Government Polytechnic Jashpur

Department of Mechanical Engineering

Notes For

Computer Aided Manufacturing and Modeling

6th Semester Diploma in Mechanical Engineering

By Dr. G. S. Bedi

UNIT 1 - ESSENTIALS OF CAD/CAM

1.1 Computer Aided Design (CAD) – Definition and Concept:

Computer aided design is the use of computer software to design and document the design process of a product. It can be defined as the use of computer systems to assist in the development, modification, analysis and optimisation of a design.

It involves creating computer models using the geometrical parameters of an object. The geometry of any object can be represented as a mathematical model. These models typically appear on a computer monitor as a two or three-dimensional representation. Once an object is represented as a mathematical model, it can be easily edited or modified, analysed and viewed under a variety of ways.

For example, a 3D model of a machine part can be created using CAD. The volume of this 3D model can now be used to calculate its volume by just giving a command on the computer as the 3D model is not only a graphical representation but also a mathematical model which let to use the data.

Thus CAD helps in mathematical modelling of engineering problems and representing them graphically to facilitate their solutions.

Need for CAD:

The continuous advances in computer technology have made the computers an essential part in every sphere of our life. Computers have emerged as a very powerful tool in design and manufacturing. With rigorous competition at the global level industries cannot survive unless they introduce new products or existing ones with:

– Better quality
– Lower cost
– Shorter lead time

The use of CAD provides following benefits:

1. Increases the productivity of the designer:
    - By helping the designer to visualise the product and is components;
    - By reducing the time required in synthesising, analysing and documentation of the design.
    - This results in reduced design cost and shorter lead time.

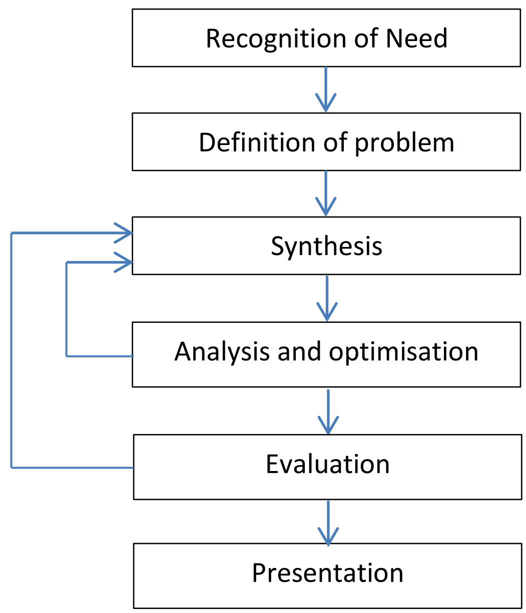
2. Improves the quality of design:
    - By facilitating a thorough engineering analysis.
    - Enables to investigate a large number of design alternatives.
    - Reduced design errors provided by greater accuracy of computer system.

3. Improved communication due to better engineering drawings, more standardisation in drawings. Better documentation, fewer drawing errors and greater legibility.

4. Creates a database for manufacturing by documenting the geometries and dimensions of the product and its components, materials specifications for components, bill of materials etc. which are required for the manufacturing of the product.

1.2 The Design Process:

The design process consists of following six steps:



Recognition of need means to identify a need and a decision to do something about it. This may be a problem with an existing design or a new idea to modify the design. The need is often a vague discontent, a feeling of uneasiness, or a sensing that something is not right. For example, the need to do something about a food-packaging machine may be indicated by the noise level, by a variation in package weight, and by slight but perceptible variations in the quality of the packaging or wrap.

The **definition of problem** is more specific and involves a thorough specification of the item. The specifications include the physical and functional characteristics, cost, quality and operating performance.

Synthesis involves conceptualising the design by preparing a rough drawing of the part, considering the types of load imposed on the part, the physical and approximate dimensions. A mathematical model of design may be prepared to synthesize the parts of design.

Analysis and optimisation involves the analysis of loading conditions and the geometric shape of the part. Stress calculations may be done and accordingly the dimensions can be recalculated. The part can further be optimized for minimum dimensions, weight, volume, the material and cost. The optimization depends on the definition of the problem and importance of a particular parameter. It may be sometimes necessary to optimize the part for certain operating parameters like efficiency, torque, etc.

Evaluation is concerned with measuring the design against the specifications established in the problem definition phase. This evaluation often requires the fabrication and testing of a prototype model to assess operating performance, quality, reliability, and other criteria.

The final phase in the design process is the **presentation** of the design. This includes documentation of the design by means of drawings, material specifications, assembly lists, and so on. Essentially, the documentation requires that a design database be created.

Functional Areas of CAD:

The various design-related tasks which are performed by a modem computer-aided design-system can be grouped into four functional areas:

1. Geometric modelling
2. Engineering analysis
3. Design review and evaluation
4. Automated drafting

Geometric modelling is concerned with the computer-compatible mathematical description of the geometry of an object. The mathematical description allows the image of the object to be displayed and manipulated on a graphics terminal. The designer constructs the graphical image of the object using commands that generates basic geometric elements such as points, lines, and circles. These geometric elements can be scaled, rotated, or subjected to other transformations. The geometric elements can be joined into the desired shape of the object being created. 2D and 3D models of the design may be created. During the geometric modelling process, the computer converts the commands into a mathematical model, stores it in the computer data files, and displays it as an image on the display screen.

Engineering analysis may involve stress-strain calculations, heat-transfer computations, or the use of differential equations to describe the dynamic behaviour of the system being designed. The computer can be used to aid in this analysis work using special programs or software. The mass properties of solid object being analysed, such as surface area, weight, volume, centre of gravity and moment of inertia can be easily calculated.

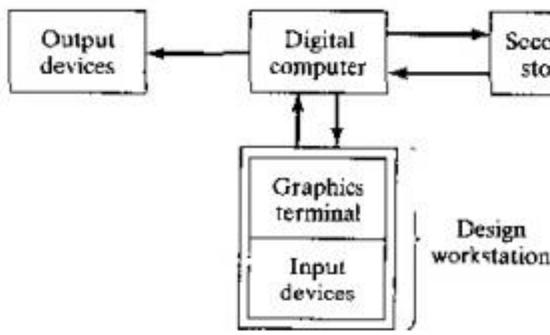
Design review and evaluation involves checking the accuracy of the design. Dimensioning and tolerancing routines can be used to reduce the possibility of dimensioning errors. The designer can zoom in on part design details and magnify the image on the graphics screen for close scrutiny. Kinematics packages can be used to animate the motion of simple designed mechanisms such as hinged components and linkages. This capability enhances the designer's visualization of the operation of the mechanism and helps to ensure against interference with other components

Automated drafting involves the creation of hard-copy engineering drawings directly from the CAD data base. CAD systems can increase productivity in the drafting function by roughly five times over manual drafting. Automatic dimensioning, generation of crosshatched sections, scaling of the drawing and the capability to develop sectional views and enlarged views of particular part details are the features of an automated drafting package.

1.3 CAD Workstation

A modern computer-aided design system is based on interactive computer graphics (ICG) and includes the following hardware components:

Digital Computer (CPU)
Secondary storage
Design workstation
Graphics terminal
Operator input devices
Plotter and other output devices



Configuration of a typical CAD system

The **Digital computer** (CPU) receives information from the work station and display the output on the graphics terminal (Display) and reads the data stored in the secondary memory storage unit.

The **Secondary memory**

- Stores files related to engineering drawing
- Stores programs required to give instruction to output devices like plotters.
- Stores CAD software
- The secondary storage unit consists of magnetic tapes and discs.

The **Design work station** is a visible part of the CAD system and performs following functions:

1. Provide interface with the central processing unit.
2. Generate a steady graphic image for the user.
3. Provide digital descriptions of the graphic image.
4. Translate computer commands into operating functions.
5. Facilitate communication between the user and the system.

It consists of a graphics terminal and operator input devices.

Operator Input Devices are provided to facilitate communication between the user and the system. These devices can be divided into three general categories:

1. Cursor control devices – Thumb wheel, Direction keys on keyboard, Joysticks, Tracker ball, Light pen, Electronic tablet/pen
2. Digitizers

3. Alphanumeric and other keyboard terminals

The cursor control devices and digitizers are both used for graphical interaction with the system. Keyboard terminals are used as input devices for commands and numerical data.

Various types of **Output Devices** are used with a computer-aided design system. These output devices include:

- Pen plotters – Drum, Flat Bed
- Hard-copy units
- Electrostatic plotters
- Computer-output-to-microfilm (COM) units

Classification of Hardware Configurations

The classification hardware configuration is as follows:

(a) Mainframe-based systems
(b) Minicomputer-based systems
(c) Microcomputer-based systems
(d) Workstation-based systems

1.4 CAD Software:

The CAD software is the collection of programs written to make it convenient for a user to operate the computer graphics system. It includes Programmes to generate images on the display screen (graphics terminal), to manipulate the images, and to accomplish various types of interaction between the user and the system. In addition to the graphics software, there may be additional programs for implementing certain specialized functions related to CAD/CAM. These include design analysis programs (e.g., finite-element analysis and kinematic simulation) and Manufacturing planning programs.

Features of a CAD software:

1. Simplicity - easy to use.
2. Consistency - operate in a consistent and predictable way to the user.
3. Completeness - no inconvenient omissions in the set of graphics functions.
4. Robustness - tolerant of minor instances of misuse by the operator.
5. Performance - Within limitations imposed by the system hardware, the performance should be exploited as much as possible and programs should be efficient and speed of response should be fast and consistent.
6. Economy - not be as large or expensive as to make their use prohibitive.

Types of CAD Software:

- 2 Dimensional: 2D CAD relies on basic geometric shapes like lines, rectangles, circles, etc. to produce flat drawings. Example AutoCAD, CATIA
- 2.5 Dimensional: In between 2D and 3D CAD is 2.5-D CAD. The models created in this type of CAD are prismatic, that is, they represent the depth of the objects. Like 2D CAD, these objects are made up of geometric objects.
- 3 Dimensional: 3D CAD allows creation of 3D images that are realistic. These images are called 3D models as they can be viewed and rotated in any direction –

X, Y or Z. You can also display views from a 3D model, such as isometrics or perspectives, from any angle using 3D CAD.

3D CAD can be further classified as:

Wire-frame models – they create skeleton like models with lines and arcs. Since they appear to be made of wires, and everything in the background is visible, they are called wire-frame models. They are not very popular anymore.

Surface models – unlike wire frames, these models are created by joining 3D surfaces. Since nothing in the background is visible, the surface models are quite realistic.

Solid models – they are considered to be the most useful CAD models. Although they appear to be the same as surface models, they also have additional properties like weight, volume and density, just like actual physical objects. These models are commonly used as prototypes to study engineering designs.

Examples: Autodesk Inventor, CoCreate Solid Designer, Pro/Engineer SolidEdge, SolidWorks, CATIA, Unigraphics NX and VX CAD.

Functions of a CAD software:

1. Generation of graphic elements
2. Transformations
3. Display control and windowing functions
4. Segmenting functions
5. User input functions

1.5 Computer Aided Manufacturing (CAM) – Definition and Concept:

Any product and its components are designed to perform certain specific functions. Once the design process establishes the specifications of the product/component, the task of manufacturing is then to produce the components according to the specifications. The components thus manufactured are assembled into the final products.

In conventional machine tools the movement of cutting tool is done by the operator either manually or by engaging the automatic feed mechanism. In addition, the operator also performs the other actions necessary for machining the component e.g. starting/stopping rotation of the spindle, changing speed and feed, turning on cutting fluid etc. Each of these actions requires the operator to exercise judgment and accordingly make a decision. The accuracy of the job depends essentially on the skill of the operator. These machines have the following disadvantages:

1. More time consumption.
2. Less accuracy.
3. Less production.
4. Require skilled labours.
5. More wastage of materials.

These disadvantages can be overcome by using computers in manufacturing. With the use of computers the decisions related to the operation of the machine are made only once at the planning stage. All the necessary movements required to machine a component are performed automatically by the machine itself according to the instructions given by the computer.

CAM can be defined as the use of computer systems to:

– Plan;
– Manage; and
– Control

the operations of a manufacturing process.

Computer can have a direct or indirect interface with the manufacturing system or process.

In the **direct interface** the computer is directly connected to the manufacturing process for monitoring and controlling. Computers are used to monitor and control the manufacturing operations.

In the **indirect interface** the computer does not control the manufacturing process but assists in carrying out manufacturing support activities like process planning, production scheduling, material requirement planning, shop floor control etc., ensure that the production resources are utilized effectively.

Purpose of CAM:

The purpose of CAM is to:

1. Make the manufacturing process faster.
2. Manufacture uniform products by reducing errors in manufacturing.
3. Ensure optimum utilisation of resources used for manufacturing.
4. Reduce the labour cost and other operating overheads.

1.6 Numerical Control (NC)

We know that for performing any task using a computer requires number of instructions. These set of instructions is, commonly, known as program. Similarly, for monitoring and controlling a manufacturing process we need to give instructions. These instructions are given using combination of letters, numbers and symbols and are known as Numerical Control.

Numerical control (NC) is a technique in which the mechanical actions of a machine tool are controlled by means of program of instructions. The instructions are given using a combination of letters, numbers and symbols.

The program:

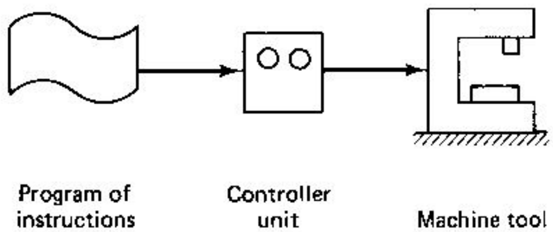
i. Defines the relative positions of the cutting tool and the work piece at all times;
ii. Enables these to be brought together for various processing operations as necessary; and
iii. Taken apart upon completion of task.

In NC, a program of instructions is designed for a particular job. When the job changes the program of instruction is also changes. This capability to change the program of instruction gives flexibility to NC.

Basic Components of an NC System:

A NC system consists of following three basic components:

1. Program of instructions
2. Controller unit, also called a machine control unit (MCU)
3. Machine tool or other controlled process



Basic Components of NC System

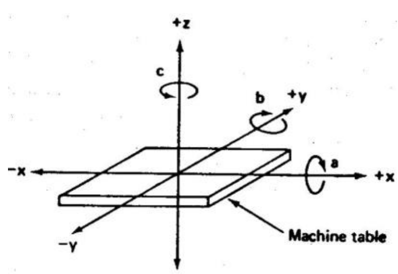
Program of instructions: The program of instructions is the detailed step-by-step set of directions which tell the machine tool what to do. It is coded in numerical or symbolic form on some type of input medium, e.g. punched tape or card, magnetic tape etc. which can be interpreted by the controller unit.

Controller Unit: The controller unit consists of the electronics and hardware that:
1. Read and interpret the program of instructions
2. Convert it into mechanical actions of the machine tool.

Machine Tool: It is the part of the NC system which performs the actual work. For example, in a NC system designed to perform machining operations, the machine tool consists of the worktable and spindle, the motors and controls necessary to drive them. It also includes the cutting tools, work fixtures, and other auxiliary equipment needed in the machining operation.

NC Coordinate Systems:

The purpose of the coordinate system is to provide a means of locating the tool in relation to the work piece. In order for the part programmer to plan the sequence of positions and movements of the cutting tool relative to the work piece, it is necessary to establish a standard axis system by which the relative positions of work piece and cutting tool can be specified. Nearly all NC/CNC machine uses a Cartesian coordinate system based on an X, Y, and Z axis. This system allows a machine to move in a specific direction along a specific plane.



Coordinate system for NC Machine

Fixed and Floating zero:

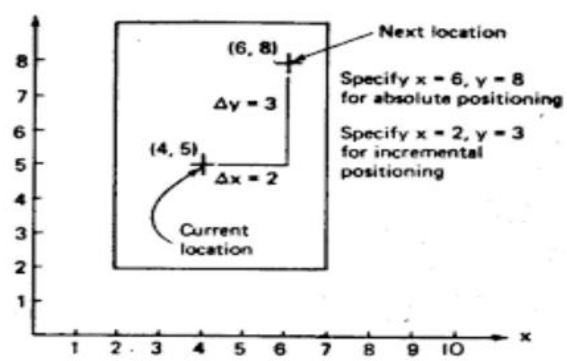
The position of the tool relative to the origin (zero point) of the coordinate system must be defined. NC machines have two methods for specifying the zero point:

1. **Fixed Zero:** The machine has a fixed zero. The origin is always located at the same position on the machine, usually, the lower left-hand comer of the table. All tool locations will be defined relative to this by positive x and y coordinates.
2. **Floating Zero:** This system in NC machines allows the machine operator to set the zero point at any position on the machine table. The part programmer is the one who decides where the zero point should be located. The decision is based on part programming convenience. For example, the work part may be symmetrical and the zero point should be established at the centre of symmetry.

Absolute positioning and Incremental positioning:

Absolute positioning means that the tool locations are always defined in relation to the zero point.

Incremental positioning means that the next tool location must be defined with reference to the previous tool location.



Absolute and Incremental Positioning

Applications of NC:

NC is mainly used for metal cutting applications like:

1. Milling
2. Drilling and related processes
3. Boring
4. Turning
5. Grinding
6. Sawing

In addition, it can also be applied to a wide variety of manufacturing operations like:

1. Press working
2. Welding

3. Tube bending
4. Flame cutting
5. Plasma arc cutting
6. Laser beam processes
7. Cloth cutting
8. Knitting
9. Riveting
10. Automated drafting

Situations in which NC should be used:

As NC systems are expensive, they should be used when:

1. Parts are to be made frequently and in small lot sizes.
2. The part geometry is complex.
3. Processing involves many operations.
4. Metal removal is high.
5. Design of part changes frequently.
6. Close tolerances must be maintained on the work piece.
7. It is an expensive part where mistakes in processing would be costly.
8. The parts require 100% inspection.

Advantages of NC:

1. Reduced non-productive time: The use of NC can increase the proportion of time the machine is engaged in the actual process. It accomplishes this by means of fewer setups, less time in setting up, reduced work piece handling time, automatic tool changes etc.
2. Reduced fixturing: NC requires fixtures which are simpler and less costly to fabricate because the positioning is done by the NC tape rather than the jig or fixture.
3. Reduced manufacturing lead time: Jobs can be set up more quickly with NC and fewer setups are generally required with NC, thereby reducing the lead time.
4. Greater manufacturing flexibility: Changes in engineering design, changes in the production schedule, and changeovers in jobs for rush orders can be easily made with use of NC.
5. Improved quality control: NC is ideal for complicated work parts where the chances of human mistakes are high. Numerical control produces parts with greater accuracy, reduced scrap, and lower inspection requirements.
6. Reduced inventory: Due to fewer setups and shorter lead times, the amount of inventory carried by the company is reduced.
7. Reduced floor space requirements: Since one NC machining center can do the production of several conventional machines, the amount of floor space required in an NC shop is usually less than in a conventional shop.

Disadvantages of NC:

1. Higher investment cost: Numerical control machine tools are expensive.
2. Higher maintenance cost: Because NC uses complex technology the maintenance becomes difficult.

3. Highly skilled and trained persons are required.

1.7 Computer Based Numerical Control Systems

Computer based numerical control system involves the replacement of conventional NC controller unit by a computer. The computer performs some or all the NC functions. The computer based NC systems are classified as:

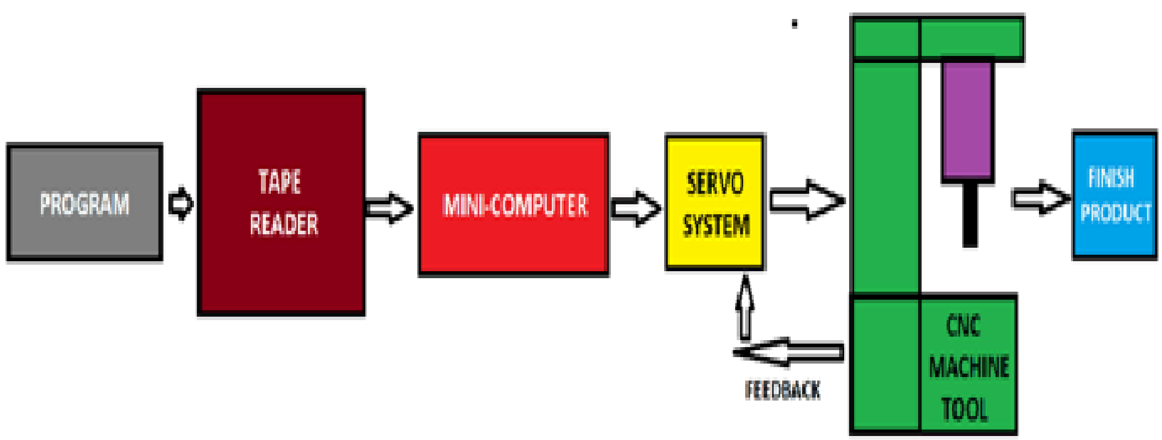
1. Computer Numerical Control (CNC)
2. Direct Numerical Control (DNC)
3. Adaptive Control (AC)

Problems with Conventional NC:

1. Part programming mistakes: Part programming mistakes are common in preparing the punched tape. The mistakes can be syntax or numerical errors.
2. Non-optimal speeds and feeds: The speed and feed cannot be changed during the machining process.
3. Punched tape: Paper is fragile and is subjected to wear and tear.
4. Tape reader: It is the most reliable element of the NC system.
5. Controller: Control features are difficult to change to make improvements.
6. The conventional NC does not provide information about machine breakdowns and tool changes.

Computer Numerical Control (CNC)

CNC is a NC system which uses a dedicated computer to perform some or all of the NC functions. The block diagram of a CNC system is shown below:



CNC System

The **main functions of CNC** are:

i. Machine tool control: This is the primary function of CNC and it involves conversion of part program into machine tool motions through the computer interface and servo system.
ii. In-process compensation: This involves dynamic correction of the machine tool motions for changes or errors which occur during the actual machining process.
iii. Improved programming and operating features: The use of computer provides the flexibility in programming and operating features. This includes:
   a. Editing of part programs to make corrections.
   b. Graphic display of the tool path to verify the punched tape.
   c. Manual data input.
   d. Local storage of part programs.
   e. Use of specially written subroutines.

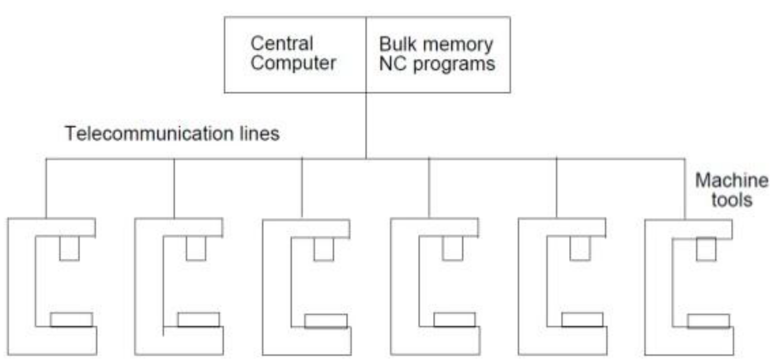
Advantages of CNC:

i. The tape and tape reader are used only once to enter the program into computer memory.
ii. The NC tape can be corrected for tool path, speeds and feeds.
iii. The tapes prepared in metric units can be changed into SI units.
iv. Flexible to introduce new control options.
v. Special programs can be written and stored as MACRO subroutines.
vi. Suitable for computerized factory-wide manufacturing.

Direct Numerical Control (DNC):

DNC is a manufacturing system in which a number of machine tools are controlled by a computer through direct connection and in real time.
The main features of a DNC system are:

- This system does not use tape reader.
- The part program is transmitted directly to the machine by the computer.
- One large computer can be used to control 100 separate machines.
- The computer can provide instructions to each machine on demand.
- DNC also provides data collection and processing from the machine tool back to the computer



DNC System

Advantages of DNC:

i. Elimination of punched tape and tape reader.
ii. Greater flexibility and computational capability.
iii. Convenient storage of NC part programs in files.
iv. Reporting of machine tool performance.
v. Establishes the framework for development of future computer automated factory.

Self-Assessment Questions

1. What are the basic components of an NC system? Write the function of each.

2. Outline the difference between absolute and incremental positioning in NC systems.

3. What are the characteristics that make NC suitable?

Answers to Self-assessment Questions

Question 1
The basic components of an NC system are:
i. the part program of instructions,
ii. the machine control unit, and
iii. the processing equipment or machine tool.

The part program consists of a detailed set of step-by-step commands/instructions that direct the actions of the processing equipment/machine tool. The machine control unit (MCU) consists of a microcomputer and control hardware for program storage. It converts the part program into a usable format and executes it in the form of the mechanical actions of the processing equipment/machine tool. The processing equipment actually performs the work of the system, once it has been provided with its instructions by the MCU.

Question 2
In the absolute system the work head is defined relative to a pre-defined origin; while in the incremental system the work head is defined relative to the previous location of the tool.

Question 3
Characteristics suitable for NC include:
i. Parts to be produced in frequent batches.
ii. Situations where the geometry of the part is complex.
iii. Situations requiring excess metal removal from parts.
iv. Situations where number of separate machining operations is to be performed on the part.
v. The part to be produced is expensive.

UNIT 2 COMPUTER AIDED SOLID MODELLING AND ASSEMBLY

2.1 Solid Modelling:

Solid or 3D models contain information about the edges, faces, and the interior of the part. They show the actual size and shape of a physical object. A solid model is a digital representation of the geometry of an existing or imagined physical object. Solid models are solid throughout, not hollow, and have a definite volume.
Solid modelling is a technique that uses computer software to create digital objects in 3D form e.g. structures, machines or components. For example, if you are designing a motor vehicle part, solid modelling allows you to find out if the part will fit correctly with the rest of the vehicle's components. It can also predict if it could withstand the stresses and temperatures the vehicle is subjected to.

Benefits of Solid Modelling

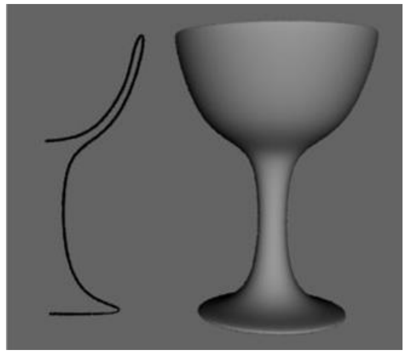
- It visualises ideas: Before manufacturing a product, a working prototype can bring an idea to life.
- It helps in detecting errors: By having an accurate 3D model, potential problems can be identified and fixed ahead of time.
- It saves time and resources.

2.2 Creating 3D Models:

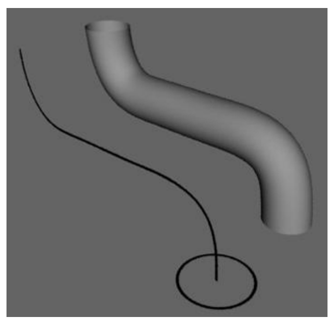
The following commands can be used to create 3D models:

1. Extrude: The most common operation in solid modelling is extrusion, i.e. "pulling" a 2D object into 3D. Extrude command converts 2D shapes into 3D shapes by adding the missing dimension through a positive or negative length. The extrusion allows the creation of a new solid while maintaining the original geometry of the 2D surface. The Extrude command lets you control the direction, height, and angle of the extrusion.

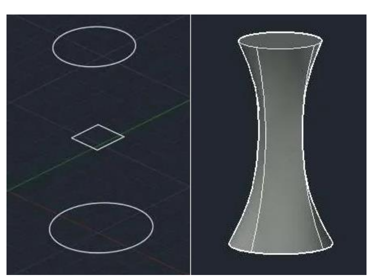
2. Revolve: The Revolve command is used to create 3D objects by rotating 2D profile around an axis. The axis of rotation must be defined before the revolve operation is carried out. The angle of revolution must be specified in order to control the shape of the volume; it means that a fully revolved solid is required or a partial one. The revolve command is very useful for creating symmetrical objects.



3. Sweep: Sweeping simply means moving along a predefined path. The SWEEP command creates a solid or surface by extending a profile shape (the object that is swept) along a specified path. The *path* can be an open or closed drawing object but must be one object. If the profile is a closed object, the sweep will create a solid. If the profile is an open object, the sweep will create a surface. Sweeps are used to create models like pipes, tubes, pipes, gasket grooves, and threads.

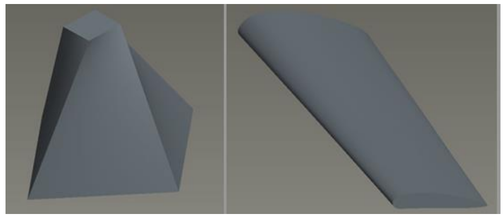


4. Loft: A *loft* is a blend or transition solid or surface that connects two or more cross sections. The cross sections, also called *profiles*, can be the same or different shapes. The cross sections can also be projected along a path. The path must be a single open or closed object. The selection of lofting order decides the final geometry and using the same set of geometries you can make different 3D shapes by simply altering lofting order.



5. Blend: Smooth transition can be made between two closed shapes with similar geometry (i.e. equal number of vertices) using the blend command. The distance

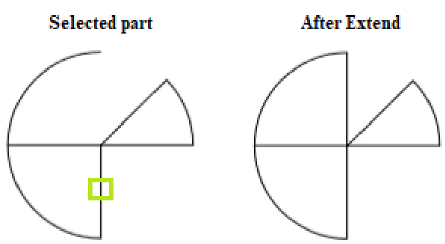
between sections must be defined. The angle of twist between sections must be specified.



2.3 Part Editing Tools:

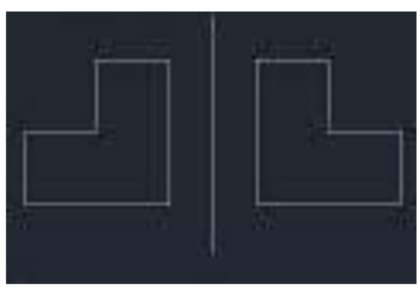
1. Trim: The Trim command allows removing unwanted parts of an object or a group of objects when working with intersecting lines or shapes, as it enables to create precise boundaries and connections between different elements in the drawing.

2. Extend: The extend command is used to extend the edges of the objects to meet the edges of other objects.



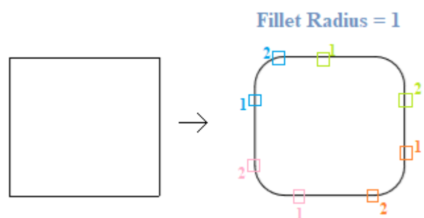
3. Erase: The erase command removes an object from the drawing.

4. Mirror: The mirror command is used to create mirror image of the selected object or drawing.

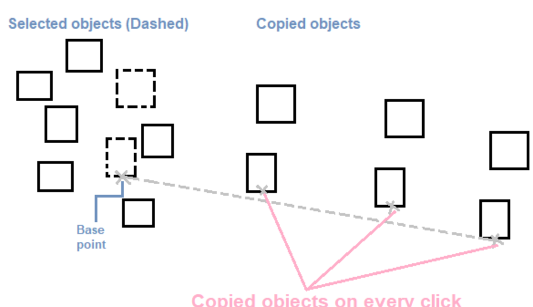


5. Chamfer: The chamfer command allows to create a bevelled edge between two intersecting lines. The distance and angles for the chamfer is required to be specified.

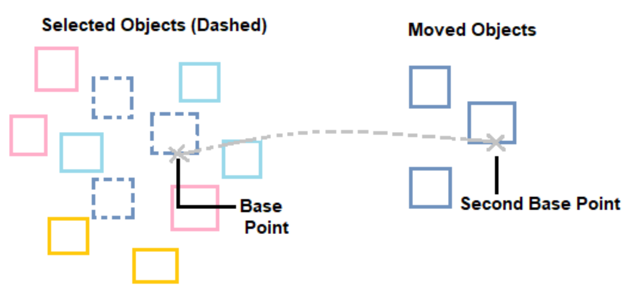
6. Round: The round or fillet command rounds the edges of two 2D objects. A round or fillet is an arc that is created tangent between two 2D objects. A fillet can be created between two objects of the same or different object types: 2D polylines, arcs, circles, ellipses, elliptical arcs, lines.



7. Copy: The copy command copies objects at specified distance in a specified direction.



8. Move: The move command moves objects at a specified distance in a specified direction.

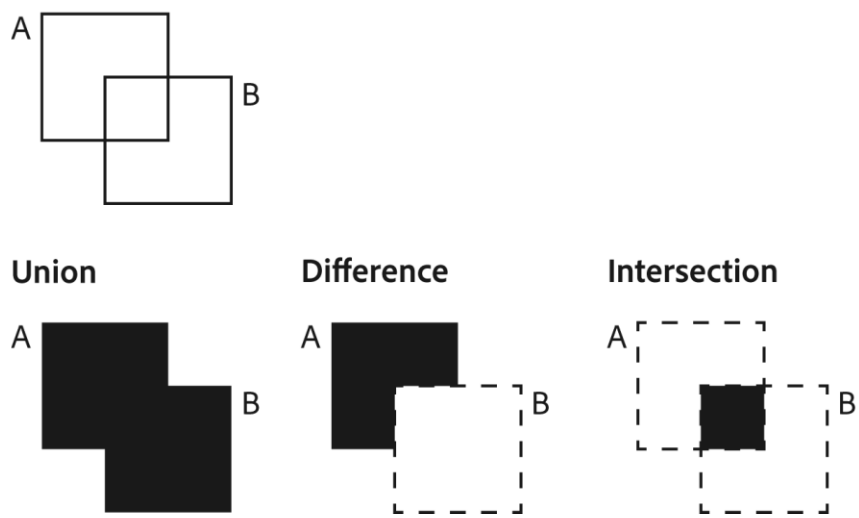


Boolean Operations:

Boolean operations are useful 3D tool and refer to a set of operations that can be performed with 3D objects:

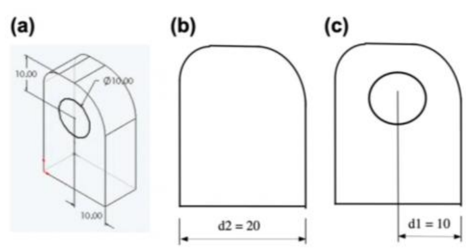
1. Union: Combine two objects together *i.e.* addition.
2. Difference or Subtract: cut away object from target objects.
3. Intersect: Delete everything except where two objects overlap.

The image below shows the results of performing each Boolean operation with shapes A and B. Note that for the Difference operation, B is the Boolean object and is subtracted from target object A.



2.4 Parametric and Non-parametric Modelling:

Parametric modelling involves specifying dimensions that define the geometry of a part and subsequently establishing and outlining the relations between the dimensions both across and within the part. Thus, the entire model will be automatically modified or rebuilt whenever one or more dimension values are changed as all the dimensions have a predefined relationship.



To better understand how parametric modelling works, let us consider the figure above. A designer wants the hole in the block shown (figure a) to remain centered even when the length of the block changes. To accomplish this, a sketch of the block (figure b) with dimension *d2* as the design variable, needs to be created.

Next, the hole must then be placed on the sketch, as shown in figure c. For this, it is necessary to specify the relationship between the center of the hole and dimension *d2.* Given the hole must remain centered even if the length is changed, the distance of the center of the hole from one edge, *d1,* should be equal to half *d2 i.e. d1 = 0.5d2.*

This is because for *d1* to be defined, *d2* must be defined first. Thus, *d1* is dependent on *d2.*

Some commonly used software for parametric modelling are:

- Creo
- Solidworks
- Autodesk Inventor
- Catia

Nonparametric modelling involves a direct approach to building 3D models without having to work with provided parameters and therefore also known as Direct modelling. It involves the creation of a model by simply manipulating its geometry. It is based on how the boundaries, namely the faces, edges, and other features, define or represent the model. As such, all one need to do is pull or push these boundary elements to achieve a given shape, similar, to working with clay. However, this time, instead of using hands to mold the clay, one just clicks the mouse cursor and moves the geometry as he wishes.

In direct modeiling, the 3D modeiling software does not store the sequence of features or geometry creation. This means that direct modelling creates a history of sequence and also does not require defining constraints, use parameters to represent the design, or provide feature-based information. The lack of these attributes makes direct modelling faster. This subsequently increases productivity and reduces development costs and design times. The designers can easily use direct modelling to edit, modify, and repurpose solid models, something that is not possible with parametric modelling.

Some commonly used software for non-parametric modelling are:

- Creo Direct
- BricsCAD
- Shapr3D
- Fusion 360

Comparison of Parametric and Non-Parametric Modelling:

Parametric Modelling	Non-Parametric Modelling
It requires pre-planning of a modelling strategy.	It does not require pre-planning of a modelling strategy.

Parametric modelling involves indirect manipulation of model geometry.	Non-Parametric modelling does not involve indirect manipulation of model geometry.
It features a less efficient way of changing the dimension – i.e., defining the relations.	It features a more efficient way of altering the dimensions – i.e., pushing and pulling.
Parametric modeling tools are not easy to use, are inflexible, and slow because the designer must consider relations between features and geometries.	Direct modeling tools are easy to use, flexible, and slow, as they are based on the principle of pushing and pulling.
Parametric modelling requires the designer to have a design intent as the paradigm is based on relationships between features and dimensions	In direct modelling, the designer must not have a design intent, as the models are not based on relationships
The designs of parametric models can only be changed by designers who are knowledgeable about the associated sequences of relationships; thus, they cannot be altered or updated by any party	The models created using this paradigm can be changed on the go.

2.5 Assembly drawings:

Technical or Engineering describe three-dimensional parts and components on a two-dimensional paper. These drawings come in various types, such as engineering detail drawings, production drawings, and assembly drawings.

Among the different types of technical drawings, assembly drawings are essential for identifying each part of a machine or system in its operating position and assembly sequence. Assembly drawings include a bill of materials (BOM), often called as parts list, sections, weight, and orthogonal plans.

Types of Assembly Drawings:

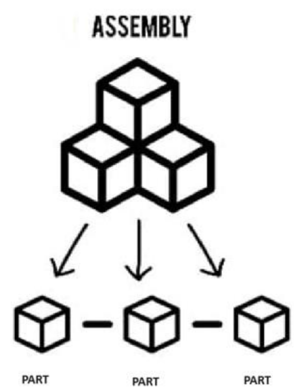
- General Assembly Drawings - identify the various components and their relationship. It contains the component's detailed drawing, the sub-assembly, and the final assembly.
- Exploded Assembly Drawing - show the components of an assembly slightly separated or suspended in space in the assembly sequence. This "exploded" view clarifies how the final product will fit together, making it easier to understand even for a layperson.
- Installation drawing - help workers with the erection or assembly of a product. They supply information on how a component will be positioned in relation to its

supporting or adjacent elements. It will also show parts, dimensional data, hardware descriptions, and a general configuration.

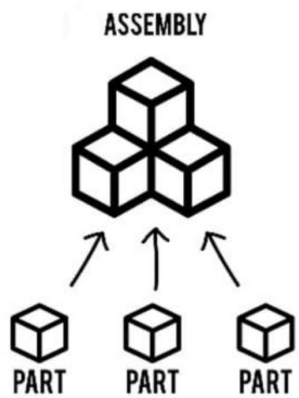
- Schematic drawing - simplified illustration of components using standardized symbols and lines. Schematic diagrams include only necessary details. For example, a schematic diagram of an electrical circuit will show how the wires and components are connected without photographs of the circuit itself.
- Machine shop drawing - for the machinist or machine operator. Because this person is unconcerned about information related to the previous stages or operations, only the information about the machined part's function is included in the part drawing.

Top=Down and Bottom-Up Approach of Assembly:

Top-Down modelling focuses on the assembly, and the individual components are modelled around the entire assembly. With Top-Down modelling, you will have a single design file and each of the components will be modelled based on the references and geometry of its surrounding components.
Top-down modelling is common within larger teams, as one designer can easily design parts knowing that it will work correctly with a different component that was designed by a different person, as well as in the context of the entire assembly.

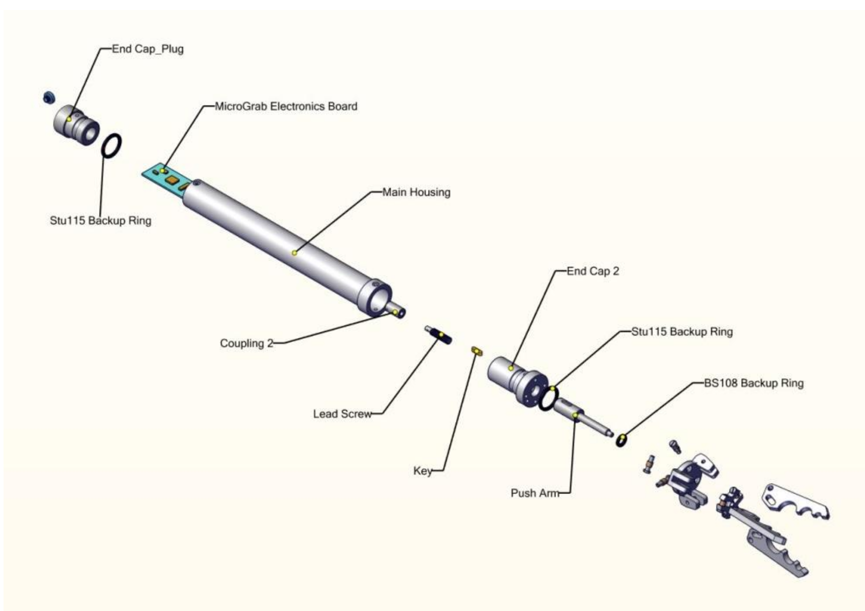


Bottom-Up modelling focuses on the individual parts that make up an assembly. Each key component is designed separately from the rest, normally in a separate design file. The geometry of one part is not typically influential on the other parts around it because they are all designed separately. (i.e. you start by designing the bottom-level components before combining them to end up with the overall assembly.) The design is completed by first designing the key components, and then assembling them together at the end.



Bottom-Up modelling is frequently used in software such as Solidworks, Inventor, and others which all have separate assembly file types. All components are modelled in separate part files, and once all the parts have been completed, an assembly file is created to join the individual components together. Bottom-Up modelling is also more common when using a large variety of standard parts because these do not need to be designed with other parts in mind and can easily be added to an assembly file at the end of the process. Bottom-Up modelling is more common when there is only a small group or single person completing the design. When a large group is involved in a bottom-up modelling task it can sometimes be more difficult to communicate the design and ensure that the entire assembly fits together correctly.
While bottom-up modelling allows for standard parts to be integrated, top-down modelling is more commonly used when a design contains all unique parts.

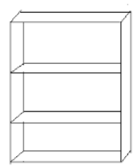
Exploded View:
An exploded view shows an assembly's components spread out, but positioned to show how the components fit together when assembled.



An exploded-view drawing is a type of drawing that shows the intended assembly of mechanical or other parts. It shows all parts of the assembly and how they fit together.

Self-Assessment Questions

1. Define solid modeling?
2. Explain the top down and bottom down approach of assembly with neat sketches?
3. Explain the function of following commands:
   i. Move
   ii. Chamfer
   iii. Fillet
   iv. Extend
4. Differentiate between parametric and non- parametric modelling.
5. List common software used for parametric and non- parametric modelling.
6. Draw the exploded view of the following object:



UNIT 3 COMPUTER AIDED DRAFTING AND PLOTTING

3.1 Orthographic, Isometric and Auxillary Projections (Views)

Orthographic projection is a common method of representing three-dimensional objects in two-dimensions. It consists of 3 orthographic views, namely, the front view, the top view, and the side view. The side view is usually the right side, but will be labelled if it is not.

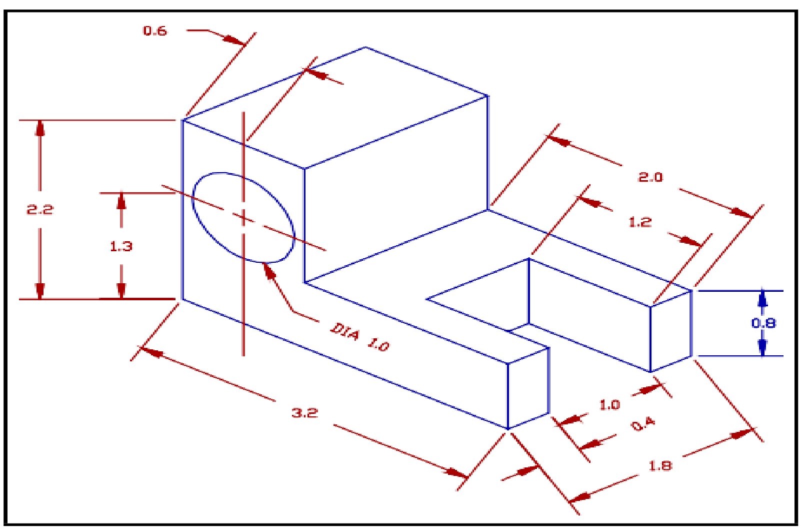
Isometric Projection is a method for visually representing three-dimensional objects in two dimensions in technical and engineering drawings. "Iso" means "equal" and "metric projection" means a projection to a reduced measure. In isometric projection the three dimensions of a solid are not only shown in one view, but also their dimension can be scaled from this drawing.

Auxiliary projection is a type of orthographic projection used to determine the true size and shape of inclined and oblique surfaces of objects.

Creating Orthographic Views:

Drawing Exercise

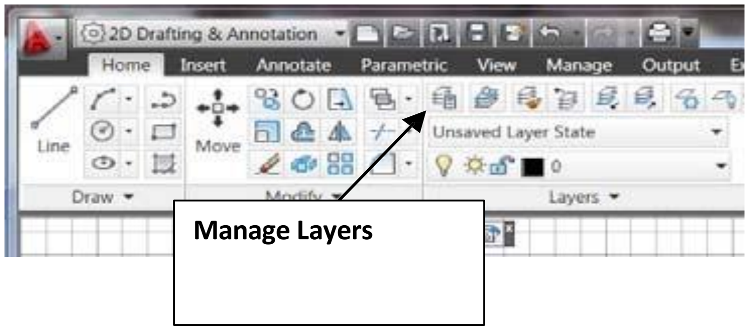
1. Draw the Orthographic views (front, top and side) of the given object.
2. Dimension all the views.
3. Print the drawing.



1. First start AutoCad and open a new file and give it a name of your choice.
2. Next set up the drawing environment. This includes:
    - Determine the units of measure needed.
    - Determine the limits of drawing.

- Decide what drawing aids will be useful.
- Create the layers needed to control the visibility, line types and colour of objects in the drawing.

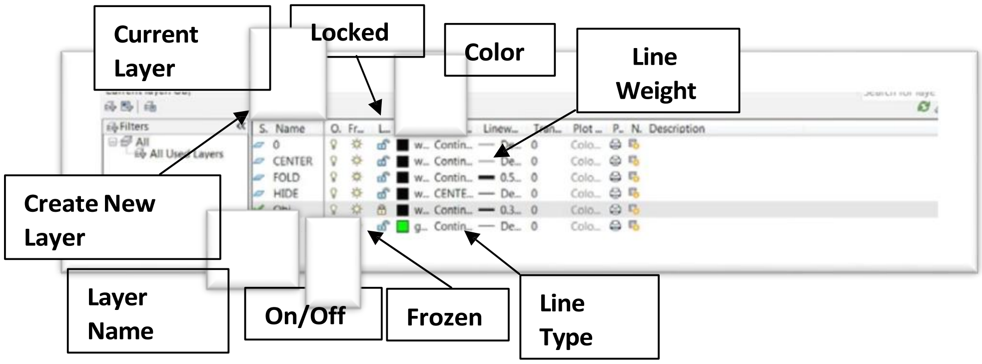
3. First create the layers. Click on the **LAYER PROPERTIES** button shown in the figure below:



Different entities like lines, circles, arcs etc. belongs to layers and each layer determines how the lines, circles, and arcs will be drawn. The layer controls the color, line type, line weight and several other attributes for the layer. These attributes are:

Layer Name	The name used for the layer. Each layer has a unique name.
On/Off	If on the lines can be seen and if off they cannot be seen if it is on it can be seen.
Frozen	This attribute is very similar to On/Off. If a layer is frozen, it cannot be seen.
Locked	The lines in the layer cannot be deleted or modified.
Color	Determines the color of the lines when they are displayed.
Line Type	Determines which line type is used to draw lines in this layer. They can be solid or dashed in many different ways.
Line Weight	This attribute controls how thick the line is when it is drawn on the screen and plotted. The varying thickness on the screen can be turned on and off using a button at the bottom of the main window.
Create New Layer	This button creates a new layer. After it is created you can change the attributes so that it meets your needs.
Current Layer	Click on one of the existing layers then click on the Current Layer check mark. All lines drawn from this point on will be in the layer marked as the Current Layer. This will remain in effect until the current layer is changed again.

This dialog box allows you to create new layers and define their attributes.



Click on the Create New Layer button and create layers with the names: OBJ, PROJ, HIDE, FOLD, and CENTER. Click on the color for the projection layer and change its color. This will make it easier to distinguish projection lines form object lines. You can choose a different color for each layer if you desire.

In this exercise, we will create several new layers. They are:

OBJ This layer is used for drawing the objects.

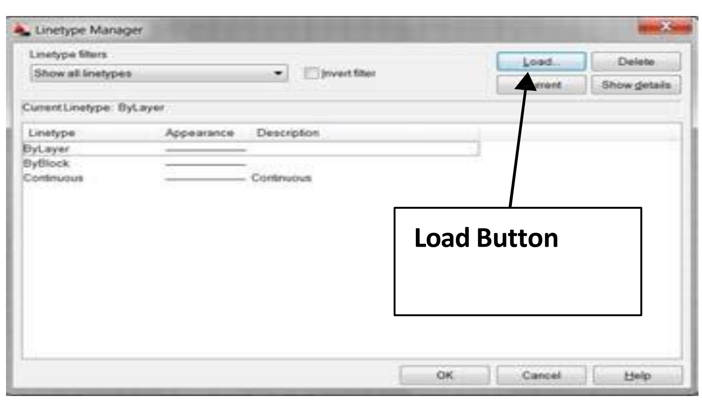
HIDE This layer is used for hidden lines.

CENTER This layer is used for centerlines.

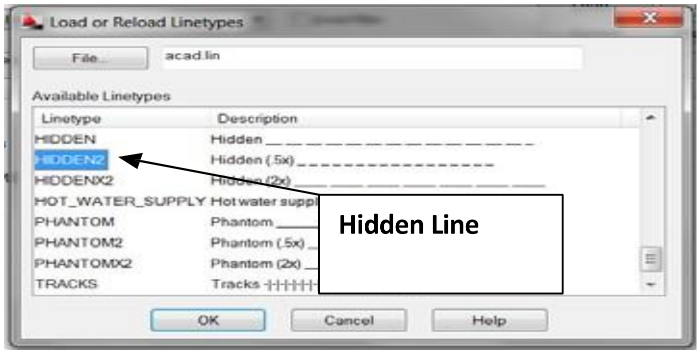
PROJ This layer is used for lines that project from one view to the next. They aid in the placement and size of the views.

FOLD Orthographic views are created by projecting views to the faces of a projection box. The lines in this layer represent the edges of that box.

4. Now assign different line types to each layer. Type LINETYPE at command line. This brings up the line type dialog box shown below:



Click on the **LOAD** button to bring up a selection of line types that can be loaded. The dialog box is shown below:



Search through the file and click on the HIDDEN2 line type. We will use this line type to draw hidden lines in our drawing. Once you have selected the HIDDEN2 name, click on the OK button.

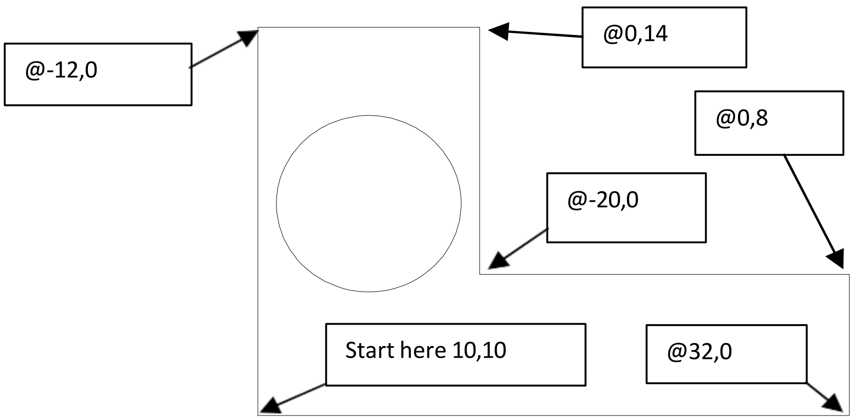
For the HIDE layer, click on the Line Type and select hidden from the list of loaded line types. Click on the line type for the CENTER layer and in the line type dialog box click on the LOAD button. Now search for CENTER2 in the line types, click on it then click on OK to load the line type also. You can now select it for the CENTER layer.

The lines drawn in each layer can have different weights. Click on the line weight for the OBJ layer and set the line weight to 0.3 mm and click on the line weight button. When you have finished that, click on the line weight for FOLD and set the line weight to 0.5 mm. We will use different line weights for the object lines and for the folding lines between views. All other layers will use the DEFAULT line weight which is the thinnest line the graphics device can produce.

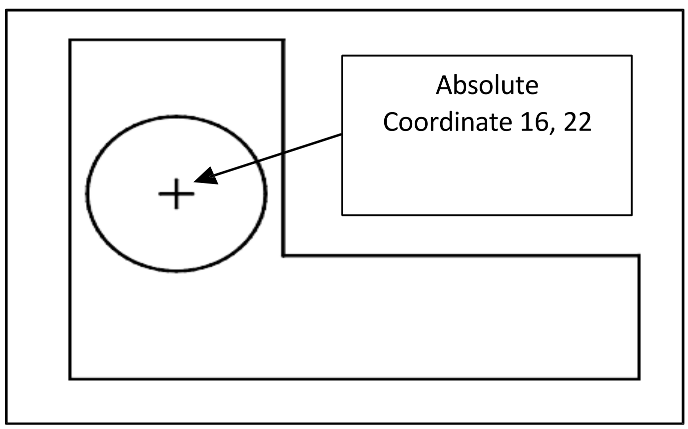
5. Start to draw the front view. Select the OBJ (object) layer by clicking on the OBJ name to highlight it and then click on the green check mark at the top of the dialog box that selects the current layer. Once you have selected the OBJ layer, click on the **X** in the extreme upper left corner of the dialog box to close the dialog box. Either type LINE or click on the line button at the top left of the screen to draw the front view of the object.

LINE

From point:	**10,10**	*{Start the drawing here for convenience}*
To point:	**@32,0**	
To point:	**@0,8**	
To point:	**@-20,0**	
To point:	**@0,14**	
To point:	**@-12,0**	
To point:	**C**	*{To close the outline of the object}*



Next, draw the circle that represents the hole through the object. Note that you started the drawing at coordinates 1, 1 so the center of the circle must be based on these coordinates. That puts the center at 1.6, 2.2.



CIRCLE
3P/2P/TTR/<Center Point>: **16, 22**
Diameter/<Radius>: **D**
Diameter: **10**

The finished drawing with the circle is shown above. This is the front view of the object.

6. Now we will draw the folding lines. Folding lines represent the corners of the projection box used to view orthographic drawings. Switch to the FOLD layer by clicking on the Layer Manage button or by typing LAYER and then selecting FOLD as the current layer. These folding lines are horizontal and vertical and you can force AutoCAD to draw only horizontally or vertical lines by typing the **ORTHO command** as shown below or by clicking on the ORTHO MODE button at the Side bar. You must do this before you draw the folding lines.

ORTHO

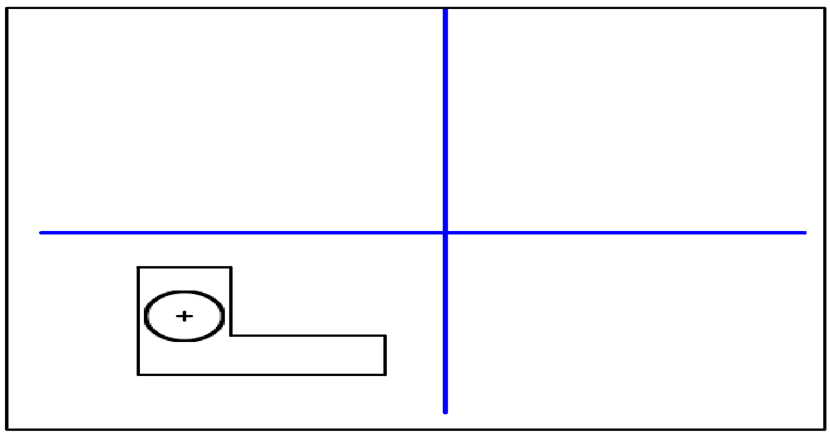
ON/OFF/<off>: **ON** *{Turn the ORTHO setting on so we can only draw horizontal and vertical lines}*

Draw the folding lines with the LINE command. Visually select a starting place for the folding lines. You could locate the lines with coordinates but that is not really necessary.

LINE

From point: *{Click on an appropriate point to draw the horizontal folding line}*

To point:*{Move the mouse to the right and extend the folding line out as far as you think will be necessary. The Ortho Mode is on so the line is horizontal.}*



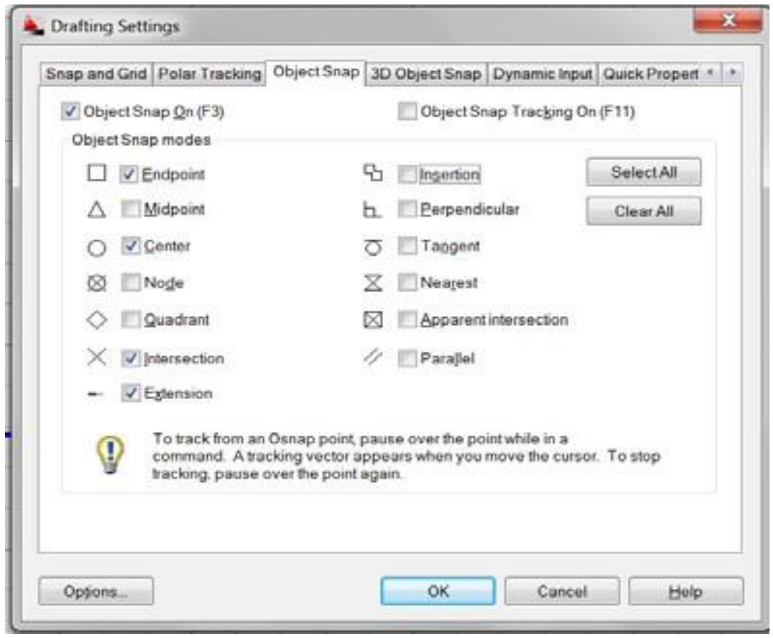
Repeat the process to draw the vertical folding line. After you have drawn this line, you are ready to draw projection lines. The projection lines will help to position and control the size of the top and right side views. The projection lines are used in much the same way as they are when drafting orthographic drawings.

7. Now, you will draw the projection lines. The projection lines are either horizontal or vertical. So the ORTHO ON setting you are current using is appropriate. The projection lines must start at the corners of the front view. You could use coordinates to place the lines in this position, but that would be difficult to do. AutoCAD provides a simpler method called OSNAP tools.

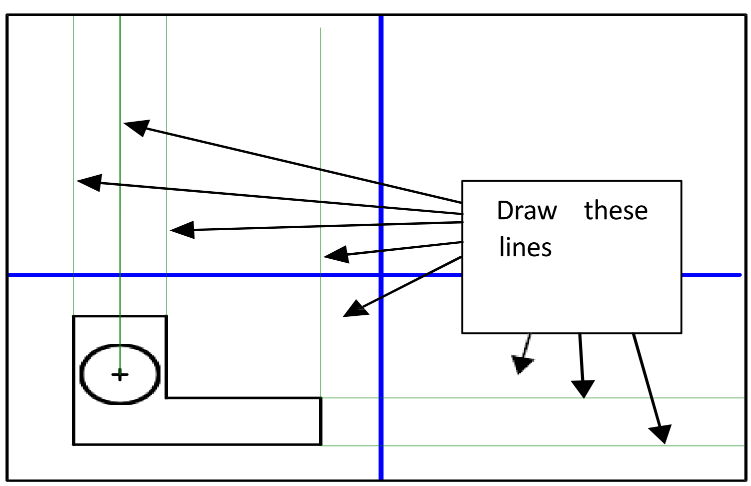
   AutoCAD OSNAP tools allow you to use the coordinates of specific points in the drawing as the coordinates of lines, circles, etc. that you are creating. To use the OSNAP tools, type;

OSNAP

This opens the dialog box shown below. It is used to determine which things the program will snap points to. For example, if the intersection on the left side of the dialog box is marked, the program will look for intersecting lines. If the mouse passes over one of these intersections, the pointer will hesitate and the program will draw a small box indicating it has found an intersection. If you click with the mouse to start a line, arc, circle or any other entity, it will start at the intersection. The dialog box on the right is used to select the things the program will snap to. Mark ***Endpoint*, *Center, Intersection*,** and ***Extension*** then click on OK at the bottom of the dialog box.



Now switch to the **PROJ** layer and use the **LINE** command to draw the projection lines from the front view into both the top and right side view (remember, you are drawing a third angle projection.) You repeat this sequence of commands and draw both vertical and horizontal projection lines. These lines are easy to draw because AutoCAD automatically finds the corners of the object and with ORTHO on you only have to draw with the mouse. When you have finished drawing these lines you can draw both horizontal and vertical projection lines from the center of the circle.

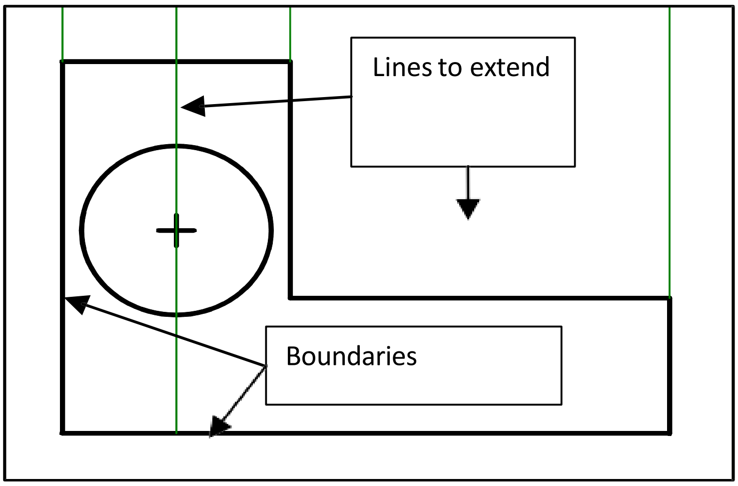


Now project the circle by drawing projection lines from the extreme edges of the circle into both the top and right side views of the object. You can do this by first extending the projection lines starting at the center of the circle through the other side of the circle then projecting the intersection points of these lines and the circle to the other two views. Start with the EXTEND command.

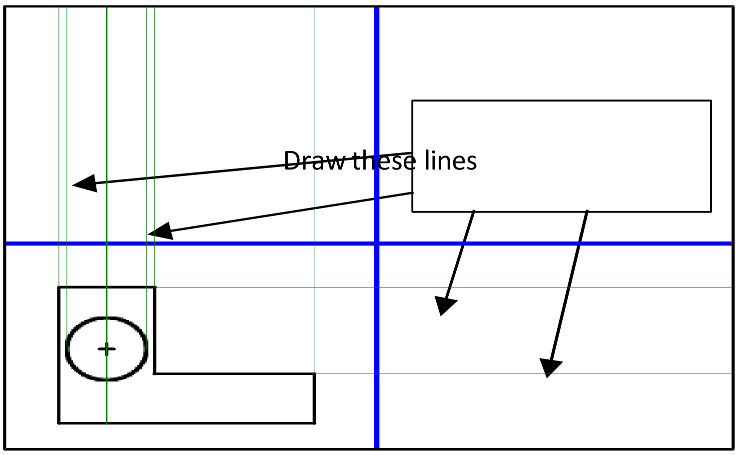
EXTEND

Select boundary edge(s)...	*{Click on the left side and bottom of the object. Press ENTER after you have selected the two edges}*
Select objects to extend:	*{Click on the two lines starting at the center of the circle. After extending these two lines press ENTER}*

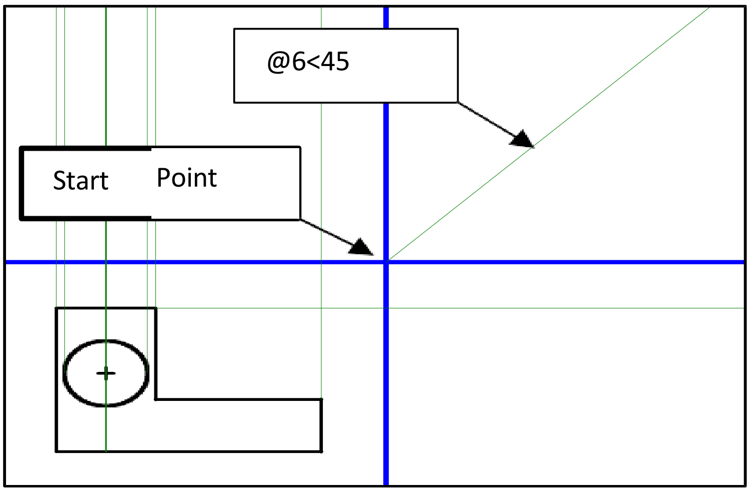
.



Now, project the lines from their intersection with the circle to the other two views. Repeat this operation to extend the projection lines into both the top and right views.



Once you are finished with these lines, draw a 45 degree line extending from the intersection of the two folding lines. AutoCAD can draw using X-Y or polar coordinates. We will use polar coordinates to create this line. Polar coordinates consist of a length and an angle. The length is entered first followed by a < symbol then an angle. The < symbol separates the length from the angle and tells AutoCAD that you are entering a polar coordinate. The angle is specified in degrees measured from the horizontal axis. A positive number is an anticlockwise rotation and a negative number is a clockwise rotation.



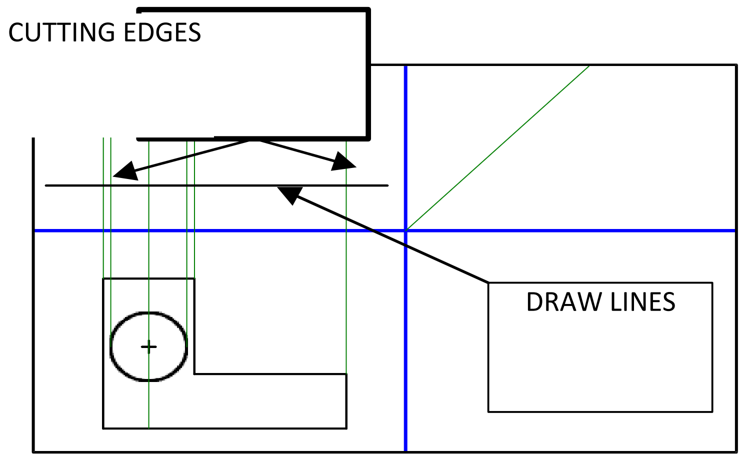
Line

Specify first point: *{click at the intersection of the folding lines}*

Specify next point: @6<45 *{Create a line that is 6 long at an angle of 45 degrees from horizontal}*

Specify next point: ***{End line command}***

8. Now, draw the top view. Use the **LAYER** command to change the current drawing layer back to OBJ and then draw a horizontal line with the mouse. This line represents the front of the object in the top view. This line should extend beyond both the left most and right most vertical projection lines. The vertical placement of the line is not important but the line should be as far from the top-front folding line as the front view is.

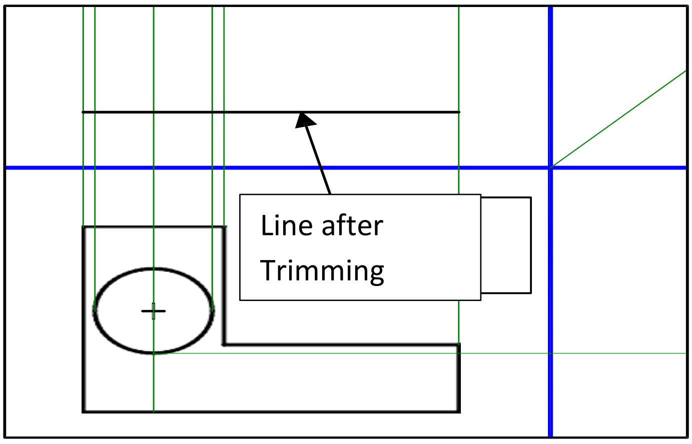


9. Use the **TRIM** command to trim the line you have just drawn so that it has the proper length.

TRIM

Select cutting edge(s) ... *{Click on the left and right projection lines. We will trim the object line to these two lines. Press Return after clicking on the two lines.}*

Select objects to trim: *{Click on the left and right ends of the line wewant to trim. It will be trimmed to the proper length.}*



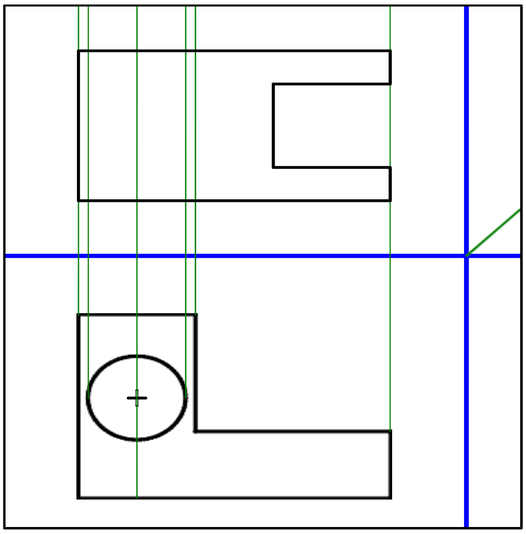
10. Now draw the rest of the top view. Start at the right end of the line you just trimmed and draw around the top view, moving in a counter clockwise direction.

LINE

From point: *{Click on the right end of the line you just trimmed}*

To point: **@0,4**
To point: **@-12,0**
To point: **@0,10**
To point: **@12,0**
To point: **@0,4**

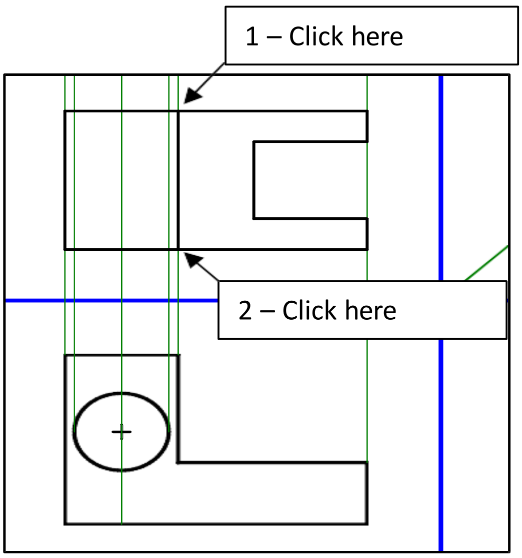
To point:	**@-32,0**
To point:	*{Click on the left end of the first line drawn to complete the outline of the object. Press ENTER to end the command.}*



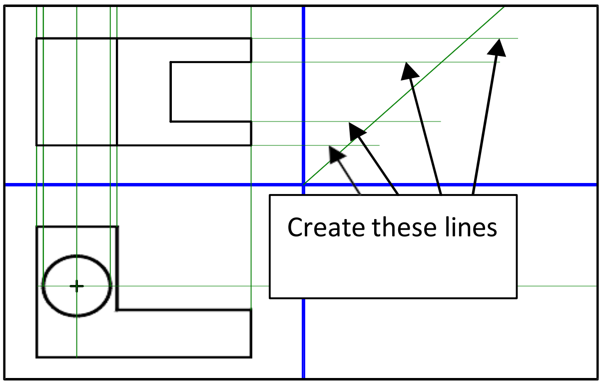
11. Next, add the vertical line defining the right edge of the rectangle containing the hole.

LINE

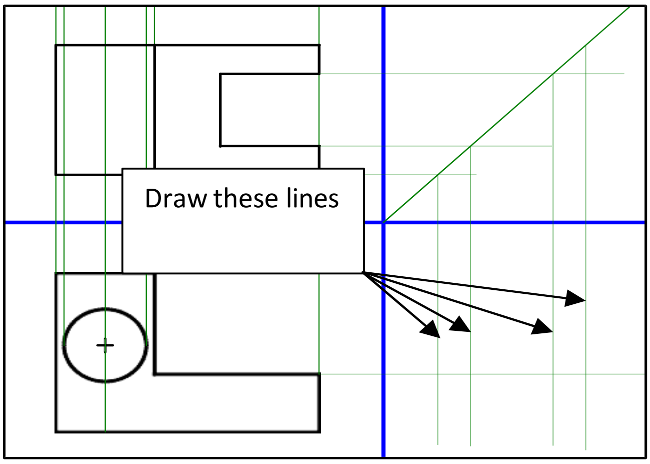
From point:	*{1 - Click on the intersection of the projection line and the front edge of the object.}*
To point:	*{2 - Click on the intersection of the projection line and the back edge of the object.}*



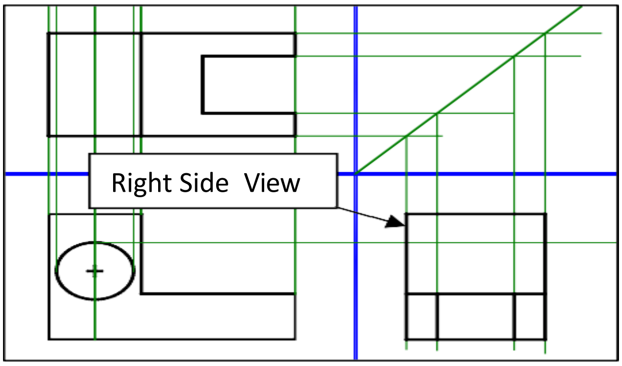
12. Now project lines from the top to draw the right side view. Switch the layer to the PROJ layer and project horizontal lines from the top view to beyond the 45 degree line to the right. This is shown in the figure below. The projection lines have no specific length so you can draw them using the mouse with ORTHO set to ON.



After drawing these projection lines, extend them down to the right side view by starting lines at the intersection of the 45 degree line and the horizontal projection lines from the top view. Draw these projection lines with the mouse as their length is not important. The ORTHO ON setting forces the lines to be vertical.

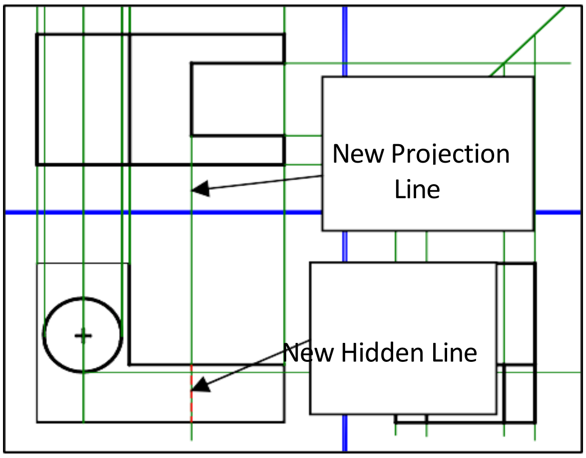


13. The next step is to draw the right view. You could do this with coordinates but it is easier to use the projection lines and let AutoCAD compute the coordinates at the intersections of the projection lines. Just draw by clicking with the mouse at the intersection of the projection lines

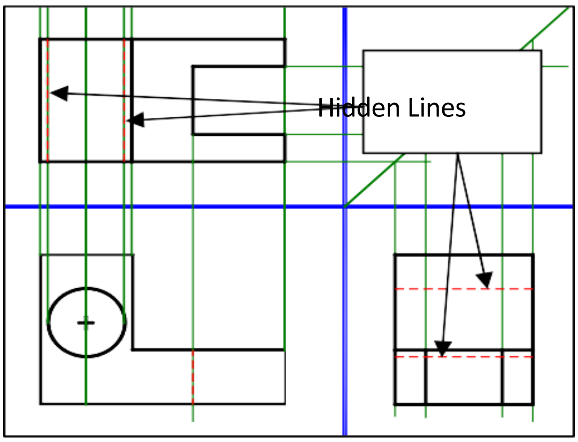


14. Now draw are a hidden line in the front view that shows the notch that is prominent on the right side of the top view and the center lines for the hole. We will draw the hidden line in the front view first. Switch the layer back to PROJ and draw a projection line from the inside of the notch in the top view down to the front view.

Next switch to the HIDE layer and draw the hidden lines in the front view showing notch. The drawing should look like the one shown below:



15. Draw the hidden lines for the hole in both the top and right side views.



16. Draw center lines for the hole. Change the layer to CENTER and draw the lines in the top and right side views using the following sequence of commands.

LINE

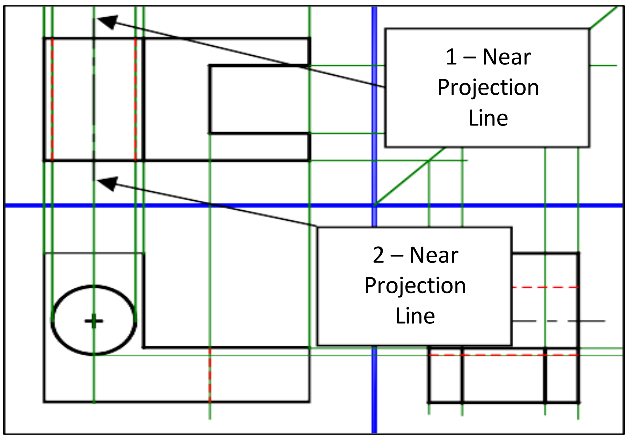
From point: **NEAR**

Of: *{1 - Click on the projection line extending through the center of the hole in the top view. Click where you want the centerline to start}*

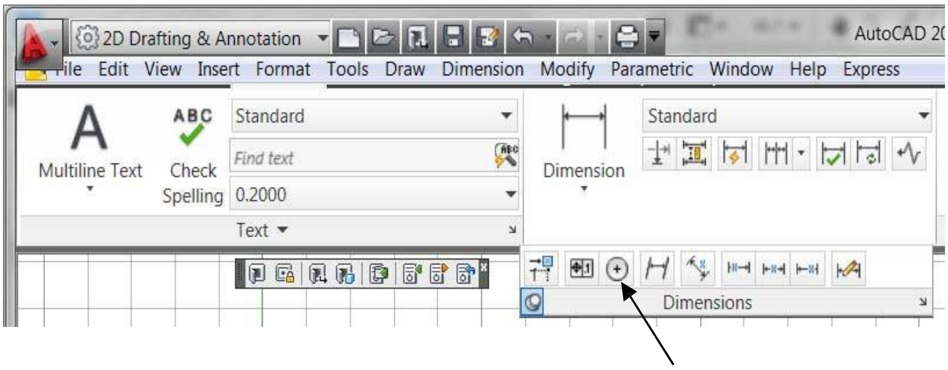
To point: **NEAR**

Of: *{2 - Click on the other end of the centerline}*

Repeat the process for drawing the centerline in the right side view.



17. Next, we will draw the center lines for the hole in the front view. It is difficult to draw the crossing type of center lines with two lines. The dashes usually do not line up correctly and the center lines will not look right. These types of center lines are best drawn using AutoCAD commands designed specifically to draw circle centers.
    Click on the Annotate tab at the top left of the AutoCAD window then on the Dimensions pull down under the dimension section of the ribbon. You should see a pull down menu like the one shown below:



Click on the Circle Center Button then click on the circle in the front view. That will put a small cross in the center of the circle. If the cross is too small or too big, you can adjust its size with the **DIMCEN** command. **DIMCEN** sets the size of the plus that is drawn at the center of the circle. It should be large enough to see but not so large that it dominates the center of the circle. Later on, we will use this circle center for dimensioning our drawings.

dimcen

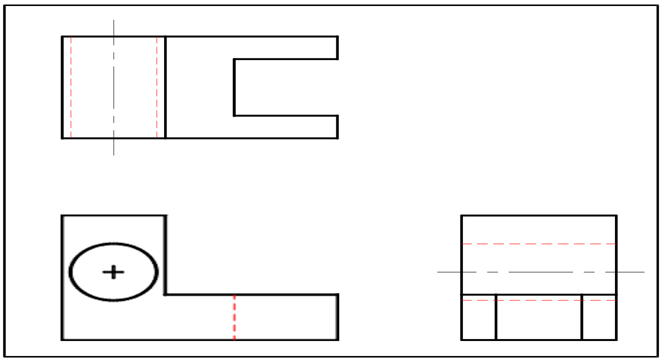
Enter new value for DIMCEN <0.0000>: **.1**

The **DIMCEN** command will not change the size of the plus after it is drawn. If you draw the plus and it is not the right size, delete it, change the **DIMCEN** size, and redraw it. Repeat this until it looks reasonable.

18. Now turn off the display of the projection and folding lines. They are no longer needed. Use the following method:

    1. Type LAYER to start the layer command
    2. Set the current layer to the OBJ layer.
    3. Click on the light bulb next the PROJ layer name.
    4. Click on the light bulb next to the FOLD layer name.
    5. Click on the X in the upper left of the layer dialog box to dismiss the layer box.

    The final drawing is shown below:



If the dashes on the center lines and hidden lines are not properly visible, you can change their length with the LTSCALE command. This command sets a scale factor for the dashes used in non-continuous line.

You can change the scale with the following command:

LTSCALE

New scale factor <1.0000> **0.5** *{Changing the line type scale to 0.5 makes the dash length ½ its current size. A factor smaller than one reduces the size of the dashes and a factor greater than one increases the size. You can experiment with different sizes until the line types look correct.}*

19. Use **DIM** command to dimension all the views.
20. Print the final drawing.

3.2 Dimensioning

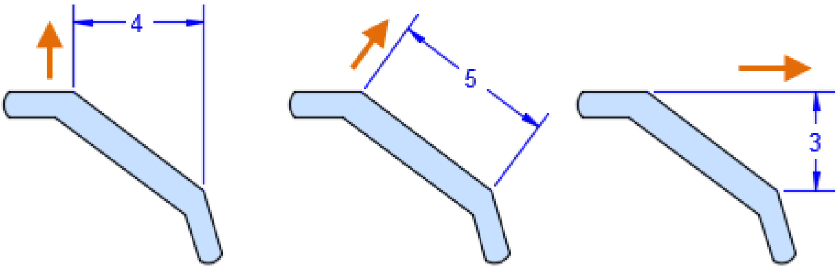
Dimensioning is the process of adding dimensions i.e. length, width, size and angle of objects. Drawing is not complete until dimensions have been added. Using AutoCAD to dimension a drawing saves time and effort since AutoCAD stores all the dimensions in a database as you create the drawing.

Types of Dimensions:

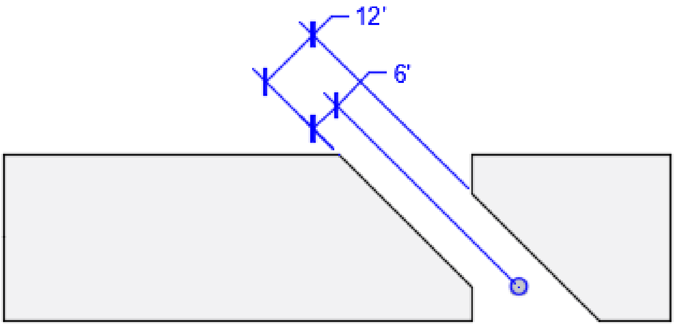
The basic types of dimensioning are linear, radial, angular, ordinate, and arc length. Use the DIM command to create dimensions automatically according to the object type that you want to dimension.

Linear Dimensions:

Linear dimensions can be horizontal, vertical, or aligned. You can create an aligned, horizontal, or vertical dimension with the DIM command depending on how you move the cursor when placing the text.

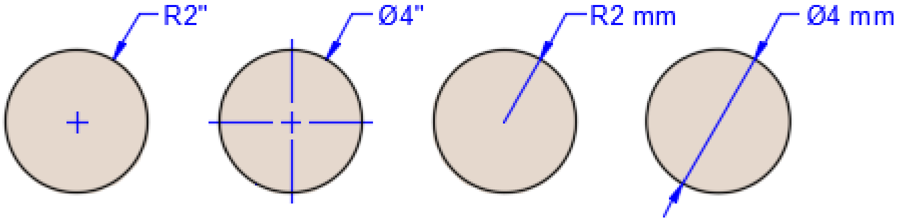


In rotated dimensions, the dimension line is placed at an angle to the extension line origin points. In this example, the angle specified for dimension rotation is equal to the angle of the slot.



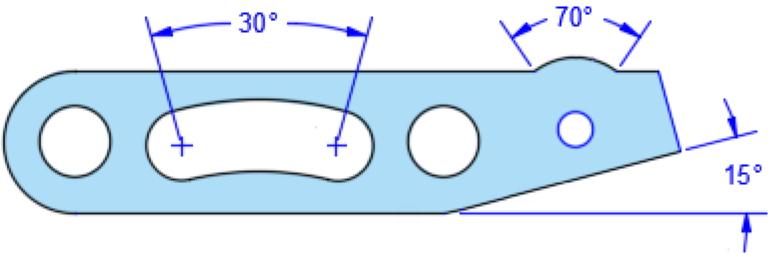
Radial Dimensions

A radial dimension measures the radius or diameter of arcs and circles with an optional centerline or center mark. Several options are displayed in the illustration.



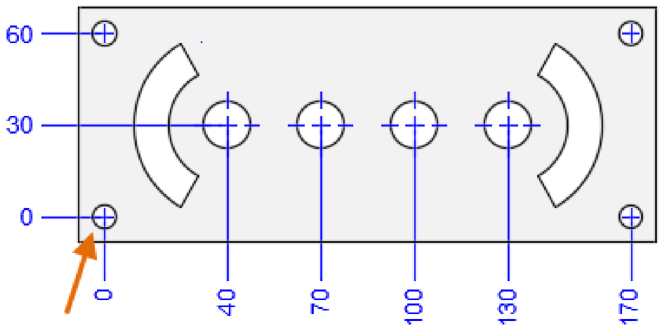
Angular Dimensions

Angular dimensions measure the angle between two selected geometric objects or three points. From left to right, the example shows angular dimensions created using a vertex and two points, an arc, and two lines.



Ordinate Dimensions

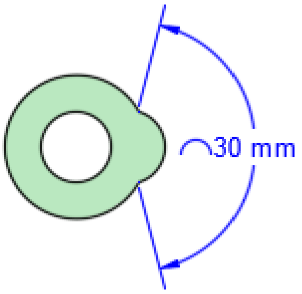
Ordinate dimensions measure the perpendicular distances from an origin point called the *datum*, such as a hole in a part. These dimensions prevent escalating errors by maintaining accurate offsets of the features from the datum.



Arc Length Dimensions

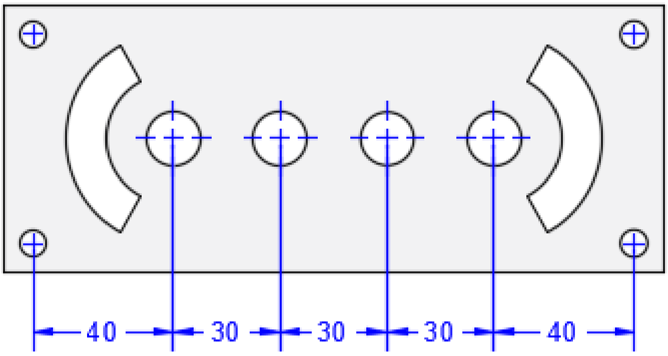
Arc length dimensions measure the distance along an arc or polyline arc segment. Typical uses of arc length dimensions include measuring the travel distance around a cam or indicating the length of a cable.

To differentiate them from linear or angular dimensions, arc length dimensions display an arc symbol by default. The arc symbol, also called a hat or cap, is displayed either above the dimension text or preceding the dimension text.

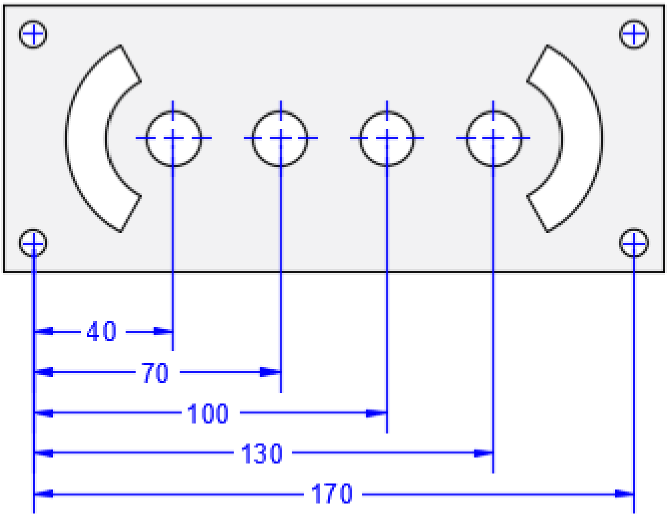


Baseline and Continued Dimensions

Continued dimensions, also called chained dimensions, are multiple dimensions placed end-to-end.



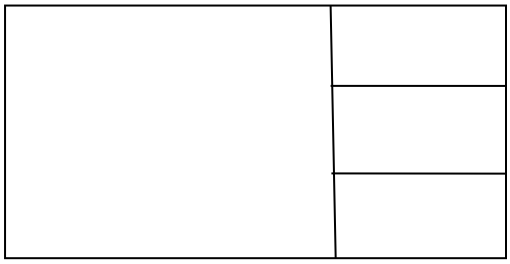
Baseline dimensions are multiple dimensions with offset dimension lines measured from the same location.



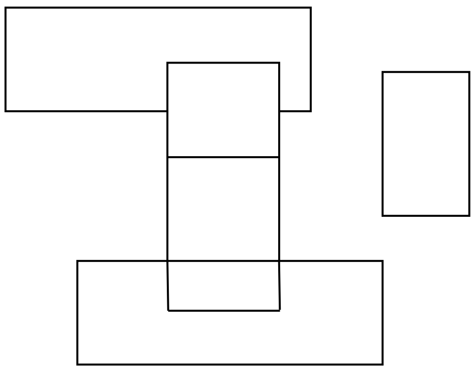
3.3 Plotting the Drawings

The terms *printing* and *plotting* can be used interchangeably for AutoCAD output.
In AutoCAD you work on drawings in Model space or Paper space.
Model space is the area where you create and edit your drawings.

Paper space is the area where you arrange views of the drawing and set up borders and title block. Using the paper space, you can print/plot a drawing with different scales and views on the same paper.
Viewports are rectangular windows that display a part/portion of a drawing's model space. AutoCAD provides two types of viewports – Tiled and Non-tiled.
Tiled viewports are used in model space to display different views side by side. Tiled viewports enable you to divide up the screen into rectangular bounding boxes. You can then show a different view of your drawing in each viewport. The purpose of tiled viewports is to help you draw. For example:



Non-tiled viewports are used in paper space. They can overlap instead of laying side by side. They are treated as objects and can be edited using commands such as Erase, Copy, Move etc.



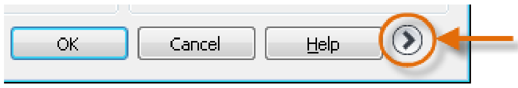
Plotting or Printing Drawing

This involves:

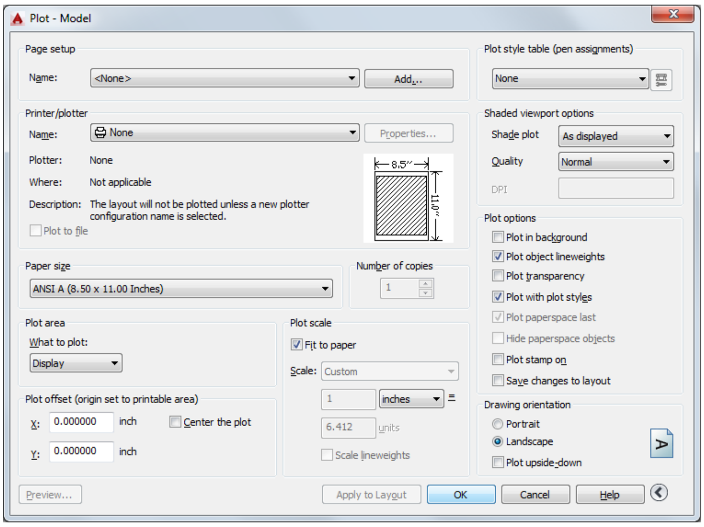
1. Selecting the correct device.
2. Deciding what portion of the drawing to plot.
3. Setting the paper size.

The command to output a drawing is PLOT and you can access it from the Quick Access toolbar.

To display all of the options in the Plot dialog box, click the More Options button.



As you can see, there are a lot of settings and options available for your use.

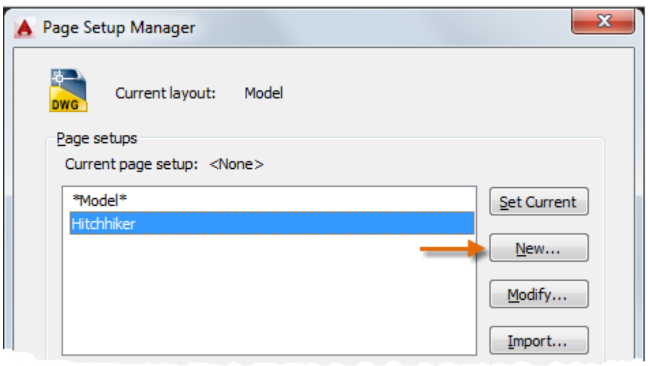


For convenience, you can save and restore collections of these settings by name. These are called *page setups*. With page setups you can store the settings that you need for different printers, printing in gray scales, creating a PDF file from your drawing, and so on.

Create a Page Setup

To open the Page Setup Manager, right-click on the Model tab or a layout tab and choose Page Setup Manager. The command is PAGESETUP.

Each layout tab in your drawing can have an associated page setup. This is convenient when you use more than one output device or format, or if you have several layouts with different sheet sizes in the same drawing.

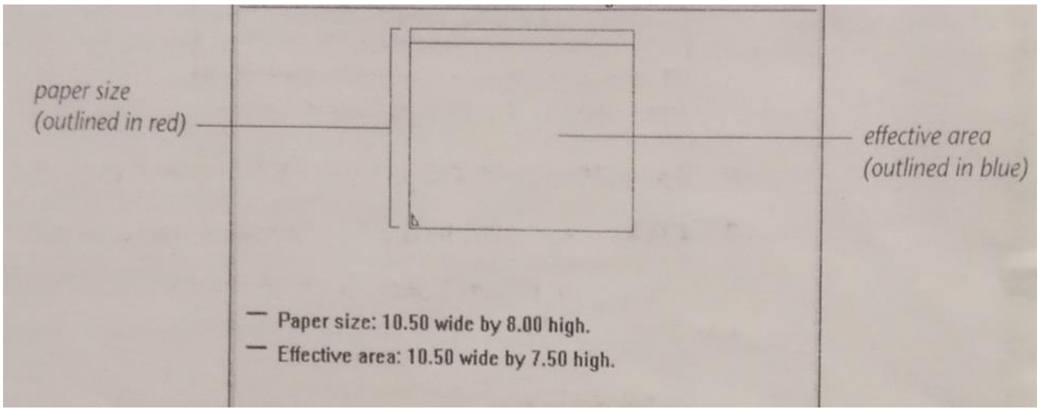


To create a new page setup, click New and enter the name of the new page setup. The Page Setup dialog box that displays next looks like the Plot dialog box. Choose all the options and settings that you want to save.

When you are ready to plot, you simply specify the name of the page setup in the Plot dialog box, and all your plot settings will be restored. In the following illustration, the Plot dialog box is set to use the Hitchhiker page setup, which will output a DWF (Design Web Format) file rather than print to a plotter.

Paper Size: is the size of the paper on which you want to print or plot your drawing. The size of the paper depends upon the printer or plotter you are using. For example if you have A4 size laser printer you can print the drawing on A4 size paper. You can print the drawings on drawing sheets using a plotter. You can also set the orientation of the paper – portrait (Prints vertically) or landscape (Prints horizontally).

Plot Area: is the area in which the drawing will get plotted on the paper. You can see how the drawing will plot on the paper and make necessary adjustments in case of problems.



Plot Offset: This setting changes based on your printer, plotter, or other output. Try centering the plot or adjusting the origin, but remember that printers and plotters have a built-in margin around the edges.

Plot Scale: Choose your plot scale from the drop-down list. A scale such as ¼" = 1'-0" is meant for printing to scale from the Model tab. On a layout tab, you normally print at a 1:1 scale.

Plot style: provides information about processing colors. Colors that look good on your monitor might not be suitable for printing. For example, you might want to create a drawing in color, but create monochrome output as you may have a monochrome printer.

UNIT 4 CNC TURNING

4.1 CNC Machines

The term **CNC** stands for '**computer numerical control**,' and CNC machining employs computer controlled machine tools to remove layers of material from a blank or work piece using various manufacturing processes to produce a custom-designed part. The CNC process is suitable for wide range of materials, including metals, plastics, wood, glass, foam and composites. CNC machining is used for manufacturing parts which require stricter tolerances. Automated nature of CNC machining enables the production with high precision and high accuracy.

[Substandard processes remove layers of material from the work piece to create custom shapes and designs, additive processes assemble layers of materials to produce the desired form, and initial processes deform and displace the stock material to the desired shape.]

A **CNC machine** is a programmable machine capable of performing CNC machining operations autonomously. On a CNC machine every step tool change/ spindle-speed/ axis-movement is controlled through CNC machine control. CNC machinists can only Start/Stop/Control-tool-cutting etc. Every component operation (tool-change/axis-movement/spindle-speed/gear-selection) is stored in the CNC machine-memory as a list of *Instructions* called as Part-Program or just a Program. CNC machining programs are first designed or prepared and then fed to a CNC machine. According to the program, CNC controls the speed and speed of machine tools.

Types of CNC Machines:

1. 3 axis machines - can move in the X, Y and Z axes simultaneously.
2. 3+1 Or 3+2 axis machines – can move along 3 linear axis and 1 rotational axis or 3 linear axis and 2 rotational axis.
3. 4 axis simultaneous: can rotate on one axis while machining in 3-axis at the same time.
4. 5 axis machines - can rotate in two separate axes, and at the same time the 3 linear axes are moving.
5. Lathe
6. Milling
7. Wire EDM
8. Water Jet Cutter
9. Plasma Cutter
10. Laser Cutter
11. 3 D Printing

Classification of CNC Machines:

The CNC machines can be classified as:

1. Based upon the motion type
   - Point to point type (Drilling, Boring Machines)
   - Continuous path or Contouring type (Milling Machine)
2. Based upon the control loops

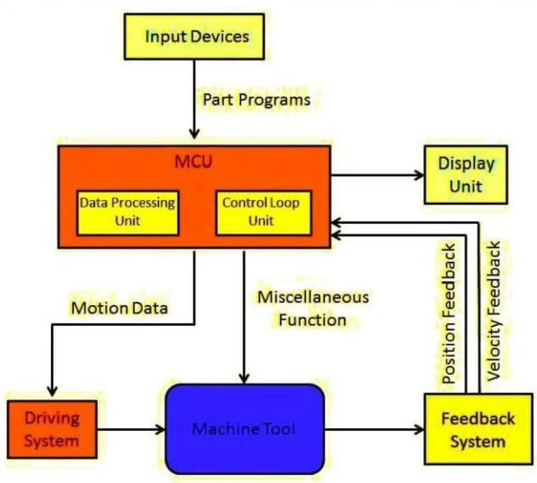
- Open loop (No feedback system, usually used in point to point type machines)
- Close loop (Employs feedback system)
3. Based upon the number of axes
   - 2 axis machines
   - 3 axis machines
   - 4 axis machines
   - 5 axis machines
4. Based upon the power supply
   - Electrical
   - Hydraulic
   - Pneumatic

4.2 Parts of CNC Machine:

The main parts of CNC machine are:

1. Input Device

These are the device that is used to input part programs in a CNC machine. There are three commonly used input devices, and these are punch tape readers, magnetic tape readers, and computers via RS-232-C communication.



CNC Block Diagram

2. Machine Control Unit (MCU)

This is the heart of the CNC machine. It reads the coded instructions from the part program and decodes them to perform all the control function of the CNC machine e.g.

(a) Displacement of slide members
(b) Angular rotation of circular tables
(c) Stop/start of the main spindle
(d) Change the spindle speed
(e) Reverse spindle
(f) Change feed rate of slide members
(g) Rotate tool turret

(h) Change tool
(i) Cutting fluid, ON/OFF
(j) Lock table in position.

3. Machine Tool

A CNC machine tool always has a sliding table and a spindle to control position and speed. The machine tables are controlled in the X and Y-axis direction, and the spindle is controlled in the Z-axis direction.

4. Driving System

The driving system of the CNC machine consists of an amplifier circuit, drive motors, and ball lead screws. The MCU feeds the signals (i.e., position and speed) of each axis into the amplifier circuit.

The control signals are then amplified (amplified) to activate the drive motors. And actuated drive motors rotate the ball lead screw to position the machine table.

5. Feedback System

It is also called a measurement system. The system consists of transducers that act as sensors. It consists of **position and motion transducers** that continuously monitor the position and speed of the cutting tool located at any given moment.

The MCU receives signals from these transducers, and it uses the difference between reference signals and response signals to generate control signals to correct position and motion errors.

6. Display Unit

A monitor is used to display programs, commands, and other useful data of the CNC machine.

7. Bed

On CNC machines, these parts bear all the weight of the machine; i.e. all other components are mounted on it. The bed component is made of hardened materials such as cast iron because the tool turret passes over them in CNC lathe machines.

8. Headstock

The headstock is one of the mains components of the CNC lathe machine. The work pieces are fixed to it. The CNC lathe features motors to help drive the main axle.

9. Tailstock

The tailstock provides additional grip to the work piece when performing operations such as threading, and turning. Support is provided on the end surfaces of the work piece.

10. Tailstock Quill

The tailstock quill helps to centralize the work piece between the headstock and the tailstock.

11. Footswitch or Pedal
The pedal is used to open and close the chuck while trying to hold the component, such as the tailstock quill is moved to the forward and reversed positions.

12. Chuck
The chuck is mounted on the main axle, which gives the tool space to fix.

13. Control Panel
Control panels are one of the important parts of CNC machines that are used to set or feed programs for the operation to be performed on the work pieces.

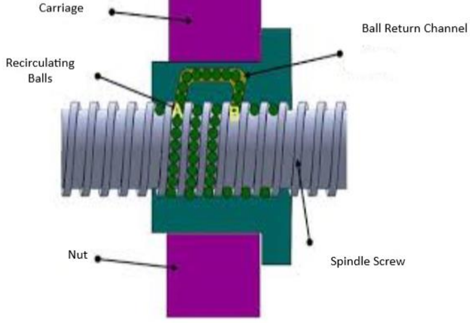
Slide ways:

A slide way is used to control the direction of translational movement of the machine table or the carriage on which the work or tool is held. The machine tools are provided with tables, slides, carriages etc., to carry the work piece or the cutting tool. These tables, slides or carriages slide on the ways fixed on the bed, column or housing of the machine tool.

The slides and slide ways of a machine tool guides the movement of tool relative to the work piece to produce a component to the required dimension. The movement generally takes place in a straight line, but is sometimes angular also. Due to movement of slides over the slide ways friction and wear occurs.
CNC machines are provided with antifriction linear motion slide ways. These slide ways:

(a) Reduce the amount of wear.
(b) Improve the smoothness of the movement.
(c) Reduce friction.
(d) Reduce heat generation.

Recirculating Ball Screw:
Recirculating ball screw is provided in CNC machines to move the tool relative to the work. It consists of a screw spindle, a nut, balls and integrated ball return mechanism as shown in the figure below.



Recirculating Ball Screw

The flanged nut is attached to the moving part of CNC machine tool. As the screw rotates, the nut translates the moving part along the guide ways. However, since the groove in the ball screw is helical, its steel balls roll along the helical groove, and, then, they may go out of the ball nut unless they are arrested at a certain spot. Thus, it is necessary to change their path after they have reached a certain spot by guiding them, one after another, back to their "starting point" (formation of a recirculation path). The recirculation parts play that role. When the screw shaft is rotating, as shown in Figure, a steel ball at point (A) travels 3 turns of screw groove, rolling along the grooves of the screw shaft and the ball nut, and eventually reaches point (B). Whenever the nut strokes on the screw shaft, the balls repeat the same recirculation inside the return tube.

Advantages of recirculating ball screw:

- They have very high efficiency (approx. 90%) No stick and slip phenomenon which results in durability
- It is virtually wear free
- They require Less starting torque
- No heat generation during operation
- Suitable for high speed operations
- Easily preloaded to eliminate backlash
- Smooth and noiseless operation
- High reliability and durability
- Load carrying capacity is more

Disadvantages of recirculating ball screw:

- Costly
- Buckling of screw is serious problem at critical speed
- Require high degree of cleanliness
- Require thin film lubrication for satisfactory operation
- Can have vibrations
- Require periodic overhauling to maintain their efficiency
- Don't have self-locking properties

Feedback Devices:

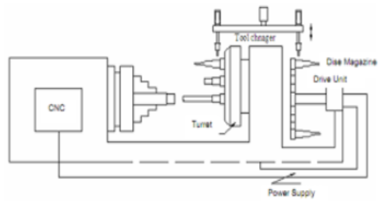
The feedback system is also referred to as the measuring system. It uses position and speed transducers to continuously monitor the position at which the cutting tool is located at any particular instant. The MCU uses the difference between reference signals and feedback signals to generate the control signals for correcting position and speed errors.

Feedback is accomplished by using linear scales and encoders.

Linear scale - measures the absolute/exact position of the slide directly. It is a graduated scale made of either glass or stainless steel along with a measuring surface and scanning unit.

Encoder - An encoder is a sensor of mechanical motion that generates digital signals in response to motion. As an electro-mechanical device, an encoder is able to provide motion control system users with information concerning position, velocity and direction. They convert rotary movement into digital signals. There are two different types of encoders - linear and Rotary. A linear encoder responds to motion along a path, while a rotary encoder responds to rotational motion.

Automatic Tool Changer:
A lot of time is spent in idle movement of tools such as tool engagement and disengagement, tool change and tool setup during the operation of a machine tool. CNC machines are expensive and are designed to perform a number of operations in a single setting of the job. It is important to reduce the idle time so as to utilise these machines to their full capacity. Automatic tool changer is equipment that reduces cycle times by automatically changing tools between cuts.
Automatic tool changer or ATC changes the tool very quickly, reducing the non-productive time. Generally, it is used to improve the capacity of the machine to work with a number of tools. It is also used to change worn out or broken tools. It is one more step towards complete automation. CNC tool changers allow a machine to perform more than one function without requiring an operator to change the tooling.



CNC Turning Machine with Integrated Disc Magazine for Automatic Tool Changing

In a CNC machine, tools are changed through program instructions. The tools are fitted in a tool magazine or drum. When a tool needs to be changed, the drum rotates to an empty position, approaches the old tool and pulls it. Then it again rotates to position the new tool, fits it and then retracts. This is a typical tool changing sequence of an automatic tool changer.

Advantages of Automatic Tool Changer:
(a) Lines changed in seconds instead of hours.
(b) Increased operator safety due to automatic tool changes.
(c) Change tools in seconds for maintenance and repair.
(d) Increases flexibility.
(e) Heavy and large multi-tools are automatically exchanged.

4.3 CNC Tooling:

CNC machines are high cost machines. Their full utilization is important in terms of actual time as well as the material removal capacity. Hence due attention is required in the selection and usage of tooling, namely tool holders, cutting tools and work holding devices. The tools for CNC machines must be quickly changeable to reduce non-cutting time, preset and reset outside the machine, high degree of interchange ability, increased reliability and high rigidity.

Requirements of CNC Tooling:

(a) To give high accuracy.
(b) For variety of operations.
(c) Interchange ability to produce same accuracy.
(d) Flexibility.
(e) Rigidity of tooling to withstand cutting forces.
(f) Rigidity to transmit the power at higher speeds.
(g) Quick changing of tools to keep the down time minimum.

Classification of Cutting Tools:

The cutting tools can be classified on the basis of following:

1. On the Basis of Setting up of Cutting Tool

(a) Preset tools.
(b) Qualified tools.
(c) Semi qualified tools.

2. On the Basis of Cutting Tool Construction

(a) Solid tools.
(b) Brazed tools.
(c) Inserted bit tools.

3. On the Basis of Cutting Tool Material

(a) High speed steel (HSS).
(b) High carbon tool steel (HCS).
(c) Cast alloy.
(d) Cemented carbide.
(e) Ceramics.
(f) Boraon Nitride.
(g) Diamond.
(h) Sialon.

Tool Presetting:

The setting of tools in advance at a place away from the machine tool or offline, in special holders is known as preset tools. A presetting device is used to preset axial and radial positions of the tool tip on the tool holder. Once this is done, the tool holder is ready to be mounted on the machine and produce a known dimension.

Since the generation of actual geometry is taken care of by the CNC part program, which is essentially the coordinates through which the cutting tool tip moves, it is important to know the actual dimensions of the tool when it is placed in the spindle. The relationship of the tool with reference to the tool holding mechanism requires a special attention during CNC machining process. The actual point to be programmed in a CNC part program is the tip of the tool whereas the axes will be moving with respect to a known point in the spindle, e.g. the centre of the spindle in case of machining centres. It becomes therefore necessary to know precisely the deviation of the tool tip from the gauge point on the spindle.

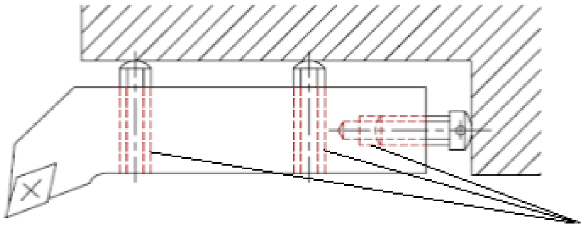
Qualified Tools:

Tool which fits into a location on the machine where its cutting edge is accurately positioned within close limits relative to a specified datum on the tool holder or slide is known as qualified tool.

Qualified Tools for CNC Machines

Semi-qualified Tools:

The qualified tools which can be adjusted to the dimensions by using several adjustable buttons on the tool shank are known as semi qualified tools. These tools demand regular maintenance and calibration for accurate dimensioning.



Semi-qualified Cutting Tool for CNC Machine

Tool Holders:

Tool holders are used for holding the tool firmly during the machining process. The tooling used in CNC machines makes use of shanks of different sizes and shapes thus requiring different types of tool holders for clamping them. This makes the tool changing and tool storage system a difficult one. So a modular tooling system is used in CNC machines so that all the tools have same holding method.

Work Holding Devices:

Fixtures are required to locate and hold the work pieces while machining on CNC machines. The work holding devices should fulfil the following requirements:

(a) Work holding devices must have required accuracy and matching reference surfaces with the reference system.

(b) Work holding devices should be capable of performing a number of operations on different faces in a single setting.

(c) Work holding devices should be capable of quick loading and unloading.

(d) Work holding devices must be fool-proof to avoid incorrect loading of the job.

(e) Work holding devices should have sufficient rigidity to fully withstand the cutting forces.

(f) Work holding devices should be safe in use and loading and unloading.

(g) Work holding devices should have sufficient clamping force for use of full roughing cuts.

(h) Work holding devices should be simple in construction as far as possible.

Chucks and vices are used for holding the job for simple components. For complex shapes special fixtures are used for quick setup of work pieces. Modular fixturing systems are widely used for holding the work pieces in CNC machines.

Automatic Pallet Changer:

A pallet is a movable and interchangeable part of a machine tool which helps to transport raw or finished parts from the machine in order to reduce downtime for part loading/unloading. These pallets simply move for interchanging their positions on the machine table. While machining is being done on a job kept on one pallet, the other pallets are accessible to the operator for clamping and unclamping raw material or finished product. This saves a lot of material handling and set up time, resulting in higher productivity.

4.4 CNC Turning Centres:

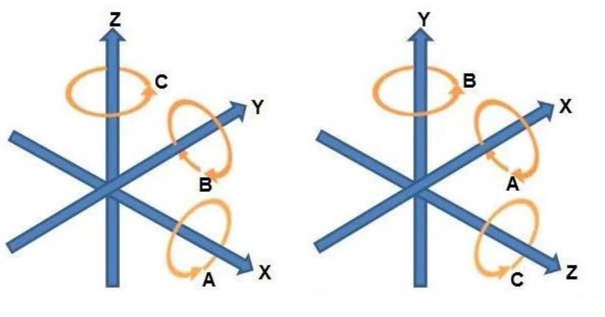
CNC turning centres are advanced computer numerically controlled machines. They can have 3, 4, or even 5 axes, along with a multitude of cutting capabilities, including milling, drilling, tapping, and of course, turning. Often these machines have an enclosed setup to ensure any cut material, coolant, and components remain within the machine.

CNC turning centres are more advanced forms of CNC lathes. Both are machine tools that rotate a bar of material, allowing the cutting tool to remove material from the bar until the desired product is obtained. However, lathes are typically only 2 axis machines and have just one spindle, whereas turning centres can have up to 5 axes and are far more versatile in terms of cutting ability.

Machine Axes in a CNC Turning Centre:

There are up to five axes that a CNC turning centre can operate on, though there are some machines that can operate on 6 axes:

- X Axis: Vertical
- Y Axis: Horizontal
- Z Axis: Depth
- A Axis: Rotation Around the X Axis
- B Axis: Rotation Around the Y Axis
- C Axis: Rotation Around the Z Axis



Vertical Turning Centre Horizontal Turning Centre

Types:

1. Turnmill Centres
   In addition to the turning operations the product may have to undergo operations like drilling cross holes, milling of flats, keyways, slots etc. Turnmill centres can perform both turning and milling operations.
2. Multiple-Axes Turning Centres
   Multiple axes machining is possible with these machines.
3. Vertical Turning Centres
   These have vertical spindle and are suitable for large and heavy work pieces.

4. Twin-Turret Turning Centres
   Two turrets are provided. Machining can be done by two tools simultaneously.
5. Multiple-Spindle Turning Centres
   Multiple spindles are provided to enable high volume production of small and medium sized parts with standard tolerances.

4.5 CNC Part Programming:

Part programming is the procedure by which the sequence of processing steps to be performed on the CNC machine is planned and documented. The part program consists of sequence of instructions that are in form of coded symbols on an input medium. These coded instructions are read by an input device and fed to the controller unit. The control unit converts the coded instructions into a sequence of machine tool actions to perform the actual machining operations.

Structure of a CNC Program:

- A CNC program consists of multiple lines of instructions.
- First line of the program contains the name of the program. The name can contain alphanumerical or numerical characters.
- Each line is called a block.
- Each block contains relevant geometric and technical information required for each machining step.
- The program ends with command M30 or M02.
- The comments, if any, are allowed within the program for identifying an operation. These comments must be set in brackets.
- Each block consists of a block number and a specific control character which informs the CNC control about the necessary action to be performed.

N75	G01	Z-10.75	F0.3	S1800	T03	M08
Number of the block	Word	Word	Word	Word	Word	Word

- A word consists of an address letter and a number with a plus or minus sign. Depending upon the address letter, the number either has a code or a value.

Example	Address	Number	Definition or Meaning
N75	N	75	For the address N, 75 is the number of the block.
G01	G	01	For the address G, 01 is a code. The command "G01" is "Moving the tool along a straight line at infeed speed".
Z-10.75	Z	-10.75	For the address Z, -10.75 is a value. Corresponding to the Command G01 of the preceding row, this means that the tool is to be moved to the position Z = -10.75.

- The sequence of the words in a block is designated as follows:

Sequence	Address	Definition
1	N	Block Number
2	G	Preparatory Functions
3	X, Z	Coordinates of the target point that are needed for travel
4	I, K	Interpolation Parameters are used to define the center of a circle for circular movements
5	F	Feed of the Tool in mm/minute
6	S	Speed of spindle in rotations per minute
7	T	Tool Position
8	M	Miscellaneous Functions

Preparatory Codes:
Preparatory codes are pre-set functions related with the movement of machine axes and the associated geometry. These are also known as G codes and are standardised by ISO. The list of commonly used G codes and their function is given below:

Code	Function
G00	Point-to-point positioning (Rapid Positioning)
G01	Linear interpolation
G02	Clockwise circular interpolation
G03	Anticlockwise circular interpolation
G04	Dwell in seconds
G20	Inch programming
G21	Metric programming
G28	Auto, Return to reference point
G70	Finishing cycle
G71	Stock removal in turning
G72	Stock removal in facing
G73	Pattern repeating cycle
G74	Peck drilling canned cycle
G81	Drilling cycle
G90	Diameter cutting cycle
G92	Threading canned cycle
G94	Facing canned cycle

Miscellaneous Functions:
These functions operate some control on the machine tool and thus affect the running of the machine. These are commonly known as M codes. A standardised list of commonly used M codes and their function is given below:

Code	Function
M02	Program end
M03	Spindle start clockwise
M04	Spindle start anticlockwise

M05	Spindle stop
M06	Tool change
M30	Program end and rewind
M98	Start of subroutine
M99	End of subroutine

Subroutine:

Also known as subprogram, subroutine is useful when repetitive machining operations e.g. making grooves, holes, contours etc. are to be done. There is no need to write part program for performing such repetitive works every time. The program can be written for performing these operations and can be saved as subroutine and can be used by simply recalling it in the main program.

Canned Cycle:

Many times a series of motions are to be repeated a number of times. For example, in a drilling operation the tool has to be positioned above the hole, then move to the required depth and then the tool has move to the top of the hole. The same actions are to be repeated for each of the hole. Instead of writing the program for each hole, a canned or fixed cycle is defined which can repeat all the motions thus avoiding to give the same instructions for each hole.

Feed Function:

The feed function is used to define the feed using the letter F. Feed is the actual speed with which the tool moves along programmed path.

F30 means that the feed rate is specified as 30 mm/minute.

Speed Function:

The speed function defines the speed of the spindle in revolution per minute using the letter S. S1000 means that spindle speed is to be set at 1000 rpm.

Tool Function:

Mostly CNC machines are equipped with turret or tool magazines with automatic tool changer to quickly position the pre-set tools. The tool function is indicated by the letter T followed by digits depending upon the capacity of turret or tool magazine. T8 causes the turret to position tool number 8 in the spindle replacing the present tool. The tool replaced from the spindle is brought back to the empty position created by tool number 8.

Format for commonly used G codes and examples:

G00 – rapid traverse or positioning

- Used to command motion at the machine’s fastest possible speed.
- Used for positioning the tool to and from cutting positions.
- Moves the tool to the position in the work piece coordinate system with an absolute or incremental format

- No machining action takes place.

  Format: G00 X Z (X and Z are the coordinates to which the tool is to be moved)

  For example if present tool position is X = 0 and Z = 0 and the required tool position is Z = -30, the program code will be written as

  G00 X30

  G00 Z-30

G01 – Used to specify straight line/linear motion; Used for actual machining and metal removal.

- Used for positioning the tool to and from cutting positions. Moves the tool to the position in the work piece coordinate system with an absolute or incremental format at a programmed feed rate in straight line.
- Double axis movement is possible with this code. Double axis movement is used for taper turning.

  Format: G01 X_Z_F_ (F is the feed rate)

G02 – Circular Interpolation Motion (Clockwise)

- Used to move the tool in a circular path in clockwise direction.

  Format: G02 X_Z_I_K_F or

  G02 X_Z_ R_F

  I = Distance between start point and centre point of arc along X-axis

  K = Distance between start point and centre point of arc along Z-axis

  R = Radius of the arc

  Example:

  To machine a radius of 10 mm on diameter of 20 mm, starting from centre point i.e. X = 0, Z = 0, up to the length of 10 mm and feed of 30 mm/min, the code will be

  G02 X20 Z-10 I0 K10 F 30 or

  G02 X20 Z-10 R10 F30

G03 - Circular Interpolation Motion (Anti-Clockwise)

- Used to move the tool in a circular path in clockwise direction.

  Format: G03 X_Z_I_K_F or

  G03 X_Z_ R_F

G04 – Dwell

- This code delays the execution of next block by given time.
- Used after the Spindle command as the spindle requires some time to reach to the specified speed.
- The cutting operation starts after this dwell period.

  Format: G04 X_

G20 – Inch programming

- This code allows the dimensions to be specified in Inch.
  Format: G20

G21 – Metric programming

- This code allows the dimensions to be specified in metric system i.e. in mm.
  Format: G21

G28 – Auto return to reference point

- This code moves the tool to the specified reference point.
- The tool movement is rapid.
- Used to take away the tool from the job to a defined reference position.
- No feed rate is specified as it is taken from internally set values.

UNIT 5 CNC MILLING

5.1 CNC Milling Machine:

Milling is the most versatile of machining processes. A CNC Milling machine is a computer controlled machine that can be used to make very precise parts. The machine works by removing material from the work piece with a rotating cutting tool. The machine does this by guiding the tool in all three directions of the Cartesian coordinate system, *i.e.* along the X, Y and Z axis. Milling has more variations in machine types, tooling, and work piece movement than any other machining method. Manual mills require the operator to set all the required machining parameters, change tools, and manually direct all table movement whereas in the CNC milling machines most of these are performed by the machine itself.

A machining centre is a machine for both milling and hole making on a variety of non- round or prismatic shapes. A CNC milling machine performs same functions as manual milling machine and creates arcs, contours and 3D surfaces. CNC machine milling rotates the tool to cut the part and thus can create more complex parts as it is the cutting tool, rather than the material, that is spinning. A machining centre is a more versatile machine tool which can perform various operations such as:

1. It control the movements along 3, 4, 5 axes.
2. It is possible to move column, spindle head, table traverse and also carry out table rotation, table tilting, tool changing, Job changing, Dimension gauging or job probing, coolant supply, by programming codes.
3. The CNC system may also move the axis in either point to point system, straight cut system or continuous path control system.
4. The basic control feature include simultaneous movements along all axes, programmable feed rates, spindle speeds and activation of fixed cycles such as, Drilling, Boring, Tapping, Mirroring, etc.
5. Additional features that increase the control capability or, machine operation include Sequence number display, Auxiliary function display i.e. spindle clamp or declamp, pallet clamp or declamp etc., Manual data input, Tool gauging and selection, Tool length and tool radius compensation, Linear and circular interpolation etc., Activation of different modes of programming / operation i.e. Block by Block, simulation, manual data input, Dry run full sequence run etc.

Construction:

A CNC milling machine basically consists of a input unit, machine control unit and the machine tool. **The input unit** makes use of a computer to prepare the part program. The part program contains the sequence of instructions that include machining parameters and positioning and machine functions.

The machine control unit consists of electronics and control hardware that interpret and execute the part program, monitor the operation and take corrective action where necessary.

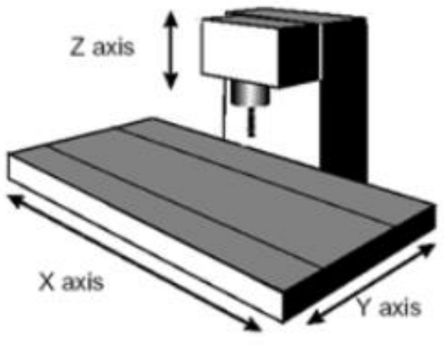
The machine tool consists of mechanical structure that performs the machining operations and the components (motors & actuators) that provide motion of each axis.

The mechanical structure mainly consists of guideways and ball screws and has been already dealt in the earlier topic – CNC Turning.

Axes of Motion:

The Axes of motion describes the relative motion that occurs between the cutting tool and the workpiece. Three main axes of motion for machine tools are referred to as the x, y, and z axes. The milling machine can be programmed on three axes:

- X axis - controls the table movement left or right.
- Y axis - controls the table movement toward or away from the column.
- Z axis - controls the vertical (up or down) movement of the knee or spindle.



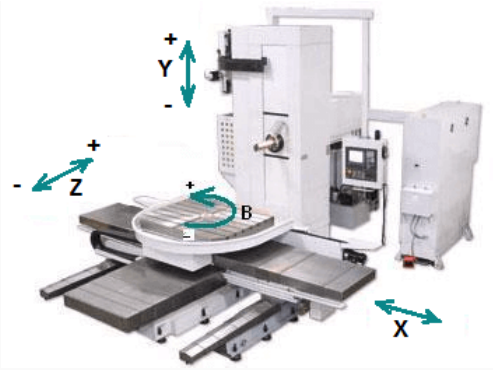
Axes of a Vertical Milling Machine

Milling Centres:

CNC milling centres are of following two types:

1. Horizontal Milling Centre

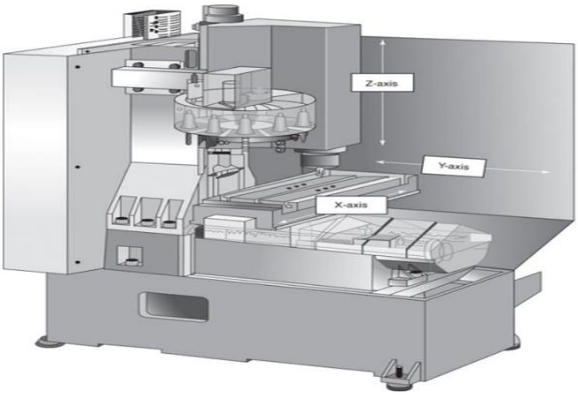
   Horizontal centres have their spindles horizontal spindles and are better suited to larger, boxy work pieces. With a horizontal spindle, a wider variety of work piece shapes are easier to mount and chips fall out of the way better. The horizontal centre's table rotates to present all four sides of a work piece to the tooling.



Horizontal Milling Centre

2. Vertical Milling Centre

   The vertical type is often preferred when work is done on a single face. With the use of rotary tables, more than one side of a work piece, or several work pieces, can be machined without operator intervention. Vertical machining centres using a rotary table have four axes of motion. Three are lineal motions of the table while the fourth is the table's rotary axis.



Vertical Milling Centre

Typical Specification of a CNC Machining Centre

SNo.	Description	Vertical Machining Centre	Horizontal Machining Centre
1	Number of Axes	3 Axes (XYZ)	4 Axes (XYZB)
2	Table Dimensions	780 x 400 mm	500 x 500 mm
3	ATC & No. of Tool	32	36
4	Maximum Travel – X axis	575 mm	725 mm
5	Maximum Travel – Y axis	380 mm	560 mm
6	Maximum Travel – Z axis	470 mm	560 mm
7	Table Indexing angle	-	0.001 degree
8	No. of Slots & Size (width x Pitch)	5nos. & 18x80mm	5nos. & 18x80mm
9	Max. allowable wt. on table	650 Kgs	650 Kgs
10	Spindle Speed	60-8000 rpm	40-4000 rpm
11	Spindle Output	AC 7.5/5.5 KW	AC 11 / 8 KW
12	Spindle nose to table distance – Z axis	150 – 625 mm	150 – 710 mm
13	Spindle Taper	BT 40	BT 40
14	Feed rate Range	2 – 10000 mm/min	2–10000 mm/min
15	Rapid Traverse Rate – XY axis	30000mm/min	30000mm/min
16	Rapid Traverse Rate -Z axis	24000mm/min	24000mm/min
17	Positioning accuracy (X, Y, Z)	+/- 0.005 mm	+/- 0.003 mm
18	Repeatability accuracy (X,Y, Z)	+/-0.002 mm	+/- 0.002 mm
19	Tool Selection	Random Memory	Random Memory
20	Maximum Tool Diameter	80 mm	80 mm
21	Maximum Tool Length	300 mm	350 mm

Tool Holding and Tool Changing Devices:

Tooling:

The milling machine tools perform the cutting process by removing material from a work piece by rotating the cutter, simultaneously, moving it into the work piece. The milling machine can hold single or multiple cutters at the same time. Different types of milling cutters, each with different purposes and cutting abilities, are used. Following milling cutters are commonly used:

1. End milling cutter: is used for producing a flat surface which may be vertical, horizontal or at an angle in reference to the table surface. The end mill is used for plunging, reaming, slotting, drilling, face milling, profile milling, etc.
2. Face milling cutter: is used for flattening and smoothing the surface of work pieces. The cutter rotates about an axis perpendicular to the work surface.
3. T-Slot Cutters: have their teeth perpendicular to the outside diameter and are used for cutting T-shaped slots.
4. Form Milling Cutter: is used for shaping irregular contours, both 2D and 3D. These cutters come in different configurations and shapes. It is ideal for creating helical gears and other complex and intricate surfaces. It is used for grooving, chamfering, and full-radius milling.
5. Side milling cutter: is used for producing a flat vertical surface on the side of a work piece.

Materials used for milling cutters:

1. Carbon steel
2. High speed steel
3. Cemented Carbide and Cermets
4. Ceramics
5. Stellite

Tool Holders:

Since CNC machine tools are versatile machines, a variety of tool holders are used to accomplish the range of machining tasks they are capable of. Some of them are listed below:

1. Collets
2. Side lock holder
3. Stub milling arbor
4. Milling reduction socket
5. Drill chuck arbor
6. Quick change chuck collet
7. Floating reamer holder
8. Keyless drill chuck
9. Split sleeve

Tool Changing Devices:

Tool set up, tool changing, tool engagement and disengagement consumes lot of time during machining operations. To minimise this idle time automatic tool changers and pellets are commonly used in CNC machines. Both of them have been explained in the last unit "CNC Turning".

Work Holding:

Work holding in CNC machines is very important as it becomes necessary to use special fixtures for quick set up of work pieces. Fixtures are used for quickly setting up the work piece. A fixture holds the work piece while being cut.

There are two components for work holding:

- The actual work holding device, such as a milling vise.
- The method of locating the work holding devices and securing that work holding device to the machine. This includes the T-Slots and the modular fixture plates.

Milling Vise

The CNC vise is precision engineered and manufactured with components ground flat and perpendicular to within .0002 inches. Once the vise is bolted to the table and aligned, parts are loaded into the vise and clamped by closing the jaws. The vise can exert tremendous force, so care is taken not to overtighten the vise and deform fragile parts. Vise pressure must be appropriate to the part being held and expected cutting forces.

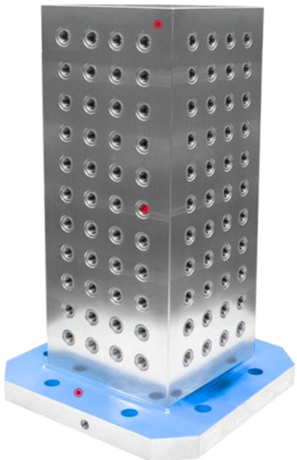
T Slots

T-Slots are the most common method of positioning and holding down the work holding device on the machine tool. They are simple and robust. T-Slot nuts and suitable studs or other fasteners used for attaching to the T slot.

Grid plates

Grid plates are ground aluminium plate that bolts to the top of the machine table. It has precisely drilled threaded holes and bushings at regular intervals to facilitate clamping on the machine table. . It protects the machine table and speeds setups by allowing clamps and other components to be quickly and precisely located anywhere in the workspace. Once installed, the grid plate, generally, remains permanently fixed on the table. Fixtures and vises are installed on top of it.

The grid plate can also in the form of a cube with four parallel faces which can all be used for clamping multiple work pieces. This fixture combined with a rotary table can be used as an indexing fixture.



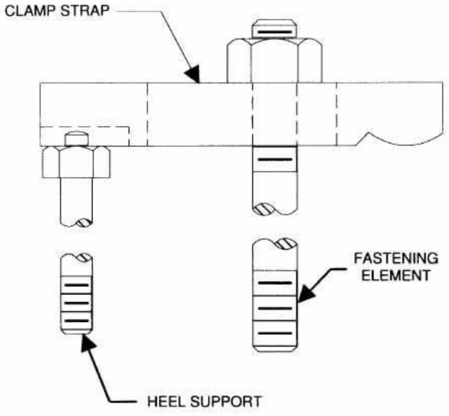
Angle Plate

An Angle plate is a precision ground steel plate that allows the part to be set on its side. Angle plates can point in a direction parallel to either the X or Y axis.

Clamps

There are different types of clamps available in various sizes. They are inexpensive, reusable, and versatile.

Strap Clamps exert downward pressure on the part. They are usually secured to the table by a special bolt that can be positioned anywhere along the T slots in the table. One end of the clamp rests on the part, and the other on a step block. To prevent galling an aluminium part, place a pad of aluminium between the clamp and part.



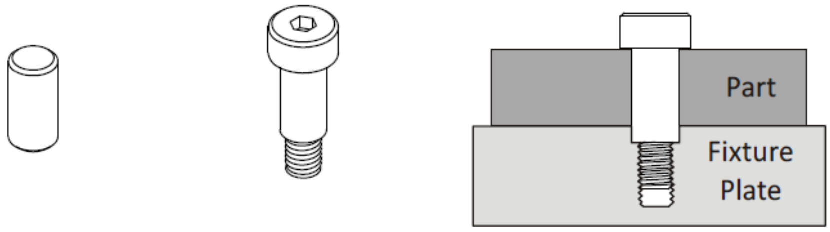
Step blocks have grooves that allow them to be stacked to different heights. Step blocks are used to support one end of the strap clamp.

Step Clamps are similar to Strap clamps, but include grooves that interlock with a single step block.

Toe Clamps bolt to the machine table. The nose of the clamp includes grooves to grip the part. It is extended by turning a screw, forcing the clamp against the part and downward.

Shoulder Bolts and Dowel Pins

Dowel pins are used to precisely locate a part. They include a small chamfer to make insertion easier. Shoulder bolts both locate and grip the part. The ground shoulder on the bolt slides into a counter bore in the fixture.



Dowel Pin Shoulder Bolt Using Shoulder Bolt

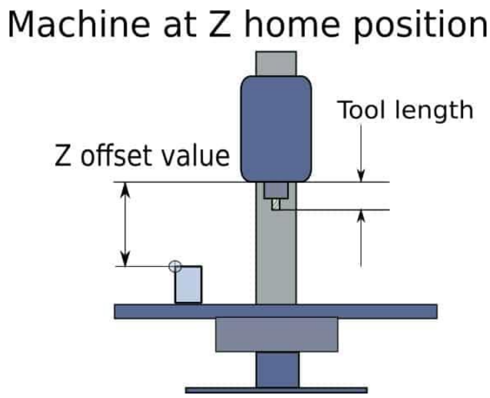
5.2 CNC Part Programming:

This has been already covered in last unit "CNC Turning". Please refer the previous unit.

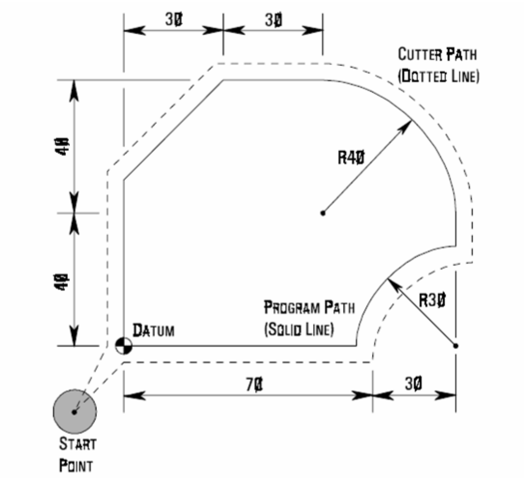
5.3 Compensations

1. **Tool Length Compensation:**

   In CNC programming when more than one tool is used the programming becomes complicated, if the programmer has to take care of the individual tool lengths. All tool lengths are measured in the assembled state using a tool pre-setter. For the tool being used, the difference in length with respect to pre-setting tool is recorded and is manually entered and stored with the associated tool number. Whenever that tool comes into action the respective compensation values are automatically taken into account in the tool motion.



2. **Cutter Radius Compensation:**
In contouring operations, it becomes necessary to calculate the tool path by offsetting the contour by an amount equal to the radius of the cutter. Whenever the cutter size changes, the program will require editing. So compensation equal to the radius of the cutter is stored in the control system. Now no change in the program will be required and it will be as if the program has been written with a cutter of zero radius.



3. **Pitch Error Compensation:**

Causes of pitch error

1. The ball screw pair is at the last stage of the transmission chain of the feed system. There are various errors in the screw and nut, such as the cumulative error of the thread pitch, the error of the thread raceway, the error of the diameter, etc. The cumulative error of the screw pitch will cause the deviation of the target value of the machine tool.
2. During the assembly process of the ball screw, due to the dual support structure, the screw is elongated in the axial direction, resulting in an increase in the error of the screw pitch and a deviation of the target value of the machine tool.
3. In the assembly process of the machine tool, the error of the parallelism between the screw axis and the machine tool guide rail will cause the deviation of the target value of the machine tool.

Pitch error compensation increases or decreases the number of pulses of the command value by adjusting the parameters of the numerical control system, so that the actual distance moved by the machine tool is close to the command movement distance to improve the positioning accuracy of the machine tool.