E-CONTENT OF ENGINEERING GRAPHICS FOR THE STUDENTS OF COMPUTER, ELECTRICAL & MECHANICAL ENGINEERING

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CHAPTER-6

GRAPHICS USING CAD

MEANING AND REQUIREMENT OF COMPUTER GRAPHICS

Computer graphics is the discipline of generating images with the help of computers. Computer graphics is responsible for displaying art and image data effectively and meaningfully to the consumer. It is also used for processing image and data received from the physical world.

COMPUTER-AIDED DESIGN (**CAD**)

It is the use of computers to aid in the creation, modification, analysis, or optimization of a design. CAD software is used to

- increase the productivity of the designer
- improve the quality of design
- improve communications through documentation
- to create a database for manufacturing

SCREEN STRUCTURE AND TOOLBARS

Toolbars contain buttons that start commands.

The **Quick Access toolbar** at the top of the application window is displayed by default. It contains frequently used commands as well as standard commands for opening and saving your file.
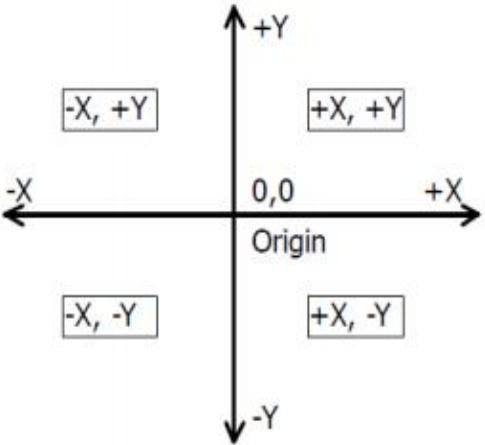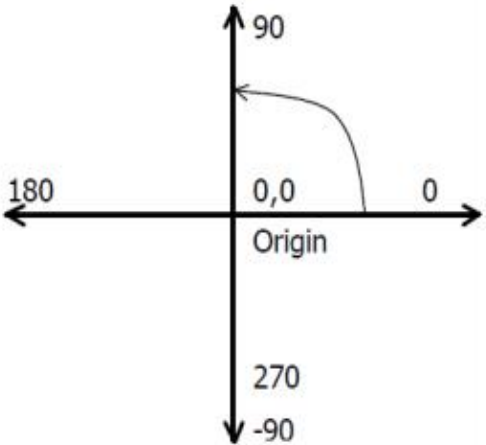
A toolbar is of two types

1. **FLOATING TOOLBAR** A floating toolbar can be displayed anywhere on screen, and it can be drag it to a new location, resize it, or dock it.

2. **DOCKED TOOLBAR** A docked toolbar can be attached to any edge of the drawing area.

AUTOCAD COORDINATE SYSTEMS

There are two different coordinate systems used in AutoCAD.

1. **Cartesian (rectangular) Coordinate System** It uses the "x" distance (left/right) and the "y" distance (up/down).

2. **Polar Coordinate System** It uses the distance between points (length or magnitude) and the angle (direction or vector).

Cartesian Coordinate System	Polar Coordinate System
Locations are measured along the x axis (horizontal) and y axis (vertical).	Locations are measured as distance and angle in degrees, with 360 degrees in all.
+Y -X, +Y +X, +Y -X 0,0 +X Origin -X, -Y +X, -Y -Y	90 180 0,0 0 Origin 270 -90
Absolute coordinates are measured from the origin or 0,0 point.	
Relative coordinates are measured from the last defined point.	

DRAWING LIMITS

It 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.

UNITS

It 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.

COMMANDS FOR DRAWING ENTITIES

Line: It allows 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: It draws a single marker/point, which is, by default, a single dot, but may be changed to something else if desired.

Circle: It 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,
(6) Starting point, ending point, and included angle,

(7) Starting point, ending point, and starting direction, and finally,

(8) 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. An ellipse is defined by a *major* (long) axis and a *minor* (short) axis.

Pline: Draws 2D polylines, which are continuous sequences of straight lines and/or arc segments with varying line length, dictated by the user.

Polygon: It 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, and 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, over scored, underscored, have symbols added, fonts changed, etc.

PRACTICE OF LINE COMMAND

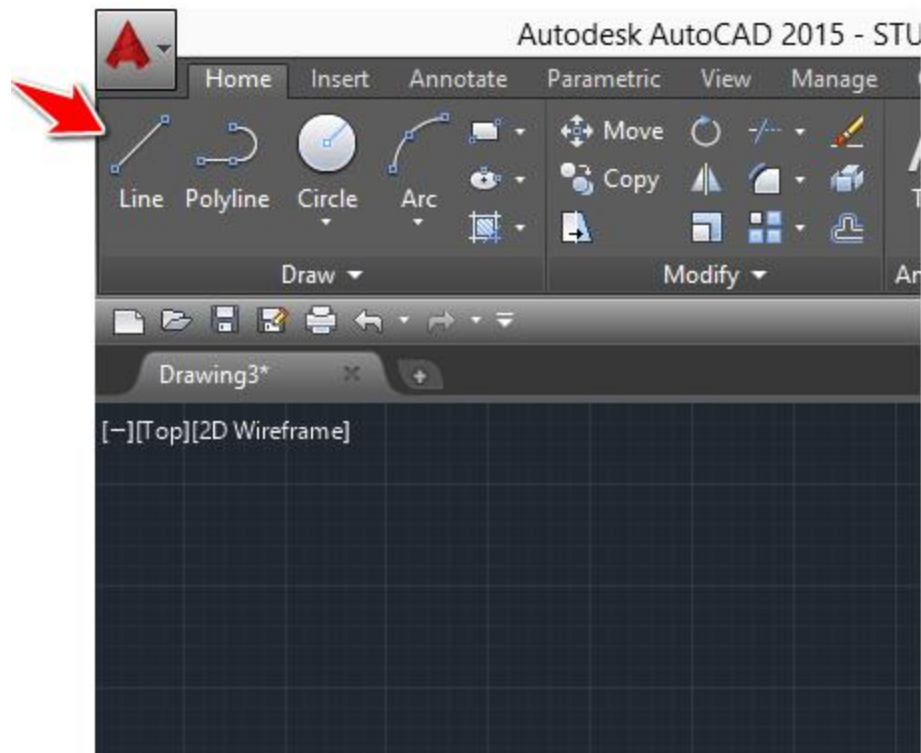To draw a straight line in AutoCAD:

1. Activate the LINE command

2. Click on the starting (Do not hold your click)

3. move your mouse

4. Click to indicate the ending point (Do not hold the click)

5. Press on the ESC key on your keyboard to end the process

For Example

To draw a vertical line of 50 units of length

1 – Click on the **LINE** icon

2 – Click once in the **drawing area**

3 – Activate the **ORTHO** mode if it is not activated

4 – Move the cursor away from the first point in the desired direction. For drawing a vertical line, you can only go either upward or downward

5 – Type **50** on your keyboard and hit ENTER on your keyboard twice

To draw a horizontal line, just change the direction in step 4 to either **left** or **right**.

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 drawings, allows enlarging and reducing views, maintaining graphic accuracy, and manipulating 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, and 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

- **Tile mode:** Switching Tile mode to off (setting the value to 0), turns AutoCAD to paper space. Setting the value to 1, AutoCAD switches to model space.

- **V ports:** Only available when Tile mode 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. It allows 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,

- **M view:** Used when Tile mode is off, M view 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.

- **M space:** Also used when Tile mode is off, this command allows the user to switch to model space, and then edit their drawing inside a paper space viewport.

- **P space:** With Tile mode 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.

- **O snap:** (Object Snap) In O snap'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 O snap 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, poly line, 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 poly line 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 part, from 2 to 32,767 parts.

- **Measure:** This measures an object, from one end to the other.

ENQUIRY COMMANDS

These 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.

Poly line is basically a composite curve which is a combination of linear and arc segments in AutoCAD.

Grid is a rectangular pattern of lines over the drawing area. Using the grid is similar to placing a sheet of grid paper under a drawing. The grid helps you align

and visualize the distance between objects. The grid does not appear in the plotted drawing.

Snap restricts the movement of the crosshairs cursor to an interval that you define. When snap mode is on, the cursor adheres or "snaps" to an invisible grid. Snap mode is useful for specifying precise points with the cursor.

Ortho is short for *orthogonal*, which means either vertical or horizontal. Like the other options on the status bar, Ortho is not really a command; it is a drawing *mode* which can either be turned on or off. Ortho mode can be *toggled* on or off in one of three ways. The quickest way is just to click on the ORTHO button on the status bar. The appearance of the button tells you whether Ortho is currently turned on or turned off. When Ortho is turned on, the ORTHO button appears pressed in.

CHAPTER-7

COMMON SYMBOLS AND CONVENTIONS USED IN ENGINEERING
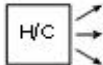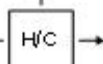
Engineering drawing abbreviations and symbols are used to communicate and detail the characteristics of an engineering drawing

ELECTRICAL ENGINEERING SYMBOLS

Name	Electrical Symbol	Function Description
Ground		A connection to earth. Used for zero potential reference and electrical shock protection.
Chassis		Connected to the chassis of the circuit.
Battery		Supplies electrical energy.
Resister		A resistor restricts the flow of current
Capacitor		A capacitor stores electric charge.
Accumulator		Accumulators are designed to increase or relieve pressure in the system.
Antenna		An antenna is a radio antenna that can be made of a simple wire, with a center-fed driven element.
Loop Antenna		A loop antenna is a radio antenna
Crystal		A crystal oscillator creates an electrical signal with a very precise frequency.
Circuit Breaker		A circuit breaker protects an electrical circuit from damage caused by overload or short circuit.
Fuse		A type of sacrificial current protection device.

Inductor		A coil of wire creates a magnetic field when current passes through it.
Explosive Squib		Explosive squib is often used on stage and film to trigger various special effects.
Surge Protectors		Surge protectors protect your electronics from power surges in your electrical system.
Instrument		The wattmeter is an instrument for measuring the electric power in watts of any given circuit.
Delay Element		Delay element provides a specified delay between the actuation of the propellant-actuated devices.
Permanent Magnet		A permanent magnet is a material or object that produces a magnetic field.
Bell		The electric bell is found in a normal house doorbell, and when activated it makes a ringing sound.
Buzzer		An electrical buzzer is similar to the bell, but instead of a single tone or bell sound it makes a constant buzz noise.
Lamp		A transducer converts electrical energy to light, which is used for a lamp providing illumination, for example a car headlamp or torch bulb.
Speaker		A speaker can take digital input and turn it into analogue sound waves.
Oscillator		Produces a repetitive electronic signal, often a sine wave or a square wave.
AC Source		Alternating Current, continually change direction.
DC Source		Direct Current, always flow in one direction.

CIVIL ENGINEERING SANITARY FITTING SYMBOLS

Symbol	Description
	Boiler, flat ends
	Boiler, curved ends
	Boiler, angled ends
	Heating/Cooling coil
	Radiant panel (face)
	Tank, open
	Tank, closed
H/C	Heater/cooler, horizontal
H/C	Heater/cooler, downward

Symbol	Description
	Pump Rotary pump, compressor or fan.
	Pump
	Radiator
	Convector
	Convector, fan
	Radiant panel (plan)
	Radiant panel (plan), double
	Water surface
	Pipe coils
	Pipe coils, fins
	Pipe coils, break
	Pipe coils, break, fins

Symbol	Description
	Sink unit
	Basin
	Toilet
	Bath
	Basin (side)
	Toilet (side)
	Bath (side)
	End view
	Washing machine
	Shower head
	Towel rail

CHAPTER-8

DEVELOPMENT OF SURFACES

The development of surface of an object means the unrolling and unfolding of all surfaces of the object on a plane.

If the surface of a solid is laid out on a plain surface, the shape thus obtained is called the development of that solid.

METHODS OF DEVELOPMENT

Parallel-Line Method

It is used for developing prisms and single curved surfaces like cylinders, in which all the edges/generation of lateral surfaces are parallel in each other.

Radial-line Method

It is employed for pyramids and single curved surfaces like cones in which the apex is taken as centre and the slant edge or generator as radius of its development.

Triangulation Method

It is used for developing transition pieces.

Approximate Method

It is employed for double curved surfaces like spheres, as they are theoretically not possible to develop. The surface of the sphere is developed by approximate method. When the surface is cut by a series of cutting planes, the cut surfaces is called a zone.

CHAPTER-9

DETAILED AND ASSEMBLY DRAWING

A detailed drawing is a production drawing, furnishing complete information for the construction or manufacturing of various parts.

This information may be classified as:

1. Shape description

This refers to the selection of number of views to describe the shape of the part. The part may be drawn in either pictorial or orthographic projection. Sectional views, auxiliary views and enlarged detailed views may be added to the drawing in order to provide a clear image of the part.

2. Size description

Size and location of the shape features are shown by proper dimensioning. The manufacturing process will influence the selection of some dimensions, such as datum feature, tolerances, etc.

3. Specifications

This includes special notes, material, heat treatment, finish, general tolerances and number required.

4. Additional information

Information such as drawing number, scale, method of projection, date, names of the parts, etc., come under additional information which is included in the title block.

WORKING ASSEMBLY DRAWING

It is usually made for simple machines or jobs, consisting of a relatively smaller number of simple parts. All the dimensions and information necessary for the manufacture of a part and for the assembly of the parts are given directly on the assembly drawing. Separate views of specific parts, in enlargement, showing the mating of parts, may also be drawn, in addition to the regular assembly drawings.

CHAPTER-10

SCREW THREADS

Screw threads are used in fasteners such as bolts and screws, and also to provide a linear motion drive which may transmit power. There are several different types of screw thread used for different purposes.

Threads are applied to many devices for various purposes such as

1. It is used to transmit power.

2. It increases the efficiency of the applied effort.

3. It controls movement in a micrometer.

4. It holds parts together as in the case of fastening.
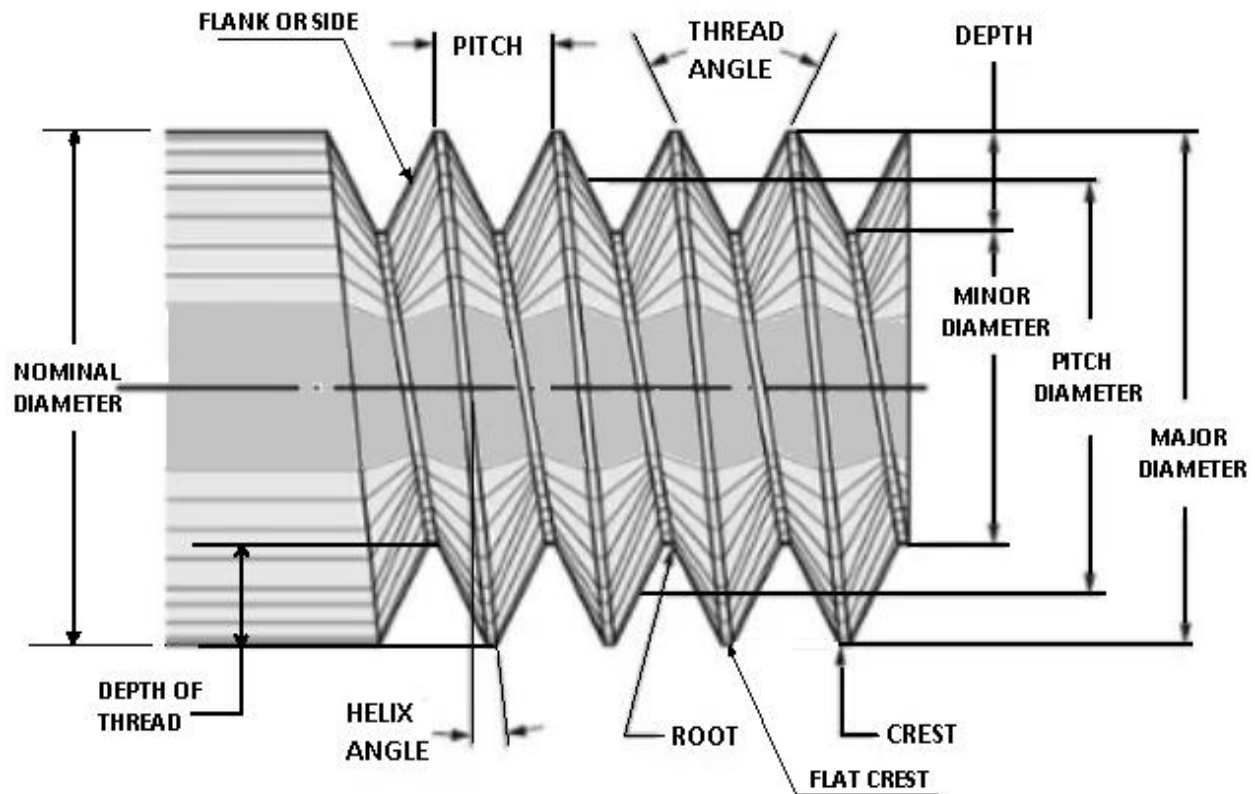
SCREW THREAD TERMINOLOGY

Description of Terms

1. Root

It is the bottom portion of the surface of a thread which joins the sides of the adjacent threads.

2. Crest

It is the top portion of the surface of a thread which joins the sides of the same threads.

3. Flank

It is the surface of a thread that connects the crest with the root.

4. Angle of the Thread

It is the angle included between the sides of the two adjacent threads and measured on an axial plane.

5. Depth of the Thread

It is the distance between the crest and the root of a thread which is measured normal to the axis on an axial plane.

6. Nominal Diameter

It is the diameter of the cylindrical rod on which the threads are cut. This diameter specifies the size of the screw.

7. Major Diameter

It is the diameter of an imaginary coaxial cylinder which bounds the crest of an external thread or the roots of an internal thread.

8. Minor, or Core, or Root Diameter

It is the diameter of an imaginary coaxial cylinder which bounds the roots of an external thread or the crests of an internal thread.

9. Pitch Diameter

It is the diameter of the thread at which an imaginary coaxial cylinder that can be passed so as to cut the thread so that the width will be equal to the width of the groove.

10. Pitch

It is the distance from a point on a screw thread to the corresponding point on the next thread, measured parallel to the axis.

11. Lead

It is the axial distance advanced by a nut for its one full turn over a threaded rod.

On a single start thread, the lead and the pitch are identical.

On a double start thread, the lead is twice the pitch

On a triple start thread, the lead is three times the pitch.

Thus, the lead may also be defined as the product of the pitch and number of starts.

TYPES OF THREAD

V-THREAD
The Bureau of Indian Standards has adopted V-thread profile recommended by the International Organization for Standards, ISO, a metric screw thread for use in our country. These types of screw threads are also known as Unified Thread. It has a 60° thread angle.
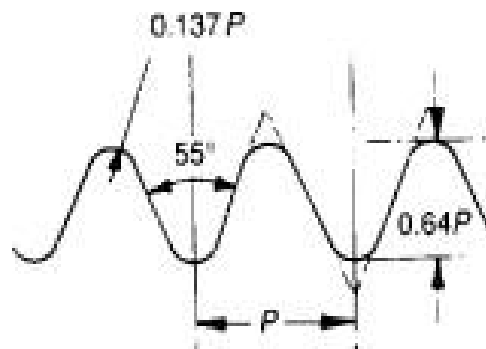
Whitworth thread

SQUARE THREAD

Since these types of screw threads are in the form of a square, it is called **Square thread**. The flanks or the sides of this thread are perpendicular to the axis of the thread. The depth and thickness of the thread are equal to half the pitch.

ACME THREAD

This thread is the modified form of a square thread. Unlike the square thread, it is easier to cut and stronger at the root. The angle of the thread is 29°.

SINGLE AND MULTI-THREADS

In a piece of work, it is possible to have several separate and independent threads running along with it. Accordingly, there are a single threaded screw and multiple or multi-start threaded screw.
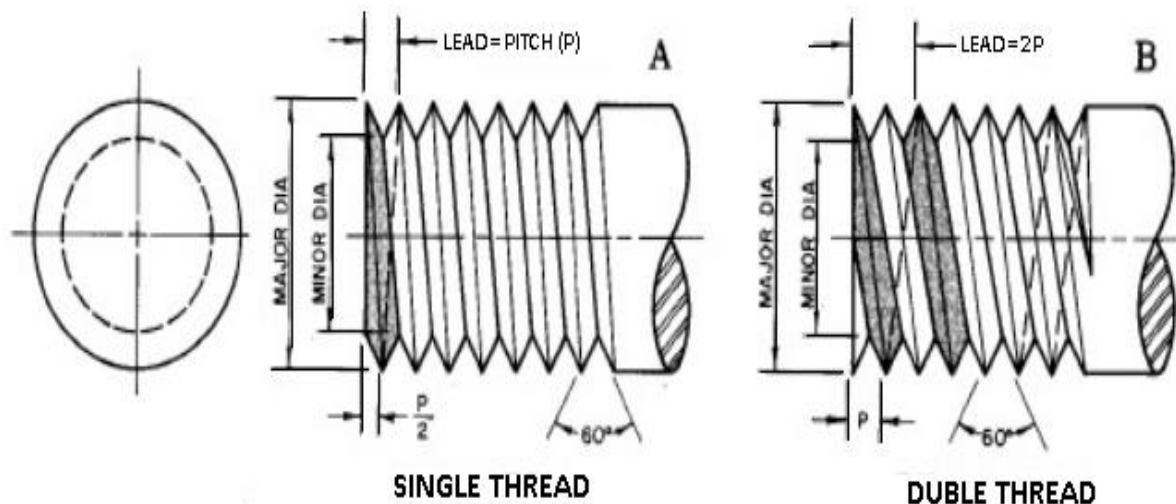
The independent threads are called starts, and we may have single-start, two-start, three-start, etc.

SINGLE THREADED SCREW

For one complete turn round the screw or bolt then there is a movement of one thread the screw is called single threaded screw.

MULTI-START THREADED SCREWS

There is a movement of more than one thread, the screw is called multiple or multi-start threaded screws.

In the case of a three-start thread, for one complete turn, the thread advances three times as far if it were a single thread. Multi-start threads are used in those cases where rapid movement or motion is required.

RIGHT-HAND AND LEFT-HAND THREADS

Screw threads may be made either right or left-handed

LEFT HANDED   Turn Counter clock wise

RIGHT HANDED   Turn Clockwise
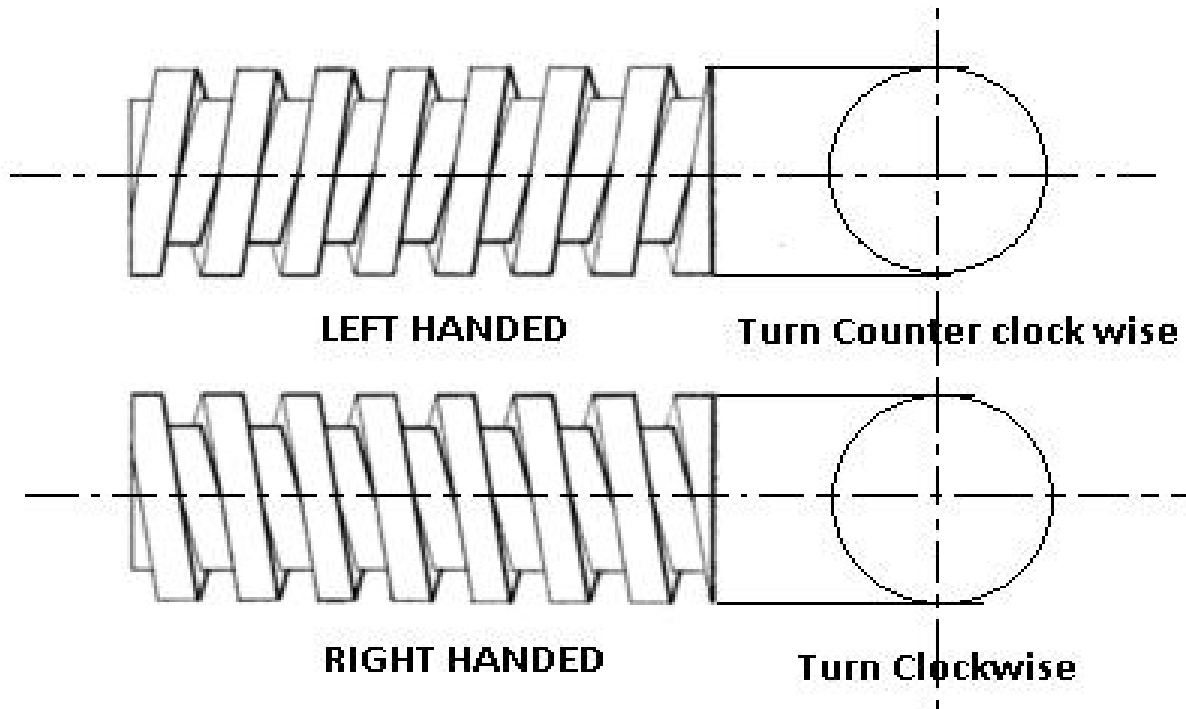
RIGHT-HAND THREAD

A right-hand thread is one in which the nut must be turned in a right-handed direction to screw on it.

LEFT-HAND THREAD

A left-hand thread being one on which the nut would be screwed on by turning it to the left.

NUTS

The bolt and nut joint or screwed fastening is a temporary joint. A bolt is used to join two assembled parts with the help of a mating part, the nut. A screw is used to join two parts by making its own thread in the joining part while screwing itself.**A nut is an internally threaded component joined to the externally threaded bolt.**

BOLT

A bolt and nut in combination is a fastening device used to hold two parts together. The body of the bolt, called shank is cylindrical in form, the head; square or hexagonal in shape, is formed by forging.

FOUNDATIONS BOLTS

Foundation bolts are used for fixing machines to their foundations. Foundation bolts are made by forging from mild steel or wrought iron rods. The bolt size depends upon the size of the machine and the magnitude of the forces that act on them when the machine is in operation.

RAG BOLT

This bolt consists of a tapered body, square or rectangular in cross-section, the tapered edges being grooved. A rag foundation bolt is set first in lead and then in cement concrete.

LEWIS BOLT

It is a removable foundation bolt. The body of the bolt is tapered in width on one side. To use this bolt, a pit is produced in cement concrete, by using a foundation block. Once the concrete sets-in, the bolt is placed in it so that the tapered bolt surface, bears against the tapered face of the pit. A key is then inserted, bearing against the straight surfaces of the pit and the bolt. This arrangement makes the bolt firm in the bed. However, the bolt may be removed by withdrawing the key

EYE BOLT

This is the simplest form of all foundation bolts. In this, one end of the bolt is forged into an eye and a cross piece is fixed in it.

CHAPTER-11

KEYS AND COTTERS

A key is employed to connect two pieces, like shaft and a gear, in such a way that there is no relative rotational movement between the shaft and the parts mounted on it. Key is always made of steel because it is subjected to shearing and torsional stresses. It is inserted in the axial direction between the shaft and the boss or hub of the mating piece. A keyway is the groove cut in the shaft as well as in the hub, to accommodate the key.

Keys may be classified as follows:

    (i)   Taper keys and (ii) parallel or feather keys.

(i)Taper keys

It is tapered (standard taper of this key is 1 in 100) in thickness but uniform in width. A taper key joint prevents relative rotational as well as axial movement between the two pieces. The types of taper keys are;

(1) Sunk taper key

It is a standard form of key and may be either of square or rectangular cross-sections. Half part of its nominal thickness is sunk in the shaft key way and the remaining half portion fits in the keyway inside the hub of the mating piece.

(2) Saddle keys

They are also made in two forms i.e. hollow and flat. A hollow saddle key has its underside hollow to fit the curved surface of the shaft.

(3)Round key (pin key)

It is of circular cross-section, usually tapered along the length and is inserted in a hole drilled partly in the shaft and partly in the hub of mating piece.

(4) Gib-head

It is a taper key is usually provided with a head, called a gib-head, to facilitate its removal.

(ii) Parallel or feather keys

When there is a requirement of sliding or axial movement of mating piece on the shaft, a feather key is used. It is a sunk key of uniform width and thickness. It may be of various cross sections like rectangular, square or dove-tail. This key may be secured to either the shaft by means of two cap-screws, having countersunk heads or the mating piece and free to slide in the keyways in the shaft. These keys are also classified as peg feather key, single headed and double headed feather key.

WOODRUF KEY

It is a sunk key in the form of a segmental part of a circular disc having uniform thickness. It is inserted into a corresponding form of a circular keyway cut in the shaft so that the some portion projects outside.
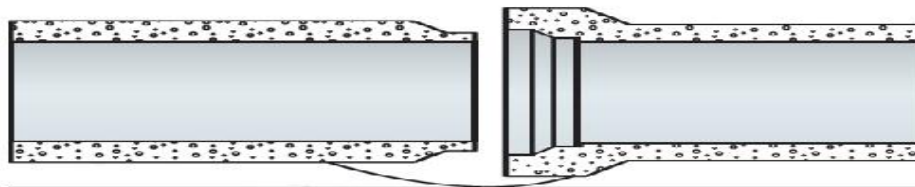
COTTER JOINTS

A cotter is a flat wedge-shaped piece of rectangular cross section and made of steel. It is used to connect two rods rigidly subjected to compressive or tensile forces and is inserted at 90^0 to the axes of the rods. It is tapering in width, generally on one side only but uniform in thickness. The bearing edge of the cotter and the bearing slots are generally made semicircular. This joint is not suitable for connecting the rotating members.

VARIOUS TYPES OF JOINTS The most commonly used types of cotter joints are

SPIGOT AND SOCKET JOINTS

A spigot joint is a type of pipe fitting connection that is inserted into another pipe fitting. The spigot end typically has the same outer diameter as the pipe and is usually   fitted   into   another   joint   called   a   bell   or   socket.

GIB AND COTTER JOINT

A gib and cotter joint is usually used in strap end (or big end) of a connecting rod In such cases, when the cotter alone (i.e. without gib) is driven, the friction between its ends and the inside of the slots in the strap tends to cause the sides of the strap to spring open (or spread) outwards.

In order to prevent this, gib are used which hold together the ends of the strap.
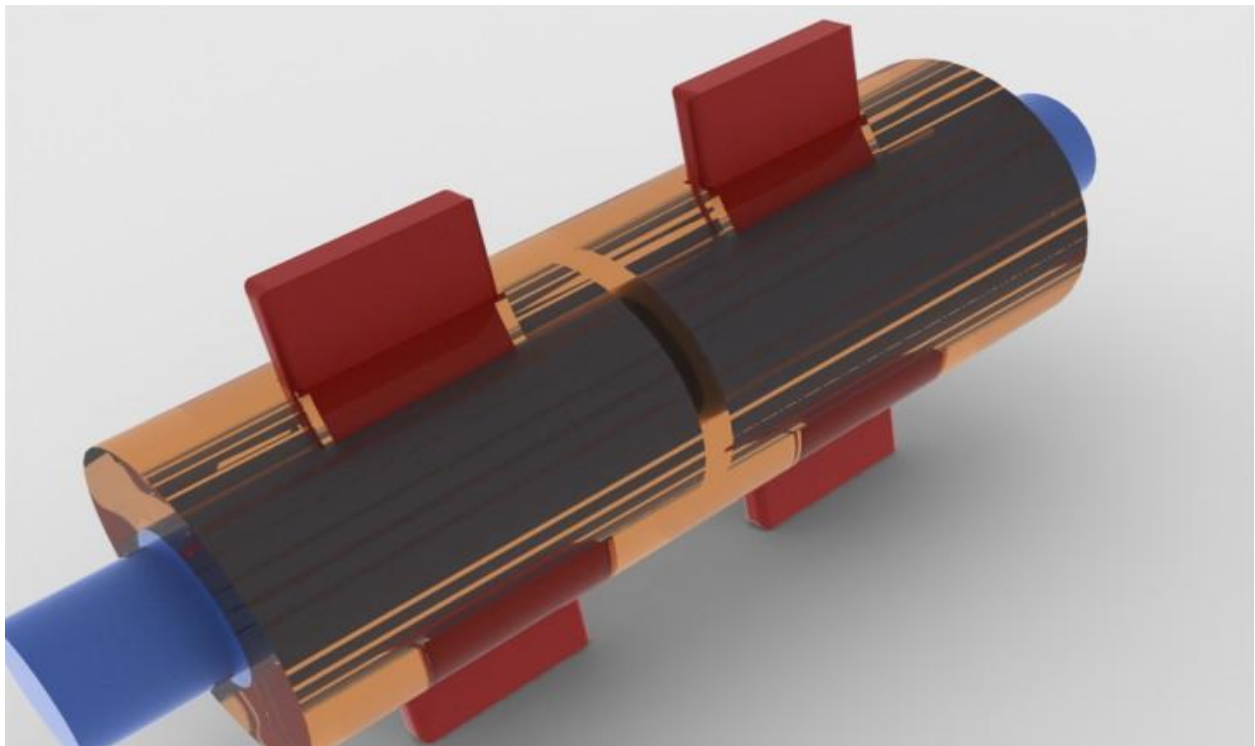
KNUCKLE JOINT

A **knuckle joint** is a mechanical joint used to connect two rods which are under a tensile load, when there is a requirement of small amount of flexibility, or angular moment is necessary. There is always axial or linear line of action of load. The knuckle joint assembly consists of the following major components:

1. Single eye.

2. Double eye or fork

3. Knuckle pin.

APPLICATIONS

1. Tie rod joint of roof truss.
2. Tension link in bridge structure.
3. Link of roller chain.
4. Tie rod joint of jib crane.
5. The knuckle joint is also used in tractor.

CHAPTER-12

INTRODUCTION TO COUPLING

A **coupling** is a device used to connect two shafts together at their ends for the purpose of transmitting power. The primary purpose of couplings is to join two pieces of rotating equipment while permitting some degree of misalignment or end movement or both.

USES

Shaft couplings are used in machinery for several purposes. A primary function is to transfer power from one end to another end (ex: motor transfer power to pump through coupling).

USES

- To provide protection

- To connect driving and the driven part

- To reduce the transmission of shock loads from one shaft to another

TYPES

MUFF COUPLING

It is the simplest form of rigid coupling, consists of a hollow cylinder whose inner diameter is same as shaft. It is fitted over the ends of the two shafts by means of gib head key. The power is transmitted from one shaft to the other shaft by means of a key and a sleeve or muff.
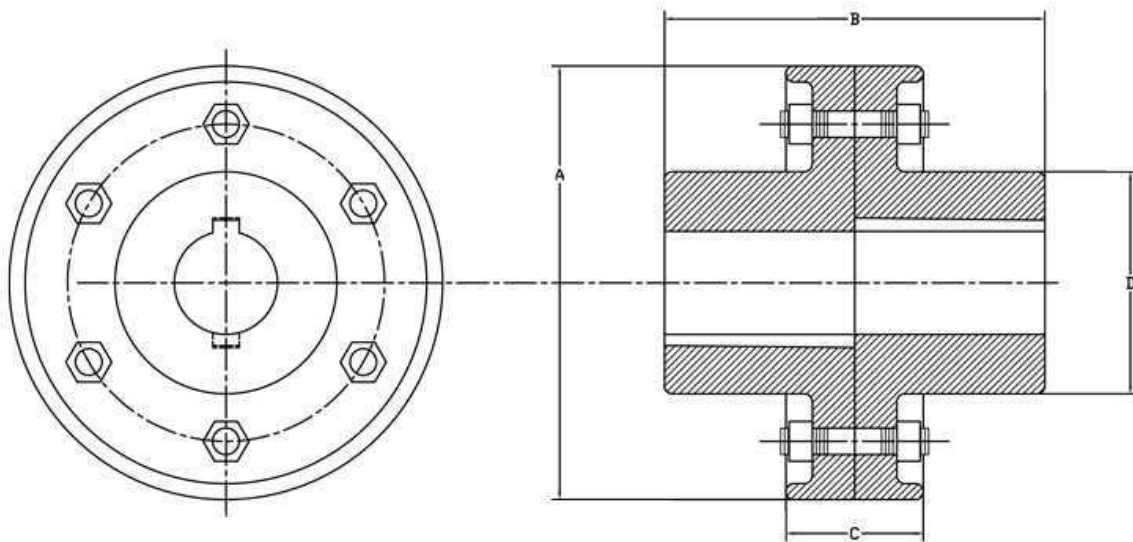

SLEEVE COUPLING / MUFF COUPLING
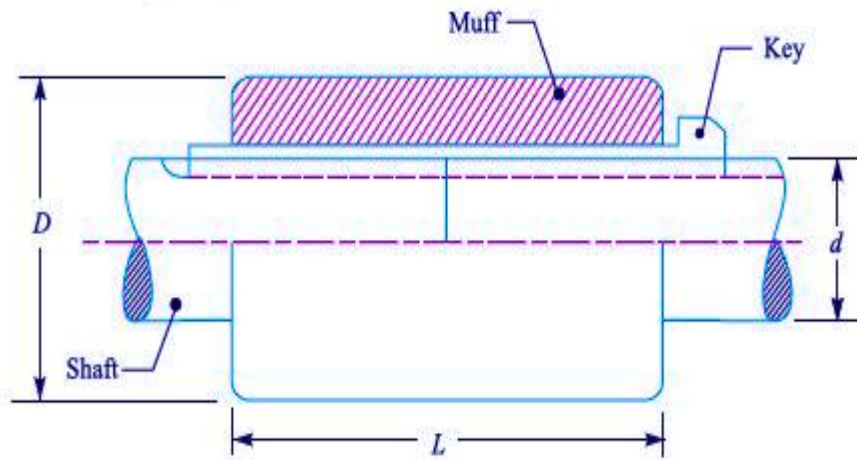
FLANGE COUPLING (PROTECTED)

Flange Coupling is a type of connector between rotating flanges that have two sets of flanges. One of these flanges is immobile at the end of each shaft. To complete the process of transferring power, both the flanges are fastened together with many nuts and screws.

FLEXIBLE COUPLING

The function of a flexible coupling is to transmit torque from the driver to the driven machine while making allowances for minor shaft misalignment and shaft end position changes between the two machines.

Shaft misalignment exists when the centerlines of two shafts joined by a coupling do not coincide.

Fig. shows the various types of misalignment and shaft end position changes that can occur.

(A) Perfect Alignment

(B) Parallel Misalignment

(C) Angular & Offset (Parallel) Misalignment

(D) Axial Position Change