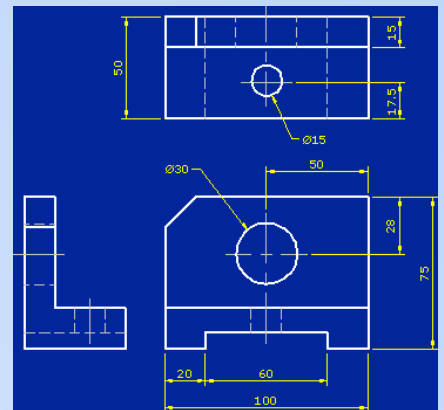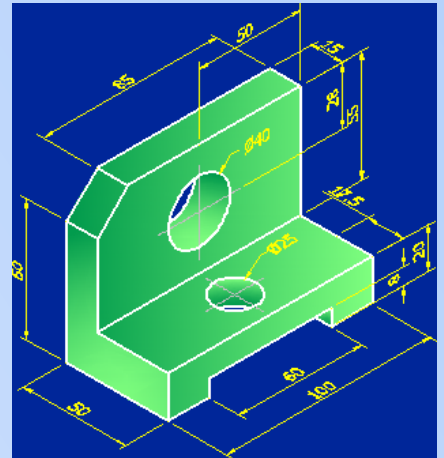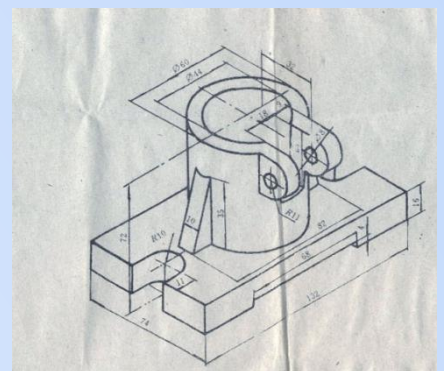# MEM30001A

Use computer aided drafting systems to produce basic engineering drawings.

Table of Contents

Topic 1 – AutoCAD Basics:

Required Skills:
On completion of the session, the participants will be able to:

- Access the AutoCAD program.
- Create a New drawing then Save the drawing to a folder.
- Draw straight lines by using Toolbar icons, Keyboard commands and the Pull Down Menu.
- Erase unwanted lines or lines drawn incorrectly.
- Retrieve the last erased entity.
- Use the Endpoint and Intersection Object snap options for drawing lines from geometric points.
- Print/plot a simple drawing.
- Exit the software program and closedown the computer

Required Knowledge:
- Identify areas of the AutoCAD screen.
- Name of the Absolute, Relative and Polar co-ordinate methods used in producing lines and entities.

General Introduction:
Prior to the introduction of computers to the workplace, nearly every drawing in the world was produced with pencil or ink on paper, linen or film.   Minor changes meant erasing and redrawing while major changes often meant recreating the drawing from the beginning.   If a change to one drawing affected other documents a draftsperson had to manually recognize the need to make the changes to the other drawings then carryout those modifications, CAD has fundamentally changed the way design is done.   Computer Aided Drafting is a tool used to produce plans used by the various manufacturing and construction disciplines, but modern programs allow for the modification on one drawing to automatically update related drawings.

CAD Drawing Types:
Computer Aided Design (or Drafting), also known as CAD, is a design tool that employs computers to create drawings and models of products while they are in the process of being created. CAD was first created in the early 1960s and today is used to design almost every product on the market in the world.

2D CAD:
Two-dimensional, or 2D, CAD is used to create flat drawings of products and structures. Objects created in 2D CAD are made up of lines, circles, ovals, slots and curves. 2D CAD programs usually include a library of geometric images; the ability to create Bezier curves, splines and polylines; the ability to define hatching patterns; and the ability to provide a bill of materials generation. Among the most popular 2D CAD programs are AutoCAD, CADkey, CADDS 5, CATIA v4 and Medusa.

3D CAD:
Three-dimensional (3D) CAD programs come in a wide variety of types, intended for different applications and levels of detail. Overall, 3D CAD programs create a realistic model of what the design object will look like, allowing designers to solve potential problems earlier and with lower production costs. Some 3D CAD programs include Autodesk Inventor, CoCreate Solid Designer, Pro/Engineer SolidEdge, SolidWorks, Unigraphics NX and VX CAD.

3D Wireframe and Surface Modeling:
CAD programs that feature 3D wireframe and surface modeling create a skeleton-like inner structure of the object being modeled. A surface is added on later. These types of CAD models are difficult to translate into other software and are therefore rarely used anymore.

Solid Modeling:
Solid modeling in general is useful because the program is often able to calculate the dimensions of the object it is creating. Many sub-types of this exist. Constructive Solid Geometry (CSG) CAD uses the same basic logic as 2D CAD, that is, it uses prepared solid geometric objects to create an object. However, these types of CAD software often cannot be adjusted once they are created. Boundary Representation (Brep) solid modeling takes CSG images and links them together. Hybrid systems mix CSG and Brep to achieve desired designs.

Raster Images:
A raster image is a representation, consisting of rows and columns of dots that form a graphics image in the memory of the computer. The value of each dot (whether it is filled in or not) is stored in one or more bits of data.   For simple monochrome images, one bit is sufficient to represent each dot, but for colours and shades of grey, each dot requires more than one bit of data. The more bits used to represent a dot, the more colours and shades of grey that can be represented. Raster images are saved with .BMP, .JPG, .PNG, .GIF or similar file extensions.

Vector Graphics:
Vector graphics use geometrical formulas to represent images by specifying a point using a distance from a datum point along horizontal and vertical axes.   Vector oriented images are more flexible than raster images because they can be resized and stretched. The various CAD software programs save their drawing files with specific file extensions.

- AutoCAD .DWG
- TurboCAD .TCW
- Solid Edge .ASM
- Microstation .DGN

Accessing the AutoCAD Program:

The method for accessing the program depends on the version installed and the installation set up. AutoCAD can be started by clicking on the Windows **Start** button at the bottom left corner of the screen, moving the mouse to **All Programs** then clicking on **Autodesk,** then **AutoCAD** and then **AutoCAD** (the word AutoCAD may be followed by the version or year).   The program can also be started from the Desktop by double clicking on the AutoCAD icon if displayed.

On entering the program, a clear graphics screen is displayed with the environment displayed as set by the previous operator; the environment can be customised to suit the individual user. A typical AutoCAD environment is shown in Figure 1.1 below.
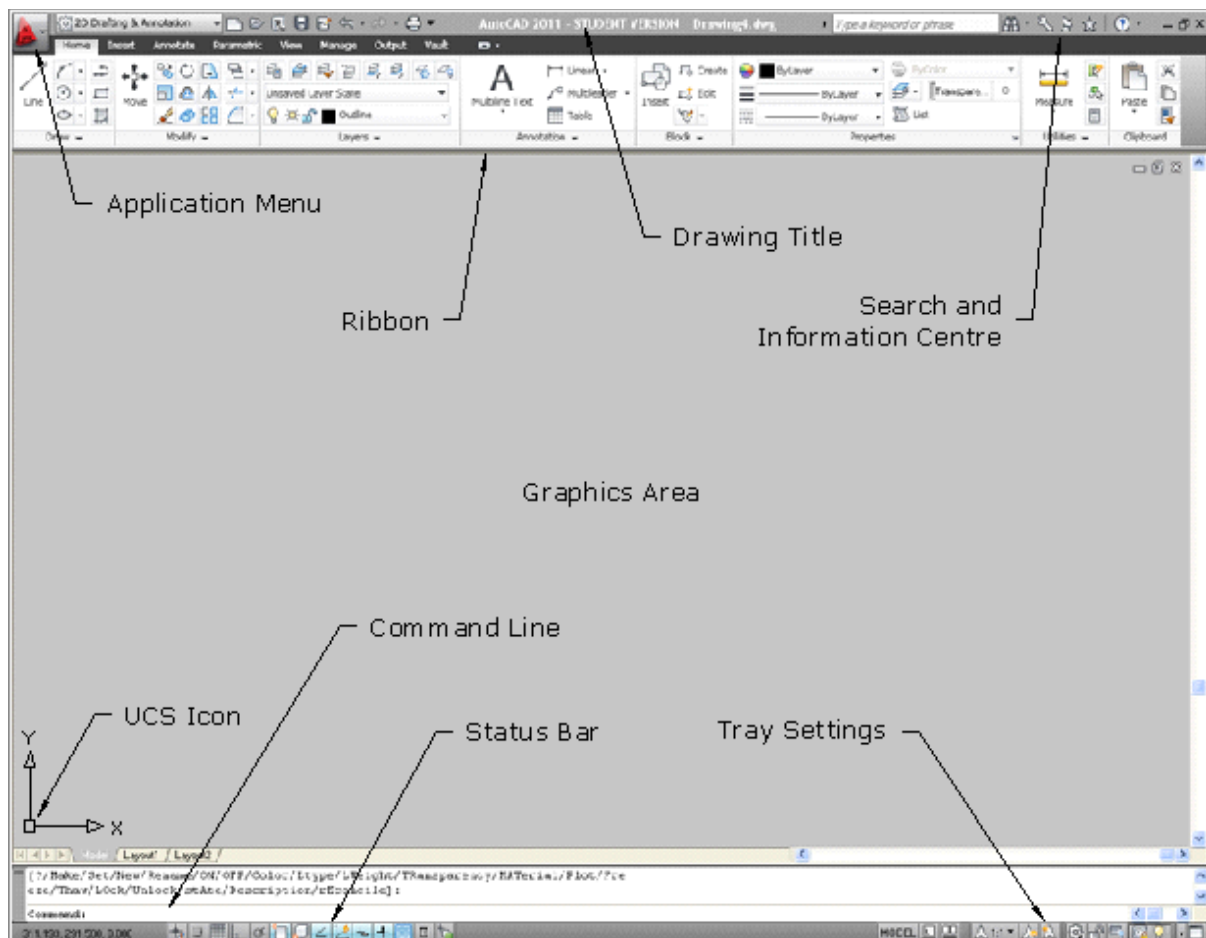
Figure 1.1

The interface can also be displayed showing the standard AutoCAD Toolbar menus that can be moved between and docked onto the horizontal and vertical edges of the graphics area as shown in Figure 1.2.   Using toolbar menus and docking them to the left and/or right vertical edges of the graphics area can provide a drawing area in proportion with the standard sheet sizes, especially on wide screen monitors.
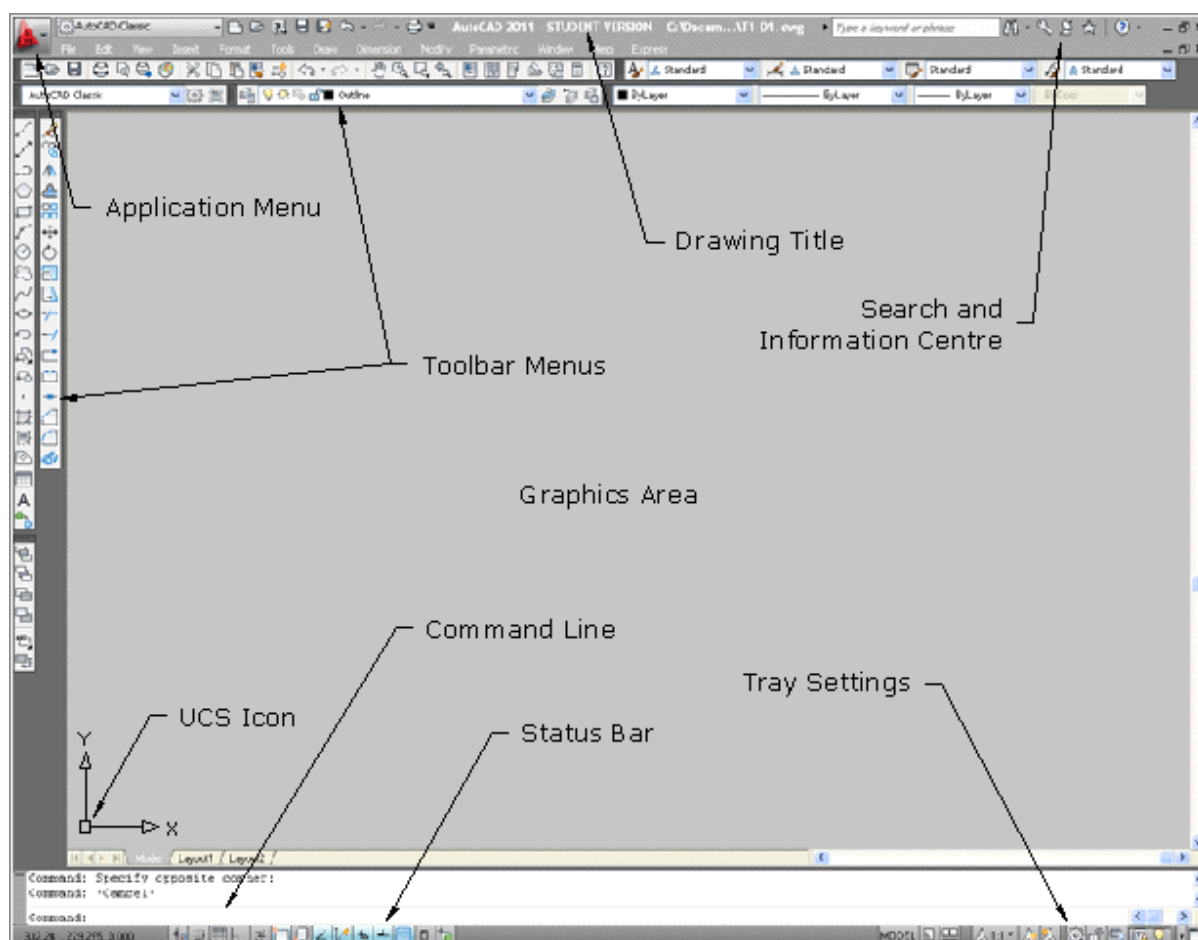
Figure 1.2

Application Menu:

The Application menuis located in the top left corner and is used to access commands and options. The Browser appears as a drop-down menu with the File, Edit, View, Insert, Format, Tools, Draw, Dimension, Modify, Express, Window and Help menus displayed. A list of the most recently opened drawings is located in the right side column.

Ribbon:

The ribbon provides a single, compact placement for operations that are relevant to the current workspace and eliminates the need to display multiple toolbars, reducing clutter in the application and maximizing the area available for work using a single compact interface. The ribbon is similar to the Dashboard in previous releases of AutoCAD and reduces the number of toolbars therefore allowing a larger area graphics area for drawing.
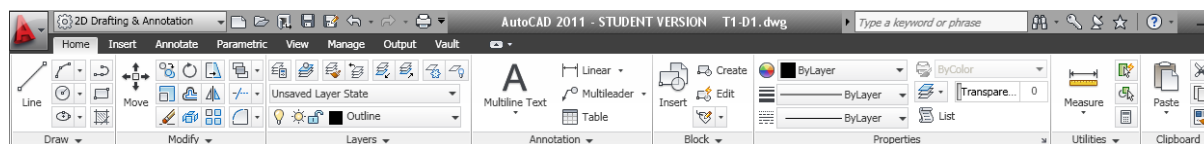
The ribbon contains a series of tabs each with separate panels; these tabs include Home, Insert, Annotate, Parametric, View, Manage, Output and Vault.the main tabs are Home, Blocks and References, Annotate Tools, View, Output and Vault. Clicking on a tab displays the panels within that tab and provides the access to the different commands.

The display of the Ribbon Panel is controlled by:

Tray Settings   - *Select option from the Context Menu*.

Application Menu   A – Tools - ▷ Palettes - Ribbon

Many Ribbon tabs can be temporarily expanded to display additional commands and icons by clicking on the triangle ◢ at the bottom-right edge of the tab name. The tab can also has on option of locked open by clicking the pin also located at the bottom-right edge of the tab name.

Toolbar Menus:

Toolbar menus contain icons that activate commands.   If the cursor is placed over an icon, the name of the icon will be displayed.   Icons displaying a small triangle in the lower right corner indicated a flyout toolbar (display additional icons).   Additional toolbars can be turned on and/or off using the Top Pull Down men - View – Toolbars or right clicking over one of the displayed toolbars and selecting one of the options available.   The toolbars can be fixed to the sides of the screen or allowed to "float" about the screen by holding the left button down over a part of the toolbar (not an icon) and dragging the cursor about the screen.   Commands are executed by left clicking on the required icon.

Command Line:

Commands can be entered using the keyboard and appear in the command line.   The computer uses the command line to request further information from the operator and give instructions for the completion of the command.   The use of keyboard commands in modern CAD programs is rapidly disappearing; **get used to using the Toolbar menus**. Most mistakes are made by operators **NOT** reading the Command Line or the Dynamic Input display next to the cursor – **constantly LOOK at the Command Line** or the **Dynamic Input display**.   Commands are executed by typing the name of the command then pressing the **Enter** key.   Many commands like Erase have shortcut keys where the letter **E** can be typed then the **Enter** key or **Space** bar pressed to execute the command.

Status Bar:

The status bar displays and enables the operator to change several important settings that affect the methods of drawing and editing the current drawing. All features within the Status Bar can be turned ON and OFF by clicking on the respective button.
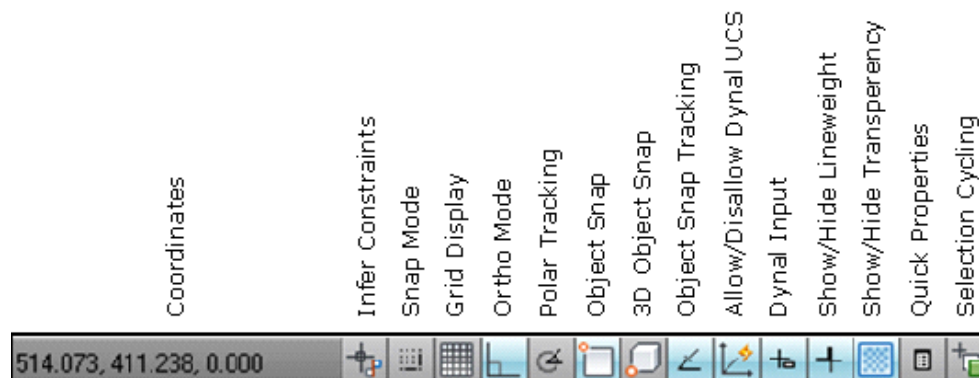
Coordinates:

The cursor coordinates readout displays the current X,Y,Z location of the cursor in the drawing area, with respect to the origin point (whose coordinates are 0,0,0). It is similar to having a GPS (Global Positioning System) device in the drawing.

If the coordinates in the lower-left corner of the screen are grayed out, then coordinate tracking is turned OFF and the coordinates will not change. The coordinates appear with dark lettering when the coordinates are turned ON and change when the cursor is moved about the drawing area.

Infer Constraints:

Constraints are used in parametric drawing to permanently position an entity of point to a corresponding point.

Snap, Grid, and Ortho:

The Snap, Grid and Ortho buttons control three of AutoCAD's tools for ensuring precision drawing and editing. **Snap** constrains the cursor to regularly spaced hot spots, enabling you to draw objects a fixed distance apart more easily. **Grid** displays a series of regularly spaced dots, which serve as a distance reference. **Ortho** constrains the cursor to horizontal or vertical relative movement.

Polar Tracking:

Polar tracking causes the cursor to prefer certain angles when you draw and edit objects. By default, the preferred angles are multiples of 90 degrees, but you can specify other angle increments, such as 45 or 30 degrees. Clicking the POLAR button toggles polar tracking on or off. Ortho and polar tracking are mutually exclusive — turning on one mode disables the other.

Object Snap and 3D Object Snap:

Object Snap is a tool used for ensuring accurate drawing and editing by locking onto the precise points of existing entities — for example, the endpoint of a line or the center of a circle.

Object Snap can be used in a *Running* mode where the individual Object Snaps are constantly turned on or in a *Temporary* mode by selecting the required Object Snap from the context menu; the temporary snap works for the selection of the one point only.

Object Snap Tracking:

Object Snap Tracking hunts in a more sophisticated way for points that are derived from object snap points.

Allow/Disallow Dynamic UCS:

Allow/Disallow Dynamic UCS – Dynamic UCS (User Co-ordinate System is used in 3D modes and will be covered in later, and more advanced units.

Dynamic Input:

Dynamic Input – Turns the dynamic input ON and OFF. Dynamic Input provides a command interface near the cursor to assist in focusing in the drafting area instead of having to read from the Command Line. When Dynamic Input is on, tooltips display information near the cursor that is dynamically updated as the cursor moves. When a command is active, the tooltips provide a place for user entry.
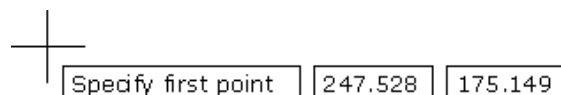
The data displayed in the tooltip area changes depending on the command and the information being input.

Show/Hide Lineweight:

Show or Hide Lineweight controls whether the display of the lineweights on the screen. The button does not control the lineweights appearing on plots which is controlled by a separate setting in the Plot dialog box.

As a drawing is zoomed in or out, the display of the lines will automatically adjust to display the thick, medium and thinner lines in proportion to the screen.

OFF

ON
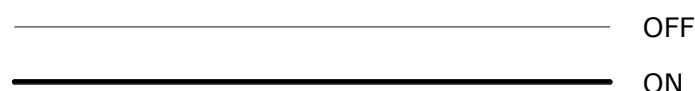
Quick Properties:

Turns the Quick Properties dialog box On and Off. Quick properties allows the operator to changes the properties (layer, colour, lineweight etc) of any entity on the current drawing.

Selection Cycling:

Selection Cycling assists when trying to select overlapping entities in a congested 2D or 3D drawing. When the Selection Cycling is turned On, a dialog box displaying all overlapping entities appears on the screen. As the cursor is passed over each object in the list, the relevant entity in the drawing is highlighted enabling the correct entity to be selected.

Tray Settings:

The Tray Settings line displays valuable information about the drawing statistics, modes and extents. The modes on the status bar can also be accessed using the appropriate Function key on the top of the keyboard.

Model or Paper Space:

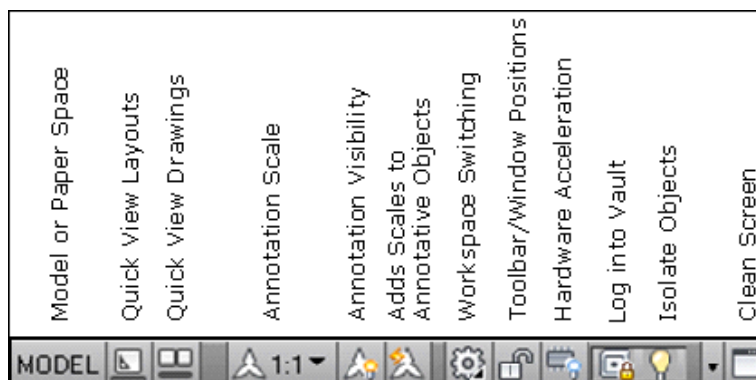The drawing area is composed of overlapping tabbed areas labeled Model, Layout1, and Layout2 by default. The Model tab displays a part of the drawing called *model space,* where most of the drawing is created. Each of the remaining tabs displays a *paper space layout,* where a plottable view with a title block is created. A completed layout will include one or more *viewports*, which reveal some or all the objects in model space at a particular scale.

Quick View Layouts:

When the Quick View Layouts button is selected, the Layouts are displayed as thumbnails. Holding the Ctrl key and scrolling the mouse wheel dynamically changes the image tile size.

Quick View Drawings:

The Quick View Drawings button displays the currently opened drawings and their Layout preview in a dynamic view thumbnail tile. As with the Quick View Layout button, the Ctrl and mouse to change the size of the thumbnails.

Annotation Scale:

Annotation scale is a setting that is saved with model space, layout viewports, and model views. When **annotative** objects are added to the drawing, they support the current annotation scale and are scaled based on that scale setting and automatically displayed at the correct size in model space.

Annotation Visibility:

All annotative objects, or only those that support the current annotation scale can be displayed in model space or a layout viewport.

Adds Scale to Annotative Objects:

The Add Scale to Annotative Objects button

When an **annotative** object is created, it supports one **annotation scale**; the annotation scale that was current when the object was created. The annotative objects can be updated to support additional annotation scales.

Workspace Switching:

Using the Workspace Switching button allows the switching between the Ribbon, Toolbar, 3D Modelling, or other available display modes shown in the Pop-up menu.

Toolbar/Windows Positions:

The Toolbar/Windows button is used to lock the layout viewports so the scale canot be accidentally adjusted.

Log into Vault:

Autodesk Vault is a data management tool and helps design teams to track work in progress and maintain version control in multi-user environments allowing members to organize and reuse designs by consolidating product information and reducing the need to re-create designs from scratch. Access is available clicking the button to ON.

Isolate Objects:

The Isolate Objects button allows the user to control the display of objects by isolating or hiding a selection set and has nothing to do with layers. Objects to be isolated or hidden can be selected individually or using a window.

Clear Screen:

Selecting the Clean Screen button provides a larger view of the graphics area by turning everything OFF on the screen except for the status bar, menu bar, and the command line. Invoking the Clean Screen command again restores the screen to the previous settings.

Navigation Bar:

The Navigation bar assists in navigating around the drawing.

View Cube	*View Cube:* The View Cube is a clickable interface that provides a reference to the current orientation of the model and allows switching between standard and isometric views.
Navigation Wheel	
Pan	*Navigation Wheel:* Provides access to the Full Navigation Wheel used when operating in the 3D mode and a smaller 2D version used in the 2D mode.
Zoom	
Orbit	*Pan:* Panning allows the operator to quickly move around the drawing area at the same magnification.
Show Motion	

Zoom:

Provides access to a range of zooming tools through the drop-down menu.

Orbit:

Allows the 3D model to be viewed at any rotated angle or position.

Navigation Tools:

Navigation tools change the orientation and view of the model and will be covered separately in future lessons.

Show Motion:

Creates animated presentations.

Basic Commands:

Commands, common to all Windows programs are also found in AutoCAD on the Quick Access Menu and the Menu Browser; they include Open, New, Save, Print, View and Spelling and can be accessed by left clicking once on the applicable icon.

Open		Opens an existing AutoCAD drawing with an extension of *.dwg*.
New		Creates a new AutoCAD. New documents are created using a template or a blank format. A template has a prepared basic structure for a drawing and contains settings such as Layers, Dimensions and Text Styles, Drawing Size and Units.
Save		Saves a new document file by requesting the path and name of the file and automatically saves a named file to the specified folder with an extension of *.dwg*.
Plot		Prints the document to a specified printer or plotter.

Tutorial Exercise TUT1-1:

Access the AutoCAD program and create a new drawing using a template file called **A4BASIC.DWT** that is stored in the CAD Drafting Template folder on the college network drive.

Procedure:

1. Enter the AutoCAD program by clicking on the AutoCAD icon 🔺 on the desktop.
2. Left click on the New drawing 🗋 button in the Quick Access Menu.
3. Expand the Look in box by clicking the triangle ▾.
4. Locate the drawing template called **A4BASIC** in your work area (or CD).
5. Click **OPEN** at the bottom of the dialog box and the new drawing is created.

Ribbon or Toolbar Menus:

The decision to use Ribbon or Toolbar Menus will be determined by the operator's experience using computer technology; many long-term users will probably continue to use the Toolbar menus while new users should probably become accustomed to using the Ribbon menu system as most other Microsoft software.

The topics in this Student Resource Book will be delivered using the Ribbon menu as the majority of users will be new while the more experienced operators will be able to continue with the use of the toolbars.

Ribbon Menu:

The AutoCAD Ribbon (including the Application menuand Quick Access Toolbar) presents a user interface similar to the Microsoft Office programs.   The ribbon provides easy access to tools through a collection of tabs and panels with each tab containing multiple panels and the panels containing multiple command buttons.   Some panels can be expanded to access additional tools by clicking the expand ▾ icon; clicking the ◢ icon allows the operator to modify the styles for the options within the panel.

Of the 8-tabs displayed in the Ribbon menu, only the Home, Insert and Annotate tabs will be explained in this Unit.

The display of the Ribbon menus can be controlled using the Minimise 🔲🔳 button next to the Vault tab.   Minimising options are Minimise to Tabs, Minimise to Panel Titles, or Minimise to Panel Buttons.

Home Tab:

The Home tab contains the most frequently used commands within the software.   Basic object and drawing tools can be found here, as well as frequently used commands like modification tools, layers, and basic annotation.   The Utilities and Clipboard panels and

some of the individual commands contained in the remaining panels will not be included in this unit. The image of the Home tab has been split into 2-separate images for clarity.
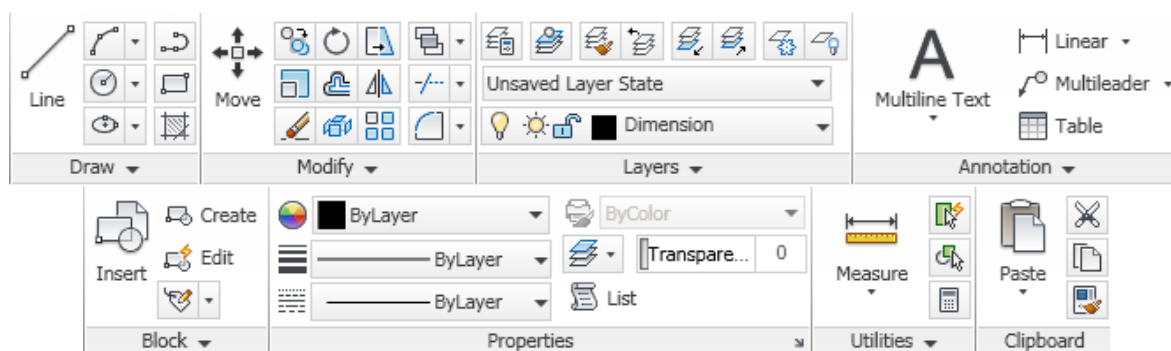

Figure 1.3 – Home Tab

Draw – Contains the main tools for creating the entities in a drawing. Commands covered in this unit include Lines, Arcs, Circles, Ellipses, Rectangles, Polylines, Polygons, Splines and Hatching.

Modify – Includes the commands for modifying the views, and entities. Commands include Erase, Trim, Extend, Copy, Move Stretch, Scale, Offset, Mirror, Fillet, Chamfer, Array (Rectangular and Polar), Break, Lengthen and Editing Hatch.

Layers – Contain the commands needed to work with various layers. Commands include Layer and Layer Properties (Visible, Freeze, Lock, Colour, Construction and Lineweight).

Annotation - Includes the commands for inserting text, leaders, and dimensions.

Block – Blocks are groups of entities (e.g. symbols and drawing sheets) that can be inserted to a drawing to reduce drawing time.

Properties – Layers are used to identify different types of lines, notes and dimensions on a drawing. The visibility and editing of various layers can be turned ON and OFF as required.

Utilities – Provides a series of commands to find the length, area, volume and other information of entities or shapes.

Clipboard – Can be used to copy and paste entities around the drawing.

Insert Tab:
The Insert tab is used to add and create various types of blocks in a drawing. During this unit only the Insert command in the Block panel will be used. The image of the Insert tab has been split into 2-separate images for clarity.
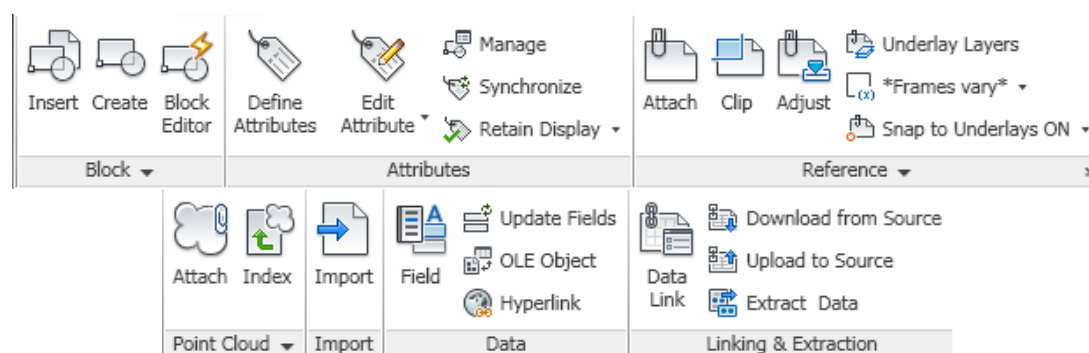

Figure 1.4 – Insert Tab

Block – Creates and inserts blocks such as symbols and drawing sheets onto the drawing.

Attributes – Inserts blocks of lines and text into a drawing.

Reference – Links a block in the drawing with a master drawing so the most current inserted drawing is displayed.

Point Cloud – Point clouds are used in 3D work and consist of a set of points created by 3D scanners.

Import – Imports files of different formats (.FBX, .WMF, .SAT, .3DS and .DGN) into the current drawing.

Data – Inserts fields and OLE objects into the drawing and creates hyperlinks.

Linking and Extraction – Links and extracts data to and from the drawing.

Annotate Tab:

The Annotate tab contains the commands used to dimension and add text to the drawing as well as commands to create multi-leaders and tables. The scale of the text and dimensions is adjusted with the annotation scales. During this unit only the Text, Dimensions and Leaders panels will be used.
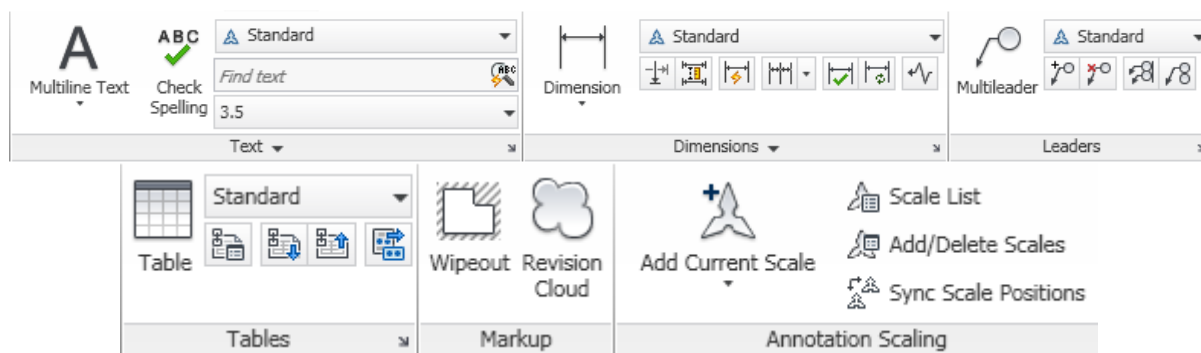
Figure 1.5 – Annotate Tab

Text – Multiple lines of text can be added to the drawing using the text command. Options within the panel allow existing text styles to be modified or new text styles created.

Dimensions – Dimensions of various styles can be applied to the drawing. Existing dimension styles can be modified and new dimension styles created.

Leaders – Places notes on the drawing and connects to the specific detail with arrows. Existing leader styles can be modified and new leaders created.

Tables – Creates and places tables on a drawing plus uploads and connects data.

Markup – Allows specified areas of the drawing to be hidden behind a white area; features that have been modified can be indicated with the Revision Cloud button.

Annotation Scaling – Modifies and applies annotation scales to entities.

Dynamic Input:

Dynamic Input provides a command interface near the cursor to assist in keeping the operator's focus in the drawing area. When Dynamic Input is on, tooltips display information near the cursor that is dynamically updated as the cursor moves. When a command is active, the tooltips provide a place for user entry. After a value is types in an input field and the Enter key pressed, the field then displays a lock icon, and the cursor is constrained by the entered value. A value can then be entered for the second input field. Alternately, if a value is typed and the Enter key pressed, the second input field is ignored and the value is interpreted as direct distance entry.

Dynamic Input is not designed to replace the command window. The command window can be hidden to increase the size of the screen area for drawing, but will need to be displayed for some operations.   The F2 key is used to hide and display command prompts and error messages as needed.   Alternately, the command window can be undocked and Auto-hide used to roll open or roll up the window.

Turn Dynamic Input On & Off:
Click the Dynamic Input button ⊟ on the status bar to turn the dynamic input on and off. The Dynamic Input can also be turned off temporarily by pressing the F12 key. Dynamic Input has three components: pointer input, dimensional input, and dynamic prompts.

The Settings to control what is displayed by each component when dynamic input is on is accessed by right-clicking on the Dynamic Input button ⊟ on the status bar.

Pointer Input:
When pointer input is on and a command is active, the location of the crosshairs is displayed as coordinates in a tooltip near the cursor. Coordinate values can be entered in the tooltip instead of on the command line. The default for second and subsequent points is relative polar coordinates (relative Cartesian for RECTANG). There is no need to type the "at" sign (@).   If absolute coordinates are to be used, the number sign (#) is used as a prefix.   For example, to move an object to the origin, for the second point prompt, enter *#0,0.*

247.528   175.149
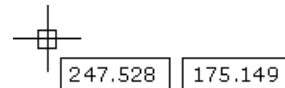
Dimensional Input:
When dimensional input is on, the tooltips display distance and angle values when a command prompts for a second point. The values in the dimensional tooltips change as the cursor is moved.   Press TAB to move to the value to be changed. Dimensional input is available for ARC, CIRCLE, ELLIPSE, LINE, and PLINE.

125.747
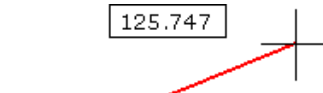
When grips are used to edit an object, the dimensional input tooltips can display the following information:
- Original length
- A length that updates as you move the grip
- The change in the length
- Angle
- The change in the angle as you move the grip
- The radius of an arc

Dynamic Prompts:
When dynamic prompts are on, prompts are displayed in a tooltip near the cursor; responses can be entered in the tooltip instead of on the command line. Press the DOWN ARROW key to view and select options.   Press the UP ARROW key to display recent input.

Specify first point
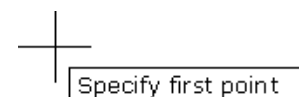
Ortho Mode:
The Ortho Mode restricts the cursor movements to horizontal and vertical for convenience and precision when creating and modifying objects. As objects are created or moved, the Ortho mode is used to restrict the cursor to the horizontal or vertical axis; as the cursor is moved, the rubber-band line follows the horizontal or vertical axis, along whichever is nearest to the cursor.   The orientation of the current user coordinate system (UCS) determines the horizontal and vertical directions. In 3D views, Ortho

mode additionally restricts the cursor to the up and down directions, in which case, the tooltip displays a +Z or -Z for the angle.

Ortho can be turned ON and OFF at any time during drawing and editing. Ortho is ignored when coordinates are entered, or an object snap specified. To turn Ortho ON and OFF temporarily, hold down the temporary override key, SHIFT. While the temporary override key is used, the direct distance entry method is not available.

The Ortho mode is turned ON and OFF by clicking the Ortho icon on the Status Bar at the bottom of the graphics area, or, pressing the F8 function key on the keyboard.

Draw a Line:

A LINE is a straight segment drawn between two points and defines the outline or shape of an object.

Ribbon Panel	Home – Draw –
Draw Toolbar	

On entering the command, the computer prompts for the first point "*_Line Specify first point:*" On picking/entering the point, the computer prompt for the next "*Specify next point or [Undo]:*" The command keeps repeating its self until no further points are picked, clicking the Right mouse button or the Enter key ends the command.

Typing the letter **C** then pressing the Enter key, or right clicking will draw a line from the last selected point to the start point of the first line in the sequence.

Tutorial Exercise TUT 1-2:
Draw a zigzag line using a series of straight lines of any length and direction.

Procedure:

1. Enter the Line command by clicking on the Line ╱ icon in the Draw Panel of the Home tab.
2. On the command prompt "*_Line Specify first point:*" move the cursor towards the left top left corner of the drawing area and click the left mouse button.
    If the line appears vertical or horizontal, click the ORTHO button in the Status bar or press the F8 key.
3. Follow the command prompt "*Specify next point or [Undo]:*" move the cursor down and to the right and again click the left mouse button.
4. Draw another line by following the command prompt "*Specify next point or [Undo]:*" move the cursor vertically down and again click the left mouse button.
5. Move the cursor horizontally to the right and again click the left mouse button.
6. Move the cursor to the left and up, then again click the left mouse button.
7. Click right on the mouse button to complete and exit the command.
8. Notice there are no further instructions waiting on the command line or Dynamic Input display.   The computer is waiting for further command instructions.

Caution must be used when using the freehand method.   Vertical and horizontal lines are drawn and then edited using a variety of Modify commands, the shape can be constructed.   To draw lines exactly vertical and horizontal, the Ortho box on the Status Line is clicked (of the **F8** function key on the keyboard can be pressed), the Ortho can be turned OFF by clicking on the Ortho box again or pressing the **F8** key).   Lines can also be drawn at exact horizontal and vertical distances by dragging the mouse in the desired direction, typing the Length on the command line, then pressing the right button or **Enter** key.

Tutorial Exercise TUT 1-3:
Draw a vertical line 50mm long and a horizontal line 100mm long.

Procedure:

1. Enter the Line command by clicking on the Line ╱ icon in the Draw Panel.
2. On the command prompt "*_Line Specify first point:*" move the cursor towards the centre of the work area and press the left mouse button.
3. Following the command prompt "*Specify next point or [Undo]:*", move the cursor up the screen and notice the line is not exactly vertical.
4. Left click on **ORTHO** 🔲 on the status bar and notice the line is now perfectly vertical.
5. Type **50** on the command line then press the **Enter** key.
6. On the command prompt "*Specify next point or [Undo]:*", move the cursor to the right and notice the line is now exactly horizontal.
7. Type **100** on the command line then press the **Enter** key.
8. On the command prompt "*Specify next point or [Undo]:*", press the right mouse button and the command is exited.

Erase an Entity:

Any entity can be erased from a drawing using the Erase command.

Ribbon Panel   Home – Modify -

Modify Toolbar

On entering the command, the crosshairs are replaced by a pick box (a small box) that allows objects (or entities) to be selected using the left mouse button; any object selected changes appearance from a continuous line to broken lines.   When all objects have been selected, the right mouse button, or the Enter key, is pressed to execute and exit the command.

Any erased entities can be restored within the next few commands by typing **OOPS** on the command line and pressing the **Enter** key; once the Erase command is used following a previous erasure, those entities cannot be retrieved using the OOPS command.

Tutorial Exercise TUT 1-4:
Erase the zigzag lines created in Tutorial Exercise TUT 1-2.

Procedure:
1.   Enter the Erase command by moving the cursor over the Erase   icon in the Modify Panel and click the left mouse button.
2.   Move the pick box over one of the lines and click the left mouse button (*Notice the selected line appears as a dashed line*).
3.   Move the pick box over another line and click the left mouse button.
4.   Repeat the above step and select the remaining lines.
5.   Click the right mouse button to erase the lines and exit the command.
6.   Type **OOPS** using the keyboard and press the **Enter** key.
7.   The lines have reappeared.
8.   Erase the lines again by repeating steps 1 to 5 above.

Drawing Lines Using Coordinate Methods:

Any drawing produced by CAD should be drawn accurately as complications can occur later causing major errors in the process of manufacturing the components or project. As stated earlier, CAD systems produce lines by specifying points using a distance from a datum point along horizontal and vertical axes.   In all there are 3 coordinate methods for inputting data to draw lines and other entities; they are Absolute, Relative and Polar Coordinate Methods.   Coordinates use a datum style grid where all points are given specific grid points and/or a combination of length and direction from the last specified point.

The coordinate Methods use values to specify a point by entering an X and Y value, the values are separated by a comma; e.g. 75,100.   The X-value (75) is the horizontal distance along the X-axis measured from the datum; the Y-value (100) is the vertical distance along the Y-axis measured from the datum.

Absolute Coordinate Method:

All points used in the Absolute Coordinate Methos are taken from the drawing's Origin Point which is located at the intersection of the X & Y axes.   The position of point A lies 75mm to the right of the datum point along the X-axis and 50mm above the Y-axis. Point B is 95mm to the left of the datum point along the X-axis and 35mm below the Y-axis.   Point C is 95mm to the left of the datum point along the X-axis and 35mm below the Y-axis.   While point D is 170mm to the left of the datum point along the X-axis and 40mm above the Y-axis.   When using Absolute Coordinates, the coordinates of the start

point should be a simple figure to be used in the calculations; e.g. 100,100. With Dynamic Input, absolute coordinates can be specified with the **#** prefix; if coordinates are added on the command line instead of in the tooltip, the **#** prefix is not used.
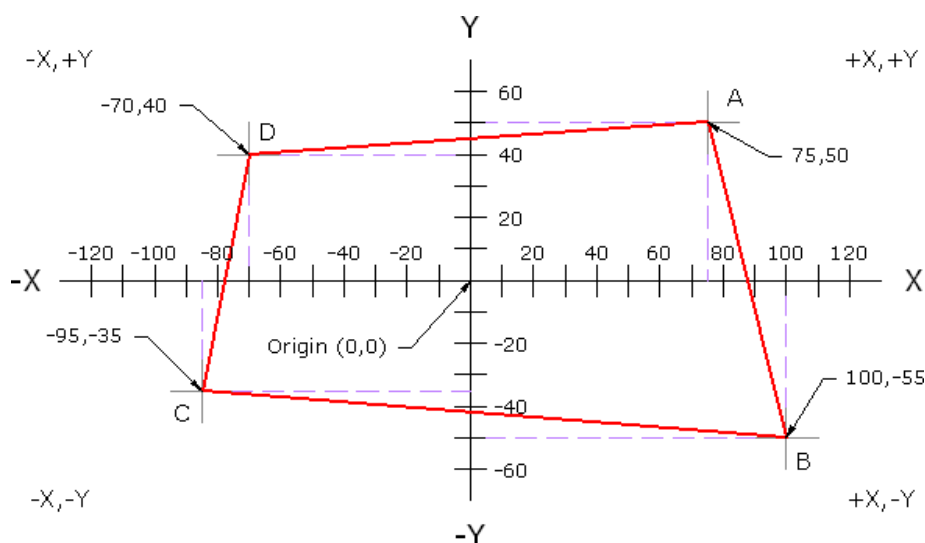

Figure 1.6 – Absolute Coordinate Method

The commands to draw the above shape (A to B to C to D to A) would be:

- Enter the Line command by clicking on the Line icon in the Draw Toolbar.
- "*_Line Specify first point*:" type **75,50** and press **Enter**
- "*Specify next point or [Undo]:*" type **100,-55** and press **Enter**
- "*Specify next point or [Undo]:*" type **–95,-35** and press **Enter**
- "*Specify next point or [Undo]:*" type **–70,40** and press **Enter**
- *Specify next point or [Undo]:*" type **C** and press **Enter** to end.

Relative Coordinate Method:

Relative Coordinates temporarily move the datum points from its normal position to the last specified point for the one command only, the datum then reverts to its normal position. Adding and ampersand sign, @, immediately before the X & Y coordinates forces the datum point to the last specified position which would normally be the endpoint of a line or the centre of a circle. A typical example of the coordinates would be @100,-75.
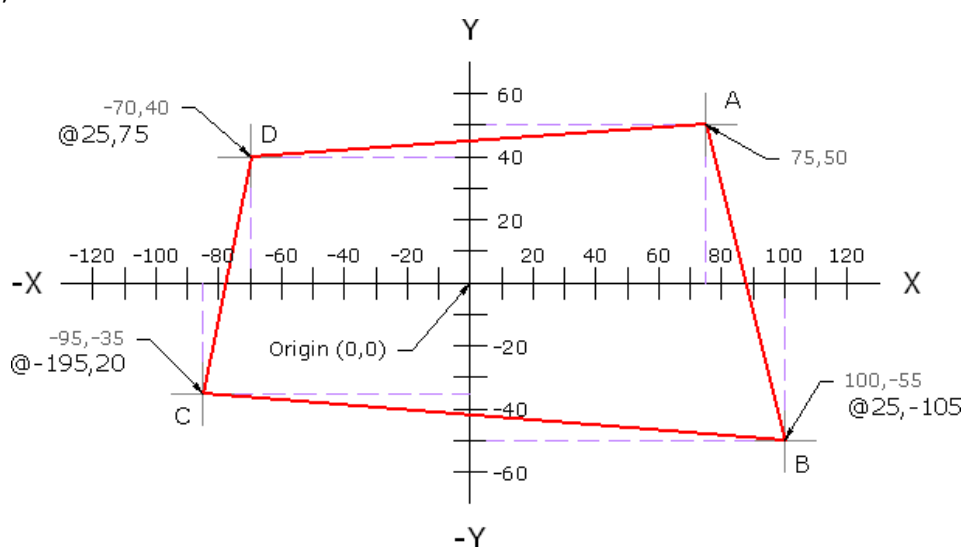

Figure 1.7 – Relate Coordinate Method

The commands to draw the above shape (A to B to C to D to A) would be:

- Enter the Line command by clicking on the Line icon in the Draw Toolbar.

- "*_Line Specify first point*:" type **75,50** and press **Enter**
- "*Specify next point or [Undo]:*" type **@25,-105** and press **Enter**
- "*Specify next point or [Undo]:*" type **@-195,20** and press **Enter**
- "*Specify next point or [Undo]:*" type **@25,75** and press **Enter**
- *Specify next point or [Undo]:*" type **C** and press **Enter** to end.

The distance between points A & B is 100-72=25 right or +ve direction on the X-axis and 50+55=105 down –ve direction on the Y-azis.   The distance between points B & C is 100+95=195 left or -ve direction on the X-axis and 55-35=20 up +ve direction on the Y-azis.   The distance between points C & D is 95-70=25 right or +ve direction on the X-axis and 35+40=75 up +ve direction on the Y-azis.

If the Dynamic Input is turned ON and the display appears next to the cursor, all distances are automatically made Absolute from the last point entered (the @ symbol is not required).

Polar Coordinate Method:

Polar coordinates require a distance and angular direction from a given or datum point. The distance and angular values are separated by the angle symbol, <.   All angles are measured from the horizontal in a counter clockwise direction with 0° at the right side of the screen.   Angles given in the clockwise direction are preceded by the –ve sign.
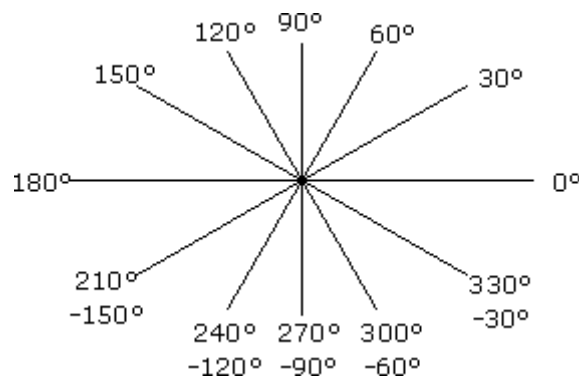
Figure 1.8 – Polar Angles

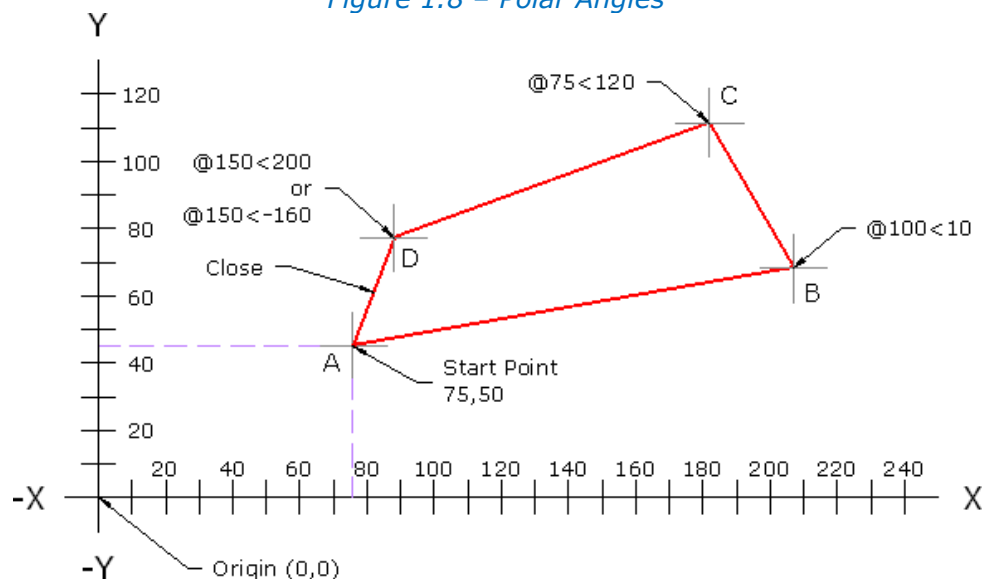*Figure 1.9 – Polar Coordinate Method*

The commands to draw the above shape (A to B to C to D to A) would be:

- Enter the Line command by clicking on the Line ✏ icon in the Draw Toolbar.
- "*_Line Specify first point*:" type **75,50** and press **Enter**
- "*Specify next point or [Undo]:*" type **@100<10** and press **Enter**
- "*Specify next point or [Undo]:*" type **@75<120** and press **Enter**

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- ▪ *"Specify next point or [Undo]:"* type **@150<200** and press **Enter**
- ▪ *Specify next point or [Undo]:"* type **C** and press **Enter** to end.

Tutorial Exercise TUT 1-5:

Create a NEW drawing using the template file called A4BASIC provided in your folder on the network drive and produce a drawing of the following Plates using the Absolute, Relative and Polar Coordinate Methods for Plate 1, Plate 2 and Plate 3 respectively.

Plate 1
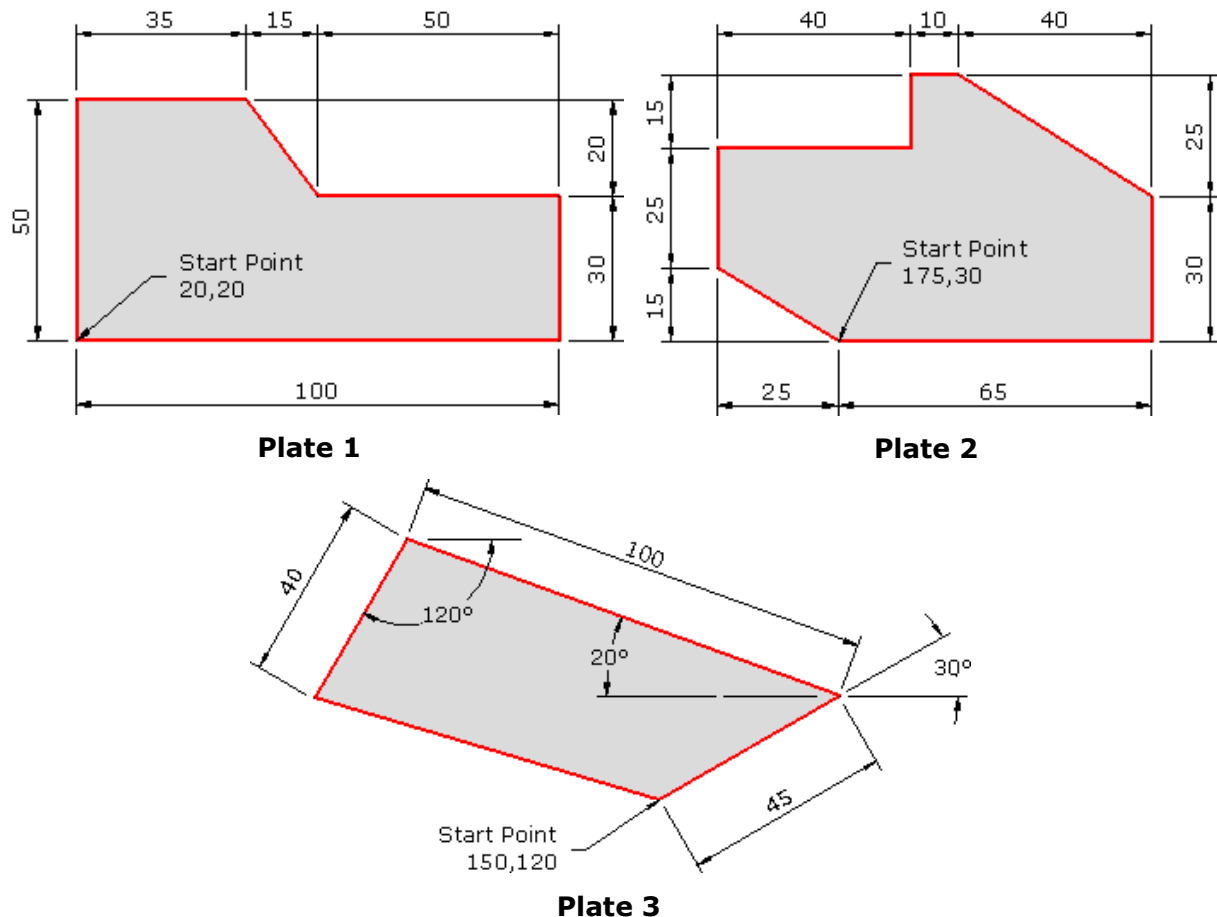
Plate 2

Plate 3

Procedure:

Create a new drawing using the BASIC template.

1. Left click on the New drawing ▢ button in the Quick Access Menu.
2. Expand the Look in box by clicking the triangle ▾.
3. Locate the drawing template called **A4BASIC** in your work area (or CD).
4. Click **OPEN** at the bottom of the dialog box and the new drawing is created.

Draw Plate 1 using only Absolute Coordinates only.

5. Turn the Dynamic Input OFF by clicking the ⊢ icon on the Status bar.
6. Enter the Line command by clicking on the Line ╱ icon in the Draw Toolbar.
7. On the command prompt **"_Line Specify first point:"**, type **20,20** for the Start Point and then press the **Enter** key.
8. The end of the horizontal line is 100 to the right, therefore the new X-coordinate is 20+100=120 and the Y-coordinate is 20+0=20.
9. *Specify next point or [Undo]:"* type **120,20** and press **Enter**.
10. Calculate the coordinates at the top of the right end; X-coordinate does not change as the line is vertical; Y-coordinate is 20+30=50 (120,50).
11. On the command prompt **"_Line Specify first point:"**, type **120,50** and then press the **Enter** key.

12. Draw the horizontal for the step.   As the line is horizontal and to the left, the X-value will be the previous X-value less the distance while the Y-value remains the same; therefore X-value is 120-50=70 and Y-value is 50 (70,50).
13. *Specify next point or [Undo]:"* type **70,50** and press **Enter**.
14. Draw the vertical line of the step. As the line is angled to the left by 15 and up by 20, the X-value becomes 70-15=55, while the Y-value becomes the last Y-value plus the step's height, 50+20=70 (55,70).
15. *Specify next point or [Undo]:"* type **55,70** and press **Enter**.
16. Draw the top of the block. The line is horizontal and drawn to the left 35 so the X-value becomes 55-35=20 and the Y-value remains the same (20,70).
17. *Specify next point or [Undo]:"* type **20,70** and press **Enter**.
18. Draw the vertical line representing the left side of the block.   The easiest method is to use the Close option within the command.
19. *Specify next point or [Undo]:"* type **C** and press **Enter**.

Draw Plate 2 using the Absolute Coordinate Method to locate the start point then the Relative Coordinate Method to draw the shape.

20. Enter the Line command by clicking on the Line ╱ icon in the Draw Toolbar.
21. On the command prompt **"_Line Specify first point:"**, type **175,30** for the Start Point and then press the **Enter** key.
22. The end of the horizontal line is 65 to the right, therefore the new X-coordinate for the end of the line is 0+65=65 and the Y-coordinate is 0+0=0.
23. *Specify next point or [Undo]:"* type **@65,0** and press **Enter**.
24. The next line rises vertical for 30 therefore the coordinate from the end of the last point is 0 for the X-value and 30 for the Y-value.
25. *Specify next point or [Undo]:"* type **@0,30** and press **Enter**.
26. The next line rises to the left therefore the coordinate from the end of the last point is –40 (to the left) for the X-value and 25 for the Y-value.
27. *Specify next point or [Undo]:"* type **@-40,25** and press **Enter**.
28. The short horizontal line is drawn 10 the left therefore the coordinate from the end of the last point is –10 for the X-value while the Y-value remains the same.
29. *Specify next point or [Undo]:"* type **@-10,0** and press **Enter**.
30. The vertical line is drawn down 15 therefore the coordinate from the end of the last point is 0 for the X-value while the Y-value becomes –15 (below the last point).
31. *Specify next point or [Undo]:"* type **@0,-15** and press **Enter**.
32. The horizontal line is drawn 40 the left therefore the coordinate from the end of the last point is –40 for the X-value while the Y-value remains the same.
33. *Specify next point or [Undo]:"* type **@-40,0** and press **Enter**.
34. The vertical line at the left end of the block is drawn down 25 therefore the coordinate from the end of the last point is 0 for the X-value while the Y-value becomes –25 (below the last point).
35. *Specify next point or [Undo]:"* type **@0,-25** and press **Enter**.
36. The last line can be drawn using one of three methods: the shape can be closed, the coordinates of the start point typed and entered, or the coordinate from the last point typed and entered.
37. *Specify next point or [Undo]:"* type **@25,-15**, press **Enter,** then **Enter** again.

Draw Plate 3 using an Absolute Coordinate to specify the stat point and then us only the Polar Coordinate Method to complete the shape.

38. Press the **Enter** key to re-enter the command.   (The previous command can be executed by pressing the Enter key instead of using the toolbars or menus if preferred).

39. On the command prompt **"_Line Specify first point**:", type **150,120** for the Start Point and then press the **Enter** key.
40. Working in a counter clockwise direction, the first line is 45 long at an angle of 30° therefore the polar coordinate is @45<30.
41. **Specify next point or [Undo]:"** type **@45<30** and press **Enter**.
42. The second line is 100 long but the angle is given as 20° to the horizontal but drawn to the left. The angle from 0° must be first be calculated. 180-20=160 therefore the polar coordinate is @100<160.
43. **Specify next point or [Undo]:"** type **@100<160** and press **Enter**.
44. The next line is 40 long at an angle of 120° in the clockwise direction. The angle can be calculated and specified from 0° in the counter clockwise direction 360-120=240 or the negative angle of –120 can be used. Errors are made during the most simplest of calculations.
45. **Specify next point or [Undo]:"** type **@40<-120** and press **Enter**.
46. The easiest method to close the shape is to use the Close option.
47. **Specify next point or [Undo]:"** type **C** and press **Enter**.

Save the Drawing:

All drawings must be saved so they can be retrieved later for editing and/or plotting. AutoCAD drawing files are saved with the extension of .DWG while a backup copy is also created using same filename but an extension of .BAK.

Quick Access Toolbar

Menu Browser   A – File - Save

Standard Toolbar

NB: SAVE ALL WORK EVERY 10 MINUTES as power failures, computer freeze or the dislodgment of power chords can occur at any time and are never planned. REGULARLY SAVE YOUR WORK. BE WARNED!!!!!

When the drawing is first saved, the Save As dialog box is displayed where the filename and location for the file can be entered.

Save In:

Drawing Figures   The Save In area displays the current folder or drive. Click the arrow to view the hierarchy of the folder path and to navigate up the path tree or to other drives, network connections, FTP locations, or web folders (either Web Folders or My Network Places, depending on the version of your operating system). Web folders can be created in Windows Explorer.

Back to Previous Folder:

Returns to the previous file location.

Up One Level:

Navigates one level up in the current path tree.

Search the Web:

Displays the Browse the Web dialog box, from which allows the operator to access and store files on the Internet.
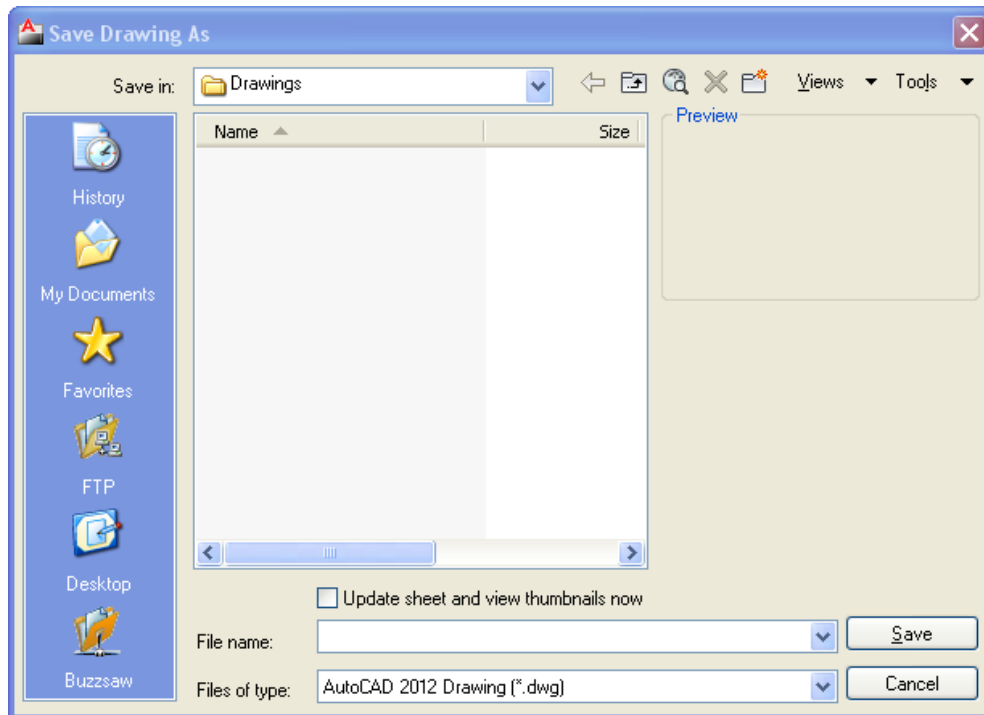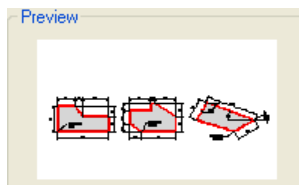
Lesson 1 – AutoCAD Basics

Figure 1.10

Delete:

   Deletes the selected file or folder.

Create a New Folder:

   Creates a new folder in the current path using a name that specified by the operator.

Preview:

   Displays a bitmap of the selected file when Views » Preview is chosen in the dialog box.   The Preview area is blank if no files are selected. To save a bitmap with a drawing file, use the Save a Thumbnail Preview Image option on the Open and Save tab in the Options dialog box.

File Name:

   Displays the name of the file selected in the Files list. If multiple files are selected, the File Name displays each selected file within quotation marks. Once a file name is entered in the File Name box, any selection is cleared.   Quotation marks must be used when entering multiple file names. Wild-card characters can be used to filter files displayed in the Files list.

Files of Type:

   The Files of Types sets the version of AutoCAD or the file type to be saved as for other applications. Drawings can be saved in other formats (or older AutoCAD Releases) by expanding the hierarchy and selecting the appropriate format.

Location of Saved Files:

Files can be saved to several places, a USB Flash Drive, CD-R or CD-WR discs, Network Drive and Internal or External Hard Drive.

Network Drive:

Drawing files and other related files can be stored on a network system where those files are available to all authorised users in the organisation or group.

Files saved on the network drive at NSW TAFE colleges are all different however may be similar to the following address: **Campus on Grf6 (P:)drive – Students – CAD_Drafting – Drafting_Students – *Your Teacher – Your Class – Your Name***.

It is recommended that all drawings saved on the network drive are backed up using another method as the folders are in the Public Domain allowing other students to accidentally or deliberately, move or delete your valuable work. Once deleted, the file cannot be retrieved. The drive may not be backed up by the section or college, so again, any work lost can never be retrieved.

USB Flash Stick:

The USB Flash Drives are available in several sizes; they are easy and quick to use and can store many complex drawings. It is highly recommended that a USB Flash Drive is purchased and used for ALL classwork.

CD-R or CD-WR:

CD-R and CD-RW discs require more time and effort to save drawings to; a drawing cannot be directly saved to the disc but must be burnt using a special program.   CD-R discs can only be written to once while CR-RW discs can be continuously written and rewritten to.   It is recommended to use CD-r and CD-RW discs for saving files as a backup once a job is 100% completed and no changes are likely.

USB Flash Stick	CD-R/CD-WR Disc	External Hard Drive

Tutorial Exercise TUT 1-6:
Save the drawing in your work area as TUT 1-6.

Procedure:

1.   Left click Save 🖫 icon.
2.   Expand the "Save in" box by clicking the down arrow and locate your folder:
          Select – **Network Drive** (Campus on Grf6 (P:)drive)

          Click – **Drive Folder** – (Students)

          Click – **Teaching Section** – CAD_Drafting

          Select – **Teacher** – *Your Teacher*

          Select – **Class** – *Your Class*

          Select – **Name** – *Your Name*

3.   Click in the "File name" box and type **TUT 1-6**.
4.   Check the "Files of type" is set to .DWG.
5.   Click on **Save**.

Object Snap:

Object snap constrains points to exact geometric locations on an entity and allows rapid positioning of the points without having to determine/calculate and then enter the particular coordinate. Running or Permanent and Temporary are the two methods used when working with Object Snap; Running, or permanent Object Snaps will be covered in a later lesson, temporary snaps will be used in the first few lessons as an introduction only and to show the different ways of entering data. Temporary Object Snaps can be selected by holding the Control key and clicking the Right Mouse button.

Any Object Snap bounded in the Context menu by a square indicates that particular snap is active when the Object Snap is running in the Permanent mode.

The most commonly used Object Snaps are Endpoint, Intersection, Centre and Node.

Temporary Object Snap Mode:

The object snap can be selected in the temporary mode by right clicking the pointing devise and selecting the required object snap icon. The selection lasts for only the next point selected.
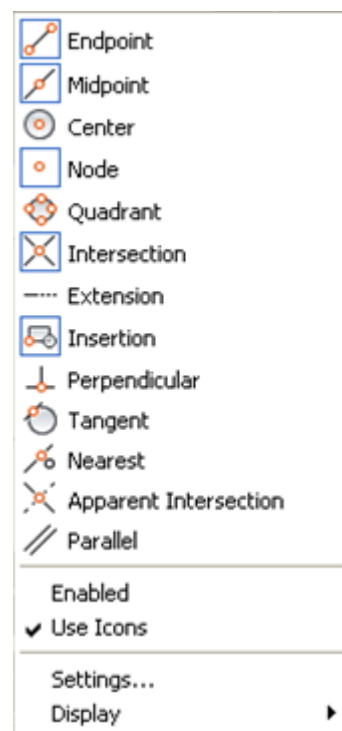

Figure 1.11

Permanently Turning an Object Snap ON or OFF:

The object snaps are turned permanently ON or OFF using the Settings menu. Right clicking on the Object Snap button in the Status Bar displays the Context menu where one object snap can be selected; selecting **Settings** displays the Drafting Settings dialog box. Check the Object Snap tab is current and then left click in the box next to the object snap to be turned ON. Left click to remove the tick and deactivate the object snap.

Endpoint:

Endpoint snaps to the exact geometric end of a straight line or arc and is displayed by a small square □ at the exact mathematical endpoint. The cursor will select the end nearest to the location of the cursor.

Intersection:

Intersection snaps to the exact intersection of straight lines, arcs and circles or a combination of lines, arcs and circles and is displayed by a cross ⨯.

Midpoint:

Midpoint snaps to the geometric midpoint of a straight line or arc and is displayed by a small triangle △. The midpoint is located by slowly moving the cursor over the line or arc and the geometric midpoint will be shown.

Tutorial Exercise TUT 1-7:

Create a NEW drawing using template file SNAP 1 provided in your work area. Draw a straight line between the top-right hand end of the green line and the midpoint of the white line, and then another straight line between the midpoint of the white line and the intersection of the yellow lines.

CTRL + RMB will be used in the following Tutorial exercise to display an Object Snap mode. Hold down the Control key and click the Right Mouse Button.

Procedure:

1.   Left click on the New Drawing icon, (Use a Template), and then click **Browse**.
2.   Expand the "Current drive or folder" directory and locate the template file called **SNAP 1** in your work area.
3.   Enter the line command by clicking on the Line icon in the Draw Panel.
4.   At the prompt "*_Line Specify first point*:" **CTRL + RMB** and click on the Endpoint icon.
5.   From the prompt "*_line Specify first point: _endp of*" place the cursor towards the right hand end of the green line (notice the endpoint square is indicated) and click the left mouse button.
6.   On the prompt "*Specify next point or [Undo]*:" **CTRL + RMB** and select the Midpoint icon.
7.   From the prompt "*Specify next point or [Undo]: _mid of* " move the cursor over any part of the line forming the white line (notice the triangular image shows on the exact point) and click the left mouse button.
8.   Follow the prompt "*Specify next point or [Undo]*:" **CTRL + RMB** and select the Intersection icon.
9.   From the prompt "Specify next point or [Undo]: _*int of* " move the cursor over the intersection of the 2-yellow lines (notice the cross is displayed at the intersection point) and click the left mouse button.
10.   Right click the mouse button to exit the command.

## **Printing/Plotting the Drawing:**

All drawing must be plotted off for the project to be constructed or component manufactured.   The drawing can be printed directly to smaller A4 & A3 document printers or larger drawings plotted out on A1 & A0 plotters.

Quick Access Toolbar

Application Menu   A – Print – Plot

Standard Toolbar

Entering the Plot command displays the Plot dialog box.
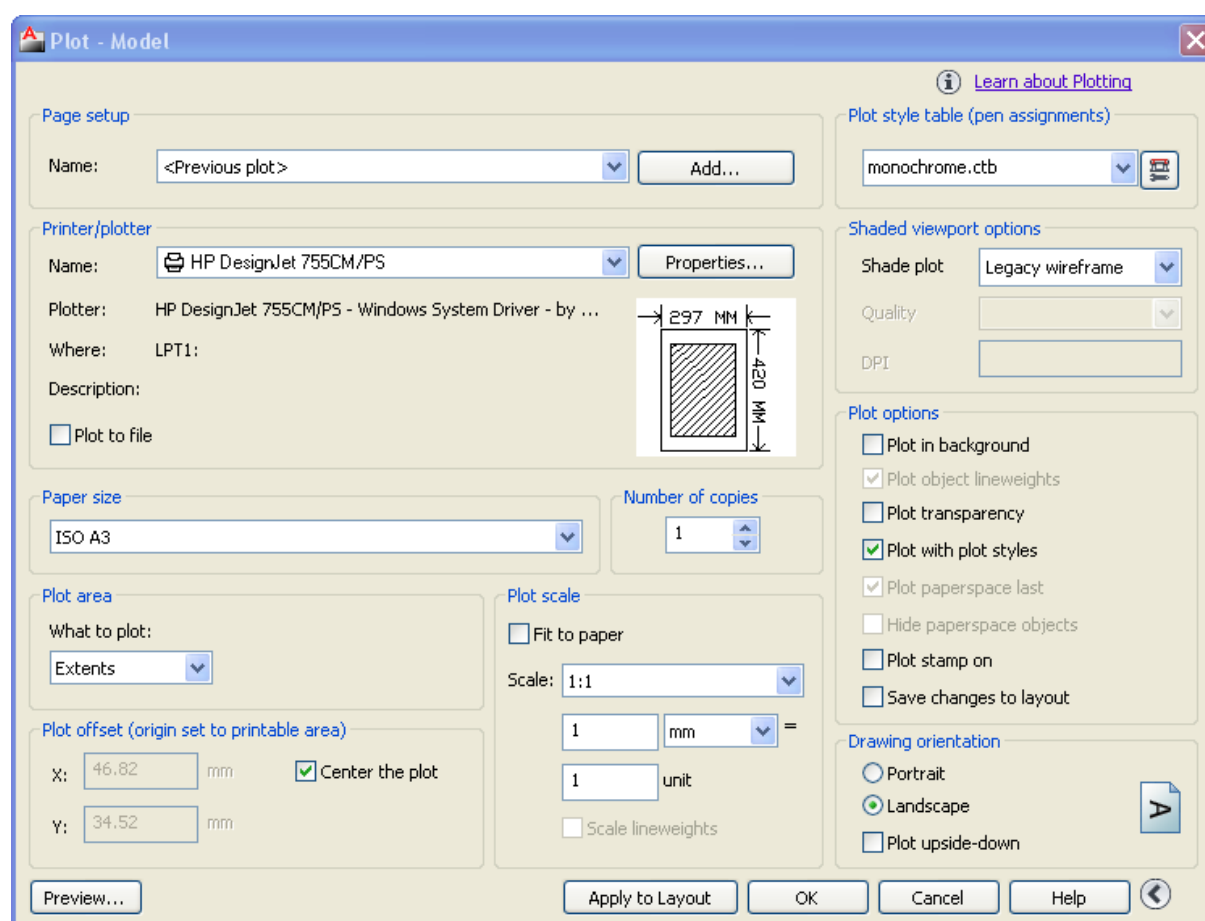
Figure 1.12 – Plot Dialog Box

The areas within the Plot dialog box that effects most prints are:

Printer/Plotter:
Selects the printer or plotter from a list available for use from the computer. Additional printers and plotters can be added using the "Add printers and faxes" function in the Start menu.

Thumbnail Preview:
The thumbnail shows the area to be printed on the sheet, relative to the paper size and printable area as a hatched rectangle; the width of the paper in the plotter should match the dimension shown above the thumbnail. Any border of the thumbnail showing red indicates the plot will not fit into the available area – the scale must be changed or a new sheet selected. Conversely, if the plot area appears too small, the sheet size or the scale must be adjusted accordingly.

If the cursor arrow is moved over the thumbnail, the dimensions for the sheet size and printable area are displayed.

Plot to File:
Sends to data output to be saved to the hard drive or other device for later printing; on clicking OK button, the Browse for Plot File dialog box. The default location for plot files is specified in the Options dialog box, Plot and Publish tab, under Default Location for Plot-to-File Operations.

All plot files as saved with a .PLT file extension.

Paper Size:
When expanded, the appropriately sized drawing sheet can be available for the stipulated plotter can be selected.

Number of Copies:

Specifies the number of prints required.

Plot Area:

Sets what area is to be plotted clicking the What to Plot box and selecting one of the options.

<u>Layout:</u>

Layout is only available while working in the Layouts. All entities within the printable area of the specified paper size will be printed, with the origin calculated from 0,0 in the layout.

<u>Limits:</u>

Limits is only available while working in the Model. Only entities within the setup limits (or grid) will be printed.

<u>Display:</u>

Plots the view in the current viewport in the selected Model tab or the current paper space view in the layout.

<u>Extents:</u>

Plots the area with the extreme outermost entities.

<u>Window:</u>

Plots any portion of the drawing specified by the operator. On selecting the Window option, the Window button becomes available. Select the two corners of the area to be plotted using the pointing device, or enter coordinate values.

Fit to Paper:

Scales the entire plot to fit within the selected paper size. The scale and units boxes are frozen but display the exact scale. The Fit to Paper option is mainly used for Checking Prints to save paper wastage.

Scale:

Sets the specific scale of the plot; the standard engineering reduction scales are 1:1, 1:2, 1:5, 1:10, 1:20, 1:50 and 1:100 etc. The engineering enlargement scales are 2:1, 5:1 and 10:1.

Plot Style Table (Pen Assignments):

Sets the plot style table, edits the plot style table, or creates a new plot style table.

When plotting a drawing using just black lines, the assignments should be set to **MONOCHROME**.

Drawing Orientation:

Drawing Orientation specifies the direction of the drawing on the paper for plotters that support landscape or portrait orientation. The paper icon represents the media orientation of the selected paper while the letter icon represents the orientation of the drawing on the page.
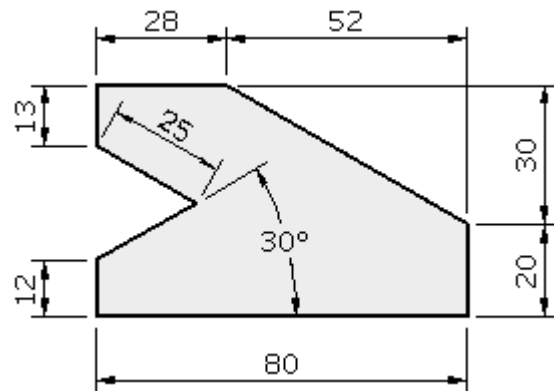
Skill Practice Exercises

Back up all your Tutorial Exercises and Skill Practice Exercises to a separate hard drive device prior to finishing all work.

Skill Practice Exercise 1-1

Create a new drawing using the template provided called A4BASIC and reproduce the drawing shown in Skill Practice Exercise 1-1.   DO NOT DIMENSION. Save the file in your work area as MEM30001-SP-0101. Plot the final drawing on an A4 sheet.

Skill Practice Exercise 1-2

Create a new drawing using the template provided called A4BASIC and reproduce the drawing shown in Skill Practice Exercise 1-2.   DO NOT DIMENSION. Save the file in your work area as MEM30001-SP-0102. Plot the final drawing on an A4 sheet.
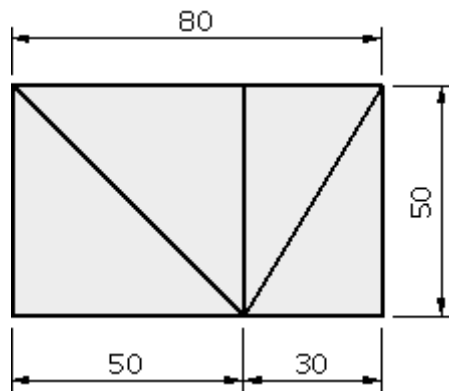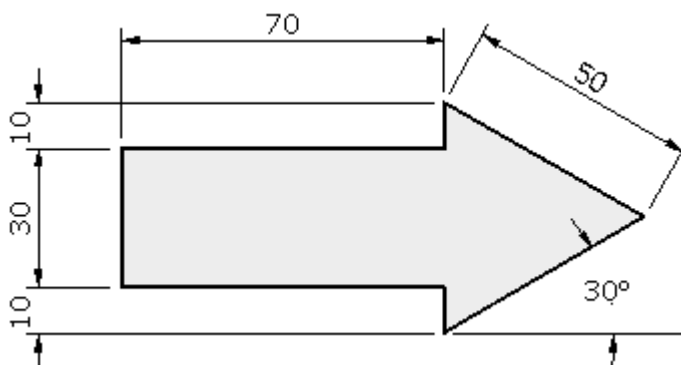
Skill Practice Exercise 1-1

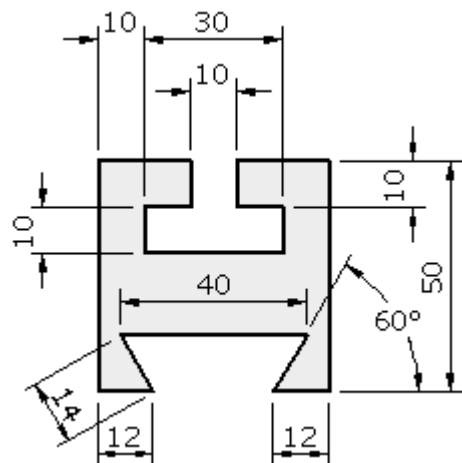Skill Practice Exercise 1-2

Skill Practice Exercise 1-3

Create a new drawing using the template provided called A4BASIC and reproduce the drawing shown in Skill Practice Exercise 1-3.   DO NOT DIMENSION. Save the file in your work area as MEM30001-SP-0103. Plot the final drawing on an A4 sheet.

Skill Practice Exercise 1-4

Create a new drawing using the template provided called A4BASIC and reproduce the drawing shown in Skill Practice Exercise 1-4.   DO NOT DIMENSION. Save the file in your work area as MEM30001-SP-0104. Plot the final drawing on an A4 sheet.

Skill Practice Exercise 1-3

Skill Practice Exercise 1-4