CHAPTER 2: COMPUTER AIDED DESIGN (CAD)

Fundamental of technical drawing

Technical drawing, also known as drafting or draughting, is the act and discipline of composing plans that visually communicate how something functions or is to be constructed. Technical drawing is essential for communicating ideas in industry and engineering. To make the drawings easier to understand, people use familiar symbols, perspectives, units of measurement, notation systems, visual styles, and page layout. Together, such conventions constitute a visual language, and help to ensure that the drawing is unambiguous and relatively easy to understand. This need for precise communication in the preparation of a functional document distinguishes technical drawing from the expressive drawing of the visual arts. Artistic drawings are subjectively interpreted; their meanings are multiply determined. Technical drawings are understood to have one intended meaning.

A drafter, draftsperson, or draughtsman is a person who makes a drawing (technical or expressive). A professional drafter who makes technical drawings is sometimes called a drafting technician.

Methods of technical drawing

1. Sketching

A sketch is a quickly executed, freehand drawing that is not intended as a finished work. In general, sketching is a quick way to record an idea for later use. Architect's sketches primarily serve as a way to try out different ideas and establish a composition before undertaking a more finished work, especially when the finished work is expensive and time consuming.
Architectural sketches, for example, are a kind of diagrams. These sketches, like metaphors, are used by architects as a means of communication in aiding design collaboration. This tool helps architects to abstract attributes of hypothetical provisional design solutions and summarize their complex patterns, hereby enhancing the design process.

2. Manual or by instrument

The basic drafting procedure is to place a piece of paper (or other material) on a smooth surface with right-angle corners and straight sides—typically a drawing board. A sliding straightedge known as a T-square is then placed on one of the sides, allowing it to be slid across the side of the table, and over the surface of the paper.
"Parallel lines" can be drawn simply by moving the T-square and running a pencil or technical pen along the T-square's edge. The T-square is used to hold other devices such as set squares or triangles. In this case the drafter places one or more triangles of known angles on the T-square—which is itself at right angles to the edge of the table—and can then draw lines at any chosen angle to others on the page. Modern drafting tables come equipped with a drafting machine that is supported on both sides of the table to slide over a large piece of paper. Because it is secured on both sides, lines drawn along the edge are guaranteed to be parallel.

In addition, the drafter uses several technical drawing tools to draw curves and circles. Primary among these are the compasses, used for drawing simple arcs and circles, and the French curve, for drawing curves. A spline is a rubber coated articulated metal that can be manually bent to most curves.

Types of technical drawings

The two types of technical drawings are based on graphical projection. This is used to create an image of a three-dimensional object onto a two-dimensional surface.

Two-dimensional representation: uses orthographic projection to create an image where only two of the three dimensions of the object are seen.

Three-dimensional representation: also referred to as a pictorial, all three dimensions of an object are visible.

Views

1. A section, or cross-section, is a view of a 3-dimensional object from the position of a plane through the object.
   A cross section is a common method of depicting the internal arrangement of a 3-dimensional object in two dimensions. It is often used in technical drawing and is traditionally crosshatched. The style of crosshatching indicates the type of material the section passes through.
   A 2-D cross-sectional view of a compression seal.
2. An elevation is a view of a 3-dimensional object from the position of a vertical plane beside an object. In other words, an elevation is a side-view as viewed from the front, back, left or right (and referred to as a front elevation, [left/ right] side elevation, and a rear elevation). It is the corollary to the concept of a "view" (which is always overhead and is therefore referred to as an overhead view).
   An elevation is a common method of depicting the external configuration and detailing of a 3-dimensional object in two dimensions. Building façades are shown as elevations in architectural drawings and technical drawings.

   Elevations are the most common orthographic projection for conveying the appearance of a building from the exterior. Perspectives are also commonly used for this purpose. A building elevation is typically labeled in relation to the compass direction it faces; the direction from which a person views it. E.g. the North Elevation of a building is the side that most closely faces true north on the compass.
   - A developed elevation is a variant of a regular elevation view in which several adjacent non-parallel sides may be shown together, as if they have been unfolded. For example, the north and west views may be shown side-by-side, sharing an edge, even though this does not represent a proper orthographic projection.
3. A plan is a view of a 3-dimensional object from the position of a horizontal plane through, above, or below the object. In such views, the portion of the object in front of the plane is omitted to reveal what lies beyond. In the case of a floor plan, the roof and upper portion of the walls may be omitted. Elevations, top (roof) plans, and bottom plans are orthographic projections, but they are not sections as their viewing plane is outside of the object.
   A plan is a common method of depicting the internal arrangement of a 3-dimensional object in two dimensions. It is often used in technical drawing and is traditionally cross-hatched. The style of crosshatching indicates the type of material the section passes through.
4. An auxiliary view is a view taken from an angle that is not one of the primary views. An auxiliary view is a view at an angle used to give deeper insight into the actual shape of the object. An auxiliary view is used to show a slanted surface in true size and shape. This is accomplished by providing a view that is perpendicular to the slanted surface.

Meaning and importance of computer-aided design

Introduction to CAD (Computer aided design)
Computer-aided design (CAD) is the use of computer systems to assist in the creation, modification, analysis, or optimization of a design. CAD software is used to increase the productivity of the designer, improve the quality of design, improve communications through documentation, and to create a database for manufacturing. CAD output is often in the form of electronic files for print, machining, or other manufacturing operations.
Computer-aided design is used in many fields. Its use in designing electronic systems is known as electronic design automation, or EDA. In mechanical design it is known as mechanical design automation (MDA) or computer-aided drafting (CAD), which includes the process of creating a technical drawing with the use of computer software.
CAD software for mechanical design uses either vector-based graphics to depict the objects of traditional drafting, or may also produce raster graphics showing the overall appearance of designed objects. However, it involves more than just shapes. As in the manual drafting of technical and engineering drawings, the output of CAD must convey information, such as materials, processes, dimensions, and tolerances, according to application-specific conventions.
CAD may be used to design curves and figures in two-dimensional (2D) space; or curves, surfaces, and solids in three-dimensional (3D) space.
CAD is an important industrial art extensively used in many applications, including automotive, shipbuilding, and aerospace industries, industrial and architectural design, prosthetics, and many more. CAD is also widely used to produce computer animation for special effects in movies, advertising and technical manuals, often called DCC digital content creation. The modern ubiquity and power of computers means that even perfume bottles and shampoo dispensers are designed using techniques unheard of by engineers of the 1960s. Because of its enormous economic importance, CAD has been a major driving force for research in computational geometry, computer graphics (both hardware and software), and discrete differential geometry.
The design of geometric models for object shapes, in particular, is occasionally called computer-aided geometric design (CAGD).

Uses/importance of CAD
Computer-aided design is one of the many tools used by engineers and designers and is used in many ways depending on the profession of the user and the type of software in question.

CAD is one part of the whole Digital Product Development (DPD) activity within the Product Lifecycle Management (PLM) processes, and as such is used together with other tools, which are either integrated modules or stand-alone products, such as:
- Computer-aided engineering (CAE) and Finite element analysis (FEA)
- Computer-aided manufacturing (CAM) including instructions to Computer Numerical Control (CNC) machines
- Photo realistic rendering and Motion Simulation.
- Document management and revision control using Product Data Management (PDM).

CAD is also used for the accurate creation of photo simulations that are often required in the preparation of Environmental Impact Reports, in which computer-aided designs of intended buildings are superimposed into photographs of existing environments to represent what that locale will be like, where the proposed facilities are allowed to be built. Potential blockage of view corridors and shadow studies are also frequently analyzed through the use of CAD.

CAD has been proven to be useful to engineers as well. Using four properties which are history, features, parameterization, and high level constraints. The construction history can be used to look back into the model's personal features and work on the single area rather than the whole model. Parameters and constraints can be used to determine the size, shape, and other properties of the different modeling elements. The features in the CAD system can be used for the variety of tools for measurement such as tensile strength, yield strength, electrical or electro-magnetic properties. Also its stress, strain, timing or how the element gets affected in certain temperatures, etc.

Types of CAD

There are several different types of CAD, each requiring the operator to think differently about how to use them and design their virtual components in a different manner for each.

There are many producers of the lower-end 2D systems, including a number of free and open source programs. These provide an approach to the drawing process without all the fuss over scale and placement on the drawing sheet that accompanied hand drafting, since these can be adjusted as required during the creation of the final draft.

3D wireframe is basically an extension of 2D drafting (not often used today). Each line has to be manually inserted into the drawing. The final product has no mass properties associated with it and cannot have features directly added to it, such as holes. The operator approaches these in a similar fashion to the 2D systems, although many 3D systems allow using the wireframe model to make the final engineering drawing views.

3D "dumb" solids are created in a way analogous to manipulations of real world objects (not often used today). Basic three-dimensional geometric forms (prisms, cylinders, spheres, and so on) have solid volumes added or subtracted from them, as if assembling or cutting real-world objects. Two-dimensional projected views can easily be generated from the models. Basic 3D solids don't usually include tools to easily allow motion of components, set limits to their motion, or identify interference between components.

Solid Modeling There are two types of solid modeling

1). 3D parametric solid modeling allows the operator to use what is referred to as "design intent". The objects and features created are modifiable. Any future modifications can be made by changing how the original part was created. If a feature was intended to be located from the center of the part, the operator should locate it from the center of the model. The feature could be located using any geometric object already available in the part, but this random placement would defeat the design intent. If the operator designs the part as it functions the parametric modeler is able to make changes to the part while maintaining geometric and functional relationships.

2). Explicit Modellers or Direct 3D CAD Modelers provide the ability to edit geometry without a history tree. With direct modeling once a sketch is used to create geometry the sketch is incorporated into the new geometry and the designer just modifies the geometry without needing the original sketch. As with Parametric modeling, Direct modeling has the ability to include relationships between selected geometry (e.g., tangency, concentricity).

Top end systems offer the capabilities to incorporate more organic, aesthetics and ergonomic features into designs. Freeform surface modeling is often combined with solids to allow the designer to create products that fit the human form and visual requirements as well as they interface with the machine.

AutoCAD

AutoCAD stands for Automatic Computer Aided Design. AutoCAD is a computer-aided drafting software program used to create blueprints for buildings, bridges, and computer chips, among other things. Discover how AutoCAD is used by drafters and other professionals.
AutoCAD is a commercial software application for 2D and 3D computer-aided design (CAD) and drafting — available since 1982 as a desktop application and since 2010 as a mobile web- and cloud-based app marketed as AutoCAD.
Developed and marketed by Autodesk, Inc. and is used across a wide range of industries, by architects, project managers, engineers, designers, and other professionals.

Career and application Information of AutoCAD

While drafters work in a number of specialties, the five most common specialization areas are as follows: mechanical, architectural, civil, electrical, and electronics.

- Mechanical drafters prepare plans for machinery and mechanical devices.
- Architectural drafters draw up plans for residential and commercial buildings.
- Civil drafters draw up plans for use in the design and building of roadways, bridges, sewer systems, and other major projects.
- Electrical drafters work with electricians to prepare diagrams of wiring electrical system layouts.
- Electronics drafters also prepare wiring diagrams for use in the making, installing, and repairing of electronic gadgets.
- Applications for technical drawing

  Architecture: To plan a renovation, this architect takes measurements, which he later enters into his computer-aided design software.
  The art and design that goes into making buildings is known as architecture. To communicate all aspects of the shape or design, detail drawings are used. In this field, the term plan is often used when referring to the full section view of these drawings as viewed from above. Architectural drawings describe and document an architect's design.
  Engineering: Engineering can be a very broad term. It stems from the Latin ingenerare, meaning "to create". Because this could apply to everything that humans create, it is given a narrower definition in the context of technical drawing. Engineering drawings generally deal with mechanical engineered items, such as manufactured parts and equipment.
  Engineering drawings are usually created in accordance with standardized conventions for layout, nomenclature, interpretation, appearance (such as typefaces and line styles), size, etc. Its purpose is to accurately and unambiguously capture all the geometric features of a product or a component. The end goal of an engineering drawing is to convey all the required information that will allow a manufacturer to produce that component.

Advantages of a CAD system

- Decrease in error percentage: As the CAD software makes use of some of the best tools, the percentage of error that occurred because of manual designing is significantly reduced.
- Decrease in effort: When it comes to the amount of effort that was needed for the sake of designing the different models, it has been reduced significantly because the software automates most of the task.
- Saves time: When you are using the computer aided design software, it will save your time and you can make better and more efficient designs in shorter time duration.
- Easy to edit: When you are making designs, you may find the need to make alterations. When you are using computer aided design software, it will be much easier to make any changes because you can fix the errors and modify the drawings easily.
- Code re-use: As the entire task is carried out with the help of computer tools, it removes the problem of duplication of labor, you can copy the different parts of code and design which can then be reused multiple times over and over again.
- Improved accuracy: There is absolutely no doubt about the fact that the kind of accuracy that CAD software will offer can never be achieved by opting for manual drawings. You have tools to measure the precision, skill and accuracy level of the designs.
- Easy to share: The CAD tools make it easier to save the files and store it in a way that you can use it time and again and send it without any unwanted hassles too.

Features of a CAD system

Key Features and Benefits

Document

AutoCAD is synonymous with documentation for good reason.

Drive your projects from concept to completion with the powerful documentation tools in AutoCAD. Work faster with automation, management, and editing tools that minimize repetitive tasks and speed your time to completion.

No matter your project‘s size or scope, y AutoCAD can help you meet the challenge with AutoCAD — continuously leading and innovating documentation for over 25 years.

Communicate

With AutoCAD, communication is a seamless operation.

Share critical design data securely, efficiently, and accurately with AutoCAD.

Experience the benefits of native DWG™ support, the worlds most widely used design data format, allowing you to keep everyone in the loop at all times.

Take your ideas to the next level with presentation-ready graphics, rendering tools, and some of the best plotting and 3D printing capabilities in the business. It‘s communication at its best.

Explore

AutoCAD gives you 3D power to explore your ideas in almost any shape imaginable.

AutoCAD and a blank canvas have a lot in common. Both give you the ability to create the previously unimaginable. But AutoCAD provides the flexibility to explore design ideas in both 2D and 3D, with intuitive tools that help your concepts become real. The world is your canvas — what will you create next?

Customize

Customize and configure AutoCAD in ways you never thought possible.

Your job is unique. Your software should be as well. Customizing AutoCAD to meet your unique needs is easier than you ever thought possible. Configure your settings, extend the software, build custom workflows, develop your own application or leverage one already built. Some think you have to choose between flexibility and power. With AutoCAD, you can have both.

Document

Feature	Benefit
Parametric Drawing	Dramatically slash your design revision time with parametric drawing. By defining persistent relationships between objects, parallel lines remain parallel and concentric circles remain centered, all automatically.
Sheet Sets	Organization isn‘t a luxury. The AutoCAD Sheet Set Manager organizes your drawing sheets, reduces steps to publish, automatically creates layout views, links sheet set information into title blocks and plot stamps, and performs tasks across a sheet set so everything is in one convenient place.
Annotation Scaling	Spend less time creating and managing multiple items across multiple layers. With annotation scaling tools, you can create a single annotative-type object that automatically resizes to reflect the current viewport or model-space scale.
Text Editing	Now you can easily manipulate text by viewing, sizing, and positioning text as you type. Adjust the text ‘s appearance as needed using familiar tools common in text-editing applications, including paragraph and column tools to achieve professional-quality formatting.
Multiple Leaders	With multileader tools, creating and editing leaders is a breeze. Define multileader styles to provide consistency across leaders, add multiple leader lines to a single leader object, and even include bubbles or blocks as leader content.
Tables	Increase your productivity by automating the tedious task of creating and maintaining tables. You can define table styles to easily apply consistent table formatting including fonts, colors, borders, and much more.
Data Extraction	Quickly and easily extract property data from objects in drawings (including blocks and attributes) and drawing information with the Data Extraction wizard. The extracted data can then be automatically output to a table or an external file.
Data Linking	Easily link Microsoft®Office Excel®data to your AutoCAD designs for consistency and efficiency. Data links can be updated in both directions, reducing the need to update tables or external spreadsheets independently. All linked information can easily be kept current and in sync automatically.
Dynamic Blocks	Save time and standardize drawings with ease. With Dynamic Blocks, you no longer have to redraw repetitive standard components, and you can reduce your cumbersome block libraries in the process. Dynamic Blocks make individual block geometry editable and reduce the need to define a new block for every variation of shape and size.
Layer Management	Create and edit layer properties faster, and reduce errors at the same time. With the Layer Dialog box, changes are instantly reflected in the drawing as they are made in the dialog.
Prompts and Editing	Focus on the design, not the tools. Dynamic Input displays a command line type of prompt right at the cursor so you can launch commands, view dimensions, and enter values without even having to look at the command line. With the Quick Properties menu you can dramatically save time by viewing and modifying relevant object properties right at your cursor.
Efficient User	Interface Working with several files doesn‘t have to be painful anymore. The Quick View feature uses thumbnails in addition to file names, so you can

	visually find and open the correct drawing file and layout even faster than before. Within the Menu Browser you can also quickly browse files, examine thumbnail images, and get detailed information about file size and file creator.

Communicate

Feature	Benefit
DWG	Save and share files with confidence. DWG technology from Autodesk is the genuine and one of the most accurate ways to store and share design data when working with anyone in the industry.
PDF Integration	Sharing and reusing designs has never been easier in AutoCAD thanks to a long list of upgrades made in the name of streamlined communication. Publish PDF files directly from AutoCAD drawings and attach and snap to PDF files as underlays.
Autodesk Design Review	With built-in tools to publish and import DWF™ files within AutoCAD, collaborating on projects is more seamless than ever. Autodesk®Design Review software is the free, integrated, digital solution for clients or vendors to view, print, and mark up designs without needing the original software.
Autodesk Impression 3 (Available to subscription customers only)	Supercharge your design presentations with a hand-drawn look. Autodesk® Impression software lets you create compelling presentation-ready graphics directly from your DWG and DWF files.
Photo-realistic Rendering	With the latest in rendering technology, you can create stunning models in less time. Capabilities include a slider control that graphically displays the trade-off between time and rendering quality.
3D Printing	Do more than just visualize your designs—make them real. Your design can now sit in the palm of your hand with your 3D printer or through a connected service provider. ShowMotion With ShowMotion®technology, you have the power to create camera animations to navigate through your design. The ShowMotion control panel displays thumbnail images of the view categories and view shots saved in the drawing.

Explore

Feature	Benefit
Free-Form Design	You now have the power to design ideas in almost any form you can imagine. Simply push/ pull faces, edges, and vertices to model complex shapes, add smooth surfaces, and much more.
Solid and Surface Modeling	Shape your ideas in 3D just like you have in 2D. With an easy-to-learn environment for creating both solids and surfaces, you can now create and edit 3D shapes with the familiarity of 2D tools.
Visualization	Visualize your ideas like never before. Choose from over 300 materials, apply photometric lighting, and control the display to create highly accurate, photo-realistic rendered images.
3D Navigation	Walk or fly through a model with the click of a button. Quickly rotate and orient any solid or surface model with the ViewCube®tool, or pan, center, and zoom on

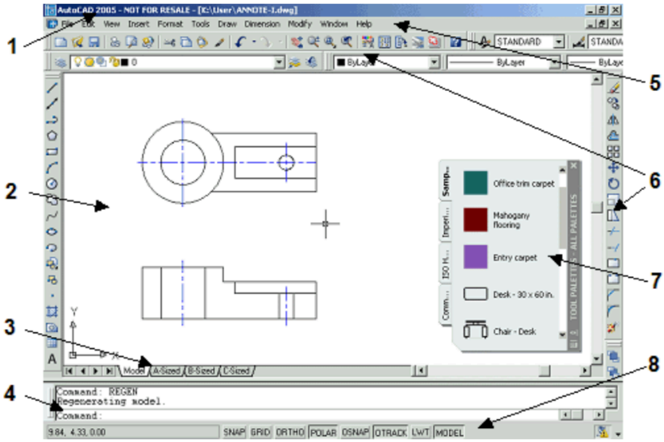
	any object with the SteeringWheels®tool.

Customize

Feature	Benefit
Programming Interface	Take advantage of our flexible development platform to boost your productivity. With direct access to database structures, the graphics system, and native command definitions, you can specialize design and drafting applications to perfectly fit your needs.
Action Recorder	Save time and increase productivity by automating repetitive tasks without requiring the assistance of a CAD manager. Now you can record tasks, add text messages, and request user input, then quickly select and play back recorded macros.
Autodesk Partner Products and Services	Get the most out of thousands of Autodesk's software partners from around the world. These partners can further enhance your software with a broad range of fully integrated and interoperable solutions no matter what you're designing.
Autodesk Developer Network	If you're creating innovative software built on AutoCAD, you'll want to join the Autodesk®Developer Network. Training and support enables your software or plug-in to tightly integrate with your workflow with the latest AutoCAD release.
Ribbon User Interface	Pump up overall drafting productivity with the ribbon interface. The ribbon is both easily customizable and expandable so that it can be optimized for each user and can help meet each company's standards.

Typical screen layout of a CAD system

AutoCAD's program window is divided into eight major areas:



1. The Title Bar is located across the top of the window. At the right end of the title bar are the standard Windows buttons for minimizing the window, resizing the window, and exiting the program.

2. The Drawing Windows are the areas of the screen where the drawings appear. Inside the drawing window, crosshairs indicate the current pointer location. The lower left corner of the window may also display a pair of arrows, called the UCS icon, that indicates the current drawing plane. Drawing windows can be resized, minimized, and maximized.

3. The Layout Tabs divide your working area into the Model, where you draw the full-sized objects, and any number of layouts, where you prepare the drawing to print. You select a tab to switch to it.

4. The Command Line is the single most important part of the AutoCAD interface. All commands and functions are issued through this small text window located at the bottom of the screen.

5. The Pull-down Menus provide easy access to AutoCAD commands and settings.

6. Toolbars contain buttons for commonly used commands. There are many toolbars, each of which contains buttons for related commands. Six toolbars are open by default: Standard and Styles (below the pull-down menus), Layers and Properties (below Standard), Draw and Modify (along the left and right edges of the screen). To open other toolbars, right-click on any tool

button to bring up a shortcut menu, and select the toolbar you want.

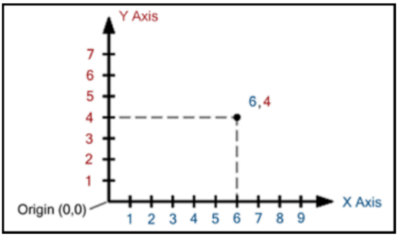
7. Tool Palettes Tool Palettes offer quick access to often-used drawing components and commands. The palette window can be turned on or off as needed. For the beginning of class, close the Tool Palettes window by picking the "x" at the top of the window.

8. The Status Bar displays and allows you to change many of AutoCAD's drafting settings. You may see an additional window called the Sheet Set Manager when you open AutoCAD. This is an advanced feature for organizing sets of drawings and sheets. For the beginning of class, close this window by picking the "x" at the top of the window.

AutoCAD's Cartesian Workspace

How AutoCAD Locates Points

AutoCAD uses Cartesian (X,Y) coordinates to indicate locations in a drawing. Points are located by designating a horizontal (X) and vertical (Y) distance as measured from the origin (0,0). (There is also a third coordinate, Z, which is used only in 3D drawing.)



Many commands in AutoCAD require that the user input a point. For example, in order to draw a line, you must tell AutoCAD where to begin and end the line. You can specify the point in one of two basic methods:

1. Picking a point on the screen with the pointing device.
2. Typing in coordinates at the Command: prompt (when it is requesting point entry) in the form X,Y. For example, the point (6,4) would be typed at the keyboard as 6,4.

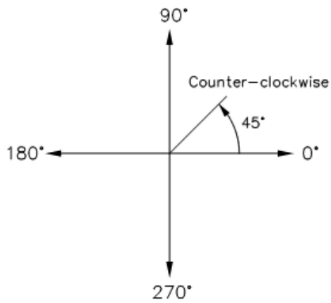
Finding Coordinates: the ID Command

AutoCAD displays the current coordinate location of the cursor in the Status Bar. You can also have AutoCAD report the coordinates of any point you pick. The ID command (Inquiry toolbar) reports on the Command Line the X, Y, and Z coordinates of any point you pick on the screen.

```
Command: ID <Enter>
Specify point: Pick a point on the screen
X = 14.5000     Y = 4.0000     Z = 0.0000
```

Drawings Are Created at Full Scale

AutoCAD's Cartesian workspace is essentially unlimited in size. Whatever the object you are creating, you will typically draw it at full scale. Whether you are drawing a building that's 100 meters by 200 meters, a city that's 10 square miles, or an IC chip that is 0.1" by 0.1", you always draw in the real units of the object. Your drawing area is a big as you need it to be. In fact, the entire solar system can be drawn at full scale in AutoCAD. (Scaling the drawing only becomes necessary when it is printed.)



Points in Cartesian Workspace

Besides being enormous, the AutoCAD drawing plane is also remarkably precise. Each point you enter in AutoCAD has an accuracy of 14 significant digits (1.0000000000000).

How AutoCAD Measures Angles

Along with the Cartesian coordinate system, you also need to understand how angles work in AutoCAD. This will be crucial for coordinate entry, rotating objects, and for working with arcs. AutoCAD measures angles in a counter-clockwise direction relative to the positive X-axis.

Entering Points in AutoCAD

You can enter points directly on the command line using three different systems.
The one you use will depend on which is more applicable for the situation. The three systems are as follows:

1. ABSOLUTE CO-ORDINATES - Using this method, you enter the points as they relate to the origin of the WCS. To enter a point just enter in the exact point as X,Y.
2. RELATIVE CO-ORDINATES - This allows you to enter points in relation to the first point you have entered. After you've entered one point, the next would be entered as @X,Y. This means that AutoCAD will draw a line from the first point to another point X units over and Y units up relative to the previous point.
3. POLAR CO-ORDINATES - You would use this system if you know that you want to draw a line a certain distance at a particular angle. You would enter this as @D<A. In this case, D is the distance and A is the angle. Example: @10<90 will draw a line 10 units straight up from the first point.

The three ways of entering co-ordinates shown above are the ONLY way AutoCAD accepts input. First decide which style you need to use, and then enter as shown.
Remember that X is always before Y (alphabetical). Don't forget the '@' symbol when you are entering relative points. Any typing error or omission will give you results you don't want. If you make a mistake and need to see what you typed, press F2 to bring up the text screen and check your typing. (Press F2 to get back to your drawing.)

AutoCAD configuration- Units and Scales

Introduction

Among the most important concepts that newcomers to AutoCAD need to get to grips with are those of drawing scale and drawing units. You cannot start creating sensible drawings with AutoCAD until you are familiar with scale, units and the commands you use to control them. This tutorial discusses these concepts, starting with the two most commonly asked questions in this subject area.

At what scale should I draw?

As a general rule, everything you draw with AutoCAD will be drawn full size. This often comes as quite a surprise to those who are new to CAD and have spent a number of years working on a drawing board. When you start drawing with AutoCAD you do not have to decide upon a drawing scale as you do when using a drawing board. When drawing on paper you must decide do draw at say, 1:20 or 1:200 depending upon the size of the object that you are drawing so that your scaled drawing will fit on the drawing sheet, be that A3 or A1. In AutoCAD you do not need to decide upon a drawing scale until you come to print the drawing and because the scaling of your drawing takes place at the printing stage, you can create drawings at a scale of 1:1. This has particular advantages because you can, for example, measure lengths, areas and volumes within an AutoCAD drawing and not need to compensate for any scale factor.

Am I drawing in metres or millimetres?

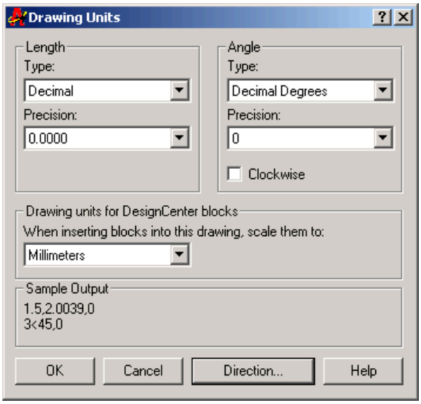
Most people who use AutoCAD, draw using decimal drawing units. What these drawing units represent is entirely up to the individual. However, you must decide what units you will use before you start drawing. One drawing unit could represent one millimetre, one centimetre, one metre, kilometre, mile, furlong or fathom. It is entirely up to you. However, in most parts of the world it is common practice to work in either millimetres or metres. Which of these two units you use will largely depend upon the type of drawing you are creating. For example, if you were creating a detail drawing of a flight of steps, you would most likely use millimetres (Architects will almost always use millimetres). If, on the other hand you are drawing a landscape masterplan, you would probably want to work in metres (Landscape Architects and Civil Engineers usually use metres).

By way of example, consider a drawing where you need to draw a footpath in plan. The footpath is two metres wide. If you are working in millimetres, the footpath would be drawn 2,000 drawing units wide but if you are working in metres, the footpath would be drawn just 2 drawing units wide. To translate this into practical terms, if you had drawn one edge of the path and you intended to draw the other edge using the Offset command, you would enter either "2,000" or "2" for the offset value depending upon whether you were using millimetres or metres respectively.

Although decimal drawing units are the most commonly used, you can configure AutoCAD to work with other types of drawing units. To change the unit type, you must use the Drawing Units dialogue box.

Units Control

Toolbar	none
Pull-down	Format ▸Units…
Keyboard	UNITS



When you start the Units command, the first thing you see is the Drawing Units dialogue box, shown on the right. The dialogue box is divided into four main sections. The upper two are "Length", which refers to linear units and "Angles", referring to angular units. Settings for linear units and angular units can be made independently and in each case, you can control both the type and precision. In addition, the Angles section also allows you to specify the direction in which angles are measured. See below for more details.

A third section, entitled Drawing units for Design Center blocks allows you to assign a specific unit to the drawing so that when blocks are inserted via the AutoCAD Design Centre, they will automatically be scaled. The final section, Sample Output, gives you a preview of the drawing units as they will be displayed using the current settings.

Linear Units

You can see from the dialogue box that there are five different linear unit types for you to choose from, one of which is "Decimal", the default. The table below shows the effect of the different unit settings on two drawing unit values to give you an idea how the various settings might be used along with a brief description.

Unit Type	1.5 Drawing Units	1500 Drawing Units	Description
Decimal	1.5000	1500.0000	Metric or SI units
Scientific	1.5000E+00	1.5000E+03	Decimal value raised to a power
Engineering	0'-1.5000"	125'-0.0000"	Feet and decimal inches
Architectural	0'-1 1/2"	125'-0"	Feet and fractional inches
Fractional	1 1/2	1500	Whole numbers and fractions

Notice that when you change the unit type, the co-ordinate display on the status bar changes to show co-ordinates using the current unit type. Changing the unit type also affects the way distances, areas and volumes are reported when using the appropriate inquiry command.

For the most part you should not need to change the unit type. Units such as "Architectural" and "Engineering" are there mainly for AutoCAD users in the USA where Feet and Inches are still in common use.

Angular Units

Looking at the Drawing Units dialogue box again, you will notice that there are also five angular unit types. The default is decimal degrees, but there are other options. The table below shows the effect of the different unit types on two angular unit values. As with the linear units, there are not many circumstances under which you would want to use anything other than the default.

Unit Type	12.5 Angular Units	180 Angular Units	Description
Decimal Degrees	12.500	180.000	Metric units
Deg/Min/Sec	12d30'0"	180d0'0"	Degrees, Minutes and Seconds
Grads	13.889g	200.000g	400 grads = 360 degrees
Radians	0.218r	3.142r	2 Pi radians = 360 degrees
Surveyor	N 77d30'0" E	W	Compass bearings

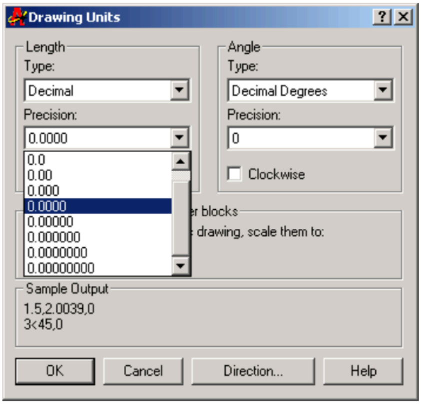
AutoCAD also allows you to control the direction in which angular units are measured and the position of the start angle. By default, AutoCAD starts with the zero angle at the 3 o'clock position (East) with angles increasing in an anti-clockwise direction. For the most part this does

not present any problems once you get used to the idea. However, there are specific situations where it may be desirable to have the zero angle at the 12 o'clock position (North) and angles increasing in a clockwise direction. For example, if you are working on a surveyors drawing or a map base, this latter situation enables you to specify angles with respect to North. To change the direction of angular measurement, use the Clockwise check box in the Angle section of the Drawing Units dialogue box. When this box is checked, positive angles are measured in a clockwise direction, when it is not checked (the default), positive angles are measured in an anti-clockwise direction.

To change the start angle, click on the "Direction…" button in the Drawing Units dialogue box. The Direction Control dialogue box appears. You can set the Base Angle to any of the circle quadrants by clicking on the appropriate radio button or you can set it to a specific angle with the "Other" option. You can enter a specific angle into the edit box or you can pick an angle using the Pick an angle button. The ability to specify an "Other" or user angle can be useful if, for example, your drawing is not oriented to North but where you still want angular measurements to be made with respect to North. To change the direction of angular measurement, simply click on the appropriate radio button.

Unit Precision

The Drawing Units dialogue box can also be used to set the precision of linear and angular units. By default, AutoCAD sets the linear unit precision to four places of decimal, so distances appear in the form 0.0000. Angular unit precision is set to whole degrees only.



To change the precision with which linear and angular values are displayed, simply click the down arrow against the appropriate drop-down list (see illustration on the left) and select the number of decimal places required. The default setting of four decimal places is usually adequate for linear units. It is, however, often necessary to change the precision for angular units. Working in whole degrees does not usually give an adequate level of detail for many drawing functions. However, you do not need to change the precision of either linear or angular units unless you have a specific reason for doing so.

Changing the unit precision does not make your drawing more accurate, it just means that the co-ordinate display on the status bar and the results from the various inquiry commands will be displayed with a higher degree of precision. The accuracy of your drawing will be determined by the values you enter for the size of objects when you draw and edit them and by the correct use of the various object snaps and drawing aids.

Basic AutoCAD Commands

These commands may be used at any time, but are mainly called upon more during the beginning of the drawing process.

Commands for Setting the Drawing Environment

These commands may be used at any time, but are mainly called upon more during the beginning of the drawing process.

- Units: Specifies the display format and precision. This command dictates whether the units are displayed as decimals or as a fraction, as well as how many decimal places the number will go. Also specifies how angles will be represented, either as decimal degrees, radians, etc., as well as how angles will be measured, i.e. clockwise vs. counterclockwise.

- Limits: Sets limits to the boundary and size of the current drawing. The user must specify the lower left-hand corner and the upper right-hand corner. This setting may be turned on or off – when on, points may not be specified outside the currently set limits.
- Qtext: Quick text – may be turned on or off, like the Limits command. When switched on, text is displayed in rectangles, showing the extent of the text. When off, just the text is displayed, without the rectangle.
- Viewres: Performs two functions: first, it lets the user disable fast zoom, which makes sense when conforming a more modern version of AutoCAD to an earlier one that lacks the fast zoom capability. Second, Viewres allows the user to control the smoothness and speed of circles and arcs drawn in the display. The user does this by choosing the number of sides circles will have. It is recommended this value be set at 2000.
- Blipmode: Blips are small crosses used to mark screen positions that the user has pointed to. They can be useful reference points, but too many of them can crowd the screen, making it difficult for the user to see. Engaging Blipmode allows the small crosses to stay up in the wake of the pointer, and turning it off makes them disappear. Blips are not part of the final drawing and are removed when the drawing is complete.
- Fill: When this command is engaged, solids, traces, wide polylines, and donuts that are drawn are then filled in with color, as opposed to just being outlines. This command does not affect the drawing's plotted output, and, when using this command, there is a trade-off between regeneration time and the image's quality.
- Status: This command simply lists, on the Text Screen, the current drawing environment, the modes, and statuses that are engaged for this drawing.

Commands for Drawing Entities

Once the drawing environment has been set, these commands are used to actually draw the entities.

- Line: Allows for the sequential drawing of one or more straight lines. Once engaged, this command elicits a prompt of "From point:", at which point the user specifies a starting point for a line, or they may press RETURN, which starts the line at the end of the previous line or arc that was drawn. Next, the prompt "To point:" is displayed, allowing the user to specify a sequence of points to which the line will extend. They may also type the letter C to close the polygon, or the letter U to undo the previous line segment, or they may simply press RETURN to complete the command.
- Point: Draws a single marker/point, which is, by default, a single dot, but may be changed to something else if desired.
- Circle: Draws a circle by letting the user specify the center point, then dictating the circle's size by entering a value for either the circle's radius or its diameter. Another option to creating a circle with this command is to specify three points on the circle's circumference, two end-points of its diameter, or its radius along with two other lines or circles to which the new circle will be tangential.
- Arc: This command draws arcs, and, like circles, may be dictated in one of several ways. The various methods for constructing arcs with this command are as follows: (1) specify three different points, (2) starting point, center, and end point, (3) starting point, center, and included angle, (4) starting point, center, and length of chord, (5) starting point, ending point, and radius, (5) starting point, ending point, and included angle, (6) starting point, ending point, and starting direction, and finally, (7) starting point and direction of previous line or arc, plus ending point.
- Ellipse: Ellipses are constructed by specifying the two end points of one of the major or minor axes, followed by a distance value defining half the length of the other axis.

- Pline: Draws 2D polylines, which are continuous sequences of straight lines and/or arc segments with varying line length, dictated by the user. You are able to close a polyline to form a polygon. It also may be helpful to know that polylines may be exploded into separate line and arc entities if necessary.
- Polygon: Draws regular polygons by entering the number of edges, then specifying the shape's center and radius, or by locating the endpoints of any of its edges. Once created, polygons are to be treated as closed polylines.
- Hatch: Within one or many closed boundaries, a cross-hatch pattern may be created with the Hatch command. The boundaries must be well-defined, otherwise, the cross-hatch may leak out unexpectedly. The user may select pre-determined patterns supplied by AutoCAD, or they may enter their own and add them to the set that's already there.
- Bhatch: A newer command in the AutoCAD quiver, it helps the user to better use the previously discussed Hatch command. It supports boundary hatching, allowing the user to pick a point that is adjacent to the boundary they wanted, and this new command lets AutoCAD search for the nearest entity, then constructs a closed boundary by tracing in a counterclockwise fashion to look for intersection points as well as connecting lines or arcs. Bhatch is convenient in that it allows the user to preview adjustments without having to start over each time.
- Dtext: Allows you to draw text dynamically, changing text height and rotation, allowing it to be moved, centered, stretched between two points, aligned, overscored, underscored, have symbols added, fonts changed, etc.

Utility Commands

These are some basic and useful commands that may be used more than other more specific commands.

- Redraw: Refreshes the program and re-displays the graphics on the screen, but without extraneous graphics, such as blips, that may have been left behind from earlier operations.
- Save: Saves all current changes and drawings to be saved to the disk. As with any important project done on a computer, it's best to get into the habit of saving regularly, especially during long drawing sessions, to prevent any work from being lost.
- End: Finishes the current session, saves the work, and takes you back to AutoCAD's main menu.
- Quit: Finishes the current session, but does NOT save the changes that were made to the current drawing, then returns you to the main menu.

Drawing and Manipulation Commands

These commands alter your drawings, allowing you to enlarge and reduce views, maintain graphic accuracy, and manipulate space and viewports, among other things.

Moving Around the Drawing Area

- Zoom Scale: Allows the user to enter a magnification or reduction factor. Numbers less than 1 will reduce the drawing, those greater than 1 will expand it. The Zoom Scale amount is applied to the entire drawing, and doesn't change the actual size of the entities, simply the magnification.
- Zoom Extents: Commands AutoCAD to display all of the current drawing's graphics, using the largest possible image, not necessarily extending it to the user-defined limits.
- Zoom All: Displays the drawing to the drawing limits.
- Zoom Window: Prompts the user to define which part of the drawing is to be magnified, by defining the lower left-hand and upper right-hand corners of the box to be zoomed in on.

- Zoom Center: The user enters a point, which the program uses as the center of an area to be magnified, then they enter a value to be applied to the new, magnified image's height.
- Zoom Left Corner: Like Zoom Center, the specified point is the lower left-hand corner of the new display.
- Zoom Previous: Commands the program to revert back to the prior view displayed. Up to five views may be stacked up for comparison.
- Zoom Dynamic: The most powerful of the Zoom options, it allows for quick movement around the drawing.
- Pan: Permits panning across the current drawing without changing the scale.
- Vpoint: Establishes a viewpoint from anywhere in space, which may be entered as a 3D point, a spherical point, or dynamic (simply press RETURN instead of entering a specific point).
- Dview: Provides a dynamic tool for viewing an object in 3D as either a parallel projection or a perspective. Using a camera along with target concept, AutoCAD is able to manipulate the viewing position, direction of view, focal length, and viewing distance.
- Plan: Puts the user back in plan view when done working in 3D.

Model Space, Paper Space, Viewports

- Tilemode: Switching Tilemode to off (setting the value to 0), turns AutoCAD to paper space. Setting the value to 1, AutoCAD switches to model space.
- Vports: Only available when Tilemode is on, it allows the user to establish up to 16 viewports on the screen, so that each one holds a different view of the drawing. You are able to work in only one viewport at a time, but may easily move among the different ones.
- View: Saves the current view under a name the user specifies, or restores a previously saved view, and may be used in model or paper space,
- Mview: Used when Tilemode is off, Mview creates and defines various viewports' characteristics while in paper space. They may be turned on and off and linked with views that have been previously saved with the previous View command.
- Mspace: Also used when Tilemode is off, this command allows the user to switch to model space, then edit their drawing inside a paper space viewport.
- Pspace: With Tilemode off and model space active, the user may switch back to paper space and edit graphics.

Drawing Tools

- UCS: The User Coordinate System (UCS) is set up to be positioned and oriented anywhere in 3D space. After the UCS is implemented, the previous 2D drawing is now done in the X-Y plane of the new UCS. The user is now able to easily draw anywhere in space, and also aids when drawing in 2D.
- Snap: Sets up a grid that is both invisible and orthogonal, square or rectangular, which all points entered with the mouse may be locked onto.
- Grid: This command sets up a visible grid of white dots that is used for referencing purposes.
- Axis: Similar to the Grid command, except the white dots are replaced with two intersecting axes with tick marks.
- Ortho: When turned on, Ortho mode makes all lines drawn with the mouse parallel to the axes.
- Osnap: (Object Snap) In Osnap's "Running Mode", it allows points to be precisely located on reference points of existing objects. They may be overridden by selecting different object snap modes for a specific entry.
- Aperture: Sets the size of the Osnap target box, where values ranging from 1-50 screen units are valid.

Deletion Commands

Only two commands are in this group, used to delete objects and entities.

- Erase: This removes a selected group of entities, which may be entered before or after the command itself is entered.
- Oops: Restores the most recently deleted object group from using the previous Erase command. This command may not be repeated, as it only restores one group of deleted objects.

Transformation Commands

The following group of commands allows the user to select a group of objects that need to be transformed in one way or another.

- Move: The user may dictate the direction and length of a move of specified objects by indicating two points which define a vector between the objects.
- Copy: Similar to the Move command, Copy does not affect the original group of objects, with the copied objects being completely independent of the original objects.
- Rotate: Providing a specific base point and angle, the user may rotate an object of their choosing with this command. Negative angles will provide a clockwise rotation, while a positive angle gives a counterclockwise rotation.
- Scale: Enlarge or shrink a selected group of objects by selecting a base point for the scaling as well as applying a factor for which to scale.
- Mirror: This command produces a mirror copy of a selected object group by specifying the two ends of the mirror line. Then, the mirrored objects may be deleted or kept, depending upon the user's preference.
- Stretch: Allows the user to move a portion of a drawing while retaining its connections to other parts of the drawing, thus stretching it out. Blocks, Hatch patterns, and Text entities may NOT be stretched.
- Array: Produces multiple copies of selected objects that are arranged in a rectangular or circular pattern.
- Offset: Constructs a new entity parallel to an existing one. This could be a single line, polyline, arc, circle, or curve.

Error Recovery Commands

These commands bring back errors made during the editing process.

- U: The U command undoes the most recent drawing or editing and may be used repeatedly, all the way back to the beginning of the session.
- Redo: This command is used immediately after an error to redo what was undone.
- Undo: This is like the U command but a bit more complicated. It is able to undo several commands at once, allows the user to set mark points and later undo back to those points, and to group operations together and undo them simultaneously.

Commands that Change Existing Entities

These commands allow different editing changes to be made to existing entities.

- Change: This command covers a lot of ground, with two basic main capabilities: first, use this command to change properties that all entities possess, for example, layer name, color, line type. Second, change the geometry and attributes of specific types of entities.
- Pedit: This is the command used to make changes to polylines, such as width, taper, closing an opening, breaking one into two, moving, adding, and deleting vertices, etc.

- Break: This command splits an already existing line, arc, circle, or polyline into two separate parts, producing an erased portion in between them.
- Trim: This command trims parts of certain objects in a drawing in order to finish them precisely at some cutting edge (or edges) that are established by one or more other objects. One or several lines, arcs, circles or polylines must first be identified to serve as cutting edges, which may be selected by any of the methods available. Next, pointing is used to select the parts of the objects that are to be trimmed. Many objects may be selected in this way for trimming, including ones that had been specified as cutting edges.
- Extend: Complementing the Trim command, Extend operates similarly, but the selected lines are extended to end exactly at the specified boundary edges.
- Fillet: This command connects two existing lines, circles, or arcs by adding an arc with a specific radius (a fillet), and allows the user to change the current default radius prior to filleting.
- Chamfer: Like the Fillet command, this one chamfers, or cuts away, as in carpentry, corners with a straight edge.
- Divide: Divides an object into a specified number equal parts, from 2 to 32,767 parts.
- Measure: This measures an object, from one end to the other.

Enquiry Commands

This final group of AutoCAD commands are used to obtain information a drawing‘s object‘s position and nature.

- List: Lists stored information about any selected entities found within the current drawing.
- Dist: Calculates and displays the distance and angle between two points in a drawing.
- Area: Calculates and displays any region’s area and perimeter, as long as it is defined by a sequence of specified points on the drawing, as long as they form a closed polygon.

Note: The command are to be used in the command line, this commands may be accessed in respective tool bars as tools

A. Tools

Drawing tools – on the drawing tool bar or Drawing menu

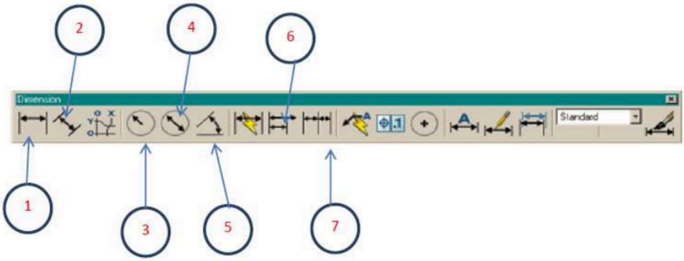
Command	Keystroke	Icon	Menu	Result
Line	Line / L		Draw > Line	Draw a straight line segment from one point to the next
Circle	Circle / C		Draw > Circle > Center, Radius	Draws a circle based on a center point and radius.
Rectangle	**RECTANGLE /** **REC**		Draw > Rectangle	Draws a rectangle after you enter one corner and then the second.

Modifying tools – on the Modifying toolbar or modifying menu

Command	Keystroke	Icon	Menu	Result
Move	**Move / M**		Modify > Move	Moves an object or objects
Copy	**Copy / CP**		Modify > Copy	Copies object(s) once or multiple times
Stretch	**Stretch / S**		Modify > Stretch	Stretches an object after you have selected a portion of it
Mirror	**Mirror / MI**		Modify > Mirror	Creates a mirror image of an object or selection set
Trim	**TRIM / TR**		Modify > Trim	Trims objects to a selected cutting edge.
Extend	**EXTEND / EX**		Modify > Extend	Extends objects to a selected boundary edge.
Offset	**OFFSET / O**		Modify > Offset	Offsets an object (parallel) by a set distance.

Rotate	Rotate / RO		Modify > Rotate	Rotates objects to a certain angle
Fillet	Fillet / F		Modify > Fillet	Creates a round corner between two lines
Chamfer	Chamfer / CHA		Modify > Chamfer	Creates an angled corner between two lines
Array	Array / AR		Modify > Array	Creates a repeating pattern of the selected objects

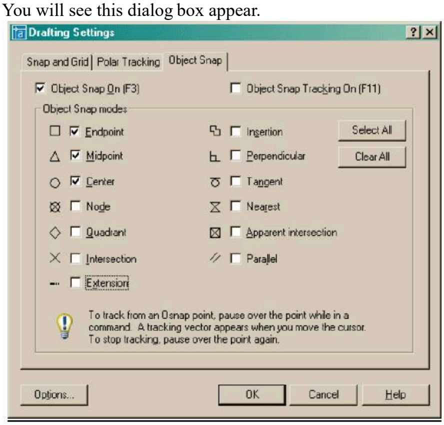
Dimensioning tools – on the Dimensioning toolbar or Dimensioning menu



1. Linear dimensions are used for dimensioning either horizontal or vertical distances.
2. Aligned dimensions will measure the actual length of an angled line parallel.
3. Radius dimensions will give you the radius of either arcs or circles.
4. Diameter dimensions are used on circles.
5. Angular dimensions will measure the angle between two lines that you pick.
6. Baseline dimensions are a special type – elements with a common baseline
7. Continued dimensions are a special type – elements that dimension from a previous dimension

B. Object Snaps

Suppose you want to draw a line from the center of the circle to the middle of the vertical line you extended earlier. AutoCAD has a feature that makes this very easy. These are the Object Snaps (or Osnaps "Oh-Snaps"). Type OS <ENTER>
You will see this dialog box appear.



Endpoint - snaps to either the beginning or the end of an object such as a line - END
Midpoint - snaps to the exact middle of a line or an arc - MID
Center - snaps to the center-point of a circle or arc - CEN
Node - snaps to 'nodes' (not covered in this course) - NOD
Quadrant - snaps to any of the four quadrants of a circle - QUA
Intersection - snaps to the point where two object cross - INT
Extension - Snaps to the phantom extension of an arc or line - EXT
Insertion - snaps to the insertion point of an object (such as a block or text) - INS
Perpendicular - will snap so that the result is perpendicular to line selected - PER
Tangent - snaps to create a line tangent to a circle or arc - TAN
Nearest - will find the closest point an object and snap to that point - NEA
Parallel -Snaps parallel to a specified line - PAR
None - temporarily turns off all Osnaps. (Pressing your F3 Key is quicker) - NON
Osnap settings - opens the Osnap dialog box.
Temporary Tracking - Creates a temporary tracking point (see Object Tracking).
From - Allows you to select a point, and then denote a new location 'from' that point using relative co-ordinates. This can save you the time of drawing (and erasing) construction lines.

C. Layers

A LAYER is a CAD feature used to organize drawings (organize drawing to sections for else of management). On format menu, click Layers then use the Layer manager window that appears to organize the drawing
i.e. Create different layers(from layers toolbar) and properties and draw on respective layers.

D. Blocks

A BLOCK is a collection of simple entities (lines, arcs, circles, text, etc.) that form a more complex entity that normally represents an object in the real world, e.g. a door, a chair, a window, a computer. Select the various drawing entities and on drawing menu, choose block to create a single entity of the drawing.

E. Hatch

Hatching in AutoCAD is a way of filling in areas of your drawing with a preformatted pattern to represent certain materials. It is usually used in sectional views.
There are two types of hatching you can use.

Command	Keystroke	Icon	Menu	Result
Boundary Hatch	**Bhatch / H**		Draw > Hatch	Covers an area with a predefined pattern
Hatch Edit	**HatchEdit / HE**		Modify > Object > Hatch...	Edits an existing Hatch

Steps
On drawing menu, choose hatch then cover an area with preformatted pattern.

3-D CAD Concept

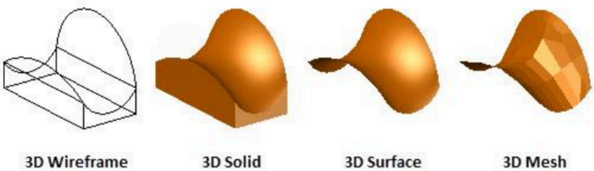
2-D	A concept of displaying real-world objects on a flat surface showing only height and width. This system uses only the X and Y axes.
3-D	A way of displaying real-world object in a more natural way by adding depth to the height and width. This system uses the X Y and Z axes.

Terminology

Elevation - The difference between an object being at zero on the Z-axis and the height that it is above zero.
Extrude - The extrude command raises the shape of a 2D outline into a 3D solid. For example, a circle would be extruded into a cylinder.
Face - The simplest true 3-D surface.
Facet - A three or four sided polygon that represents a piece (or section) of a 3-D surface.
Hidden line removal - A way of hiding lines that would not be visible if you were viewing the actual object you have drawn in AutoCAD. (Command: HIDE)
Region - A 2-D area consisting of lines, arcs, etc.
Thickness - A property of lines and other objects that give them a 3-D like appearance.
Z-Axis - The third axis that defines the depth.
UCS - The User Co-ordinate System. This is defined by the user.
WCS World Co-ordinate System is define as the default CAD origin

Types of 3D Models

Several types of 3D modeling are available in AutoCAD. Each of these 3D modeling technologies offer a different set of capabilities.



Wireframe Models:

1) Consists of lines, arcs, and curves that define the object
2) There are no surfaces. Objects appear as outlines only
3) Time consuming to make it part

Surface Models

1) Represent a thin layer or shell of the shape of an object
2) Made up of edges and surfaces
3) Surfaces models are created by using sweeping, lofting, or revolving 2D lines or arcs

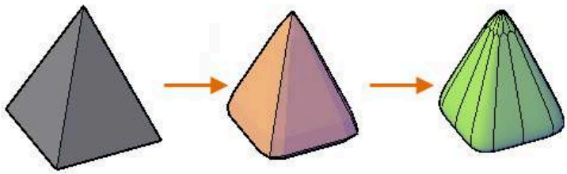
Solid Models

1) Are made up of surfaces and the volume inside
2) Solid models have properties of mass, volume, center of gravity, and moments-of-inertia
3) You can easily create 3D primitives such as boxes, cones, cylinders, and spheres or create 3D models from sweeping, lofting, or extruding 2D closed objects or profiles.

Mesh Models
1) Consists of vertices, edges, and faces that use triangles and quads to define the 3D shape
2) Meshes have to mass properties
3) Meshes allow greater ability to manipulate and deform surfaces
4) You can covert meshes into solid models

- Wireframe modeling is useful for initial design iterations and as reference geometry, serving as a 3D framework for subsequent modeling or modification.
- Solid modeling is efficient to use, easy to combine primitives and extruded profiles, and offers mass properties and sectioning capabilities.
- Surface modeling offers fine control over curved surfaces for precise manipulation and analysis.
- Mesh modeling provides freeform sculpting, creasing, and smoothing capabilities.

A 3D model can include combinations of these technologies, and you can convert between them. For example, you can convert a primitive 3D solid pyramid to a 3D mesh to perform mesh smoothing. You can then convert the mesh to a 3D surface or back to a 3D solid to take advantage of their respective modeling features.



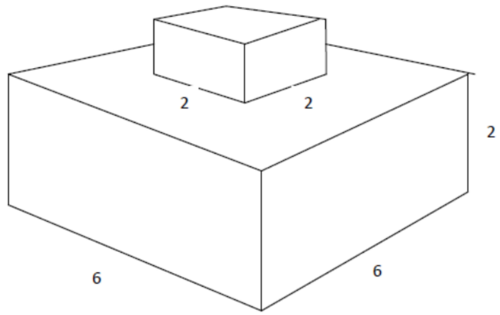
3D AutoCAD (Advantages of 3D models)

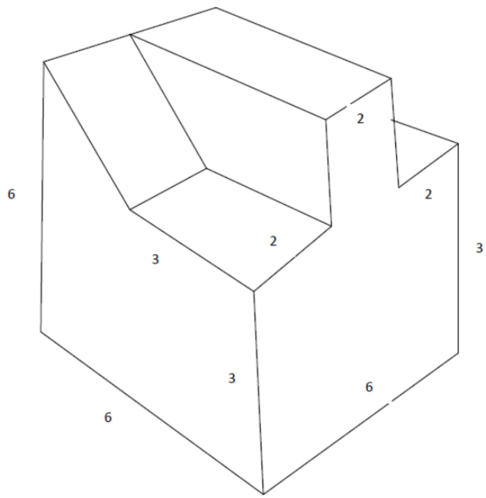
- Models can be rotated or viewed from any position
- You can generate section and auxiliary views
- You can remove hidden lines and do realistic shading
- You can run part interference checks for engineering analysis
- You can add lighting and realistic rendering to models
- You can create animations
- You can extract manufacturing data for making the part

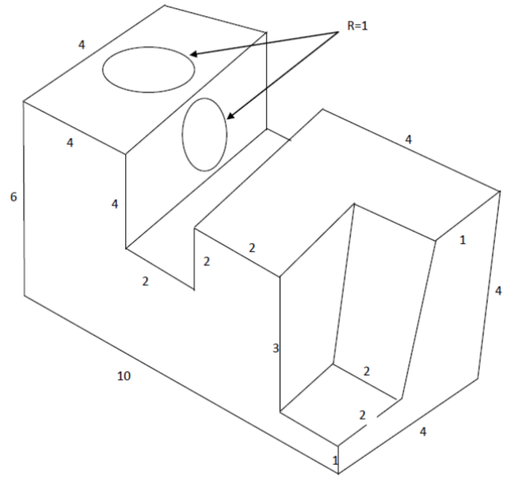
Developing a 3-D object

To start developing a 3D object, we have two options

1) Develop an object in 2D (circle, rectangle, polygon, but not lines) then activate the property of the object find thickness property and add some value for thickness (extrude)
2) Go to view menu – 3D views and choose either of the isometric views and start developing the 3-D object along all its axis (X, Y, Z)







Projection

When three-dimensional graphics are displayed on a flat computer screen or printed on paper, they must be projected onto the viewing plane. A projection is a way of converting positions in 3D space into locations in the 2D viewing plane. Engineering supports two types of projections-parallel and perspective-for each view.

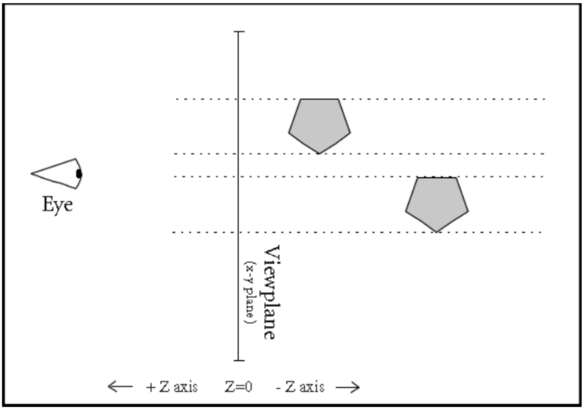
Types of Projections

The projections are classified according to the method of taking the projection on the plane. Classifications of projection are: Perspective and Parallel

Parallel Projections

A parallel projection projects objects in 3D space onto the 2D viewing plane along parallel rays. The figure below shows a parallel projection; note that two objects that are the same size but at different locations still appear to be the same size when projected onto the viewplane.

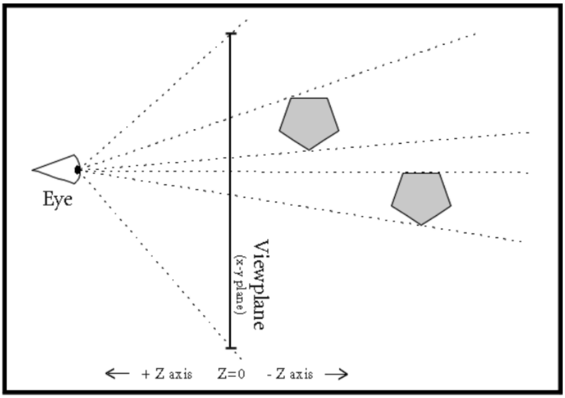
Figure : In a Parallel Projection, Rays Do Not Converge at the Eye



Perspective Projections

A perspective projection projects objects in 3D space onto the 2D viewing plane along rays that converge at the eye position. The figure below shows a perspective projection; note that objects that are farther from the eye appear smaller when projected onto the viewplane.

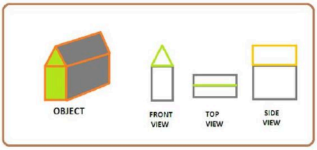
Figure: In a Perspective Projection, Rays Converge at the Eye



Parallel projection

When you look at a machine, you see that the machine has planes based on parallel lines. For example, a lathe machine has a rectangular base, which is a combination of parallel lines. If parallel lines are drawn to represent the parallel lines actually present on the machine, we call it a parallel projection. This type of projection is widely used by draftsman and architects to make blueprints and schematics.

1. Orthographic Projection



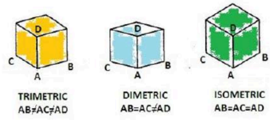
Orthographic projection is the most common parallel projection due to its simplicity. It is represented using three views; front view, side view and top view. In graphical terms, you need to draw the object in XY plane, YZ plane, and ZX plane, separately.
To draw the front view you just imagine how the object looks at front. Don't think about slope of the object, just draw whatever it looks. Similarly, the top and side view of the object can be drawn (refer the image).

Through this projection, accurate measurement can be obtained because all the views are drawn using same scale. However, it doesn't provide a realistic projection of a 3-D model; besides, it requires multiple views to feel the object design.

The orthographic projection can be drawn two ways, first angle projection and third angle projection. Both of them have differences only in the positioning of various views. In first angle projection, the front view is placed onto first quadrant; top and side view are placed onto the forth and second quadrants, respectively. In the third angle projection, the third quadrant is used for front view; whereas top and side view are placed onto the second and forth quadrants.

2. Axonometric Projection

Axonometric projection is a type of parallel projection. Drawing the object in this method is somewhat complex because it requires only one image to draw the 3-D structure of the object onto a plane paper. Suppose you are using a projector and an object is placed in front of projector lines. Now, you can see the image of 3-D object onto a 2-D plane just behind the object. This projection is nothing but an axonometric projection.



Axonometric Projection

Axonometric projection is classified into three categories depending upon orientation of the object.

The first and most convenient type is isometric. In this type of projection, angles between the three axes are equal. As the diagram says, if we project a cube onto a 2-D surface, you see all the three sides AB, AC, and AD are equal.

The second type is diametric projection, in which only two angles between the axes are equal. You can see the diagram in which only two sides AB and AC are equal.

The third is trimetric projection that can be drawn using three axes having different angles between them. It's the most common type of axonometric projection and the object can be placed anywhere with respect to the observer.

All these methods are used for furniture and structural design. Axonometric projection is good for rectangular structures rather than curved objects.

3. Oblique Projection

Oblique projection is comparatively easier than axonometric projection. It requires only one image and traditional equipments to draw any object. In this method, first you need to draw the front view or side view and then draw rest of the object with respect to it.

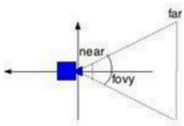
- Draw the front or side view of the object just like in orthographic projection.
- The measurements drawn backwards must be in the scale equal to half of the actual measurement.
- The lines drawn backwards should have the angle in between 30^0 to 45^0; however it is convenient if you use 45^0 angles.

Oblique projection is further divided into two types based on scaling.

The first one is cavalier projection, in which the length in X and Z axis are drawn in scale 1:1 and there is no required scaling for the length of the Y-axis. It is very easy to draw and often used for drawing an object when you can use only your hands such as drawing a cube on the blackboard. The most common use of this type of projection is in military fortification.

The other type is cabinet projection. Unlike cavalier projection, this method uses the 2:1scale to draw lengths in X and Z direction, respectively. Besides, lengths in Y-axis must also be scaled properly. This projection is very useful in furniture industries.

Perspective Projection



Perspective projection doesn‘t use parallel lines to project an object, instead it is a projection along the lines emerging from a single point. Due to this, the nearer part appears bigger than distant part. This is much like the working of our eyes in respect to depth perception. For example, when we see a railway line, it appears converging towards a single point called the vanishing point. View of the object feels more realistic using this projection.

You can use any type of projection for drawing an object onto plane surface. However, it requires good imagination power so as the object would become more realistic. The scaling must be done properly to avoid distortion. Always use clips to place the drawing sheet rigidly on the desk. Furthermore, use clean equipment to draw a neat and clean image.