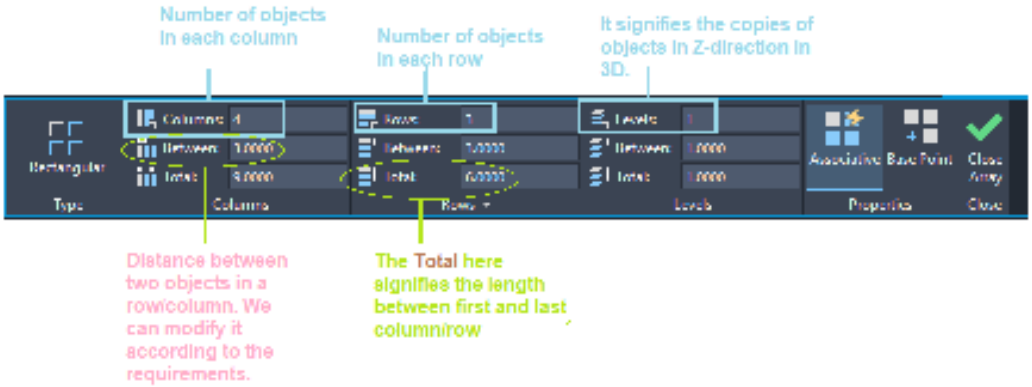


Figure:2.91(a). Array Elements Modification

j) Let's modify with the values: **Column**= 3, **Rows**= 5, **Between** under column= 1.5, **Between** under row= 1.2. The figure will now look like the given image:

Figure:2.91(b). modify with the values of Array

To randomly add rows, columns, and spacing, we can use the blue arrow and square block, as shown in the below image:

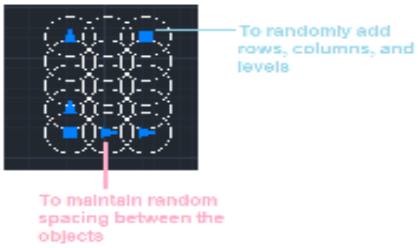


Figure:2.91(c). Arrow and square block

k) Press the **Esc** button on the top left of the keyboard or press the **Enter** to exit the array command.

2. Path Array: The path array is defined as the arrangement of copied objects along the specified path. The path can be closed or opened. The Path array command positions the items on the path. We can also use a polyline to create a path.

Example 1: The steps to create **path arrays** are listed below:

a. Open the AutoCAD software.
b. Draw a path. Let's create a simple path shown in the below image:

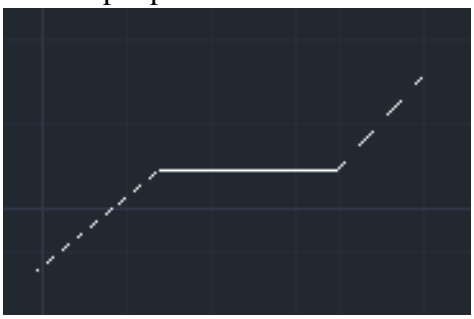


Figure:2.92(a). Draw a path

Here, we have used **polyline** to create a path. We can create any path according to the requirements.

c. Draw an object to be copied on the path. Let's create a small circle. The steps to draw a circle are:

- Click on the Circle icon on the Ribbon Panel, as shown in the below image:

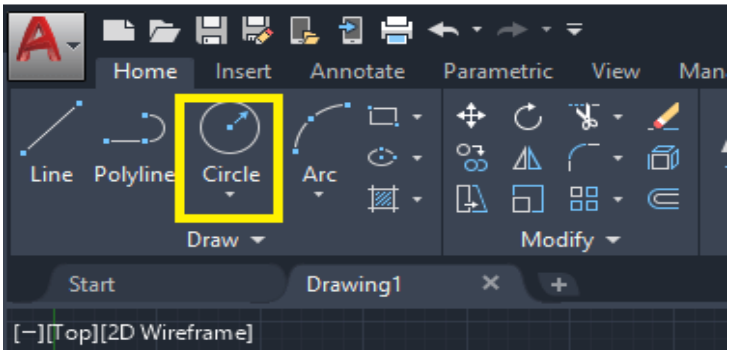


Figure:2.92(b). Circular icon

We can also write **Circle** or C on the command line and Press **Enter**.

- Specify the center point to draw a circle.

Draw the circle at the starting point of the path, as shown in the below image:

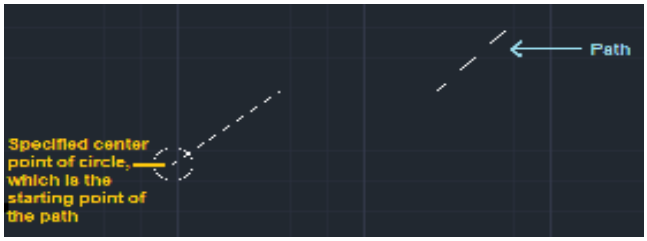


Figure:2.92(c). Draw the circle at the starting point of the path

d. Select the **Path Array** from the drop-down list of the rectangular array on the ribbon panel.
e. Select the object (circle) with the small square cursor and press **Enter** or **Spacebar**.
f. Now, select the path.
g. The path array will be created, as shown in the below image:

Figure:2.92(d). Created path array

To modify the array further, follow the below steps:

h. Go to the top row of the screen, which will look like the given image:

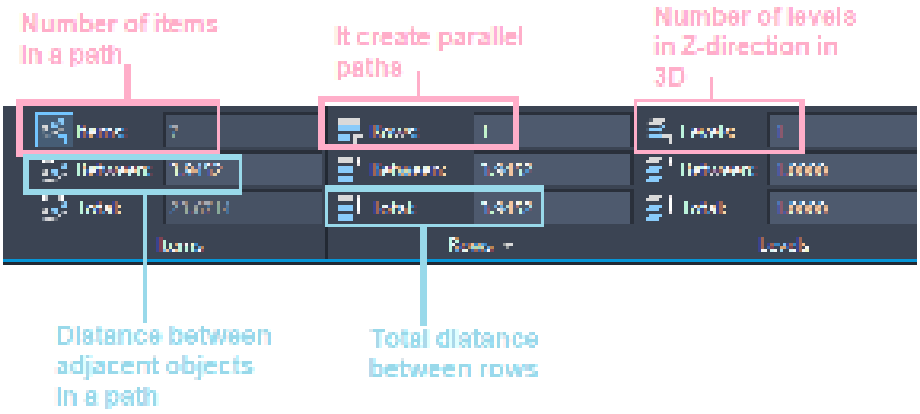


Figure:2.92(e). modify the array

i. We can modify the values according to the requirements.

1) Let's modify the values: **Rows** = 1, **Between** under **Items=2**. The value of **Items** will be adjusted according to the distance. The **figure** will now look like the given image:

Figure:2.92(f). Modified the array

2) Press **Esc** or **Enter** or **Spacebar** to exit.

Role of the blue arrow in an array: The blue arrow is used to increase or decrease the copied objects of an array, as shown in the below image:

Figure:2.92(g). Increase or decrease the copied objects

Here, the arrow in the figure signifies the direction.

Role of Blue Square in an array the blue square is used to increase rows of an array, as shown in the below image:

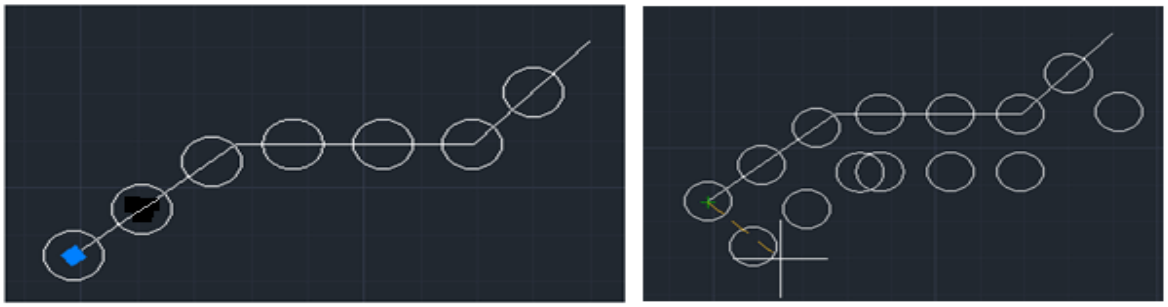


Figure:2.92(h). Increase or decrease the copied objects

Example 2: The steps to create an array are listed below:

a) Select the **polyline** command from the ribbon panel or type **polyline** on the command line and press **Enter**.

b) Draw the path.

c) Draw the rectangle at the starting point of the path, as shown in the below image:

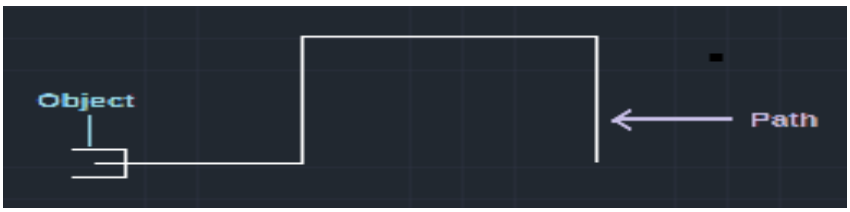


Figure:2.93(a). Draw the rectangle at the starting point of the path

d) Select the Path Array from the drop-down list of the rectangular array on the ribbon panel.

e) Select the object(rectangle) with the small square cursor and press **Enter** or **Spacebar**.

f) Now, select the path.

g) The path array will be created, as shown in the below image:

Figure:2.93(b). Created the path array

We can adjust the gap between the rectangles and rows according to the requirements.

3. Polar Array: It is also called as a circular array. The polar array is defined as the arrangement of copies of an object in a circular form.

Example: to draw small circles around a big circle. The steps to create a polar array are given below:

a) Open the AutoCAD software

b) Click on the **Circle** icon on the Ribbon Panel, as shown in the below image:

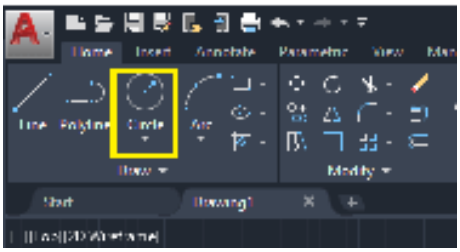


Figure:2.93(c). Circle icon

c) We can also write **Circle** or C on the command line and Press **Enter**.

d) Specify the center point on the workspace or drawing area to draw a small circle.

e) Specify the diameter for a first circle (example- **1**) and Press **Enter**.

f) Now, again select the **Circle** icon or type **C** on the command line and specify a center point for a big circle near the small circle.

g) Specify the diameter for the big circle (example- **5**) and Press **Enter**. The circles here will appear as the below image:

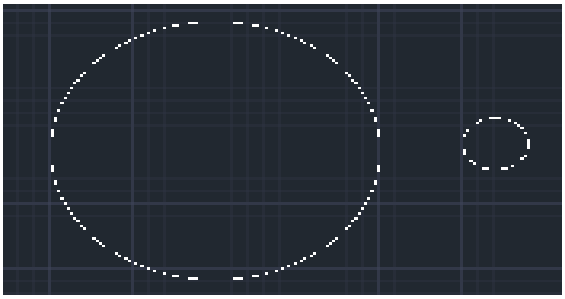


Figure:2.93(d). Circle

We can determine any value of diameter according to the requirements.

h) Select the **Polar Array** from the drop-down list of the rectangular array on the ribbon panel.

The polar array icon will look like the given image:

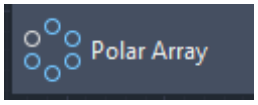


Figure:2.93(e). Polar Array

h) Select the object (small circle) with the small square cursor and Press **Enter**, as shown in the below image:

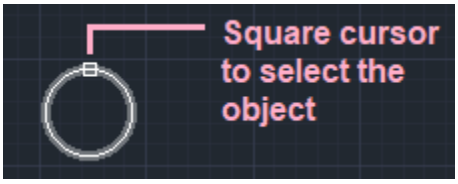


Figure:2.93(e). Select the object

i) Now, specify the center to create the array. Here, we can specify the center of the big circle.

j) We can also specify the center of any object according to the requirements.

k) The default polar array will be created, as shown in the below image:

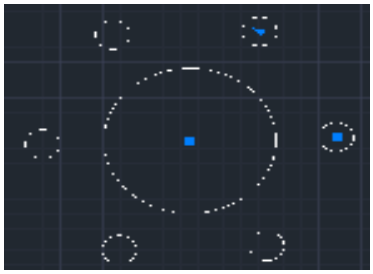


Figure:2.93(f). Created default polar array

We can also type **AR** -> select the small object with the square cursor- > select **Polar** - > select the center axis -> and then Press **Enter** on the command line.

Elements Modification: To modify elements, follow the below steps:

l) Go to the top row of the screen, which will look like the below image:

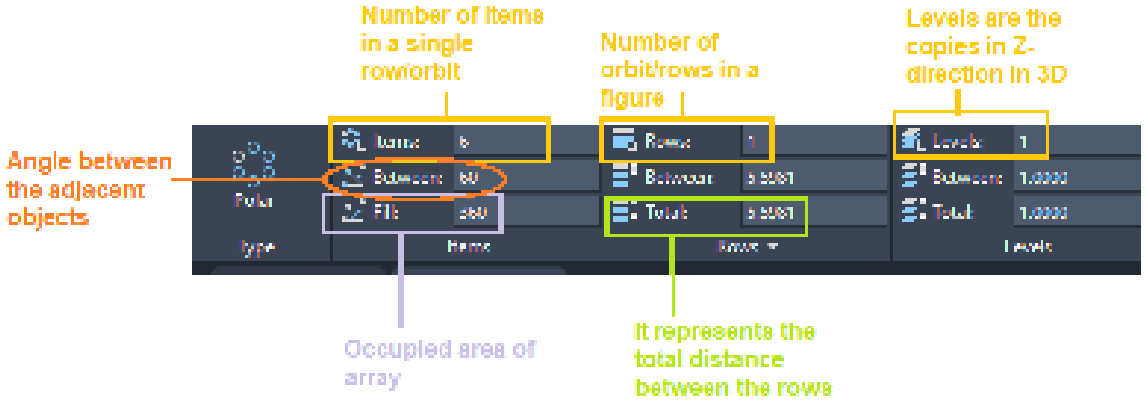


Figure:2.93(f). Modify elements

14. **Explode (X):** Breaks down complex objects or blocks into their individual components.
15. **Join (J):** Combines selected lines, arcs, or polylines to create a single object.
16. **Break (BR):** Breaks a selected object into two separate objects at a specified point.

17. **Align (AL):** Adjusts the position of selected objects relative to a specified alignment point or line.

2.4.2. Editing and modifying existing objects

Editing and modifying existing objects in a digital design or drafting software involves making changes to the properties, characteristics, or geometry of objects that are already present in the design. This process allows users to refine and customize their drawings according to their specific requirements.

To edit and modify an existing 2D drawing, you can follow these general steps:

1. **Open the drawing**: Launch your preferred software and open the 2D drawing you want to edit.
2. **Select the object(s) you want to edit:** Use the selection tool (usually represented by an arrow cursor) to select the object(s) you want to modify. Click on the object(s) to select them. You can select multiple objects by holding down the Shift key while clicking.
3. **Access the editing tools:** Look for the toolbar or menu that contains the editing tools. These tools are typically labelled with icons representing the specific editing actions.
4. **Choose the desired editing command:** Depending on the software, the available editing commands may include move, rotate, scale, mirror, trim, extend, fillet, chamfer, and more. Select the appropriate command for the type of modification you want to make.
5. **Specify the editing parameters:** Once you have selected the editing command, the software may prompt you to provide additional parameters. For example, if you are using the move command, you may need to specify the distance and direction of the movement.
6. **Apply the edit:** After specifying the parameters, click on a specific point or select a reference object to apply the edit. The selected object(s) will be modified based on the provided parameters.
7. **Review and refine:** Take a moment to review the changes made to the object(s). If necessary, you can further refine the modifications using additional editing commands or by repeating the same command with different parameters.
8. **Save your changes:** Once you are satisfied with the modifications, save the edited 2D drawing to preserve your changes.

Please note that the specific steps and commands may vary depending on the software you are using. It is recommended to consult the software's documentation or seek additional resources for more specific instructions tailored to your chosen software.

2.5. Dimensioning

Dimensioning is the process of adding measurements to a drawing to indicate the size, shape, and location of features on an object. Annotation is the process of adding text and symbols to a drawing to provide additional information about the object or drawing.

Dimensioning is essential for manufacturing and construction, as it allows workers to accurately produce and assemble objects. Dimensions are typically displayed as a line with two arrows at the ends. The line is placed between the two points that are being measured, and the arrows indicate the direction of the measurement. The dimension value is placed above or below the dimension line

2.4.2. Dimensions, text, and annotations

In technical drawings, dimensions, text, and annotations are used to provide vital information about the size, shape, and other characteristics of objects.

- **Dimensions:** Dimensions are numerical values that convey the measurements of objects or features within a drawing. They are typically represented by dimension lines, arrowheads, extension lines, and numerical values. Dimensions can indicate lengths, widths, angles, radii, or clearances.
- **Text:** Text elements are used to provide additional information, labels, or notes within a drawing. They can be used for annotations, part numbers, material specifications, or assembly instructions. Text is typically placed adjacent to the relevant objects or features in the drawing.

2.4.3. Dimension styles and settings:

Dimension styles and settings allow you to define the appearance and behavior of dimensions in a drawing. They provide a way to standardize and control the formatting of dimensions across the drawing. 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:

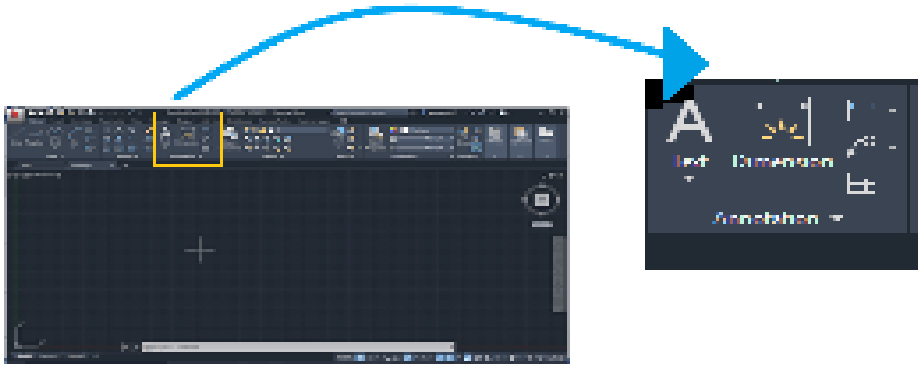


Figure:2.94(a). Dimension area

There are different types of dimensions. It will appear on the drop-down list of the dimension, as shown in the below image:

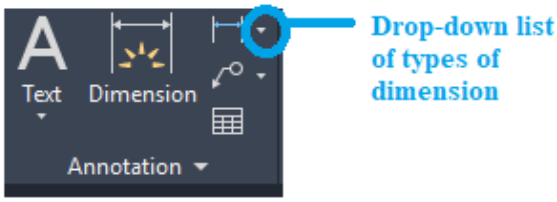


Figure:2.94(b). Drop-down list of the dimension

The drop-down list will look like the below image:

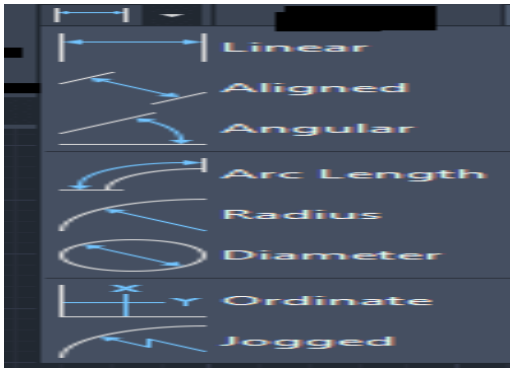


Figure:2.94(b). list of the dimension

Key aspects of dimension styles and settings include:

- **Dimension Style:** A dimension style is a collection of settings that define the appearance of dimensions, such as text height, arrow style, extension line style, and units of measurement. By creating and applying dimension styles, you can ensure consistency throughout the drawing.

- **Dimension Settings:** Dimension settings include options for precision, tolerance display, dimension lines, and other parameters. These settings allow you to customize how dimensions appear and behave in the drawing.

The Dimension Style Manager is used to create, modify, override, and compare the new styles and dimensions in AutoCAD. The **Standard** dimension style is considered as the default dimension style. To open the Dimension Style Manager,

1. Type **D** or **DIMSTY** on the command line or command prompt and press **Enter**. The dialog box will appear, as shown below:

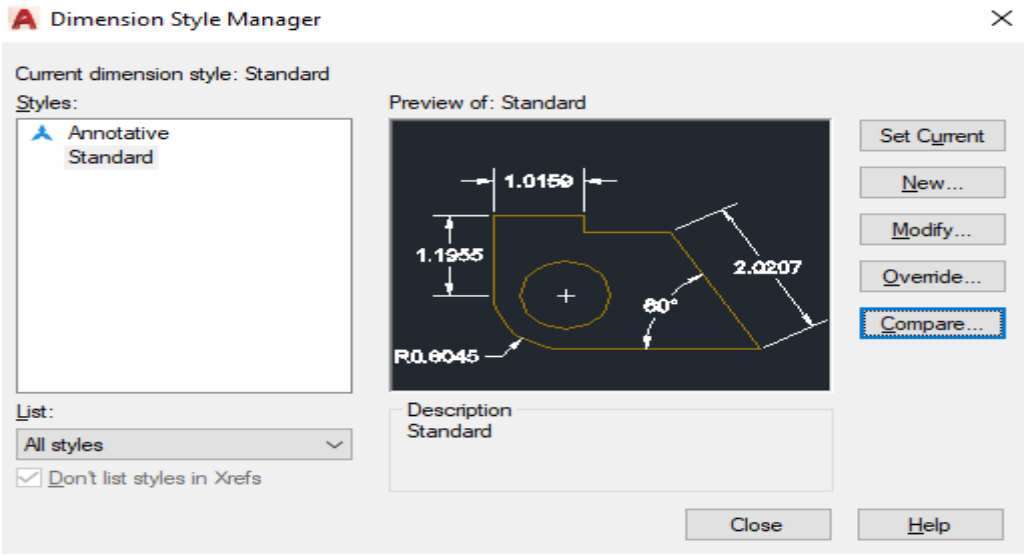


Figure:2.95. Dimension style manager

The properties given on the right side of the Dimension Style Manager are listed below:

2. **Set Current**: It applies the current style to the dimensions created by us. The selected style will be set as the current style.
3. **New**: It is used to create a new dimension style. The steps to create a new dimension style are listed below:
    a. Click on the '**New**' button. A dialog box will appear, which will look like the below image:

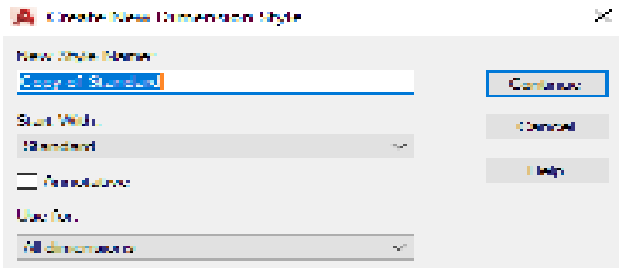


Figure:2.95(a). Create a new dimension style

b. Specify the **New Style Name** and click on the **Continue** button. A dialog box will appear, as shown below:

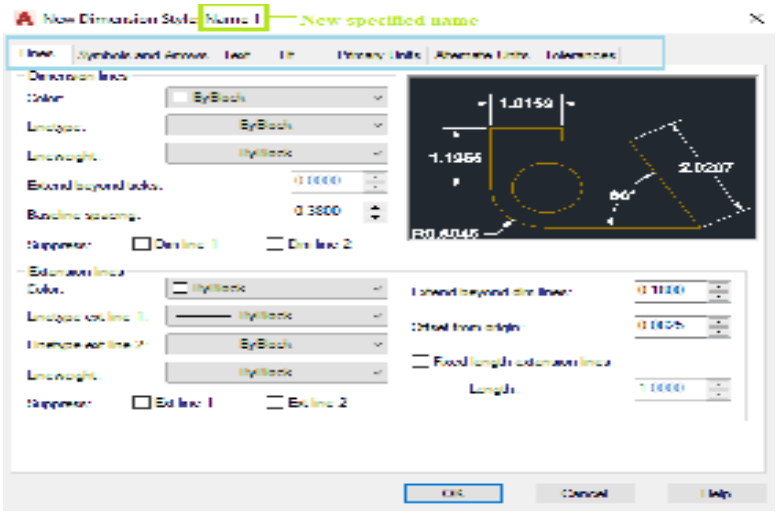


Figure:2.95(b). New dimension style

The top row marked consists of the different characteristics of Dimensions. We can modify the values accordingly. We need to click on the **OK** and **Close** to apply the dimensions on the drawing.

4. **Modify**: The **modify** option will open the modification dialog box, which is the same as the New Dimension Style dialog box. Here, we can modify the different characteristics of dimensions, according to the requirements.
   - To modify the size of the arrow, Click on the **'Symbols and Arrows'** option, which will modify the value below the **Arrow size**, as shown below:

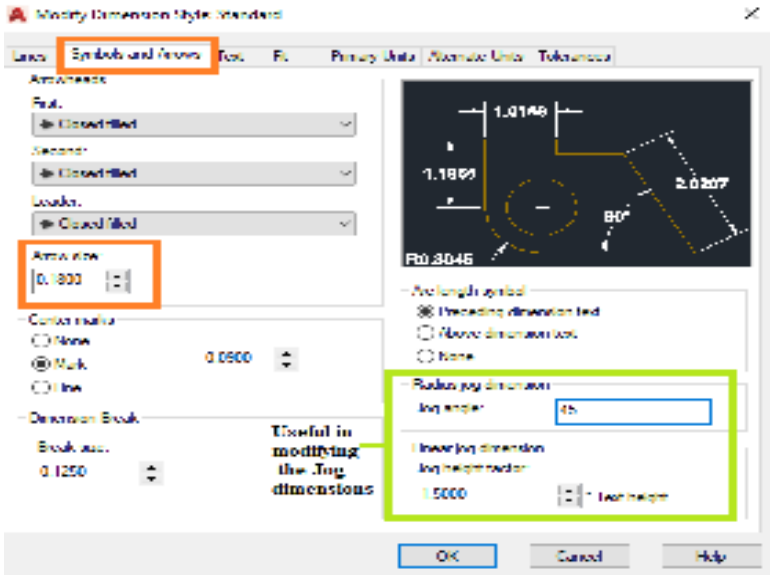


Figure:2.95(c). Symbols and arrows

- To modify the text size, Click on the '**Text**' option and modify the value in front of the **Text Height**, as shown below:

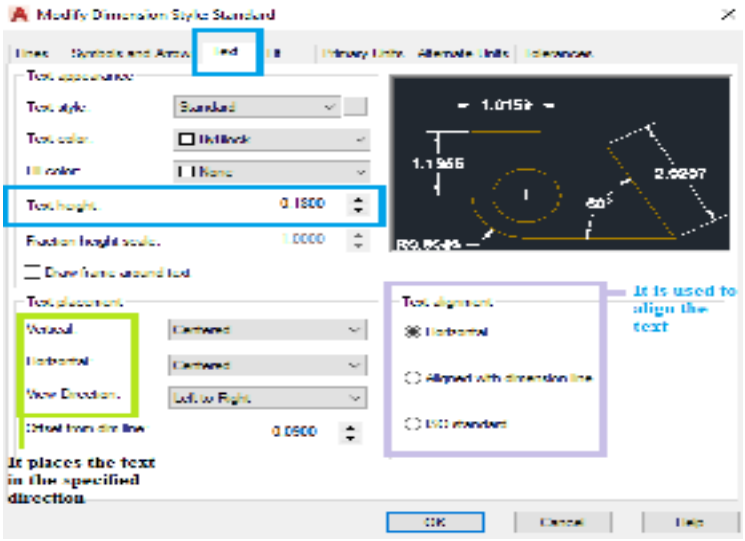


Figure:2.95(d). Text Height

- To change the **Units type** and the **precision** value,

Click on the '**Primary Units**' option, modify the value in front of the **Precision** and **Unit Format**, as shown below:

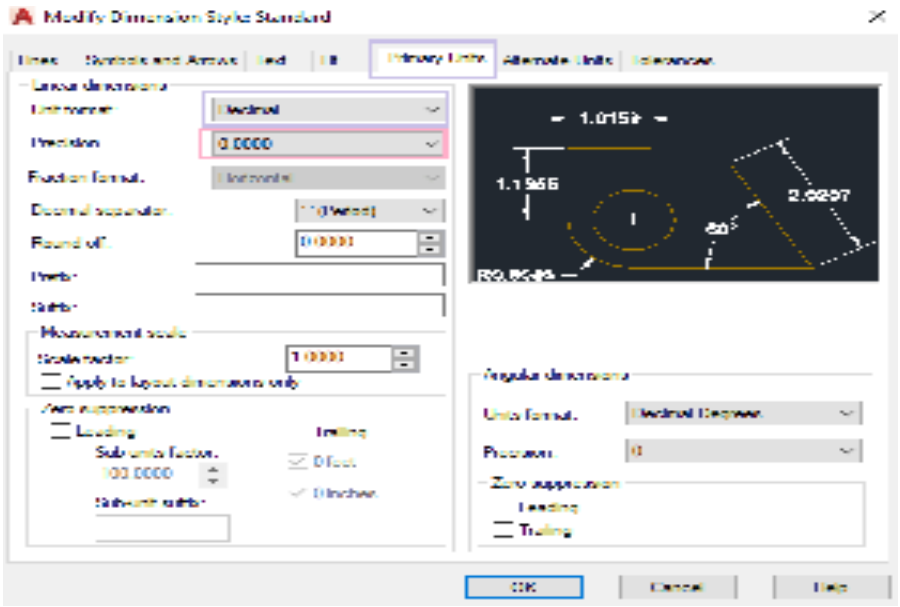


Figure:2.95(e). Precision

The precision format is shown in the below image:

Precision	Example
0	1
0.0	1.2
0.00	1.20
0.000	1.198
0.0000	1.1966
0.00000	1.19660
0.000000	1.196601
0.0000000	1.1966010
0.00000000	1.19660148

Figure:2.95(f). Precision format

2.4.4. Creating and editing multileader objects

Multileader objects are used to create annotations or callouts with multiple lines of text and associated leaders. They are particularly useful for providing detailed explanations or instructions in a drawing. Creating and editing multileader typically involves the following steps:

- **Select the Multileader tool:** In your CAD software, choose the Multileader tool from the toolbar or menu.
- **Define the leader landing:** Specify the starting point of the leader, indicating where the text or annotation will be placed.
- **Enter the text:** Type or paste the desired text into the leader. You can format the text, adjust its alignment, and add additional lines as needed.
- **Adjust the leader lines:** Modify the leader lines to connect the text or annotation to the relevant objects or features in the drawing. You can adjust the length, angle, or appearance of the leader lines.

The **Multileader** option on the ribbon panel. The arrow after the **MLEADER**, consists of a drop-down list, as shown in the below image:

Figure:2.96(a). Multileader icon

The list consists of the operation that can be carried on the Multileader. Consider the below figure showing various Multileader.

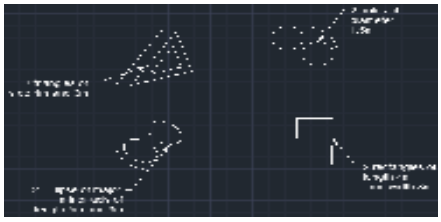


Figure:2.96(b). showing various Multileader

We will perform different operations in the above figure.

Add Leader: The add Leader commands add a Multileader to a selected Multileader object. We can move the cursor left or right to place the new Multileader line on the desired position.

The steps to implement the Add Leader are listed below:

Select the **Add Leader** from the drop list of Multileader on the ribbon panel. Or Type **AIMLEADEREDITADD** on the command line or command prompt and press **Enter**.

a) Select a Multileader with a small square cursor. Here, we will select the Multileader of the Rectangle, as shown in the below image:

Figure:2.96(c). Select a Multileader

Move the cursor to place the Multileader in the left or right direction, as shown below:

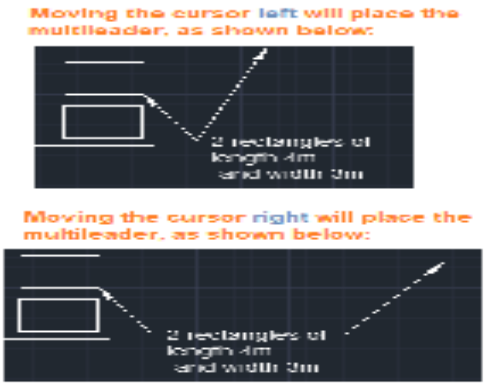


Figure:2.96(d). move the cursor left or right

b) We can move the cursor left or right according to our choice.

c) To add more Multileader, continue moving the cursor in the desired direction. For example, the addition of **three** leaders in the left direction will now appear as the below image:

Figure:2.96(e). Addition of three leaders

d) Press **Esc** or **Enter** to exit.

2.6. AutoCAD in building drawings

2.6.1. Introduction to Building Drawings

Building drawings refer to technical drawings that represent various aspects of a building, such as floor plans, elevations, sections, and details. These drawings provide information about the design, dimensions, construction, and materials used in a building.

In basic 2D drawing and drafting, these elements and techniques are used to create accurate representations of objects, structures, or buildings. They form the foundation of the drafting process and are essential for communicating design intent, dimensions, and specifications.

2.6.2. Floor plan

A floor plan is a scaled diagram or drawing that illustrates the layout of a space as seen from above. It provides a bird's-eye view of a building, apartment, or room, showing the arrangement and relationship of rooms, walls, doors, windows, and other architectural elements. Floor plans are commonly used in architecture, interior design, real estate, and construction industries.

Floor plans typically include the following information:

- **Room layout:** The floor plan shows the layout and dimensions of each room, including bedrooms, living areas, kitchen, bathrooms, and other spaces.
- **Walls**: The walls are depicted as lines on the floor plan, indicating their length, thickness, and location.
- **Doors and windows**: Doors and windows are represented by symbols or rectangles, indicating their size, shape, and location within the walls.
- **Dimensions:** Measurements are included to indicate the size of rooms, walls, and other elements.
- **Furniture and fixtures:** In some cases, floor plans may include furniture and fixtures to provide a better understanding of how the space can be utilized.

- **Annotations:** Labels and annotations may be added to indicate room names, usage, or specific features.

To draw a floor plan on AutoCAD, follow these steps:

1. Open AutoCAD and start a new drawing by clicking on "New" or using the shortcut Ctrl + N.
2. Set up the drawing units by clicking on the "Units" button in the bottom right corner of the AutoCAD window. Choose the appropriate units for your floor plan, such as inches or feet.
3. Set the drawing area by specifying the limits. Type "Limits" in the command line, then enter the desired coordinates to define the boundaries of your floor plan.
4. Begin drawing the walls of your floor plan using the Line command. Type "Line" in the command line, then click on the starting point of the wall and click on the subsequent points to define the shape and length of the wall. Repeat this process for each wall in your floor plan.
5. Use the Offset command to create interior walls. Type "Offset" in the command line, then specify the distance between the existing wall and the new interior wall. Click on the existing wall, then click on the side where you want to create the new wall.
6. Add doors and windows using the appropriate AutoCAD commands. For doors, you can use the Rectangle or Polygon commands to draw the door shape, then use the Trim command to remove the excess lines. For windows, you can use the Rectangle or Polygon commands as well, and then trim the excess lines.
7. Label and dimension your floor plan using the Text and Dimension commands. Type "Text" in the command line to add labels for rooms, doors, and windows. Use the Dimension command to add accurate measurements for walls, doors, and windows.
8. Save your floor plan by clicking on the "Save" button or using the shortcut Ctrl + S. Choose a location and name for your file.

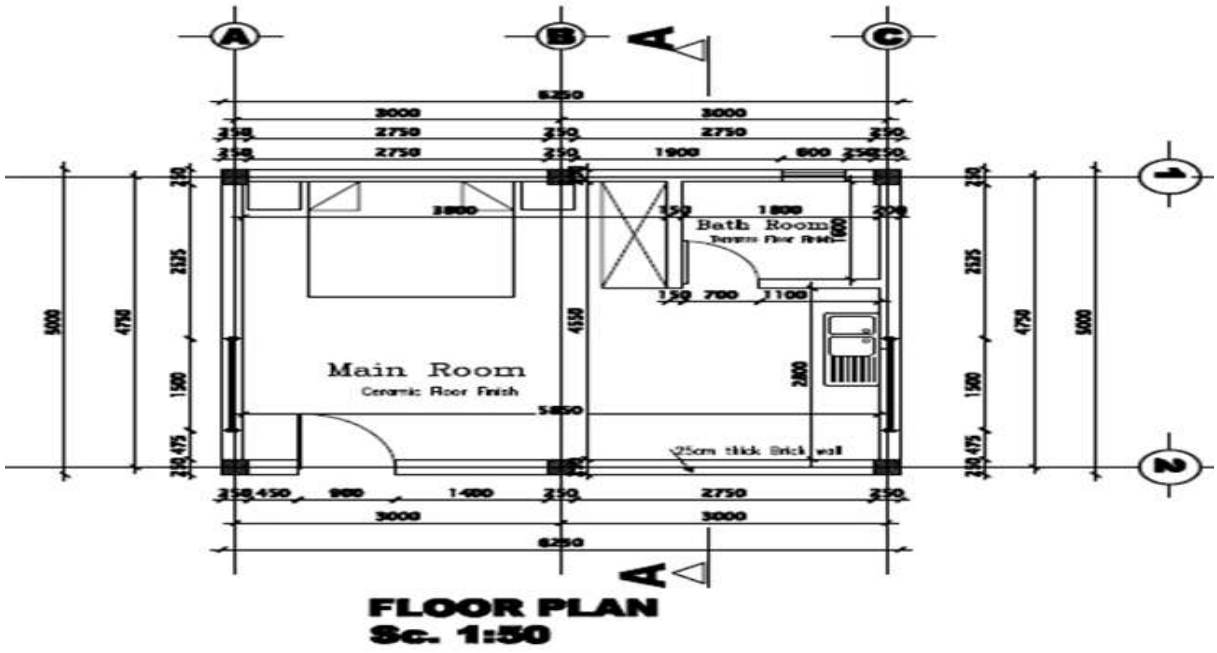


Figure:2.97. Floor plan

2.6.3. Floor plan with Electrical Plan

The electrical plan is sometimes called as electrical drawing or wiring diagram. It is a type of technical drawing that delivers visual representation and describes circuits and electrical systems. It consists of **electrical symbols** and lines that show case the engineer's electrical design to its clients. In short, an electrical plan describes the position of all the electrical apparatus.

An electrical drawing may include all of these essential details described below:

- Interconnection of electrical wires and other parts of the system
- Connection of different components and fixtures to the system
- Power lines with details such as size, voltage, rating, and capacity
- Power transformers and also their winding connections
- The main switches, tiebreaker, and fused switches
- Other essential equipment such as solar panels, batteries, generators, air conditioning, and so on

➢ To draft an electrical plan with CAD, you can follow these steps:

- Draw the floor plan of the building. You can use a CAD template to help you with this.
- Add the electrical symbols to the floor plan. You can find CAD electrical symbols online or in libraries.

- Draw the electrical circuits. Use different colors to represent different circuits.
- Add a legend to the drawing. The legend should explain the electrical symbols and colors used in the drawing.
- Label the electrical devices.
- Dimension the drawing.
- Save and print the drawing.

Example:

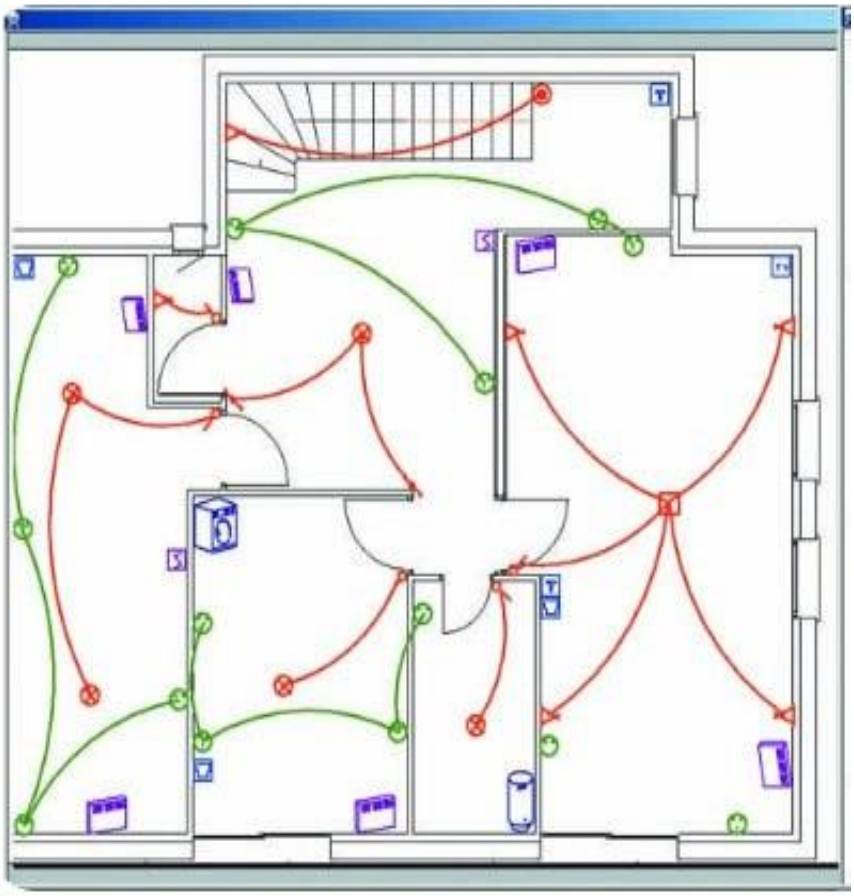


Figure:2.98. Electrical plan

Self-check 2

I. Multiple-Choice Questions

1. Which AutoCAD command is used to draw a straight-line segment?
   a. Line b. Circle b. Arc b. Polyline
2. What is the purpose of layers in AutoCAD?
   a. To control the visibility and organization of objects in a drawing
   b. To create 3D objects
   c. To apply materials and textures to objects
   d. To dimension and annotate drawings
3. Which AutoCAD command is used to create a circle?
   a. Circle b. Line c. Arc d. Polyline
4. What is the purpose of the "Zoom" command in AutoCAD?
   a. To change the display scale of the drawing
   b. To navigate and view different areas of the drawing
   c. To modify the shape of objects
   d. To apply dimensions and annotations
5. Which AutoCAD command is used to create a dimension in a drawing?
   a. Dimension b. Line c. Circle d. Polyline

II. Short Answer Questions

1. What are some basic parameters in AutoCAD that can be set before starting a drawing?
2. Explain the process of modifying and editing drawings in AutoCAD.
3. How can you add dimensions and annotations to a drawing in AutoCAD?

Operation sheet 1: Produce 2D Electrical plan

Operation title: Produce 2d floor plan & put electrical symbols

Purpose: To practice and demonstrate the skill, knowledge and attitude required to produce 2d drawing.

Instruction: Use the given tools and equipment produce 2d electrical plan including the Building electrical installations field. For this operation you have given 3 Hour each electrical plan and you are expected to provide the answer on the given table.

Tools and requirement:

- ✓ Auto CAD software
- ✓ This operation sheets
- ✓ 2d electrical plan

Precautions: when produce 2d electrical plan the drawing clear and understandable. Use auto cad software. These are general procedures for creating a 2D electrical plan in AutoCAD. The specific commands and techniques may vary depending on the version of AutoCAD you are using.

Procedures in doing the task

Step1: Launch AutoCAD: Open AutoCAD software on your computer.

Step2: Set up the drawing environment

Step3: Start a new drawing

Step 4: Set the drawing limits

Step 5: Draw basic geometry

Step 6: Modify and edit objects

Step 7: Apply layers

Step 8: Add dimensions and annotations

Step 9: Apply hatching or shading

Step 10: Create blocks and symbols

Step 11: Put electrical symbols

Step 12: Save your electrical plan

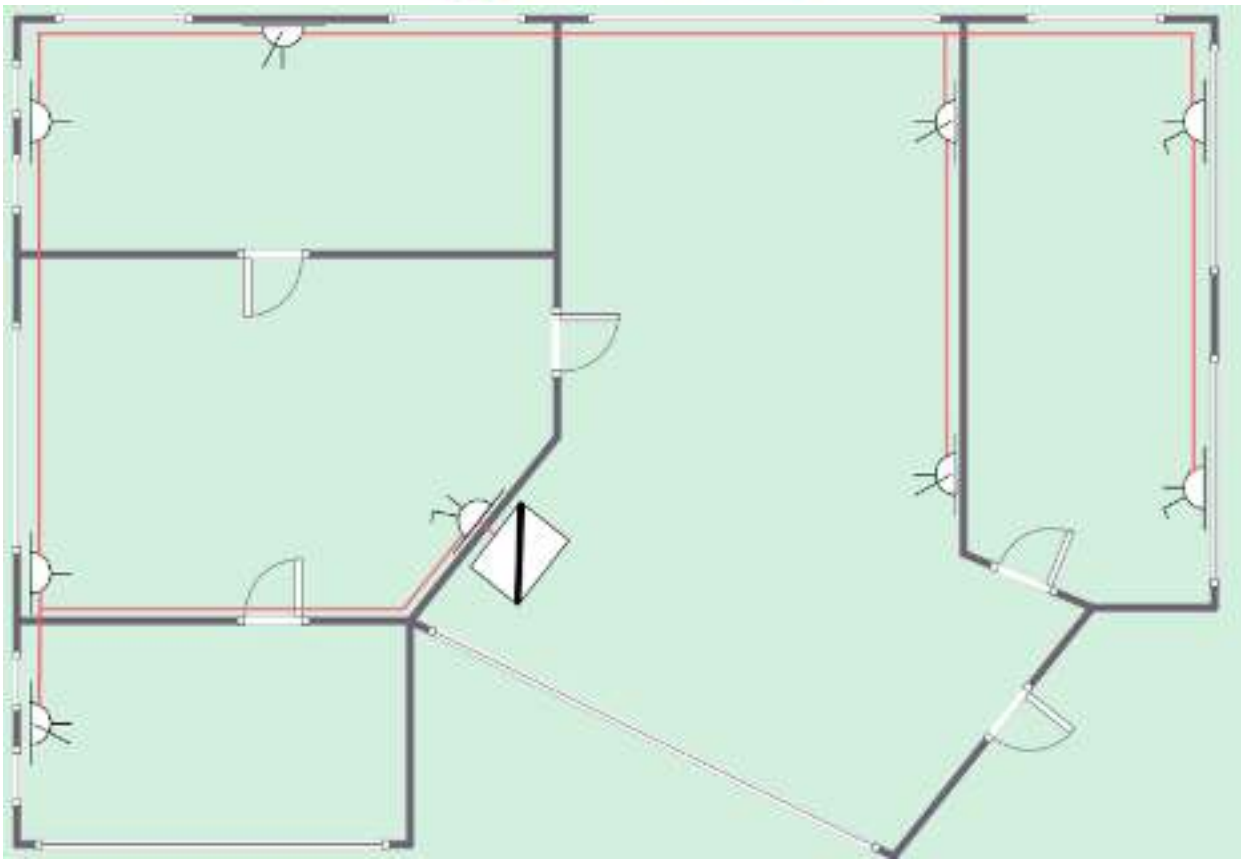


Figure:2.100. 2d drawing

Operation sheet 2: Produce 2d floor plan with electrical plan

Operation title: Produce of building electrical plan

Purpose: To practice and demonstrate the skill, knowledge and attitude required to produce building electrical plan

Instruction: Use the given auto cad software to produce building electrical plan including the finishing construction field. For this operation you have given 2Hour each electrical plan and you are expected to provide the answer on the given table.

Tools and requirement:

- ✓ Computer
- ✓ Auto cad software
- ✓ Printer
- ✓ This operation sheets

Precautions: when produce 2d modeling, the electrical plan clear and understandable These procedures provide a general guideline for creating building drawings in AutoCAD. However, it's important to note that the specific commands and techniques may vary depending on the version of AutoCAD you are using.

Procedures in doing the task

Step 1: Gather reference information

Step 2: Set up the drawing environment

Step 3: Start a new drawing

Step 4: Create the floor plan

Step 5: Set up layers

Step 6: Add dimensions and annotations

Step 7: Create symbols and blocks

Step 8: Create sections

Step 9: Develop the site plan

Step 10: Apply hatching and shading

Step 11: Add dimensions and annotations

Step 12: Review and refine

Step 13 Save your electrical plan

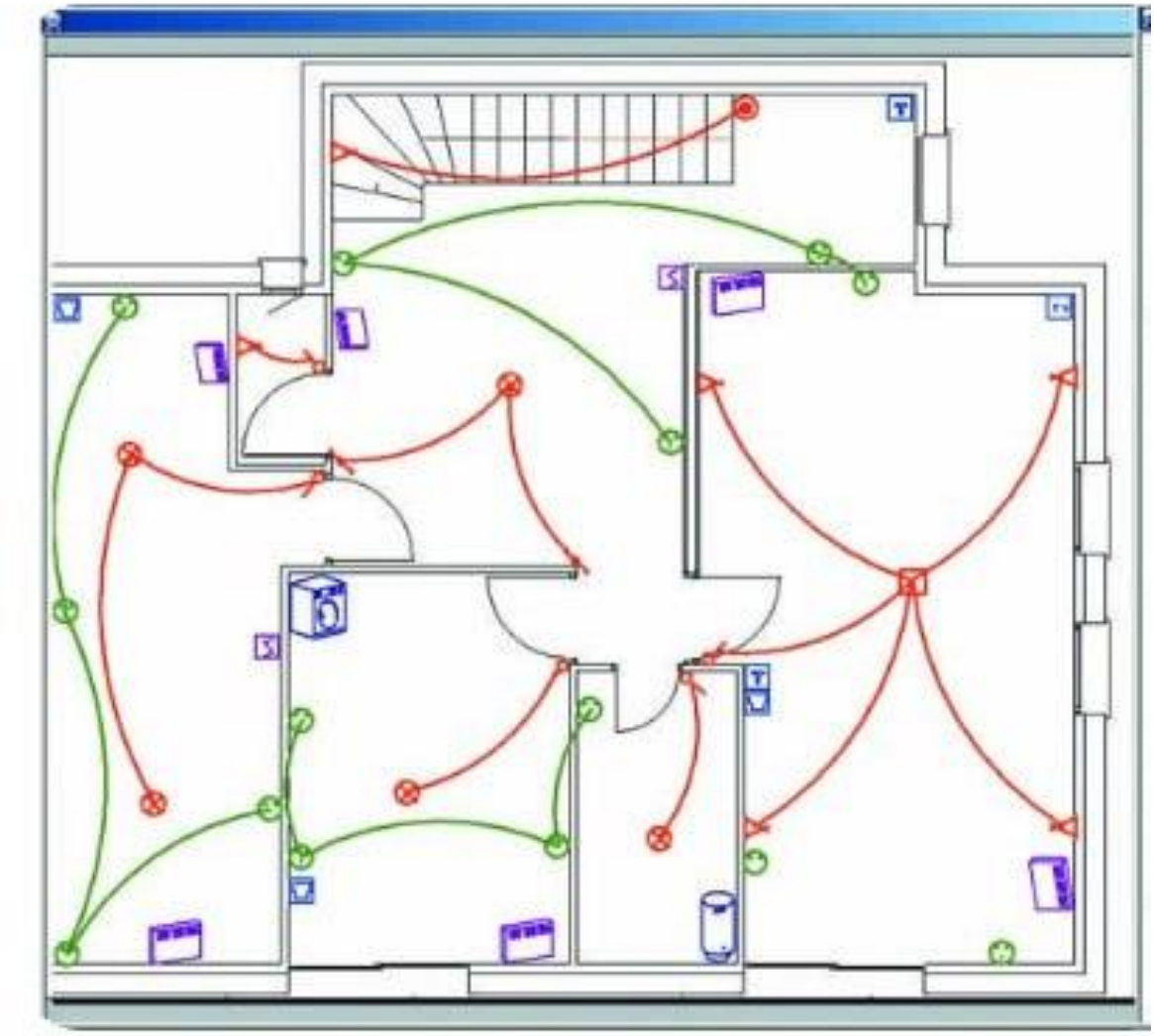


Figure:2.101. Electrical Plan on floor pan

LAP Test: Practical Demonstration

Name: _____ Date: _____

Time started: _____ Time finished: _____

Instructions: Given necessary templates, tools and materials you are required to perform the following tasks within 3 hours each task.

Task 1: Produce 2d floor plan & put electrical symbols

Task 2: Produce 2d floor plan with electrical plan

This unit is developed to provide you the necessary information regarding the following content coverage and topics:

- Concepts of 3D modelling
- Creating and manipulating 3D objects
- Modifying and editing 3D objects
- Materials and textures

This unit will also assist you to attain the learning outcomes stated in the cover page. Specifically, upon completion of this learning guide, you will be able to:

- Understanding the concepts of 3D modelling
- Create and manipulate 3d objects
- Modify and edit 3D objects
- Apply materials and textures on designed modelling

3.1. Concepts of 3D modelling

AutoCAD is a powerful computer-aided design (CAD) software widely used in various industries for creating detailed and precise 2D and 3D models. In the context of 3D modeling, AutoCAD provides a comprehensive set of tools and features to design, visualize, and document

three-dimensional objects and environments. Whether you're an architect, engineer, product designer, or involved in any field that requires precise 3D representations, AutoCAD offers a robust platform for your modeling needs.

With AutoCAD's 3D modeling capabilities, you can create complex three-dimensional objects with realistic textures, materials, and lighting effects. The software supports both surface modeling and solid modeling techniques, providing flexibility and versatility in your design process.

Surface modeling in AutoCAD allows you to create smooth, organic shapes by defining boundary curves, extruding 2D shapes, or using advanced surfacing tools. This technique is particularly useful for creating complex curves, intricate designs, and irregular shapes.

Solid modeling, on the other hand, enables you to create 3D objects with volume. Using predefined 3D primitives or combining surfaces, you can construct solid models that accurately represent physical objects. Solid modeling in AutoCAD supports advanced operations like Boolean operations for combining or subtracting objects, as well as filleting and chamfering to create rounded or bevelled edges.

AutoCAD provides a range of tools for editing, modifying, and refining your 3D models. You can manipulate individual vertices, edges, and faces, as well as move, rotate, or scale objects in 3D space. Precise measurements and dimensional constraints ensure accuracy and adherence to design specifications.

To enhance the visual realism of your 3D models, AutoCAD allows you to assign materials and textures to surfaces, define lighting conditions, and apply rendering techniques. This enables you to simulate different materials, surface finishes, and lighting scenarios, resulting in highly realistic and visually appealing renderings.

AutoCAD also offers powerful viewing and navigation tools, allowing you to explore your 3D models from different angles, adjust the perspective, and zoom in and out for detailed examination.

With its compatibility with various file formats, AutoCAD facilitates collaboration and data exchange with other software applications.

Generally, AutoCAD's 3D modeling capabilities provide a comprehensive solution for creating, modifying, and visualizing intricate three-dimensional designs. Whether you're working on architectural structures, mechanical components, or any other 3D project, AutoCAD empowers you to bring your ideas to life with precision and accuracy.

AutoCAD is a popular software used for 3D modeling in the field of computer-aided design (CAD). While many of the general 3D modeling concepts apply to AutoCAD, there are some specific concepts and features that are important to understand.

Some key concepts related to 3D modeling in AutoCAD:

- **Coordinate Systems:** AutoCAD uses a Cartesian coordinate system to define the position of objects in 3D space. The three axes are X, Y, and Z, representing the horizontal, vertical, and depth dimensions respectively.
- **Wireframe Modeling:** AutoCAD allows you to create 3D models using a wireframe representation. In this approach, you create lines and curves to define the edges and surfaces of the model. It provides a basic representation without any solid or surface properties.
- **Surface Modeling:** AutoCAD includes tools for creating 3D surfaces. Surfaces are generated by defining boundary curves or by extruding 2D shapes along a path. Surfaces can be used to represent complex shapes and can be further modified and manipulated.
- **Solid Modeling:** AutoCAD supports solid modeling, which involves creating 3D objects with volume. Solid models are created by combining surfaces or by using predefined 3D primitives such as cubes, cylinders, and spheres. Solid modeling enables advanced operations like Boolean operations (union, subtract, intersect) and filleting.
- **Editing and Modifying:** AutoCAD provides a wide range of tools for editing and modifying 3D models. These tools allow you to move, rotate, scale, mirror, offset, and align objects in 3D space. we can also modify individual faces, edges, and vertices of solid and surface models.
- **Materials and Textures:** AutoCAD allows you to assign materials and textures to 3D objects. Materials define the visual properties of surfaces, such as color, reflectivity, and transparency. Textures can be applied to surfaces to add realistic details like wood grain or fabric patterns.
- **Lighting and Rendering:** AutoCAD provides lighting tools to simulate different lighting conditions in a 3D scene. You can add point lights, spotlights, and distant lights to illuminate your models. Rendering options allow you to create realistic images by simulating light interactions and material properties.

- **Viewing and Navigation:** AutoCAD offers various viewing modes and navigation tools to help you work with 3D models. You can switch between different viewpoints, adjust the perspective, zoom in and out, and rotate the model to view it from different angles.

3.2. Creating and Manipulating 3D Objects

Primitives: 3D modeling software often provides primitive shapes like cubes, spheres, cylinders, and cones to create basic objects. These primitives can be modified, combined, or duplicated to form more complex shapes.

Extrusion: Extrusion involves extending a 2D shape along a path to create a 3D object. This technique is commonly used to create objects with depth, such as walls or beams.

Creating and manipulating 3D objects involves using various tools and techniques to shape and modify the geometry of objects. There are Some common methods and operations used in 3D modelling:

1. **Extrude:** This operation involves extending a 2D shape along a path to create a 3D object. It can be used to create features like walls, cylinders, or extruded text. The **Extrude** command in AutoCAD 3D is used to create a surface or solid of a 2D surface or a 3D curve.

Example 1: Consider the below 2D surface.

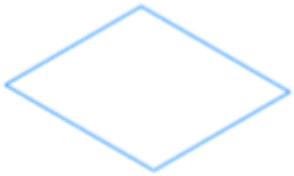


Figure:3.1(a). 2D surface

Here, we have created a rectangle.

To create a solid of the above figure, follow the below steps:

a) Select Extrude command from the ribbon panel, as shown below:

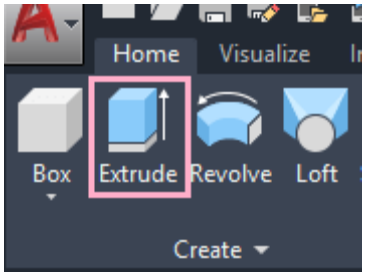


Figure:3.1(b). Extrude

Or Type **Extrude** on the command line < press **Enter**.

b) Select the object, which we want to Extrude. Here, we have selected the rectangle.

c) Press **Enter**.

d) Now, specify the **height** of the Extrude. We can specify the height in **+Z** or **–Z** direction.

We can either determine the height with the cursor or by the value. Here,

we have specified the value = **5**. The object will now look like the below image:

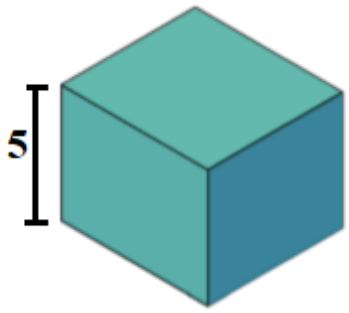


Figure:3.1(c). specify the height of the Extrude

To create a surface of the 2D rectangle, follow the below steps:

a) Type **Extrude** on the command line < press **Enter**.
b) Type **Mode** or **M** on the command line < press **Enter**.
c) Type **Surface** or **SU** on the command line < press **Enter**.
d) Select the object to Extrude.
   Here, we have selected the rectangle.
e) Press **Enter**.
f) Specify the **height** of the Extrude.
   Here, we have specified the value = **3**.
   The object will now look like the below image:

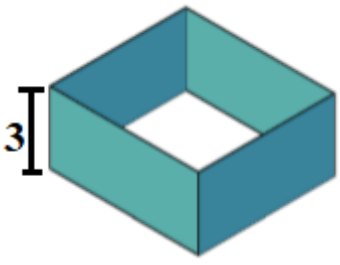


Figure:3.2. Create a surface of the 2D rectangle

To Create using **Direction/ path/ Taper Angle/ Expression**, follow the below steps:

a) Type **Extrude** on the command line < press **Enter**.
b) Select the object, which we want to Extrude.
c) Select one option from **Direction/ path/ Taper Angle/ Expression.**
   Here, we have selected Taper Angle.
d) Press **Enter**. The figure will now appear as:

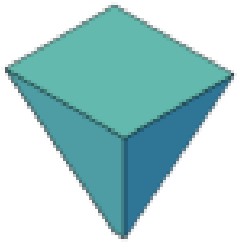


Figure:3.3. Direction/ path/ Taper Angle/ Expression

Example 2: Consider the 2D object shown below:

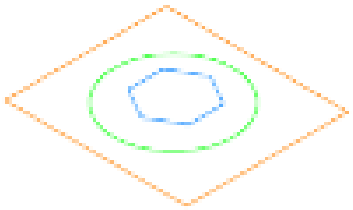


Figure:3.3(a). 2D object

Here, we have drawn three objects.

The steps are listed below:

a) Type **Extrude** on the command line < press **Enter**.

b) Select the object.
Here, we will first select the inner **polygon**.

c) Press **Enter**.

d) Specify the height. Here, we have entered height = **4**.
The object will appear as:

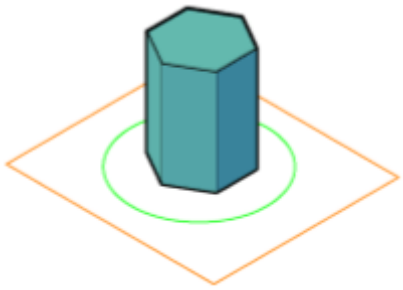


Figure:3.3(b). Extrude polygon

e) Again, Type **Extrude** on the command line < press **Enter**.

f) Select the object. Here, we will select the **circle**.

g) Press **Enter**.

h) Specify the height. Here, we have entered height = **5**. The object will appear as:

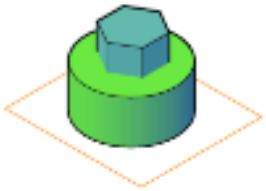


Figure:3.3(c). Extrude circle

i) Again, Type **Extrude** on the command line < press **Enter**.

j) Select the object.
Here, we will select the **outer rectangle**.

k) Press **Enter**.

l) Specify the height. Here, we have entered height = **1**.

The object in **3D** will now look like the below image:

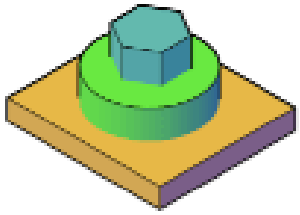


Figure:3.3(c). Extrude the object

2. **Revolve:** Revolve allows you to create a 3D object by rotating a 2D shape around an axis. It is useful for creating symmetrical objects like vases, cups, or columns.

The Revolve command in AutoCAD 3D is used to create a 3D solid or surface by sweeping the object through its axis at a predefined angle. The paths and objects that can be revolved are:

- Planar
- Non-planar
- Open
- Closed
- Solid
- Surfaces
- Circles
- Arcs

The positive direction of rotation is determined by the right-hand thumb rule. We can rotate the object with respect to **X-axis, Y-axis,** and **Z-axis**. We are required to use polyline to create a base for revolve command. We can either join the lines or segments before revolving.

Example 1: To revolve with respect to X, Y, or Z-axis. Consider the below 2D object:

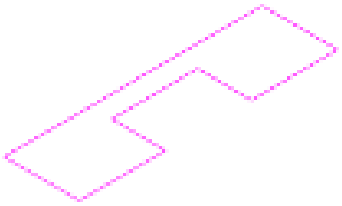


Figure:3.4(a). 2D object

Make sure to activate the **ortho mode.**

The above object is drawn in **SE Isometric** Mode.

The above object was first joined.

Place the **UCS** icon on the object.

The steps are listed below:

a. Create the above figure.

b. Select **Revolve icon** from the ribbon panel, as shown below:

Figure:3.4(a). Revolve icon

Or Type **REV** or **revolve** on the command line or command prompt and press **Enter**.

c. Select the object to revolve with a small square cursor, as shown below:

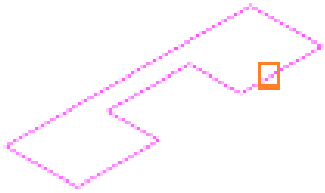


Figure:3.4(b). Select of the object

d. Press **Enter**.

e. Specify the **X, Y,** or **Z axis**.

If **X-axis** is selected:

Figure:3.4(c). Revolve the object in X-axis

If **Z-axis** is selected:

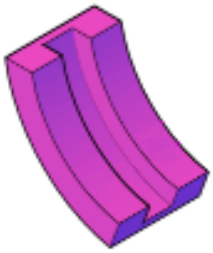


Figure:3.4(d). Revolve the object in Z-axis

f. Press **Enter**.

g. Specify angle of revolution.

h. Press **Enter**.

To revolve according to the axis line, follow the below steps:

a. Type **REV** or **revolve** on the command line or command prompt and press Enter.

b. Select the object to revolve with a small square cursor, as shown below:

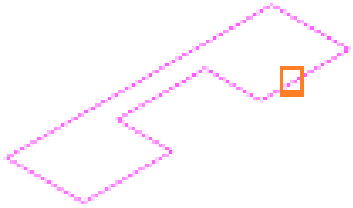


Figure:3.5(a). Select the object to revolve

c. Press **Enter**.

d. Specify axis start point of the object to be revolved.

e. Specify axis endpoint.

f. Specify angle of revolution.

If the specified angle = **30**, the object will look like the below image:

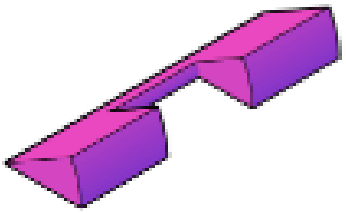


Figure:3.5(b). Revolve the object by 30^0

If the specified angle = **360**, the object will look like the below image:

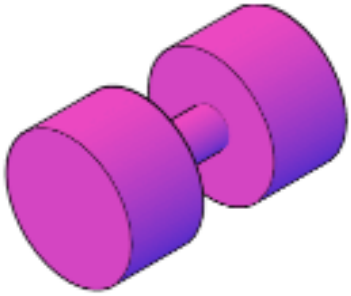


Figure:3.5(c). Revolve the object by 360^0

If the specified angle is **95**, the object will look like the below image:

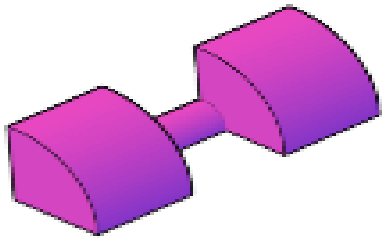


Figure:3.5(d). Revolve the object by 95^0

g. Press **Enter**.

3. **Loft:** Lofting is used to create a smooth transition between two or more 2D shapes or profiles. It is often used to create complex organic shapes, like car bodies or architectural curves.

The **loft** command in AutoCAD is used to create 3D solid or surface. The 3D solid or surface is formed within the space between various cross sections. The cross-sections determine the outer shape of the solid or surface. To create a 3D object using LOFT, we are required to specify at least two cross-sections.

Example 1: Consider the below image:

Figure:3.6(a). SE Isometric drawing

The above drawing is created in **SE Isometric**.

We can also change the **View control** accordingly.

The steps are listed below:

a. Create the three circles, as shown above.

b. Move the circles separately in the direction of Z-axis using the MOVE command, as shown below:

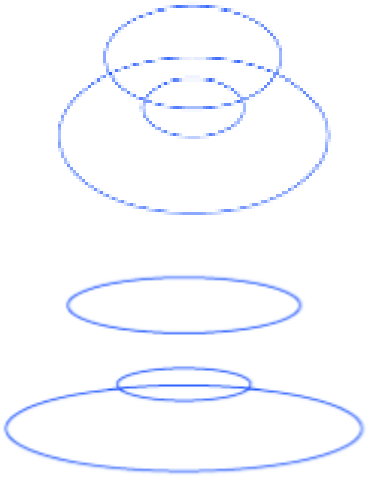


Figure:3.6(b). Move the circles separately in the direction of Z-axis

c. Select the **LOFT** icon from the ribbon panel, as shown below:

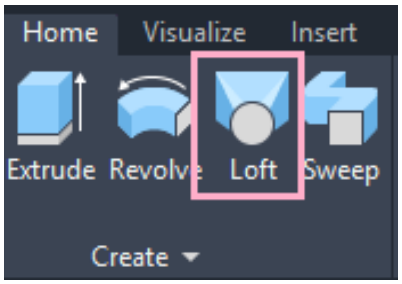


Figure:3.6(c). loft icon

Or Type **Loft** on the command line < press **Enter**.

d. Select the cross-section (circles) to apply loft, as shown below:

Figure:3.6(d). Select the circles

The order is shown above.

e. Press **Enter**.

The loft will be created, as shown below:

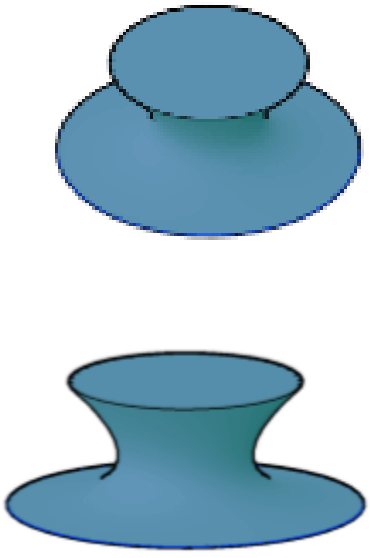


Figure:3.6(e). lofted object

4. **Sweep:** The sweep operation involves sweeping a 2D shape along a path to create a 3D object. It is commonly used to create features like pipes, rails, or mouldings. The 2D sub-object or object is swept along an open or closed path to create a 3D surface or solid using sweep command. The object enclosing the area is swept to create a 3D solid or surface, but an open-ended object is swept to create only a 3D surface.

Consider the below image:

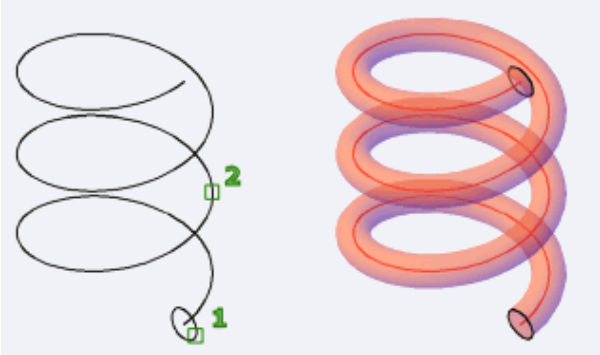


Figure:3.7. Sweep object

It clearly shows the concept of sweep.

The path and objects that can be swept to create a 3D object are:

- Arcs
- Solid
- Polylines
- Lines
- Splines
- Elliptical Arc
- Regions
- Edge sub-objects
- Trace

We can select the sub-objects by pressing the **Ctrl** key.

Example: Consider the below 2D object:

Figure:3.8(a). Create the line

Create the object using the Polyline command or join the lines.

The steps are listed below:

a) Create a circle at the endpoint of the object, as shown below:

Figure:3.8(b). Create a circle at the endpoint

b) Select the **Sweep icon** from the ribbon panel, as shown below:

Figure:3.8(c). Sweep icon

Or Type **SWEEP** on the command line or command prompt and press **Enter**.

c) Select the object to Sweep. Here, we will select a small circle.
d) Press **Enter**.
e) Select the sweep path. Here, we will select the object created using the polyline.
f) Press **Enter**.
g) The sweep will be created, as shown below:

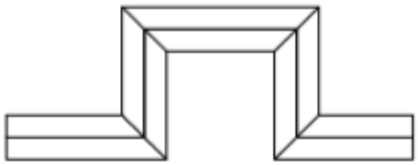


Figure:3.8(d). Created sweep

h) Now, convert the view control to **SW Isometric** and Visual control to **Conceptual**.

The object will now look like the below image:

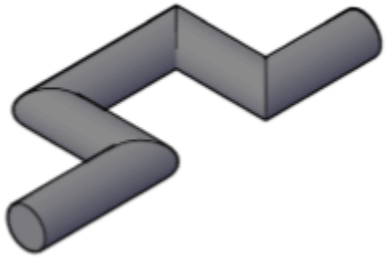


Figure:3.8(e). Created sweep

5. **Press/Pull:** Press/Pull allows you to push or pull a face or edge to extrude or resize it. It is a versatile tool for quickly modifying the shape of an object. The Press/pull command in AutoCAD 3D is used to create an area formed by the closed boundary. It is used to pull an enclosed boundary. It can also be used to create a 3D solid in case of the selected circle or interior of the bounded area. The objects, such as arc, can be used to create a 3D surface. The Press/pull command is also used to cut a part from the whole object. For example, creating holes, etc. Consider the below image:

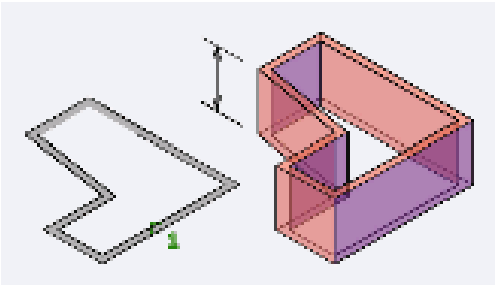


Figure:3.9. Press/Pull object

Example 1: Consider the below 2D surface.

Figure:3.9. (a) 2D surface

Here, we will extrude the boundary of the hexagon.

The steps are listed below:

a) Select the **Press/pull** icon from the ribbon panel, as shown below:

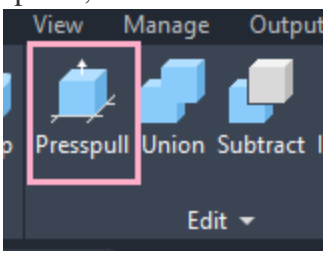


Figure:3.9. (b) Press/pull icon

Or Type **PRESSPULL** on the command line and < press **Enter**.

b) Select the object or bounded area. Here, we will select the bounded area between two edges of hexagon, as shown below:

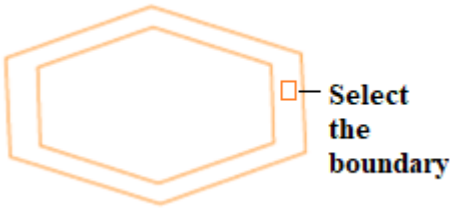


Figure:3.9. (c) select the boundary

c) Specify the **Extrusion height**. Here, we have specified height= 3. The object will now look like the below image:

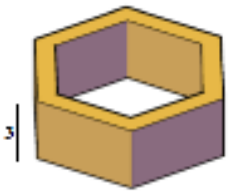


Figure:3.9. (d) Extrusion height

To remove a part of an object, Use the **Press/pull** command < **select** the area to be removed < **pull** that selected area in outer direction < press **Enter**.

6. **Boolean Operations:** Boolean operations involve combining or subtracting multiple objects to create new shapes. Operations like union, difference, or intersection are used to merge or cut objects to achieve desired results. There are three Boolean operations defined in AutoCAD 3D, which are given below:

- **Union**
- **Subtract**
- **Intersect**

These three icons are present on the ribbon panel, as shown below:

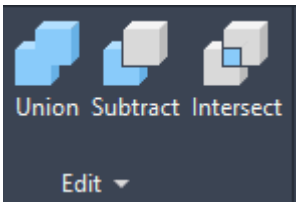


Figure:3.10. Boolean operations icon

I. **Union:** The Union command is used to combine the selected 2D regions or 3D

solids by the addition. The selected parts are combined to form a single region, surface, or solid. But we are required to select the parts of the same object. We can select two or more parts to combine it into a single region, solid or surface. Consider the below image:

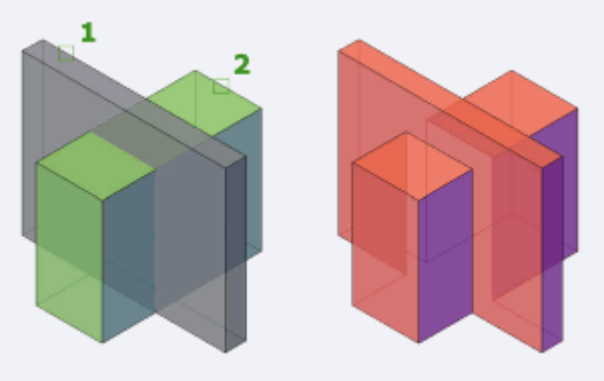


Figure:3.11. Union of the objects

II. **Subtract**: The Subtract command is used to subtract the selected 2D regions or 3D solids by the subtraction. Consider the below figure:

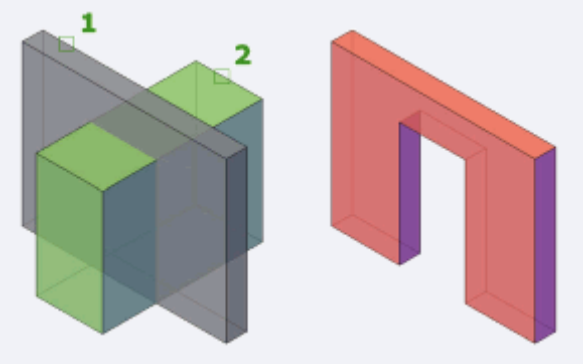


Figure:3.12. Subtract of the objects

III. **Intersect:** The intersect command is used to create a 2D region or 3D solid formed by two overlapping regions or solids. Consider the below image:

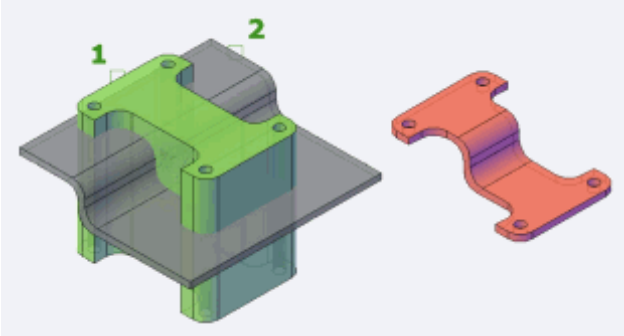


Figure:3.13. Intersect of the objects

7. **Fillet and Chamfer Edges:** Filleting rounds off sharp edges by adding a curved transition, while chamfering creates a bevelled edge. These operations add realism and smoothness to objects.

Fillet Edge: The Fillet command in 3D is used to create smooth edges. Consider the below image:

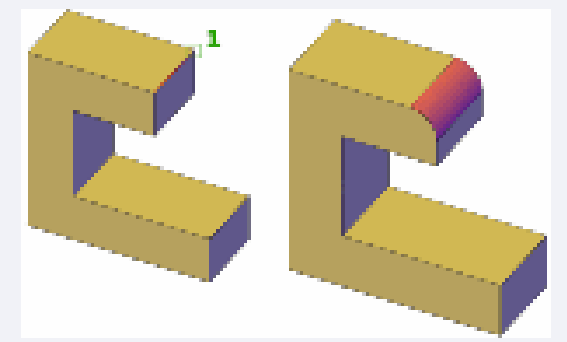


Figure:3.14. Fillet Edge

It shows the concept of Fillet in 3D.

Example:

The steps are listed below:

a) Type **FILLETEDGE** on the command line or command prompt and press **Enter**.

b) Type **R** or **radius** on the command line.
c) Press **Enter**.
d) Type **1** (radius value).
e) Press **Enter**.
f) Select the Edge to apply Fillet on it. Here, we have selected the edge, as shown below:

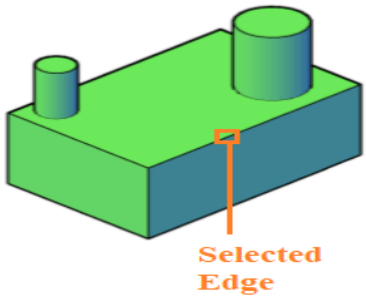


Figure:3.14. (a) selected Edge

g) Press **Enter**.
The object after fillet will look like the below image:

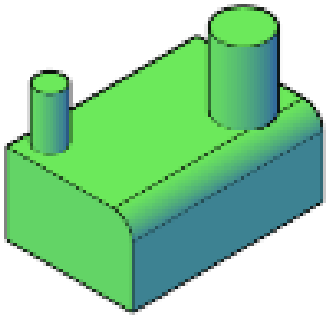


Figure:3.14. (b) filleted Edge

Chain fillet: Chain fillet is used to continue applying FILLET on other edges with the same radius, follow the below steps:

a) Type **FILLETEDGE** on the command line or command prompt and press **Enter**.
b) Type **C** or **Chain** on the command line.
c) Press **Enter**.
d) Type **R** or **radius** on the command line.
e) Press **Enter**.
f) Type **1** (radius value).
g) Press **Enter**.
h) Continue selecting the Edges to apply Fillet on it.
i) Press **Enter**.

Chamfer: The Chamfer command in AutoCAD 3D is used to create Slanted edges.

Example: The steps are listed below:

a) Type **CHAMFEREDGE** on the command line or command prompt and press **Enter**.
b) Type **D** or **Distance** on the command line.
c) Press **Enter**.
d) Specify Distance1. For Example, **2** (distance value 1).
e) Press **Enter**.
f) Specify Distance2. For Example, **2** (distance value 2).
g) Press **Enter**.

h) Select the Edge to apply Chamfer on. Here, we have selected the edge, as shown below:

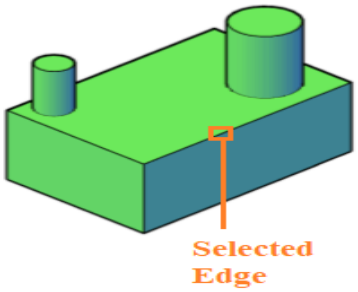


Figure:3.15. (a) Selected Edge

The object after using Chamfer command will look like the below image:

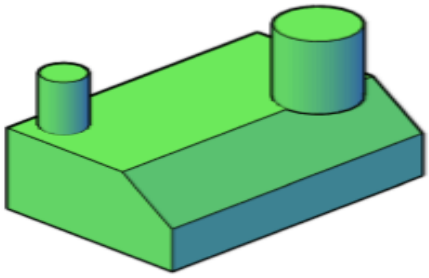


Figure:3.15. (b) Chamfered Edge

i) Press **Enter**.

j) Press **Enter**.

Loop Chamfer: To continue apply CHAMFER on other edges with the same Distance1 and Distance2, follow the below steps:

a) Type **CHAMFEREDGE** on the command line or command prompt and press **Enter**.

b) Type **L** or **Loop** on the command line.

c) Press **Enter**.

d) Type **D** or **Distance** on the command line.

e) Press **Enter**.

f) Specify Distance1. For Example, **2** (distance value 1).

g) Press **Enter**.

h) Specify Distance2. For Example, **2** (distance value 2).

i) Press **Enter**.

j) Continue selecting the Edges to apply Chamfer on.

k) Press **Enter**.

8. **Solid Editing:** Solid editing tools allow you to modify the shape of a solid object by adding or subtracting volumes, splitting or joining objects, or performing advanced operations like shelling or thickening.
9. **3D Array**: A 3D array tool allows you to create multiple copies of an object in a linear or radial pattern. It is useful for creating repetitive elements like fences, windows, or columns.

10. **Section Plane:** A section plane is a tool that allows you to cut through an object and view its internal structure. It is helpful for inspecting and modifying the inside of complex objects.

Type plae

Specifies a plane, slice, boundary or volume as the parameter when creating the section plane. Once you choose a style, the command reverts back to the first prompt, with the selected type set as default.

- **Plane.** Allows you to specify a planar segment of a 3D solid, surface, mesh, or point cloud and place the section plane.

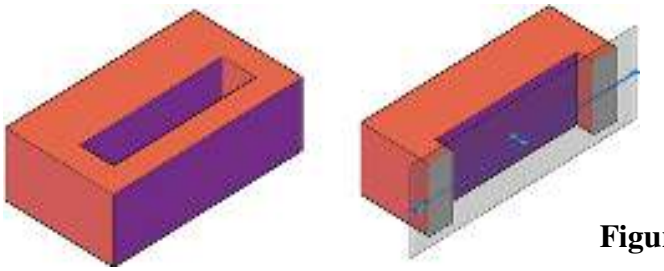


Figure:3.16. (a) Plane

- **Slice.** Allows you to select a planar segment with depth of a 3D solid, surface, mesh or point cloud to place the section plane.

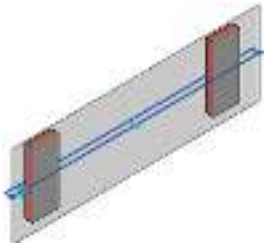


Figure:3.16. (b) Slice

Note: The slice cannot contain any jogs and the draw selection option is disabled.

- **Boundary.** Allows you to select the boundary of a 3D solid, surface, mesh, or point cloud and place the section plane.

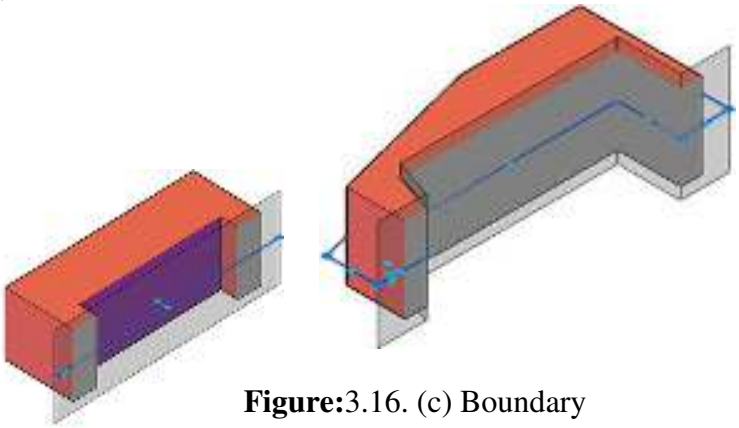


Figure:3.16. (c) Boundary

- **Volume.** Allows you to create a bounded volume section plane.

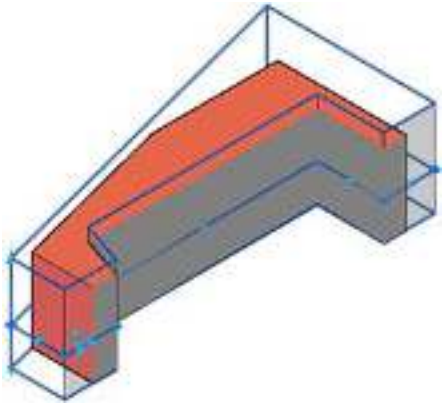


Figure:3.16. (d) Volume

11. **3D Orbit:** 3D orbit is a navigation tool that allows you to rotate, pan, and zoom around your 3D model. It helps you view and manipulate objects from different angles and perspectives.

These tools and operations are commonly found in 3D modeling software, but their availability and functionality may vary depending on the software you are using. It's important to consult the documentation or tutorials specific to your software to learn how to use these tools effectively. Creating and manipulating 3D objects in AutoCAD involves various techniques and commands. Step-by-step guide on how to create and manipulate 3D objects:

- **Launch AutoCAD:** Open AutoCAD on your computer and start a new drawing or open an existing one.
- **Set the Workspace:** Ensure that you are in a 3D workspace. You can switch to a 3D workspace by selecting the appropriate workspace from the Workspace drop-down menu on the Quick Access toolbar.
- **Select the Drawing Plane:** Choose the plane on which you want to create your 3D object. You can select the XY, XZ, or YZ plane as your working plane by using the "UCS" command or by clicking on the UCS icon in the lower-left corner of the AutoCAD window.
- **Create a Basic 2D Shape**: Start by creating a basic 2D shape using drawing tools such as lines, circles, arcs, or polylines. Use the appropriate commands and tools to draw the desired shape on your chosen working plane.
- **Extrude the 2D Shape:** Once you have the 2D shape, you can extrude it to give it depth and convert it into a 3D object. Use the "Extrude" command (shortcut: EX) and select the 2D shape. Specify the height or distance by which you want to extrude the shape.
- **Revolve the 2D Shape:** Another method to create a 3D object is by revolving a 2D shape around an axis. Use the "Revolve" command (shortcut: REVOLVE), select the 2D shape, and specify the axis of rotation and the angle. This will create a 3D object by revolving the shape.
- **Use Loft or Sweep:** The "Loft" and "Sweep" commands enable you to create complex 3D shapes by blending or sweeping between multiple 2D profiles or along a path. Use these commands to create organic or intricate shapes based on your design requirements.

- **Perform Solid Editing**: AutoCAD provides various solid editing commands to modify and manipulate 3D objects. Use commands like "Union" (shortcut: UNION), "Subtract" (shortcut: SUBTRACT), "Intersect" (shortcut: INTERSECT), or "Slice" (shortcut: SLICE) to combine, subtract, intersect, or cut through 3D objects.
- **Apply Fillet and Chamfer:** To add rounded or beveled edges to your 3D objects, use the "Fillet" (shortcut: FILLET) and "Chamfer" (shortcut: CHAMFER) commands. Select the edges of the object and specify the desired fillet or chamfer radius to create the desired effect.
- **Use 3D Array:** The "Array" command (shortcut: ARRAY) allows you to create multiple copies of objects in a specified arrangement. Use this command to create linear, rectangular, or polar arrays of your 3D objects.
- **Navigate and View in 3D:** To navigate and view your 3D objects from different angles, use the "3D Orbit" command (shortcut: 3DORBIT). This command allows you to rotate, pan, and zoom your view in 3D space for better visualization and editing.

3.3. Modifying and Editing 3D Objects

Modifying and editing 3D objects is a fundamental aspect of 3D modeling. It involves making changes to the shape, size, position, or appearance of objects to achieve the desired result. Whether you are creating a complex architectural structure, designing a character, or crafting a product prototype, the ability to modify and edit 3D objects is essential.

There are various tools and techniques available in 3D modeling software that enable you to modify and edit objects with precision and creativity. These tools range from basic transformation tools like move, rotate, and scale, to more advanced modeling tools like extrude, bevel, and boolean operations. Additionally, sculpting tools allow for organic and detailed modifications, while modifier stacks provide non-destructive editing capabilities.

Modifying and editing 3D objects also involves working with materials and textures. By adjusting the properties of materials, such as colour, reflectivity, and transparency, you can achieve the desired look and feel of objects. Textures, on the other hand, add intricate details and patterns to surfaces, enhancing the realism and visual appeal of the objects.

Through the process of modifying and editing 3D objects, you have the flexibility to refine and perfect your designs. It allows you to iterate, experiment, and make adjustments until you achieve the desired outcome. With practice and experience, you can develop your skills in manipulating objects, creating complex shapes, and transforming ideas into tangible 3D models.

Whether you are a hobbyist, a professional designer, or an architect, understanding the concepts and techniques of modifying and editing 3D objects opens up a world of creative possibilities in the realm of 3D modeling.

Modifying and editing 3D objects involves making changes to their shape, size, position, or appearance. The common techniques and tools used for modifying and editing 3D objects:

- **Selection:** Before making any modifications, you need to select the object or objects you want to edit. In most 3D modeling software, you can select objects by clicking on them in the viewport or using selection tools like lasso or box selection.
- **Transformation Tools:** Transformation tools allow you to modify the position, rotation, and scale of objects. Common transformation tools include:
  - Move Tool: Allows you to move objects along the X, Y, or Z axis.
  - Rotate Tool: Enables you to rotate objects around a specific axis or freely in 3D space.
  - Scale Tool: Lets you resize objects uniformly or along specific axes.
- **Modeling Tools:** Modeling tools provide more advanced editing capabilities to modify the shape and geometry of objects. Some common modeling tools include:
  - Extrude: Creates new geometry by extending a face or edge.
  - Bevel: Adds a chamfer or roundness to edges or corners.
  - Bridge: Connects two or more edges or faces to create a bridge between them.
  - Boolean Operations: Combines or subtracts objects using operations like union, difference, or intersection.
  - Subdivision Surfaces: Divides the object into smaller polygons to achieve smoother surfaces.
- **Sculpting Tools:** Sculpting tools are used for more organic and detailed modifications. With sculpting tools, you can push, pull, smooth, or add details to the surface of objects. Sculpting is commonly used for character modeling or organic shapes.

- **Modifier Stack:** Many 3D modeling software offers a modifier stack or history panel. This allows you to apply non-destructive modifications to objects. You can stack modifiers like bend, twist, or mirror to create complex effects and easily adjust or remove them later.
- **Material and Texture Editing:** Modifying the appearance of objects involves adjusting materials and textures. You can change colors, adjust reflectivity, add patterns, or apply images to the surface of objects. Material and texture editors within the software provide options to customize these properties.

3.4. Materials and Textures

Materials and textures play a crucial role in 3D modeling as they add realism, detail, and visual appeal to objects. Overview of materials and textures in 3D modeling:

Materials: Materials define the visual properties of 3D objects, such as color, reflectivity, transparency, and shininess. They can be assigned to different parts of an object to simulate different materials like metal, plastic, or glass.

- **Diffuse Colour:** This is the base colour of an object. It determines how light is absorbed and reflected by the surface.
- **Specular Reflection:** Specular reflection controls how shiny or reflective an object appears. Higher values create a more reflective surface, while lower values result in a duller appearance.
- **Roughness:** Roughness determines the smoothness or roughness of an object's surface. Lower values make the surface smoother and more reflective, while higher values make it rougher and less reflective.
- **Transparency:** Transparency controls how much light passes through an object. It is useful for creating materials like glass or water.
- **Bump/Normal Maps:** Bump or normal maps add the illusion of detail to a surface without actually changing its geometry. They create the appearance of bumps, scratches, or other surface imperfections.

Textures: Textures are 2D images applied to the surface of 3D objects to add detail and realism. They can be used to simulate patterns, surface imperfections, or complex materials like wood or

fabric. Texture mapping techniques, such as UV mapping, are used to apply textures accurately to the 3D object's surface.

- **Color Texture:** Color textures are images that are applied to an object's surface to add color variations, patterns, or images. They enhance the visual appeal and realism of the object.
- **Normal Maps:** Normal maps simulate surface details by encoding information about the orientation of surface normal. They create the illusion of bumps, creases, or other surface irregularities.
- **Specular Maps:** Specular maps control the intensity and distribution of specular reflections on an object's surface. They define areas of high or low reflectivity.
- **Roughness Maps:** Roughness maps determine the roughness or smoothness of an object's surface in different areas. They allow for more realistic and detailed rendering of materials.
- **Displacement Maps:** Displacement maps modify the geometry of an object by displacing the vertices based on the values in the map. They can create intricate surface details like wrinkles or grooves.
- **Opacity Maps:** Opacity maps control the transparency or opacity of an object. They define areas that are fully opaque, partially transparent, or completely transparent.
- **Ambient Occlusion Maps:** Ambient occlusion maps simulate the darkening of crevices or areas where objects come into contact, adding depth and realism to the model.

These materials and textures can be applied to specific parts or the entire surface of an object, depending on the desired effect. Most 3D modeling software provides a range of material and texture editing tools to adjust properties like color, reflectivity, transparency, or mapping coordinates.

Experimenting with different materials and textures can greatly enhance the visual quality of your 3D models and bring them to life. It's important to consider the lighting setup and environment when applying materials and textures to ensure a realistic and cohesive appearance.

To apply materials and textures to your 3D drawing, follow these steps:

- **Select the object:** Choose the object or objects in your 3D drawing that you want to apply materials and textures to. This can typically be done by clicking on the object in the viewport or using selection tools like lasso or box selection.
- **Open the material/texture editor:** In your 3D modeling software, locate and open the material or texture editor. This panel or window allows you to customize the properties of the materials and textures.
- **Choose a material or texture:** In the material or texture editor, browse through the available options or create your own. You can choose from pre-made materials or textures provided by the software, or create custom ones by adjusting various parameters.
- **Apply the material or texture:** Once you have chosen the material or texture, apply it to the selected object. This can typically be done by dragging and dropping the material or texture onto the object, or by using a specific tool like the paint bucket tool.
- **Adjust properties:** After applying the material or texture, you can adjust its properties to achieve the desired look. This may include modifying the color, reflectivity, transparency, roughness, or any other relevant properties. Use the options and controls provided in the material or texture editor to make these adjustments.
- **Mapping and scaling:** In some cases, you may need to adjust the mapping and scaling of the material or texture to ensure it fits correctly on the object's surface. This can be done by manipulating mapping coordinates or using tools like UV mapping or texture scaling.
- **Preview and fine-tune:** As you make changes to the materials and textures, it's important to preview the results in the viewport. This allows you to see how the object looks with the applied materials and textures and make any necessary adjustments to achieve the desired result.
- **Repeat for other objects:** If you have multiple objects in your 3D drawing that require materials and textures, repeat the above steps for each object, applying the appropriate materials or textures to each.

Example: of the 3D modeling a building drawing using AutoCAD

1. **Set up the drawing:** Open AutoCAD and create a new file. Set the units and scale appropriate for your project.

2. **Import or create a base plan:** Import a 2D drawing or create a basic outline of the building's footprint using AutoCAD's drawing tools. Ensure that the drawing is accurately scaled.
3. **Extrude walls:** Use the "Extrude" command to give thickness to the walls. Select the lines representing the walls and specify the desired height for each wall. Repeat this step for all the walls in the building.
4. **Insert doors and windows:** Use the appropriate commands (such as "Insert" or "Block") to add door and window components to the walls. Ensure that they are accurately placed and aligned according to the architectural design.
5. **Add floors and ceilings:** Create floors and ceilings for each level of the building using the "Extrude" command or by drawing closed polylines and extruding them to the desired height.
6. **Include additional architectural elements:** Add other architectural features such as stairs, columns, beams, and roofs using AutoCAD's drawing and editing tools. Refer to your reference materials and design specifications for accuracy.
7. **Apply materials and textures:** Assign appropriate materials and textures to different components of the building using AutoCAD's material editor. You can define materials with specific properties like color, reflectivity, and transparency.
8. **Incorporate details:** Add finer details to the model, such as interior elements, furniture, lighting fixtures, and textures. You can either create these details within AutoCAD or import pre-made 3D models from external sources.
9. **Fine-tune the model:** Review the 3D model and make any necessary adjustments or revisions to ensure accuracy and alignment with the intended design. Adjust dimensions, proportions, and details as needed.
10. **Render the model:** If desired, use AutoCAD's rendering capabilities or export the model to a rendering software to generate high-quality images or animations of the 3D model. Experiment with lighting, materials, and camera angles to achieve the desired visual output.
11. **Review and iterate:** Carefully review the final 3D model and make any necessary revisions or modifications based on feedback or design changes. Iterate as needed to achieve the desired result.

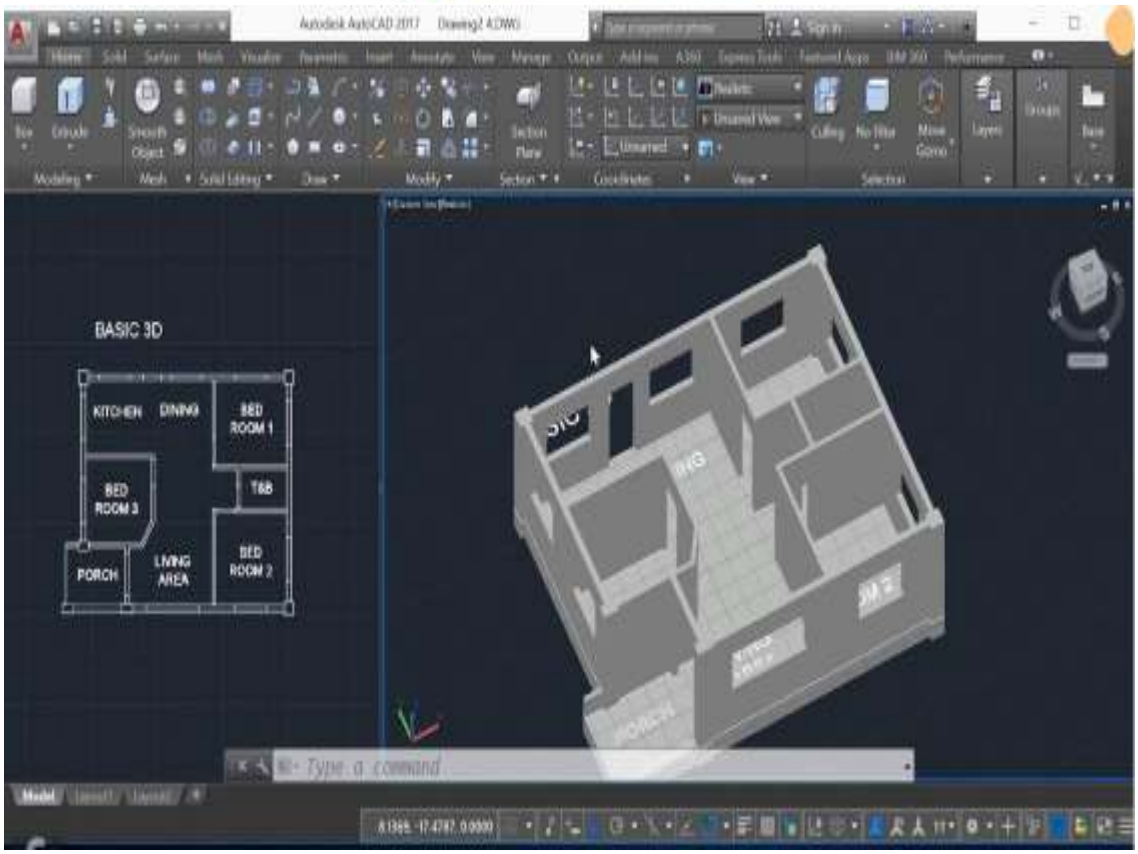


Figure:3.17. 2D plan to 3D modeling

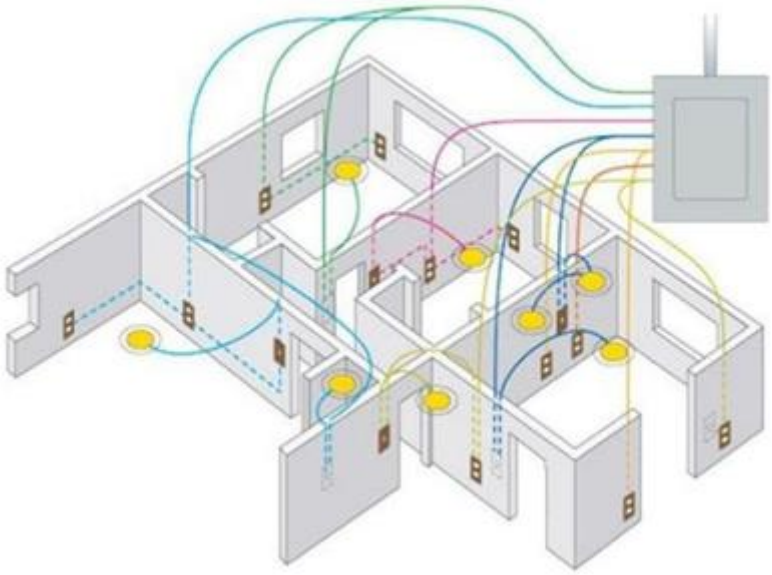


Figure:3.18. 3D modeling with Electrical wiring

Self-check 3

Multiple-Choice Questions:

1. Which of the following is NOT a common method for creating 3D objects in AutoCAD?
   A. Extruding 2D shapes
   B. Revolving 2D shapes
   C. Lofting between 2D profiles
   D. Scaling 2D shapes
2. Which AutoCAD command is used to modify the shape of a 3D object by removing or adding material?
   A. Subtract
   B. Extrude
   C. Fillet
   D. Chamfer
3. Which of the following is NOT a common type of 3D object modification in AutoCAD?
   A. Scaling
   B. Rotating
   C. Stretching
   D. Flattening
4. In AutoCAD, what is the purpose of materials?
   A. To control the appearance of 3D objects
   B. To create complex 3D shapes
   C. To define the lighting in a scene
   D. To simulate physical properties, such as weight and density
5. Which AutoCAD feature is used to apply realistic surface characteristics, such as color, texture, and reflectivity, to 3D objects?
   A. Material Editor
   B. Texture Mapping
   C. Lighting Manager
   D. Rendering Engine

Short Answer Questions:

1. What are the key principles of 3D modeling in AutoCAD?
2. Explain the process of applying materials to 3D objects in AutoCAD.
3. How can you modify the shape of a 3D object in AutoCAD?

Operation sheet 1: produce 3D modeling

- **Operation title:** Procedures of 3d modeling
- **Purpose:** To practice and demonstrate the skill, knowledge and attitude required to produce 3d modeling.
- **Instruction:** Use the given tools and equipment produce 3d modeling including the building electrical installation field. For this operation you have given 2 Hour each drawing and you are expected to provide the answer on the given table. The given drawing below drawing Procedures in doing the task
- **Tools and requirement:**
  - ✓ Auto CAD software
  - ✓ This operation sheets
  - ✓ Printer
  - ✓ Computer
- **Precautions:** Based on the given 2d Multiview drawing produce 3d modeling. when produce 3d modeling the drawing clear and understandable. Use auto cad software. These are general procedures for creating a 2D drawing in AutoCAD. The specific commands and techniques may vary depending on the version of AutoCAD you are using.
- **Procedures in doing the task**

  Step1: Launch AutoCAD: Open AutoCAD software on your computer.

  Step2: Set up the drawing environment

  Step3: Start a new drawing

  Step 4: Set the drawing limits

  Step 5: Draw basic geometry

  Step 6: Modify and edit objects

  Step 7: Apply layers

  Step 8: Add dimensions and annotations

  Step 9: Apply hatching or shading

  Step 10: Create blocks and symbols

  Step 11: Save your drawing

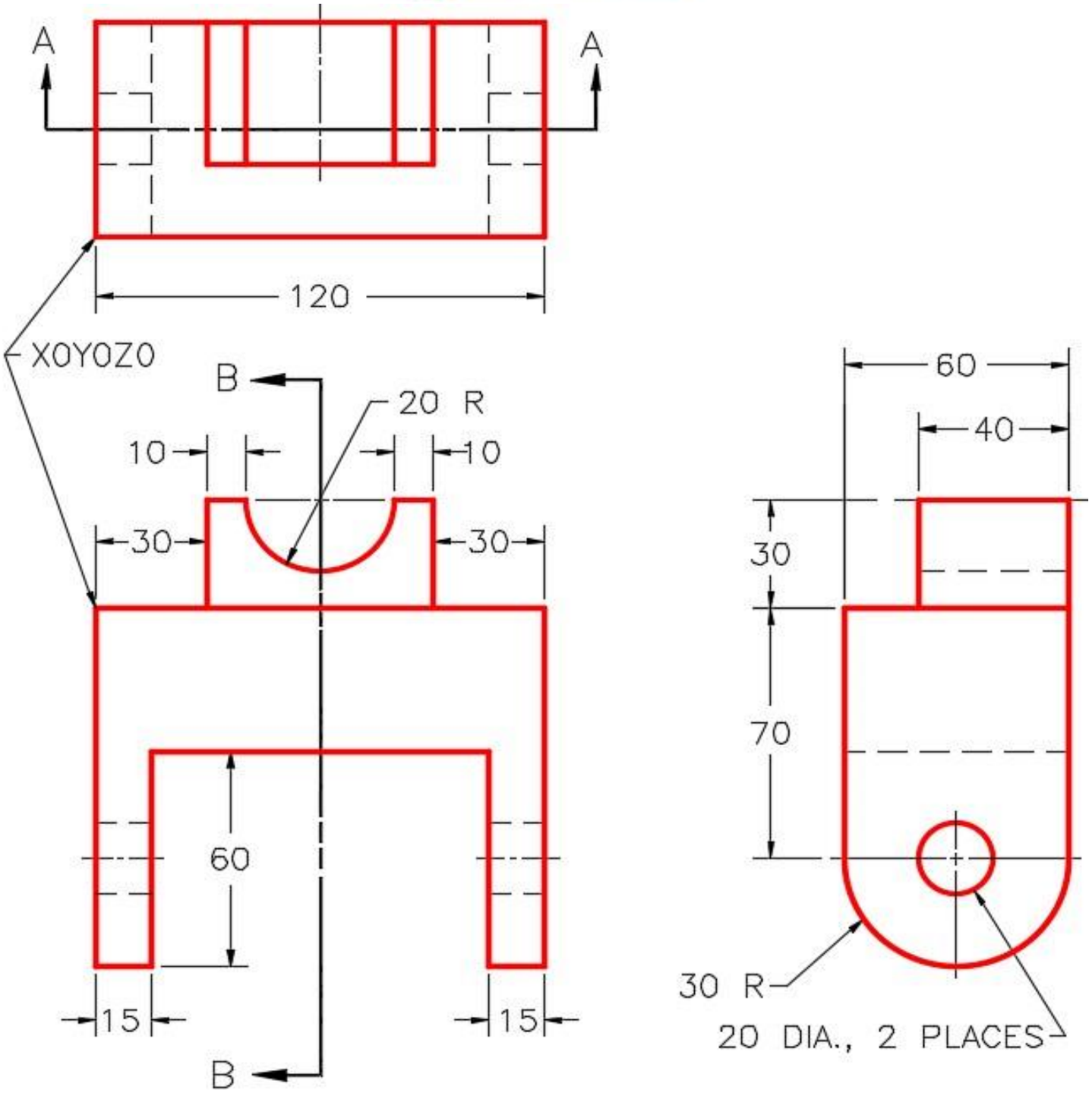


Figure:3.18. 2D drawing

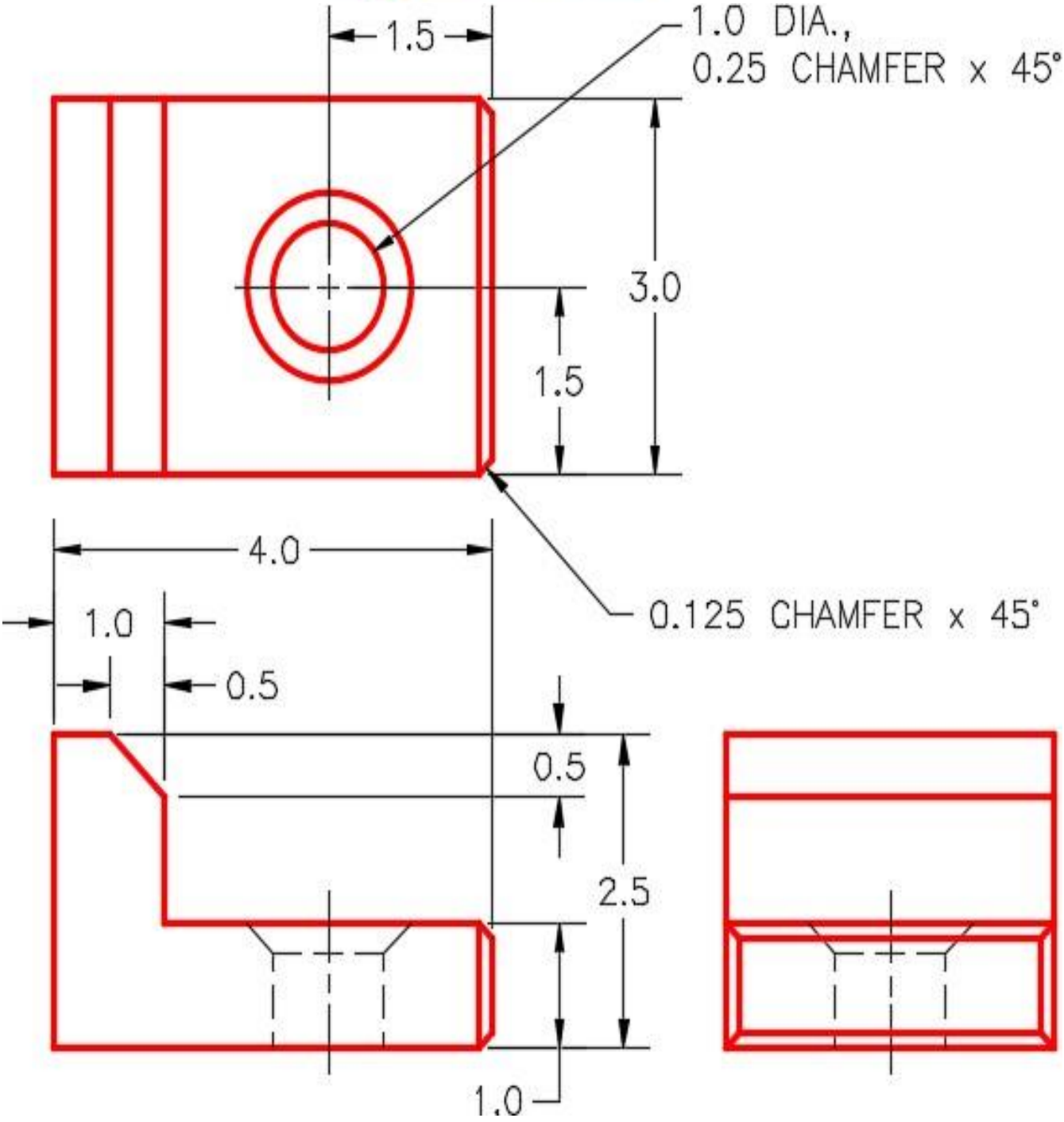


Figure:3.19. 2D drawing

Operation sheet 2: Produce 3d building modeling in auto CAD

- **Operation title:** Produce of 3d building drawing
- **Purpose:** To practice and demonstrate the skill, knowledge and attitude required to produce 3d building drawing.
- **Instruction:** Based on the given floor plan produce 3d modeling. Use the given auto cad software to produce 3d building drawing including the building electrical installation field. For this operation you have given 2:30Hour each drawing and you are expected to provide the answer on the given table.
- **Tools and requirement:**
  - ✓ Computer
  - ✓ Auto cad software
  - ✓ Printer
  - ✓ This operation sheets
- **Precautions:** When produce 3d modeling, the drawing clear and understandable. These procedures provide a general guideline for creating 3d building drawings in AutoCAD. However, it's important to note that the specific commands and techniques may vary depending on the version of AutoCAD you are using.
- **Procedures in doing the task**

  Step 1: Gather project requirements

  Step 2: open AutoCAD software

  Step 3: Set up the workspace

  Step 4: Create the floor plan

  Step 5: Extrude the floor plan

  Step 6: Add details

  Step 7: Apply textures and materials

  Step 8: Fine-tune the model

  Step 9: Set up lighting and cameras

  Step 10: Set up lighting and cameras

  Step 11: Render the final image

  Step 12: Review and revise

  Step 13: one bellow floor plan put electrical symbols & connect the wiring systems

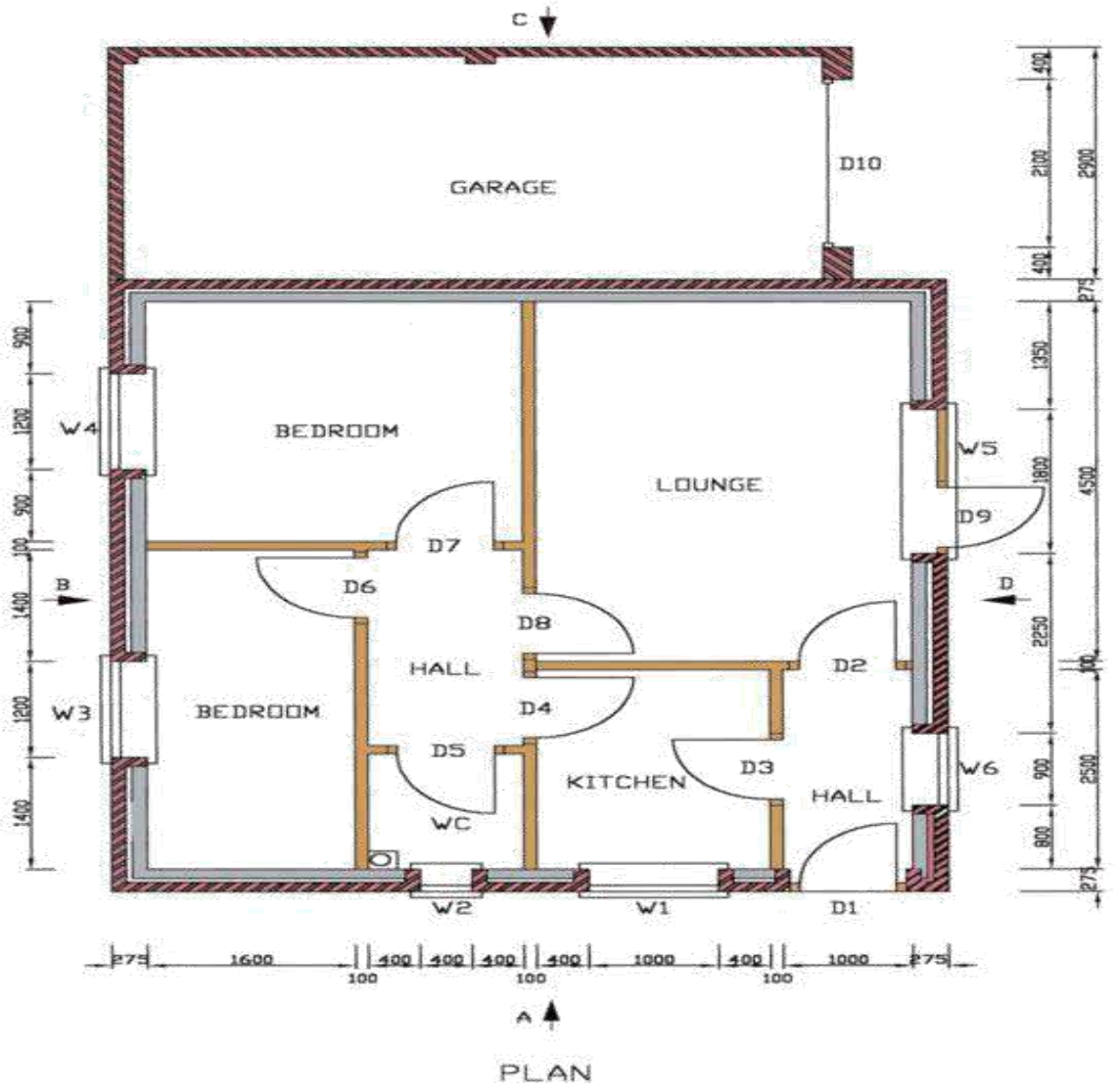


Figure:3.19. floor plan

LAP Test: Practical Demonstration

Name: _____ Date: _____

Time started: _____ Time finished: _____

Instructions: Given necessary templates, tools and materials you are required to perform the following tasks within 5 hours each tasks.

Task 1: Produce 3d modeling

Task 2: Produce 3d building modeling with electrical plans

Unit Four: Layout and Printing

This unit is developed to provide you the necessary information regarding the following content coverage and topics:

- Saving drawing files
- Creating and managing layout tabs
- Setting up viewports and paper space
- Plotting and publishing drawings

This unit will also assist you to attain the learning outcomes stated in the cover page. Specifically, upon completion of this learning guide, you will be able to:

- Saving drawing files
- Create and manage layout tabs
- Setting up viewports and paper space
- Plot and publish drawings

4.1. Saving Drawing Files

When working in AutoCAD, it is crucial to save your drawing files regularly to ensure that your work is not lost. To save a drawing file, follow these steps:

1. Click on the "Save" icon on the Quick Access toolbar or go to the "Application" menu and select "Save" or use the shortcut Ctrl + S.
2. In the "Save Drawing As" dialog box, specify the location where you want to save the file.
3. Enter a name for the file in the "File name" field.
4. Choose the desired file format from the "Save as type" drop-down menu. AutoCAD typically uses the DWG file format.
5. Click the "Save" button to save the file.

4.2. Creating and Managing Layout Tabs

Layout tabs in AutoCAD allow you to create and manage multiple views of your drawing on separate pages. How you can create and manage layout tabs:

1. To create a new layout tab, right-click on any existing layout tab at the bottom of the drawing area and select "New Layout" from the context menu. Alternatively, you can use the "Layout" command and choose the "New" option.
2. In the "Create Layout" dialog box, enter a name for the new layout and specify its size and orientation.

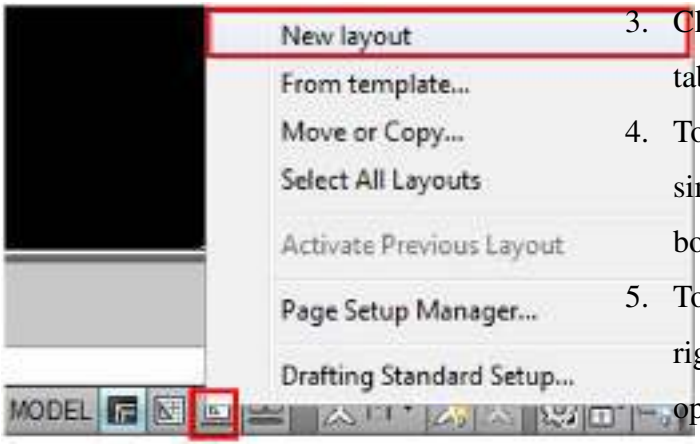


Figure:3.20. Layout

3. Click "OK" to create the new layout tab.
4. To switch between layout tabs, simply click on the desired tab at the bottom of the drawing area.
5. To manage layout tabs, you can right-click on a layout tab and use options such as "Rename," "Delete," "Copy," or "Move" to rearrange or modify the layout tabs.

4.3. Setting up Viewports and Paper Space

A viewport is a "window" created in a Paper space layout that displays the information generated in the Model space to a specified scale. A layout can have multiple viewports of different shapes. Viewports can be created from the Ribbon **View** tab > **Viewports** panel.

Figure:4.1. Viewport

Command-line: To start the **Viewport** tool from the command line, type "**MV**" and press [**Enter**].

A. Creating Viewports

Expand the **Rectangular** drop-down to see the various viewport options.

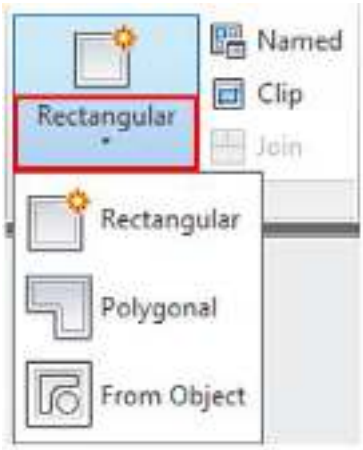


Figure:4.2. Creating Viewports

Rectangular: Creates a rectangular viewport by specifying the two opposite corners.

Figure:4.3. Rectangular viewport

Polygonal: Creates a polygonal viewport with multiple lines and arc segments.

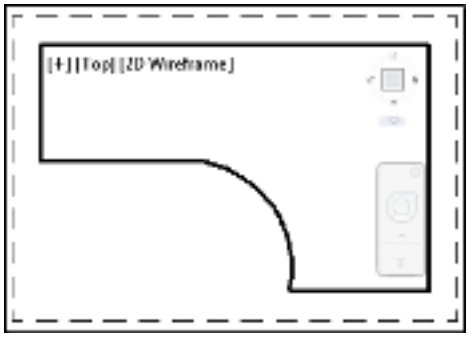


Figure:4.4. Polygonal viewport

From Object: Creates a viewport from an existing closed polyline, ellipse, spline, region, or circle.

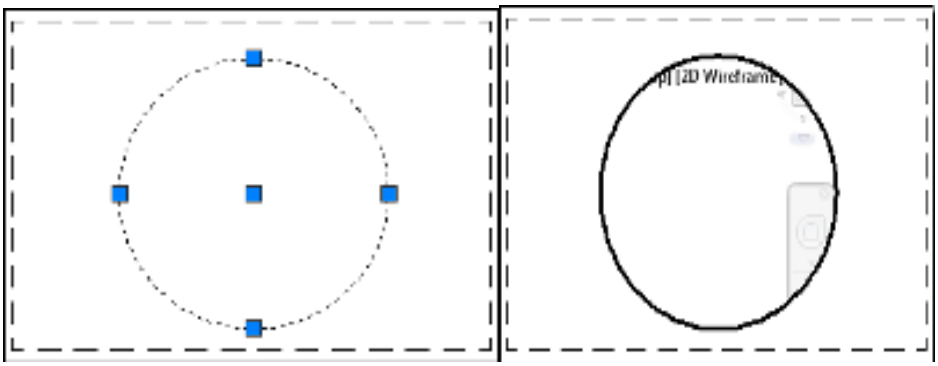


Figure:4.5. Creates a viewport from existing object

B. Activating Viewports

A viewport can be 'activated' by double-clicking inside the viewport. This allows the view within the viewport to be adjusted. Objects in Model space can be selected and modified. Zooming in or out in an active viewport will adjust the scale of the viewport. It is possible to tell if a viewport is active by the viewport border. A thick border indicates it has been activated. You will also see the View Cube and Navigation Bar inside the viewport.

Inactive Active

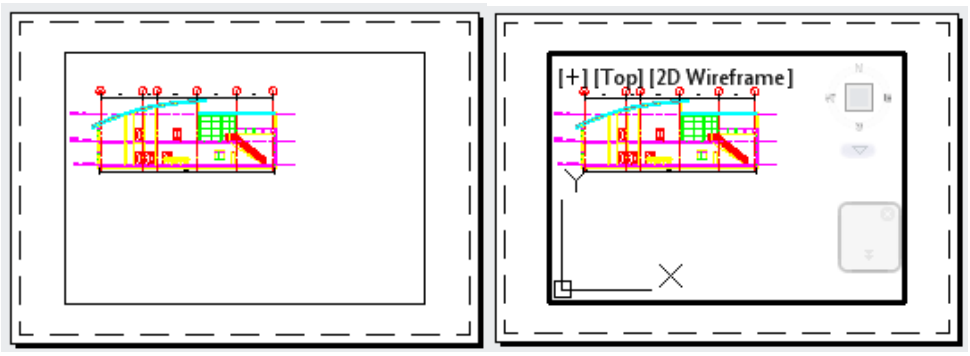


Figure:4.6. Activating Viewports

C. Viewport Scale

Objects in Model space is typically created full-size. The viewport scale controls the scale at which these objects are displayed in the viewport. For example, a 1:100 plan would be drawn (or referenced) at full size in Model space, but be displayed in a Paper space viewport with a viewport scale of 1:100. The scale of the viewport can be modified from the **Viewport Scale** drop-down of the **Application Status Bar**.

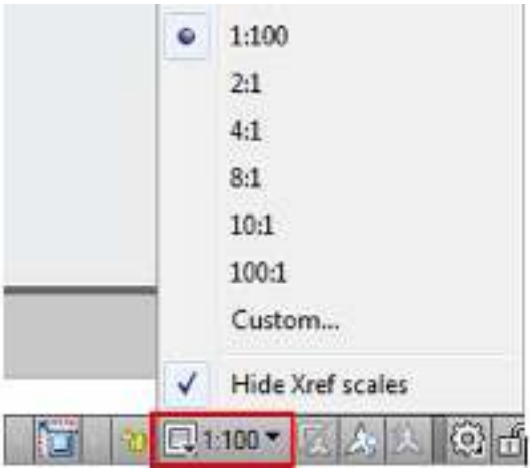


Figure:4.7. Viewports scale

To change the scale, first, click on the viewport to select it.

Change the **Viewport Scale** from the **Application Status Bar**. The viewport will zoom in or out to the chosen scale.

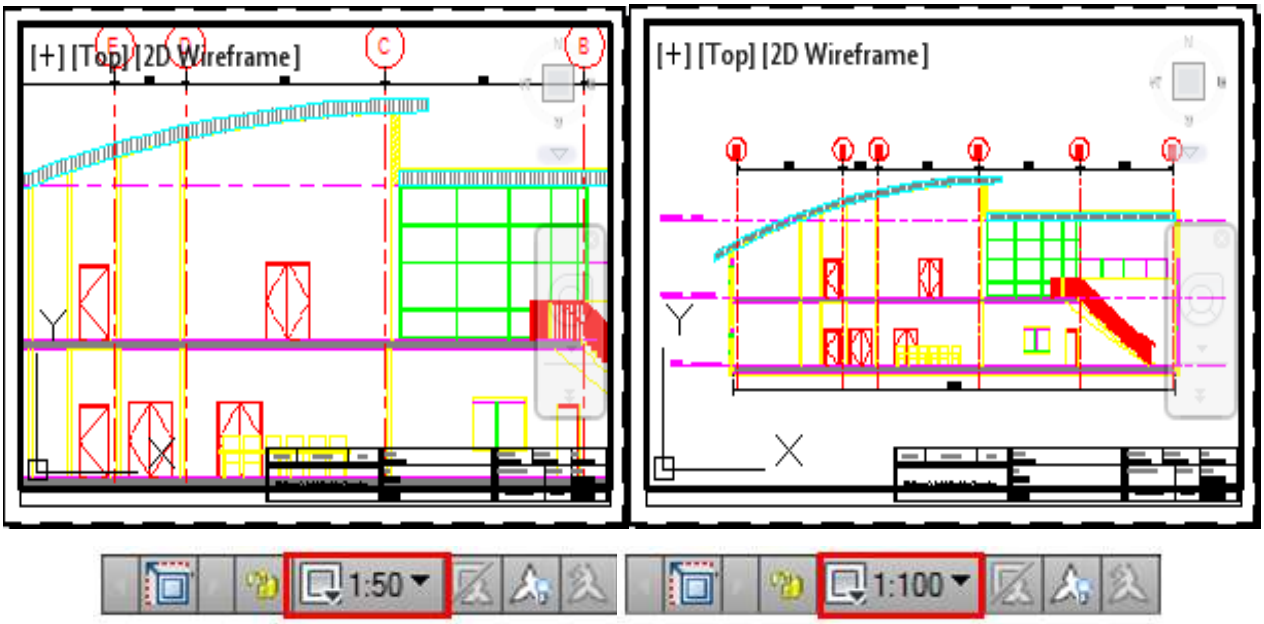


Figure:4.8. Viewports scale (Application Status Bar)

A. Locking Viewports

Once a viewport has been configured you may wish to lock it to prevent accidental changes to the view or scale. To lock a viewport, click on the viewport to select it. Click the **Lock Viewport** button on the **Application Status Bar**. It is still possible to activate the viewport but zooming in or out will zoom in Paper space, not in the viewport. Once locked, the **Viewport Scale** is no longer adjustable.

Figure:4.9. locking viewports

B. Working In Model Space from Paper Space

It is possible to make adjustments to content in Model space without switching back to the Model tab. Activate a viewport by double-clicking inside it. Click the **Maximize Viewport** button on the **Application Status Bar** to expand the activated viewport to the full screen.

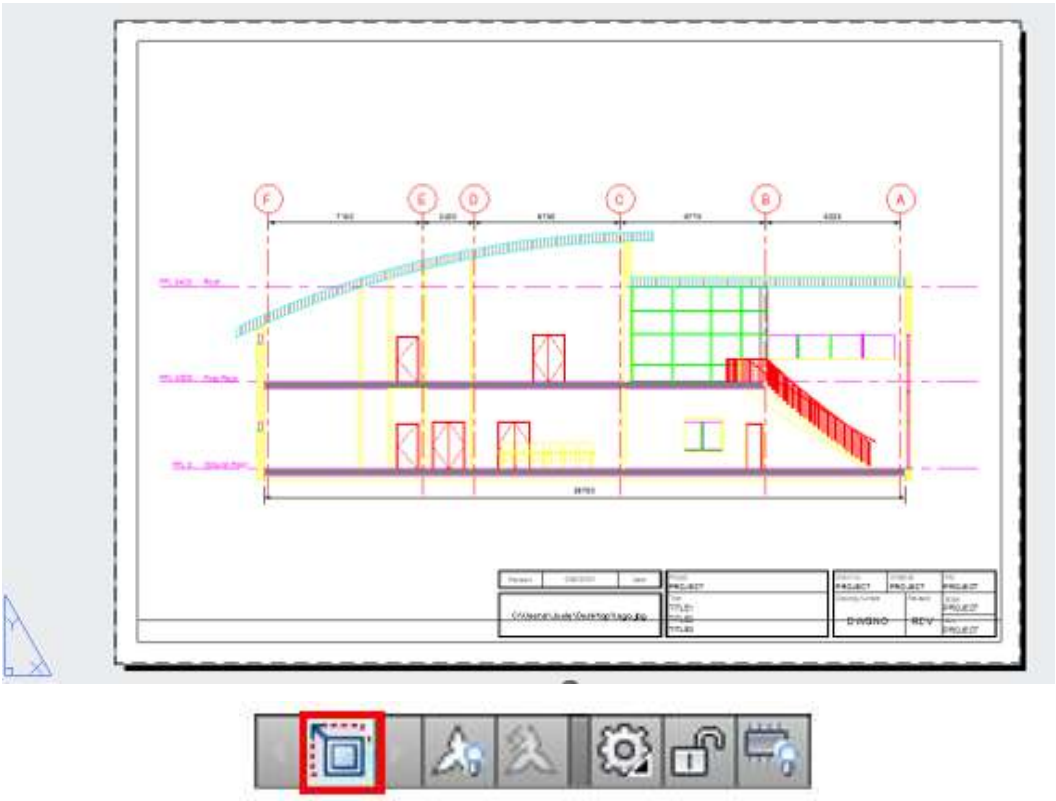


Figure:4.10. Maximize viewports

The viewport will be enlarged with a red border to indicate it is maximized. Maximizing the viewport and working on the Model space in this way allows you to zoom in and make changes without affecting the scale or position of the view within the viewport.

Figure:4.11. Enlarge viewports

Once finished, click the **Minimize Viewport** button on the **Application Status Bar** to return to Paper space with the scale and position of the view in the viewport unaffected.

C. Controlling Layers Within Individual Viewports

Layers can be turned on and off and displayed differently in individual viewports. Once a viewport is activated, from the Ribbon **Home** tab > **Layers** panel click the **Layer Properties** button. In addition to the usual properties.

Following properties are available and control the individual viewports appearance.

- **VP Freeze** controls the appearance of a Layer in the activated viewport.
- **VP Color** allows you to override the printed color of elements drawn on this layer for the activated viewport. This can be especially important if using CTBs in order to get elements to print different values between viewports.
- **VP Line type** overrides the plotted Line type of elements drawn on this layer for the activated viewport.
- **VP Line weight** overrides the plotted Line weight of elements drawn on this layer for the activated viewport.
- **VP Transparency** overrides the plotted transparency of elements drawn on this Layer for the activated viewport.
- **VP Plot Style** overrides the style applied to the layer in the activated viewport.

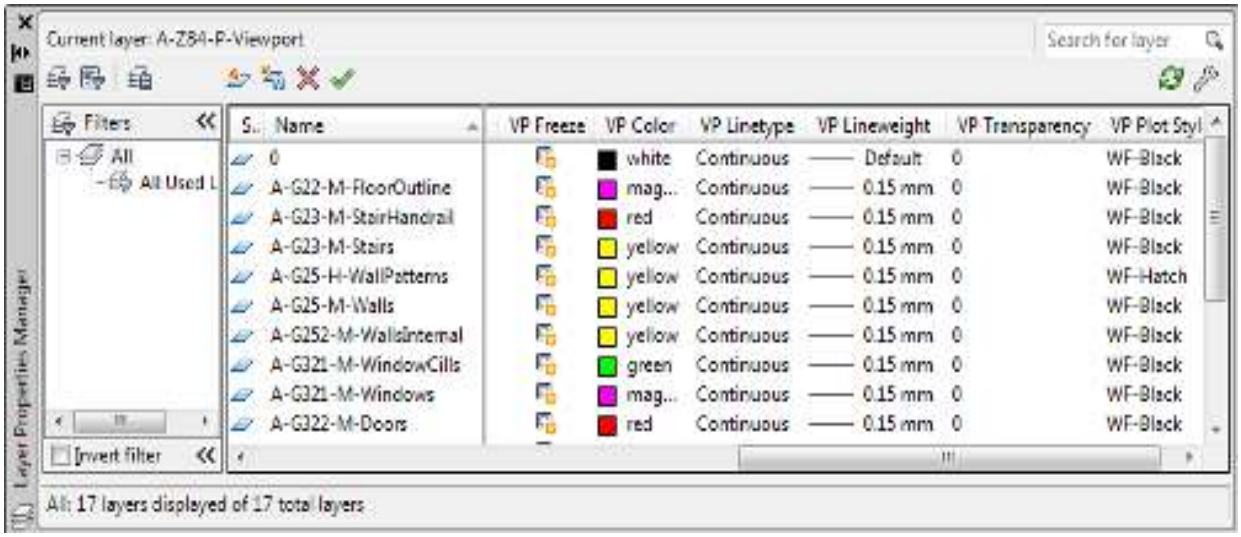


Figure:4.12. Controlling Layers

4.4. Plotting and Publishing Drawings

Plotting and publishing drawings are two different ways of outputting the drawings. Plotting is the process of creating a physical copy of your drawing on a plotter or printer. When you plot a drawing, you can specify the paper size, scale, and other settings.

Publishing is the process of creating an electronic copy of your drawing in a format that can be viewed or shared with others. When you publish a drawing, you can specify the file format, quality, and other settings.

Table 4.1. The key differences between plotting and publishing drawings:

Feature	Plotting	Publishing
Output type	Physical copy	Electronic copy
Common uses	Creating blueprints, construction drawings, and other documents that need to be printed	Sharing drawings with others, creating PDF files, and publishing drawings to websites
File formats	Not applicable	PDF, DWF, JPEG, PNG, and other image formats

How to plot a drawing

To plot a drawing, follow these steps:

1. Open your drawing file in your drawing software.
2. Go to the File menu and select Plot.
3. In the plot dialog box, select the plotter or printer that you want to use.

4. Specify the paper size, scale, and other settings.
5. Click Plot to send the drawing to the plotter or printer.

How to publish a drawing

To publish a drawing, follow these steps:

1. Open your drawing file in your drawing software.
2. Go to the File menu and select Export.
3. In the export dialog box, select the file format that you want to save the drawing as.
4. Specify the name and location of the exported file.
5. Click Export to save the file.

Self-check 4

I. Multiple-Choice Questions

1. Which AutoCAD workspace is commonly used for layout and printing tasks?
   a. Model
   b. Drafting & Annotation
   c. 3D Basics
   d. Layout
2. What is the purpose of viewports in AutoCAD layout?
   a. To display multiple drawings simultaneously
   b. To control the display of layers in a drawing
   c. To create 3D objects
   d. To manage printing settings
3. Which command is used to create a new layout in AutoCAD?
   a. LAYOUT
   b. NEWLAYOUT
   c. CREATETAB
   d. CREATENEWLAYOUT
4. In AutoCAD, which file format is commonly used for printing and sharing drawings?
   a. DWG
   b. DWF
   c. PDF
   d. DXF
5. Which AutoCAD feature is used to control the scale of objects and text in a layout viewport?
   a. Zoom
   b. Pan
   c. Plot Style
   d. Annotation Scale

II. Short Answer Questions

1. What is the purpose of paper space in AutoCAD layout?
2. Explain the difference between model space and paper space in AutoCAD.
3. What are some common plot settings you can configure when printing a drawing in AutoCAD?

Reference

- AutoCAD 2022 and AutoCAD LT 2022 Essentials by Scott Onstott
- Mastering AutoCAD by George Omura and Brian C. Benton
- AutoCAD 2022: A Problem-Solving Approach by Sham Tickoo
- AutoCAD 2022 and AutoCAD LT 2022 Bible by Ellen Finkelstein and Lee Ambrosius
- AutoCAD 2022: A Power Guide for Beginners and Intermediate Users by CAD Artifex
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- AutoCAD 2022: Architectural Design by Munir Hamad
- Residential Design Using AutoCAD 2022 by Daniel John Stine

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