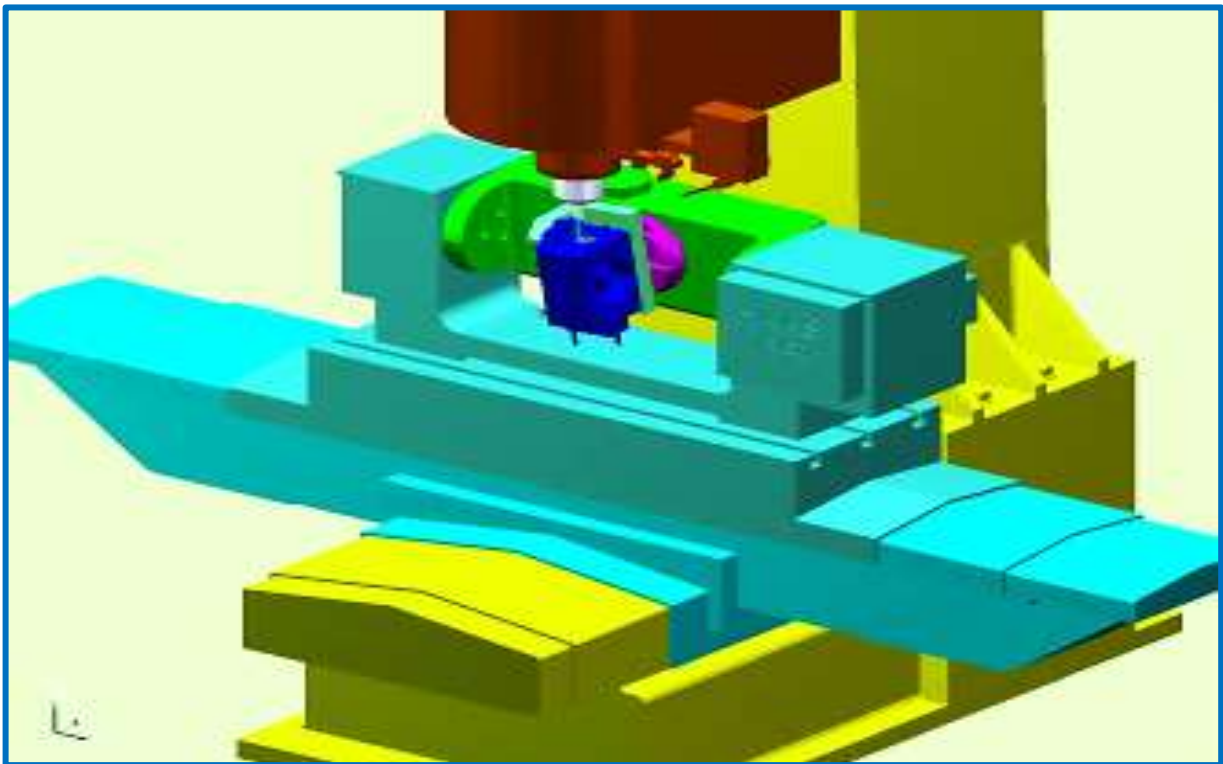**PDHonline Course M501 (8 PDH)**

CNC Machining Technology Fundamentals

Instructor: Jurandir Primo, PE

2013

PDH Online | PDH Center

5272 Meadow Estates Drive
Fairfax, VA 22030-6658
Phone: 703-988-0088

CNC MACHINING TECHNOLOGY – FUNDAMENTALS

CONTENTS:

OBS.: This is a professional didactic manual. It´s highly recommended to download and print the course content for your study, before answering the quiz questions.

INTRODUCTION:

CNC milling machines are computer controlled horizontal or vertical machines, also called machining centers, with ability to move the spindle along an X, Y, and Z-axis. This freedom permits their use in common or difficult machining, die sinking, engraving, and manufacturing 3D surfaces, such as moldings and sculptures. When combined with the use of conical tools or ball nose cutters, it significantly improves the milling precision without impacting speed, providing a cost-efficient alternative to most machining works.

Advanced CNC milling machines or multiaxis machines have two more axes and may have B and C axis, allowing mounted workpieces to be rotated on a rotating table, with asymmetric and eccentric turning. The fifth axis (or B axis) controls the tilt of the tool itself. When all these axes are used in conjunction with each other, extremely complicated geometries, even organic geometries, such as, a human head can be made with relative ease with these machines.

Five-axis machining center with rotating table and computer interface

These machines have been developed from former basic NC (Numerical Control) machines, a computerized form of CNC machines, used in industry for more than 40 years. The Numerical Control (NC) is a method of automatically operating a manufacturing machine based on a code of letters, numbers, and special characters. A group of instructions (called programs) is used to guide the machine for a set of desired operations.

The NC machines can be programmed manually, and the programs are translated to corresponding electrical signals for input to motors that run the machine. If a computer is used to create a program, the process is known as **Computer-Aided Programming**. Traditionally, NC systems have been composed of the following components:

Tape punch: Converts written instructions into a corresponding hole pattern, punched into a tape. Older units used a typewriter device called *Flexowriter*, and later devices included a microcomputer coupled with a tape punch unit.
Tape reader: Reads the hole pattern on the tape and converts into a corresponding electrical signal code.

Controller: Receives the electrical signal code from a tape reader and subsequently causes the NC machine to respond.

NC machine: Responds to programmed signals from the controller. Accordingly, the machine executes the required motions to manufacture a part (spindle rotation on/off, table and or spindle movement along programmed axis directions).

A real breakthrough was achieved around 1965, when Numerical Control machines were fitted with mini-computers and introduced the name Computer Numerical Control (CNC). The basic steps in the process of implementing automation in any manufacturing industry are to manufacture parts or components through automation using machines and machine tools with little human intervention.

The CNC is nothing more than a microprocessor based control system that accepts a set of program instructions, processes and sends output control information to a machine tool, accepts feedback information acquired from a transducer placed on the machine tool and based on the instructions and feedback, assures that proper motion, speed and operation occur. Some of the important parts of CNC machines are Machine structure, guide ways, feed drives, spindle and Spindle bearings, measuring systems, controls, software and operator interface, gauging, tool monitoring.

Advantages of CNC Compared with NC:

1. Reduction in the hardware necessary to add a machine function. New functions can be programmed into the MCU as software.
2. The CNC program can be written, stored, and executed directly at the CNC machine.
3. Any portion of an entered CNC program can be played back and edited at will. Tool motions can be electronically displayed upon playback.
4. Many different CNC programs can be stored in the MCU.
5. Several CNC machines can be linked together to a main computer. Programs written via the main computer can be downloaded to any CNC machine in the network, known as direct numerical control or DNC.
6. Several DNC systems can also be networked to form a large distributive numerical control system.
7. The CNC program can be input from flash or floppy disks or downloaded from local area networks.
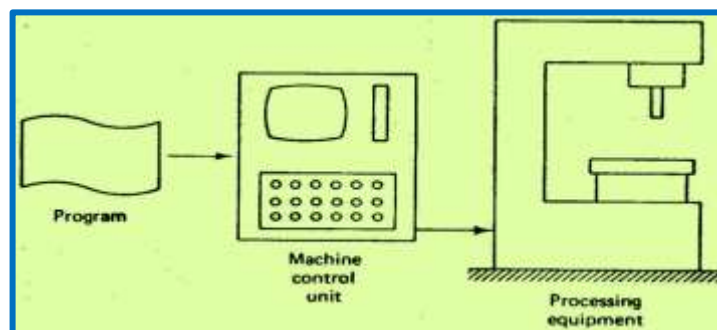
CNC MACHINING CHARACTERISTICS:

Thus, CNC machines are NC machines with the added feature of an onboard computer, often referred to as the **Machine Control Unit (MCU)**, usually hardwired, which means that all machine functions are con-

trolled by physical electronic elements built into the controller. The onboard computer, on the other hand, is "soft" wired, which means the machine functions are encoded into the computer at the time of manufacture, and they will not be erased when the CNC machine is turned off.

The computer memory that holds such information is known as **ROM** (Read-Only Memory). The MCU has an **alphanumeric keyboard** for direct or Manual Data Input (**MDI**) of part programs. Such programs are stored in a memory called **RAM** (Random-Access Memory), part of the computer that can be played back, edited, and processed by the program control.

All programs residing in RAM, however, are lost when the CNC machine is turned off. These programs can be saved on auxiliary storage devices such as, a punched tape, magnetic tape, or magnetic disk. Newer MCU units have graphics screens that can display not only the CNC program but the cutter paths generated and any errors in the program. The information stored in the computer can be read by automatic means and converted into electrical signals, which operate the electrically controlled servo systems.

This electrically controlled servo systems permits the slides of a machine tool to be driven simultaneously and at the appropriate feeds and direction, so that, complex shapes can be cut, often with a single operation and without the need to reorient the work piece. Computer Numerical Control can be applied to Milling machines, Lathe machines, Grinding machines, Boring machines, Flame cutters, Drilling machines, etc. A CNC system basically consists of the following:

(a) Central processing unit (CPU);
(b) Servo control unit;
(c) Operator control panel;
(d) Machine control panel;
(e) Programmable logic controller;

Central Processing Unit (CPU):

The Central Processing Unit (CPU) is the brain of the MCU which manages the other components, based on software contained in its main memory (RAM). The control section retrieves commands and data from memory and generates signals to activate other components, that is, it sequences, coordinates, and regulates all the activities of the MCU computer. All the compensation required for machining acquires (as lead screw pitch error, tool wear out, backlashes) are calculated by the CPU depending upon the corresponding inputs made available to the system.

The CPU is the heart of a CNC system. It accepts the information stored in the ROM memory as part of the program. This data is decoded and transformed into specific position control and velocity signals. So, it also oversees the movement of the control axis or spindle and when this does not match with the programmed values, a corrective action is taken.

The Central Processing Unit or CPU which contains all the circuitry that the computer needs to manipulate data and execute instructions. The CPU is amazingly small given the immense amount of circuitry it contains. The circuits of a computer are made of gates, with tiny components called transistors, and a modern

CPU has millions and millions of transistors in its circuitry. The **CPU** is composed of **five basic** components: RAM, Registers, Buses, the ALU, and the Control Unit, shown in the diagram below, a top view of a simple CPU with 16 bytes of RAM. This image, [Intel 2000] shows just how compact a CPU can be.
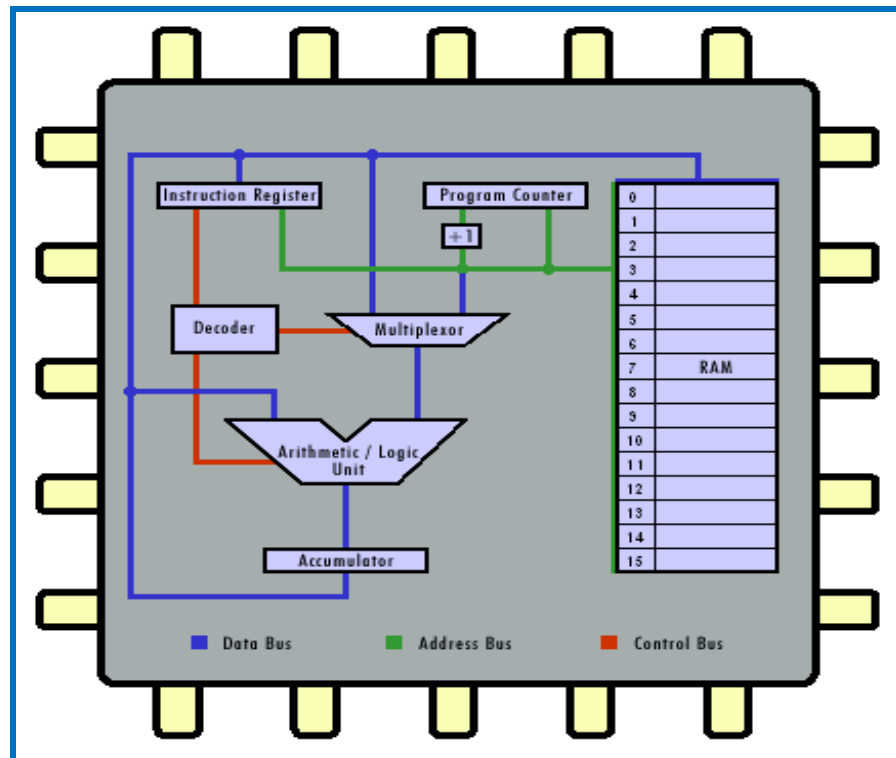
RAM: This component is created from combining latches with a decoder. The latches create circuitry that can remember while the decoder creates a way for individual memory locations to be selected.

Registers: These components are special memory locations that can be accessed very fast. Three registers are shown: the Instruction Register (IR), the Program Counter (PC), and the Accumulator.

Buses: These components are the information highway for the CPU. Buses are bundles of tiny wires that carry data between components. The three most important buses are the address, the data, and the control buses.

ALU: The Arithmetic Logic Unit performs all the mathematical calculations of the CPU. It is composed of complex circuitry similar to the adder presented in the previous lesson. The Arithmetic Logic Unit (ALU) consists of the circuitry to perform various calculations (addition, subtraction, and multiplication), counting, and logical functions required by software residing in memory. It is connected to main memory of the system data bus.

Control Unit: This component is responsible for directing the flow of instructions and data within the CPU. The Control Unit is actually built of many other selection circuits such as decoders and multiplexors. In the diagram above, the Decoder and the Multiplexor compose the Control Unit.

Servo Control Unit:

The Servo Control Unit receives a command signal, generated by the CPU for the axis movement, amplifies the signal, and transmits the electric current to a servo motor in order to produce a movement proportional to the command signal. Typically the command signal represents a desired velocity, but can also represent a desired torque or position. The servo control compares the actual motor status with the commanded motor status and alters the voltage frequency, or pulse width, to the motor to correct for any deviation from the commanded status.
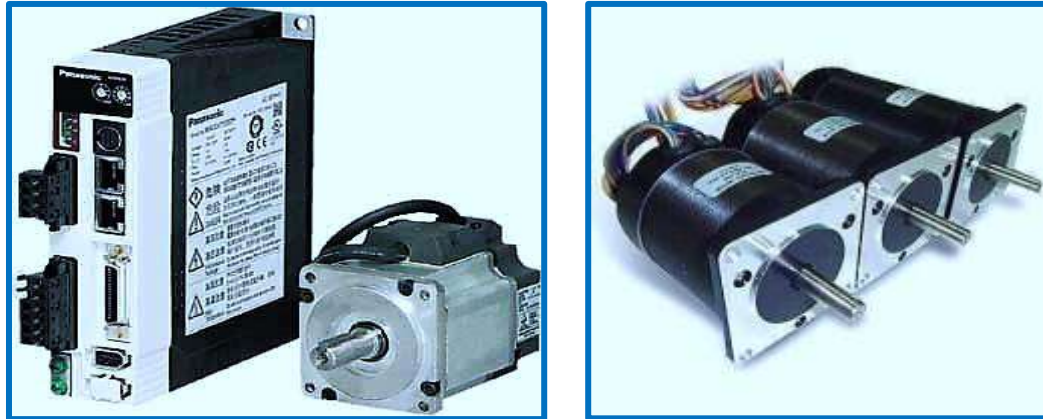
In a properly configured control, the servo motor rotates at a velocity, very closely to the velocity of the signal being received by the servo from the control system. Several parameters, such as stiffness (also known as proportional gain), damping (also known as derivative gain), and feedback gain, can be adjusted to achieve this desired performance. The process of adjusting these parameters is called performance tuning. The servo control unit also receives the position feedback signals for the actual movement of the machine tool axes, from feedback devices (as linear scales, rotary encoders, revolvers, etc.).

Operator Control Panel:

Is the group of controls on a CNC machine that allow an operator to control machine components manually, sometimes called the Machine Panel, which provides the user interface to facilitate a two way communication between the user, CNC system and a machine tool, commonly consisting of two parts: Video display unit and Keyboard.

The operator control panel is an advanced warning, communication, and automation device of "all in one" type, which also serves for interaction of the system with the operating staff, manufactured in three versions: Basic, Professional, and Enterprise. They differ in functions, methods of control, communication interfaces, and options to place additional orders or additional communication modules.

Machine Control Panel:

It is the direct interface between the operator and the NC system, enabling the operation of the machine through the CNC system. Every industrial process control room is provided with a monitor panel, which displays and allows the desired setting data storage, in a great variety of options of widths, length, and height, manufactured for any range of application control purpose.

In CNC machining the control panel monitors the axis the motion, spindle function or tool function on a machine tool, depending upon the part program stored in the memory. Prior to the starting of the machining process, the machine tool should first be prepared with some specific tasks such as, establishing a correct reference point, loading the system memory with the required part program, loading and checking of tool offsets, zero offsets, etc.
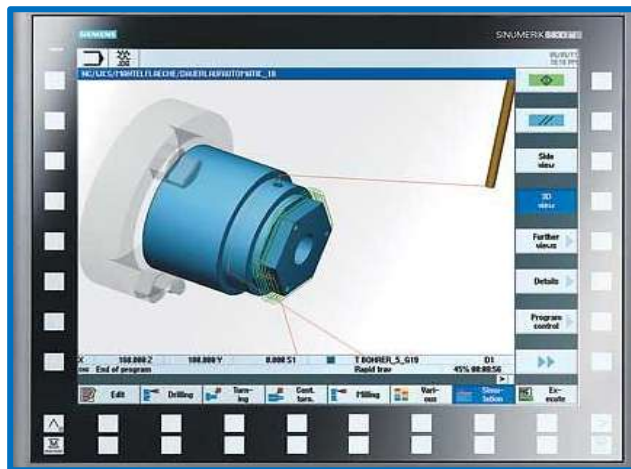
Programmable Logic Controller (PLC):

A Programmable Logic Controller, PLC or Programmable Controller is a digital computer used for automation of electromechanical processes, such as control of machinery on factory assembly lines or light fixtures. The PLC is designed for multiple inputs and output arrangements. PLC programs are typically written for a special application on a personal computer, then downloaded by a direct-connection cable or over a network to the PLC, stored either in a battery-backed-up RAM or some other non-volatile flash memory.

A PLC matches the NC to the machine, as a replacement for hard wired relay control panels. PLC's are now available with increased functions, more memory and larger input/output capabilities. In the CPU, all the decisions are made relative to controlling a machine or a process. The CPU receives input data, performs logical decisions based upon stored programs and drives the output connection to a computer for hierarchical control are done through CPU.
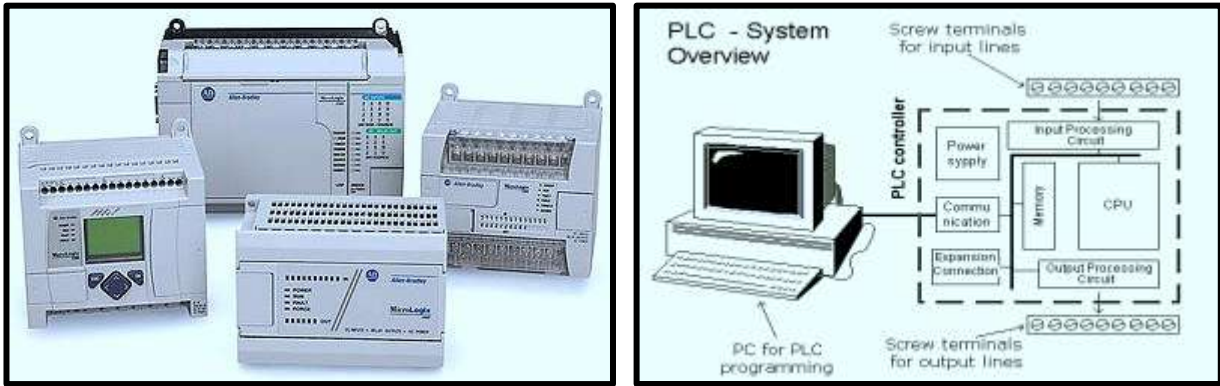
The MCU also consists of subsystems interconnected by means of a system bus, which communicates data and signals among the components of a network. The subsystems are:

(1) Memory;
(2) Input/output Interface;
(3) Controls for Machine Tool Axes and Spindle Speed;
(4) Sequence Controls for Other Machine Tool Functions.

Memory:

The CNC memory can be divided into two categories: primary memory, and secondary memory. Main memory (also known as primary storage) consists of a **ROM** (Read-Only Memory) and **RAM** (Random Access Memory) devices. Operating system software and machine interface programs are generally **stored in ROM** and these programs are usually installed by the manufacturer of the MCU.

Numerical **control part programs** are stored in RAM devices, and **can be erased** and replaced by new programs **as jobs are changed**. Common among the secondary memory devices, are the hard disks and portable devices which have **replaced** most of the punched paper tape, traditionally used to store part programs. Hard disks are high-capacity storage devices permanently installed in the CNC machine control unit. The CNC secondary memory is also used to store part programs, macros, and other software.

Input/Output Interface:

The I/O interface provides communication software between the various components of the CNC system, other computer systems, and the machine operator. As its name suggests, The I/O interface transmits and receives data and signals to and from external devices. The control panel is the basic interface by which the machine operator communicates to the CNC system.

This is used to **enter commands** related to part program editing, MCU operating mode (e.g., program control vs. manual control), **speeds and feeds, cutting fluid pump on/off and similar functions**. Either an **alphanumeric keypad or keyboard** is usually included in the operator control panel. The I/O interface also includes a display [may be a Cathode Ray Tube (CRT) or Light Emitting Diode (LED monitors)] for communication of data and information from the MCU to the machine operator.

The display is used to indicate current status of the program as it is being executed and to warn the operator of any malfunctions in the CNC system. Programs can also be entered manually by the machine operator or stored at a central computer site and transmitted via local area network (LAN) to the CNC system.

Controls for Machine Tool Axis and Spindle Speed:

These are hardware components that control the position and velocity (feed rate) of each machine axis as well as the rotational speed of the machine tool spindle. The control signals generated by MCU must be converted to a form and power level suited to the particular position control systems used to drive the machine axes. Positioning systems can be classified as open loop or closed loop, and different hardware components are required in each case.

Depending on the type of machine tool, the spindle is used to drive either (1) workpiece or (2) a rotating cutter. Turning exemplifies the first case, whereas milling and drilling exemplify the second. Spindle speed is a programmed parameter for most CNC machine tools. Spindle speed components in the MCU usually consist of a drive control circuit and a feedback sensor interface. The particular hardware components depend on the type of spindle drive.

Sequence Controls for Other Machine Tool Functions:

Beyond control of table position, feed rate, and spindle speed, several additional functions are accomplished under part program control. These auxiliary functions are generally on/off (binary) actuations, interlocks, and discrete numerical data. To avoid overloading the CPU, a programmable logic controller is sometimes used to manage the I/O interface for these auxiliary functions.
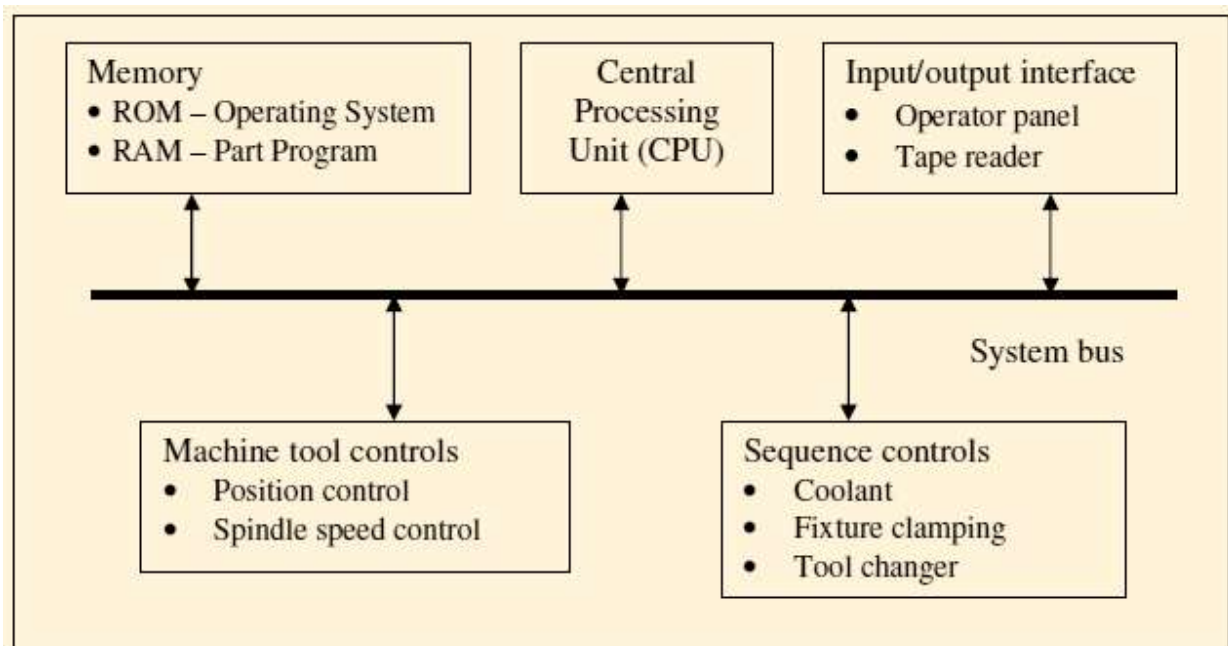
The same CPU takes care of the generation of control signals for the axis movement, and some basic safety checks are built into the system through this unit, and continuous necessarily corrective actions are

also provided by the CPU unit. Whenever the situation goes beyond control of the CPU, it takes the final action of shutting down the system and the machine.

Machine or Instruction Cycle:

The steps performed by the computer processor for each machine language instruction received. The machine cycle is a 4 process cycle that includes reading and interpreting the machine language, executing the code and then storing that code. The machine or instruction cycle is also called **fetch-and-execute cycle, fetch-decode-execute cycle, or FDX**, is the basic operation cycle of a computer.

It is the process by which a computer retrieves a program instruction from its memory, determines what actions the instruction requires, and carries out those actions. This cycle is repeated continuously by the central processing unit (CPU), from boot up to when the computer is shut down, as can be seen graphically shown below. The four steps of a machine cycle are:

Fetch: Retrieve an instruction from the memory. The next instruction is fetched from the memory address that is currently stored in the program counter (PC), and stored in the instruction register (IR). At the end of the fetch operation, the PC points to the next instruction that will be read at the next cycle.

Decode: Translate the retrieved instruction into a series of computer commands. The decoder interprets the instruction. During this cycle the instruction inside the IR (instruction register) gets decoded.

Execute: Execute the computer commands. The control unit of the CPU passes the decoded information as a sequence of control signals to the relevant function units of the CPU to perform the actions required by the instruction such as, reading values from registers, passing them to the Arithmetic Logic Unit (ALU) to perform the mathematical or logic functions, writing the result back to a register. If the ALU is involved, it sends a condition signal back to the Control Unit (CU).

Store: Sand and write the results back in memory. The result generated by the operation is stored in the main memory, or sent to an output device. Based on the condition of any feedback from the ALU, the Program Counter (PC) may be updated to a different address from which the next instruction will be fetched.
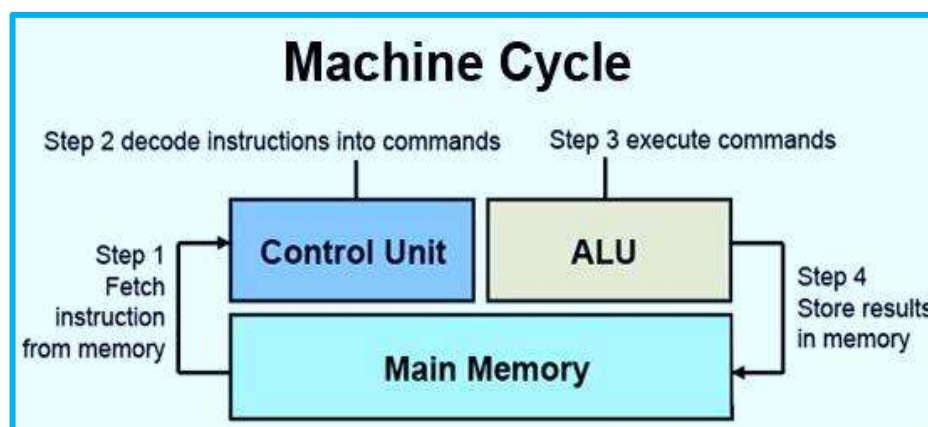

The circuits used in the CPU during the machine cycle are:

- ➢ **Program Counter (PC)**: Incrementing counter that keeps track of the memory address of the instruction that is to be executed next.
- ➢ **Memory Address Register (MAR)**: Holds the address of a memory block to be read.
- ➢ **Memory Data Register (MDR)**: Two-way register that holds data fetched from memory (and ready for the CPU to process) or data waiting to be stored in memory.
- ➢ **Instruction Register (IR)**: Temporary holding ground for the instruction that has just been fetched from memory.
- ➢ **Control Unit (CU)**: Decodes the program instruction in the IR, selecting machine resources, as a data source and a particular arithmetic operation, and coordinates activation of those resources.
- ➢ **Arithmetic Logic Unit (ALU)**: Performs mathematical and logical operations

THE CNC SYSTEM:

A CNC system may be characterized in terms of **three** major elements: **hardware, software and information**. The **hardware** includes the microprocessors that effect control system functions and peripheral devices for data communication, machine tool interfacing and machine tool status monitoring. The **software** includes the programs that are executed by the system microprocessors and various types of software associated with CNC.
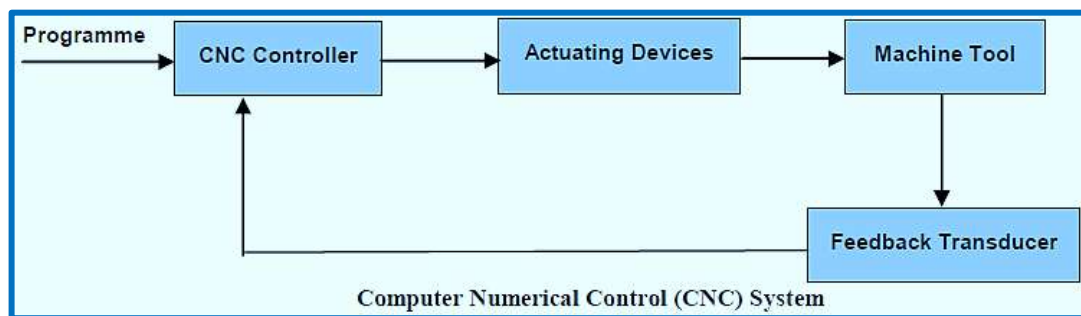
It will be useful for the reader to become acquainted with the basic concept of **binary number code** prior to discussing input and storage methods. Internally, computers and the MCU operate by a system of electrical switches. A 1 (one) is processed as an *open* switch and a 0 (zero) is processed as a *closed* switch. All numbers, letters, and special characters are represented in terms of a unique set of zeros and ones. The *only* code the computer and MCU can work on is the *binary.* All externally generated code must first be translated into binary before these devices can act.

The binary code from the computer and MCU must also be translated back into a code operators can understand. The translation process is automatically executed by devices inside the computer. The information necessary to perform CNC operations could be entered manually into the control unit, but this is a long and inefficient process. Modern CNC technology uses an array of new devices for storing and loading programs written with the aid of a microcomputer or larger mainframe computer.

The machine tool operator guides a cutting tool around a work piece by manipulating hand wheels and dials to get a somewhat finished part. In this procedure many judgments of **speeds, feeds, mathematics and sometimes even tool configuration** are his responsibility. The number of judgments the operator

has to make, usually depends on the type of stock he is working and in the kind of organization he works for. If his judgment comes to an error, it results in rejects or to be reworked or repaired in some way.

Direct Numerical Control (DNC):

A DNC system is nothing more than a computer that is networked with one or more CNC machines. Until only recently, rather crude serial communications protocol (RS-232c) had to be used for transferring programs. Newer controls have more current communications capabilities and can be networked in more conventional ways (Ethernet, etc.). Regardless of methods, the CNC program must of course be loaded into the CNC machine before it can be run.

Then, Direct Numerical Control (DNC) can be defined as a type of manufacturing system in which several NC or CNC machines are controlled remotely from a host/main frame computer or by a single (mainframe) computer through direct connection.
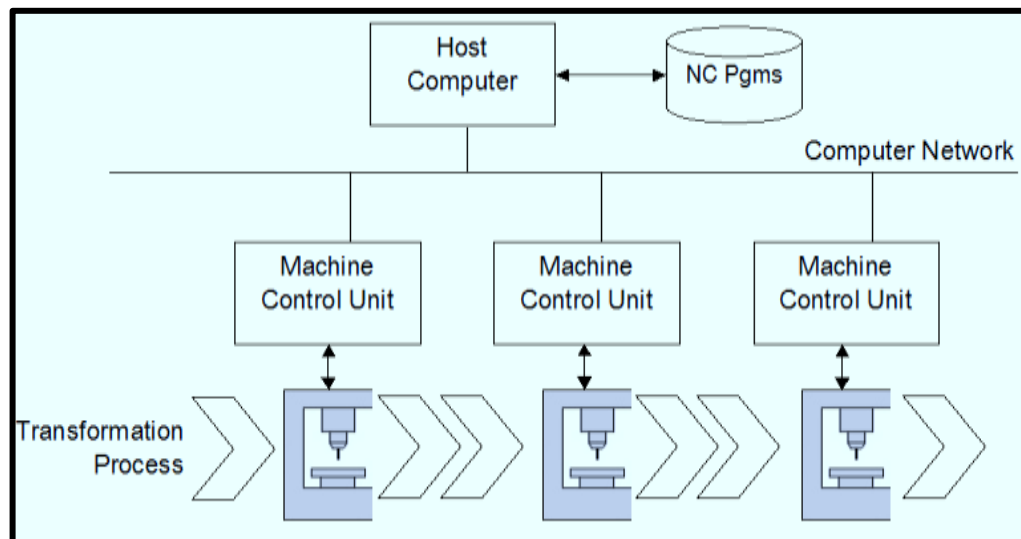
The part-program is downloaded to the machines directly (thus omitting the tape reader) from the computer memory, using the basic system: Main frame computer, memory, communication network, NC machine tool. The communication network can be done either through connecting the remotely located computer, with lengthy cables to the individual machine control directly or connecting the main frame computer with a small computer at individual operator's station known as satellite computer.

DNC system **is expensive** and is preferably used in large organizations. The combination of DNC/CNC makes possible to eliminate the use of programs as the input media for CNC machines. The DNC computer downloads the program directly to the CNC computer memory. This reduces the amount of communication required between the central computer and each machine tool.

The program consists of precise instructions about the methodology of manufacture as well as movements. For example, what tool is to be used, at what speed, at what feed and to move from which point to which point in what path. Since the program is the controlling point for product manufacture, the machine

becomes versatile and can be used for any part. All the functions of a NC machine tool are therefore controlled electronically, hydraulically or pneumatically. In NC machine tools, one or more of the following functions may be automatic, such as:

a. Starting and stopping of machine tool spindle;
b. Controlling the spindle speed;
c. Positioning the tool tip at desired locations and guiding it along desired paths by automatic control of motion of slides;
d. Controlling the rate of movement of tool tip (feed rate);
e. Changing of tools in the spindle.

Features of CNC Machining:

Computer NC systems include additional features beyond what is feasible with conventional hard-wired NC. These features, many standards on most CNC Machine Control units (MCU), include the following:

Storage of more than one part program: With improvements in computer storage technology, newer CNC controllers have sufficient capacity to store multiple programs. Controller manufacturers generally offer one or more memory expansions as options to the MCU.

Various forms of program input: Whereas conventional (hard-wired) MCUs are limited to punched tape as the input medium for entering part programs, CNC controllers generally possess multiple data entry capabilities, such as punched tape, magnetic tape, floppy diskettes, RS-232 communications with external computers, and manual data input (operator entry of program).

Program editing at the machine tool: CNC permits a part program to be edited while it resides in the MCU computer memory. Thus, a part program can be tested and corrected entirely at the machine site, rather than being returned to the programming office for corrections. Editing also permits cutting conditions in the machining cycle to be optimized. After the program has been corrected and optimized, the revised version can be stored on punched tape or other media for future use.

Fixed cycles and programming subroutines: The increased memory capacity and the ability to program the control computer provide the opportunity to store frequently used machining cycles as macros that can be called by the part program. Instead of writing the full instructions for the particular cycle into every program, a programmer includes a call statement to indicate that the macro cycle should be executed. These cycles often require that **certain parameters** to be defined, for example, a **bolt hole circle**, in which the diameter of the bolt circle, the spacing of the bolt holes, and other parameters must be specified.

Interpolation: Some of the interpolation schemes are normally executed only on a CNC system because of computational requirements. Linear and circular interpolation is sometimes hard-wired into the control unit, but helical, parabolic, and cubic interpolations are usually executed by a stored program algorithm.

Positioning features for setup: Setting up the machine tool for a given workpart involves installing and aligning a fixture on the machine tool table. This must be accomplished so that the machine axes are es-

tablished with respect to the workpart. The alignment task can be facilitated using certain features made possible by software options in the CNC system. Position set is one of the features. With position set, the operator is not required to locate the fixture on the machine table with extreme accuracy. Instead, the machine tool axes are referenced to the location of the fixture using a target point or set of target points on the work or fixture.

Cutter length and size compensation: In older style controls, cutter dimensions had to be set precisely to agree with the tool path defined in the part program. Alternative methods for ensuring accurate tool path definition have been incorporated into the CNC controls. One method involves manually entering the actual tool dimensions into the MCU. These actual dimensions may differ from those originally programmed. Compensations are then automatically made in the computed tool path. Another method involves use of a tool length sensor built into the machine. In this technique, the cutter is mounted in the spindle and the sensor measures its length. This measured value is then used to correct the programmed tool path.

Acceleration and deceleration calculations: This feature is applicable when the cutter moves at high feed rates. It is designed to avoid tool marks on the work surface that would be generated due to machine tool dynamics when the cutter path changes abruptly. Instead, the feed rate is smoothly decelerated in anticipation of a tool path change and then accelerated back up to the programmed feed rate after the direction change.

Communications interface: With the trend toward interfacing and networking in plants today, most modern CNC controllers are equipped with a standard RS-232 or other communications interface to link the machine to other computers and computer driven devices. This is useful for various applications, such as: (1) Downloading part programs from a central data file; (2) Collecting operational data such as workpiece counts, cycle times, and machine utilization; and (3) Interfacing with peripheral equipment, such as robots, that unloads and loads parts.

Diagnostics: Many modern CNC systems possess a diagnostics capability that monitors certain aspects of the machine tool to detect malfunctions or signs of impending malfunctions or to diagnose system breakdowns.

Basic Length Unit (BLU): Each BLU unit corresponds to the position resolution of the axis of motion. For example, **1 BLU = 0.0001"** means that the axis will move 0.0001" for every one electrical pulse received by the motor. The BLU is also referred to as Bit (binary digit). Pulse = BLU = Bit

Other peripheral devices:

Portable Hard Drives: The palm top devices, store data in the form of a magnetic pattern on a spinning disk, connected directly to any USB port on a PC or workstation and can dramatically increase the amount of digital data to be stored. The smaller pocket hard drive units have a storage capacity of between 2.5 GB and 5 GB. The slightly larger portable units can store data starting in the 40 GB range all the way up to 120 GB. In essence, they act as additional hard drives.
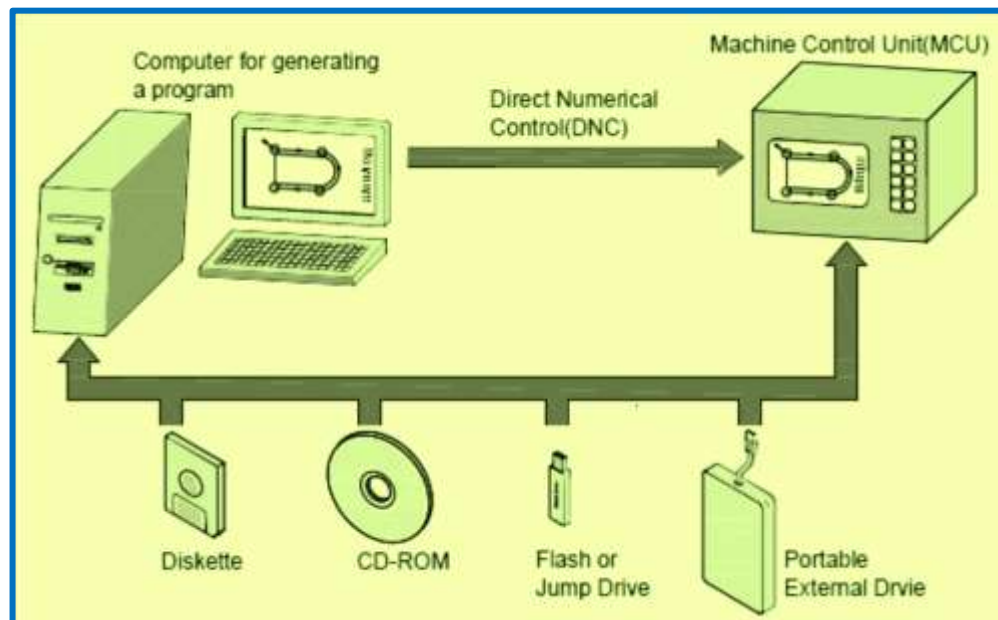
Flash or Jump Drives: A flash or jump drive is a small, portable unit about the size of a stick of gum that plugs directly into a computer's USB port. Flash memory stores data with the use of a special type of **Electrically Erasable Programmable Read-Only Memory** (EEPROM) chip. Data is written by applying an electric field to the entire chip, or to predetermined sections of the chip called blocks. Flash drives are available in various storage sizes: 512 MB, 2 GB, 4 GB, and 8 GB. The advantages of using flash devices instead of CDs or external hard drives are listed as follows:

- No fragile moving parts that can break if dropped.
- Far smaller in size.
- Do not require time-consuming configuration to connect to the computer.
- USB ports are more common than CD or DVD burners.
- Flash devices are automatically re-writable and do not require a special drive.

Classification based on the motion type:

Point-to-Point Systems: Are those that move the tool or the workpiece from one point to another and then the tool performs the required task. Upon completion, the tool (or workpiece) moves to the next position and the cycle is repeated. The simplest example for this type of system is a drilling machine where the workpiece moves. In this system, the feed rate and the path of the cutting tool (or workpiece) have no significance on the machining process. The accuracy of positioning depends on the system's resolution in terms of **BLU** (basic length unit) which is generally between **0.0001" and 0.001"**.

- ✓ It is used in some CNC machines such as drilling, boring and tapping machines, etc.
- ✓ The control equipment for use with them is known as point-to-point control equipment
- ✓ Feed rates need not to be programmed.
- ✓ In these machine tools, each axis is driven separately.
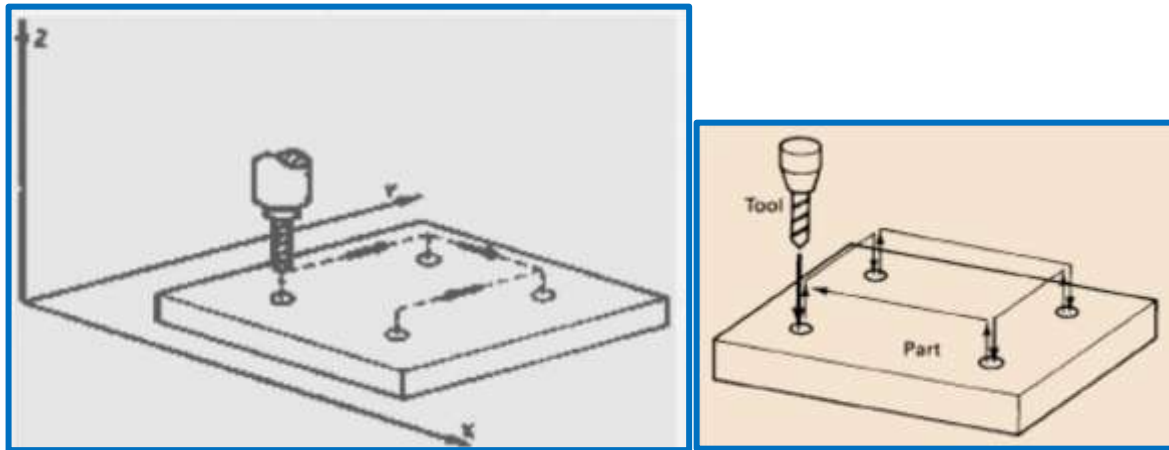
Example 1:

The **XY** table of a drilling machine has to be moved from the point (1, 1) to the point (6, 3). Each axis can move at a velocity of **0.5"/sec**, and the **BLU is 0.0001"**, find the travel time and resolution.

- Travel time in X-axis is (6-1)/0 .5 = **10 sec**, in Y-axis is (3-1)/0 .5= **4 sec**.;
- Travel time = **10 sec.**;
- Resolution = **BLU = 0.0001**.

Contouring Systems (Continuous Path Systems): These systems provide continuous path such that the tool can perform while the axes are moving, enabling the system to generate angular surfaces, two-dimensional curves, or three dimensional contours. Example is a milling machine where such tasks are accomplished. Each axis might move continuously at a different velocity. Velocity error is significant in affecting the positions of the cutter.

- ✓ It is used in CNC machine tools such as milling machines.
- ✓ The controls required for their control are known as contouring control.
- ✓ These machines require simultaneous control of axes.
- ✓ Contouring machines can also be used as point-to-point machines, but it will be uneconomical to use them unless the work piece also requires having a contouring operation to be performed on it.
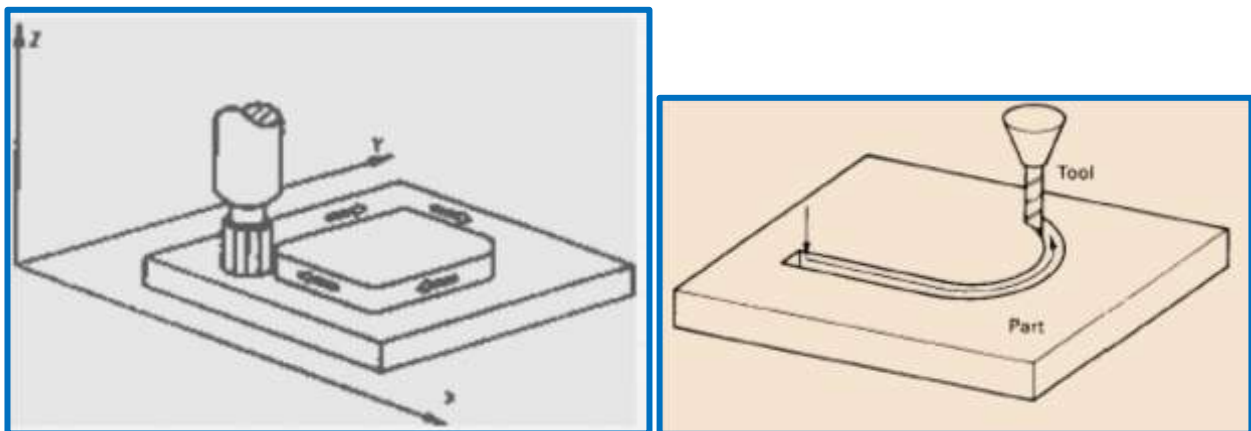
Example 2:

A CNC milling machine has to **cut a slot** located between the points (0, 0) and (4, 3) on the XY-plane, where the dimensions are in inches. If the velocity (v) along the slot is to be 0.1 in/sec, find the cutting time and axial velocities. The calculation is:

- Distance traveled along the slot = $(X^2 + Y^2)^{1/2}$ = $(16 + 9)^{1/2}$ = **5"**;
- Cutting time = 5 / 0.1 = **50 sec.**;
- Velocity in vX = $X(v) / (x^2+y^2)^{1/2}$ = 4(0.1) / 5 = **0.08 in/sec**;
- Velocity in vY = $Y(v) / (x^2+y^2)^{1/2}$ = 3(0.1) / 5 = **0.06 in/sec**.

If the velocity is Y-axis is off by 10%, what would be the new position?
New velocity in y is 0.9 x 0 .06 = **0.054 in/sec**
In 50 sec, the y- will move a distance [50(0.054)] = **2.7 in.**

Classification based on the control loops:

Open Loop Systems: The open-loop control means that there is no feedback and uses stepping motors for driving the lead screw. A stepping motor is a device whose output shaft rotates through a fixed angle in response to an input pulse. The accuracy of the system depends on the motor's ability to step through the exact number. The frequency of the stepping motor depends on the load torque. The higher the load torque, lower would be the frequency. Excessive load torque may occur in motors due to the cutting forces in machine tools. Hence this system is more suitable for cases where the tool force does not exist (Example: laser cutting).
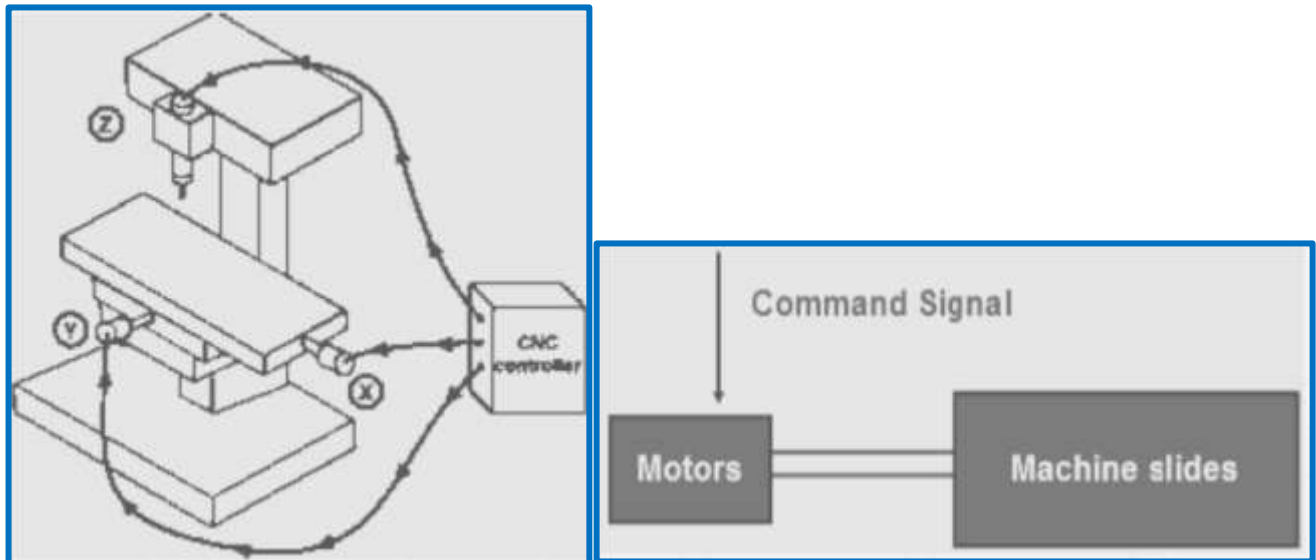

The stepping motor is driven by a series of electrical pulses generated by the MCU. Each pulse causes the motor to rotate a fraction of one revolution. The fraction is expressed in terms of the step angle, given by a = 360/N, degrees where N = number of pulses required for one revolution If the motor receives "n" number of pulses then the total angle, A = n (360/N), degrees.

In terms of the number of revolutions, it would be (n/N) if there is a 1:1 gear ratio between the motor and the lead screw, and then the lead screw has (n/N) revolutions. If the pitch of lead screw is p (in/rev), then the distance traveled axially, x, x = p(n/N) can be used to achieve a specified x-increment in a point-to-point system.

The pulse frequency, f, in pulses/sec determines the travel speed of the tool or the workpiece. 60 f = N (RPM) where N = number of pulses per revolution, RPM = RPM of the lead screw. The travel speed, V, is then given by V = p (RPM) where p pitch in in/rev.

- Programmed instructions are fed into the controller through an input device.
- These instructions are then converted to electrical pulses (signals) by the controller and sent to the servo amplifier to energize the servo motors.
- The primary drawback of the open-loop system is that there is no feedback system to check whether the program position and velocity has been achieved.

Configuration for an open loop system.
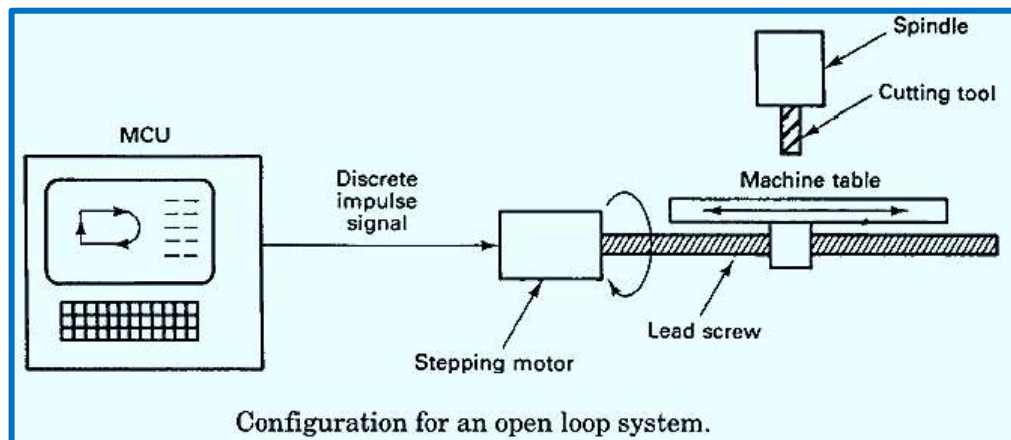
Example 3:

A stepping motor has N = 150, p = 0.2"/rev; If n = 2250 pulses, what is the distance traveled in x-direction. What should be the pulse frequency for a travel speed of 16 in./min?

- x = (0.2) (2250)/150 = **3"**
- 16' = 0 .2 (RPM), from which, RPM = **80**
- f = (150) (80)/60 = **200 Hz**

Example 4:

A stepping motor of 200 steps per revolution is mounted on the lead screw of a drilling machine. If the pitch is 0.1 in/rev, What is the BLU? If the motor receives a pulse frequency of 2000 Hz, what is the speed of the table?

BLU = 0 .1/200 = **0.0005"**
Table speed = (p) (RPM) = (0 .1) (60) (2000)/200 = **60 in/min**

Closed Loop Systems: The closed-loop NC system can be very appropriate, when there is a force resisting the movement of the tool and the workpiece. Milling and turning are typical examples. In these systems, the DC servomotors and feedback devices are used to ensure that the desired position is achieved. The feedback sensor used is an optical encoder as shown below.
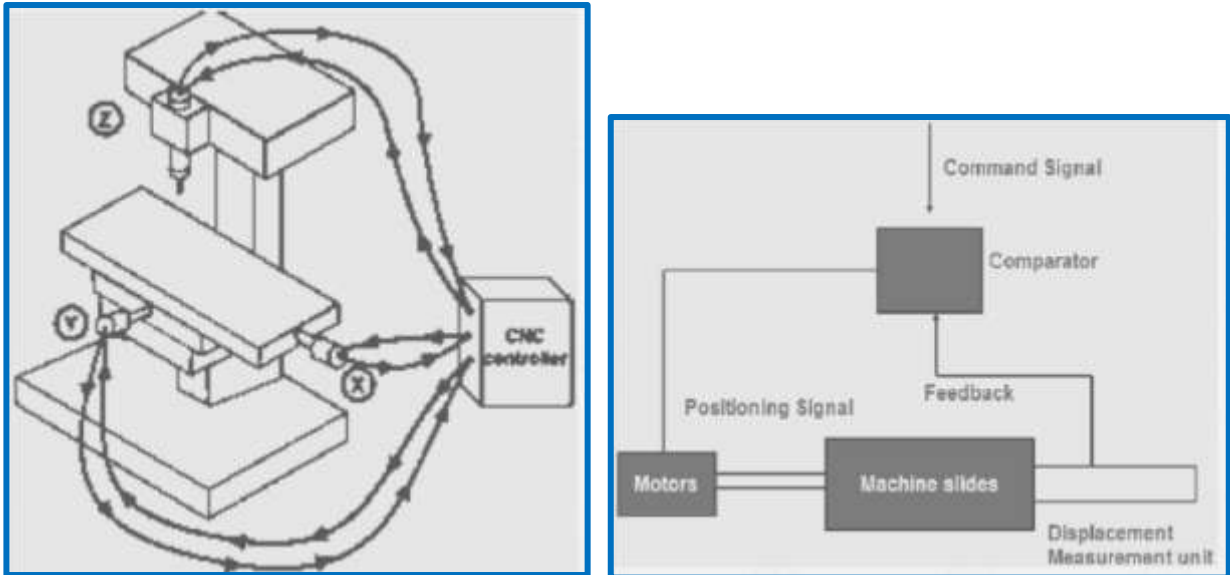
The **encoder** consists of a light source, a photo detector, and a disk containing a series of slots. The encoder is connected to the lead screw. As the screw turns, the slots cause the light to be seen by the photo detector as a series of flashes which are converted into an equivalent series of electrical pulses which are then used to characterize the position and the speed. The equations remain essentially the same as open-loop except that the angle between the slots in the disk is the step angle.

Configuration of a closed loop system.
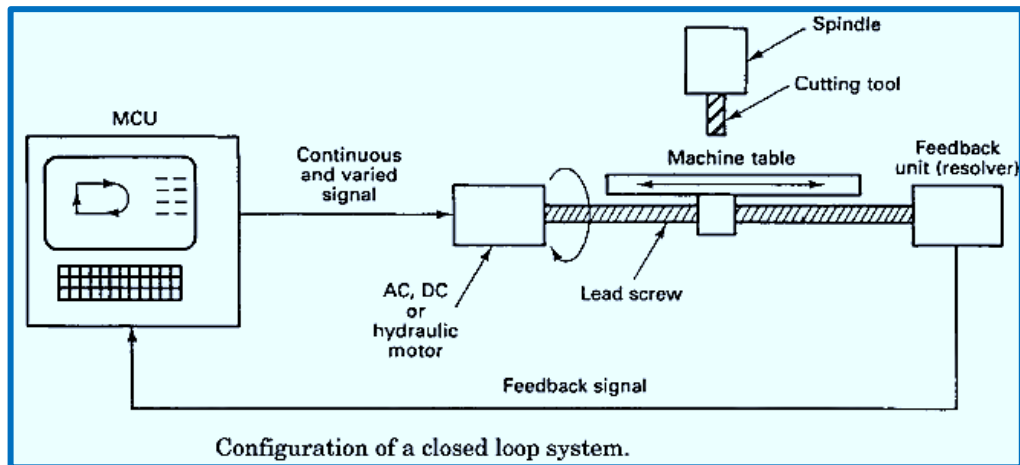
Both the input to the control loop and the feedback signals are a sequence of pulses, each pulse representing a BLU unit. The two sequences are correlated by a comparator and gives a signal, by means of a digital-to-analog converter, (a signal representing the position error), to operate the drive motor (DC servomotor). The closed-loop system has a feedback subsystem to monitor the actual output and correct any discrepancy from the programmed input. These systems use position and velocity feedback, such as:

- The feedback system could be either analog or digital. The analog systems measure the variation of physical variables such as position and velocity in terms of voltage levels.
- Digital system monitors output variations by means of electrical pulses.
- The closed-loop systems are very powerful and accurate because they are capable of monitoring operating conditions through feedback subsystems and automatically compensating for any variations in real-time.

Example 5:

Consider a CNC worktable driven by a closed-loop control system consisting of a servomotor, lead screw, and optical encoder. The lead screw has a pitch, p = 0.2" and is coupled to the motor shaft with a screw to motor gear ratio of 1:4. The encoder generates 150 pulses per revolution of the lead screw. If the number of pulses and the pulse rate received by the control system are 2250 and 200 Hz respectively, calculate:

a. Table speed
b. Motor speed in RPM
c. Distance traveled by the table

Answer:

a. V= p(RPM) = 0.2(RPM) = (0 .2) (60 f)/N = (0 .2) (60) (200)/150 = **16 in/min**
b. RPM of the lead screw = (60) (200)/(150) = **80**
RPM of the motor = **320**
c. x = p(n/N) = (0.2) (2250)/150 = **3"**

Example 6:

A do servomotor is coupled to a lead screw which drives the table of a CNC machine tool. A digital encoder, mounted at the end of the screw, emits 500 pulses per revolution. If the pitch is 5 mm per rev, and the motor rotates 600 rpm (1:1 gear ratio), calculate the:

a. Table speed
b. BLU
c. Frequency of pulses transmitted by the encoder

Answer:

a. V = p(RPM) = 5 (600) = 3000 mm/min = **3 m/min**
b. BLU = 5/500 = **0.01 mm**
c. RPM = 600 = (60f)/N = 60 f/500 from which f = **5000 Hz**

Classification based on the number of axis:

2 & 3 axis CNC Machines: CNC machines may be under **machines with two axis**, where the move-

vement takes place. The saddle moves longitudinally on the bed (Z-axis), and the cross slide moves trans-versely on the saddle (along X-axis). In **3-axis machines are one more axis**, perpendicular to the above two axis. By the simultaneous control of all the 3 axis, many **complex surfaces** can be machined.

4 & 5 axis CNC machines: multi-axis machining may come with **standard 3-axis** CNC tool path move-ments. The **4 and 5 axis CNC machines** provide multi-axis machining capabilities beyond the standard 3-axis CNC. The **5-axis milling centers** include the three X, Y, Z axis more the **A-axis** which is a rotary tilting of the spindle and the **B-axis**, which is a rotary index table.

A 5-axis machining operation

Importance of higher axis machining: Reduced cycle time by machining complex components using a single setup. In addition to time savings, improved accuracy can also be achieved as positioning errors between setups are eliminated, considering the following:

✓ Improved surface finish and tool life by tilting the tool to maintain optimum tool to part contact all the times.

- ✓ Improved access to under cuts and deep pockets. By tilting the tool, the tool can be made normal to the work surface and the errors may be reduced as the major component of cutting force will be along the tool axis.
- ✓ Higher axes machining has been widely used for machining sculptures surfaces in aerospace and automobile industry.
- ✓ Reduced cycle time by machining complex components using a single setup.
- ✓ In addition to time savings, improved accuracy can also be achieved as positioning errors between setups are eliminated.
- ✓ Improved surface finish and tool life by tilting the tool to maintain optimum tool to part contact all the times.
- ✓ The amount of human labor is reduced, if the piece would otherwise have to be turned manually during the machining.
- ✓ A better surface finish can be obtained by moving the tool tangentially about the surface.
- ✓ More complex parts can be manufactured, particularly parts with curved holes.

Electric, hydraulic & pneumatic systems: Mechanical power unit refers to a device which transforms some form of energy to mechanical power which may be used for driving slides, saddles or gantries forming a part of machine tool. The input power may be of electrical, hydraulic or pneumatic.
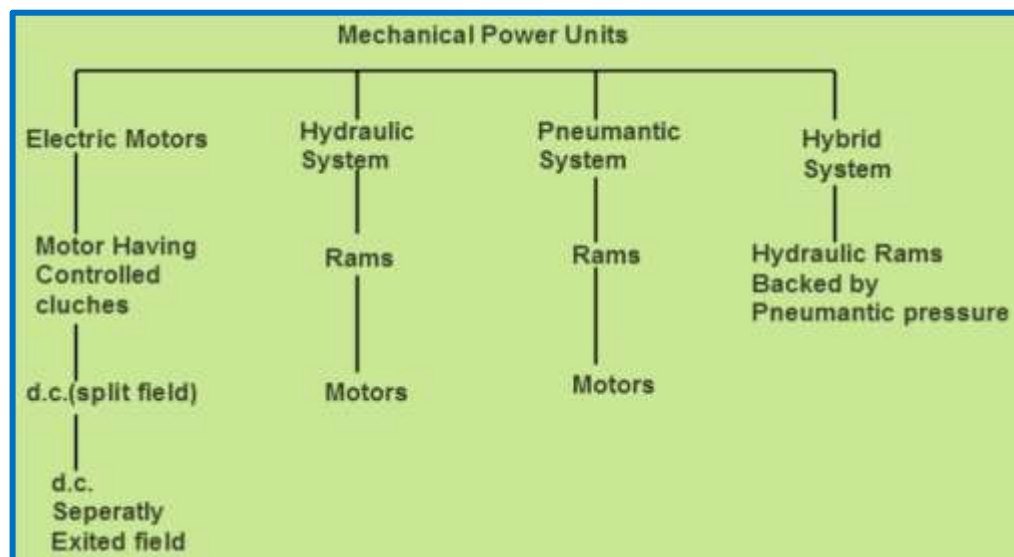
CAD/CAM SYSTEMS:

CAD means Computer-Aided Design and CAM is its subsequent Computer-Aided Manufacturing, are the use of computer software to control machine tools and related machinery in the manufacturing of workpieces. A CAM system is a **software program** that runs on a computer to help the CNC programmer with the programming process. In many companies the CAM system commonly works with the Computer Aided-Design (CAD) drawings, developed by the company's design engineering department.

The CNC programmer will simply specify the machining operations to be performed and the CAM system creates the CNC program automatically. The Computer-Aided Engineering (CAE), is then a model gene-

ated in CAD, which can be input into CAM software to control the machine tools. To simplify the programming process, a Computer-Aided Manufacturing (CAM) system can be used.

The CNC program can be developed manually for simple applications (like drilling holes). That is, a programmer will sit down to write the program armed only with pencil, paper, and calculator. Again, for simple applications, this may be the very best way to develop CNC programs. As applications get more complicated, and especially when new programs are required on a regular basis, writing programs manually becomes much more difficult.

The primary purpose of the CAD/CAM system programming is to create a faster production process, components and tooling with more precise dimensions and material consistency, which in some cases, uses only the required amount of raw material (thus minimizing waste), while simultaneously reducing work time and consecutively, energy consumption.

The CAD/CAM Software:

Computer Aided Design (CAD) and Computer-Aided Manufacturing (CAM) allow a product to go from initial design stages to production in as little as an hour. Most people use the software to scan the shape with a digitizer, perfect the shape on a computer, and then use CAM software to thermoform and cast a product such as a socket, or to guide computerized woodworking tools.

Essentially, the Computer-Aided Design software allows the owner to create a G-code file which can be used to control numerically controlled machines to perfectly create anything, since a wrench socket to a machine part. CAD/CAM can be used with Laser Cutting, engraving, 3D printing, sheet metal cutting, dentistry, and much more, depending on the buyer´s type of software and the CNC purchased.
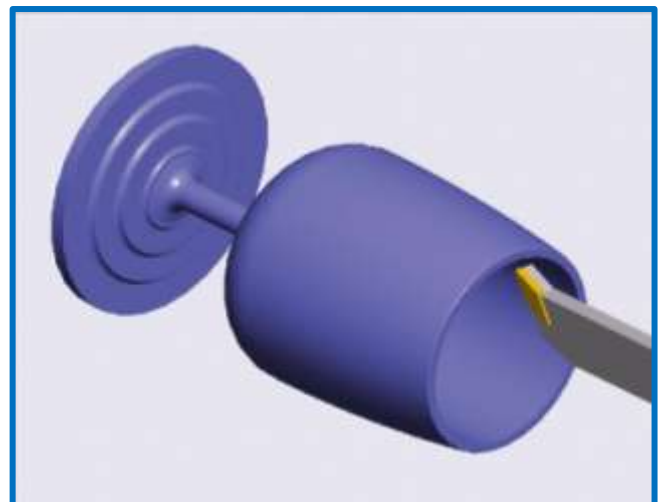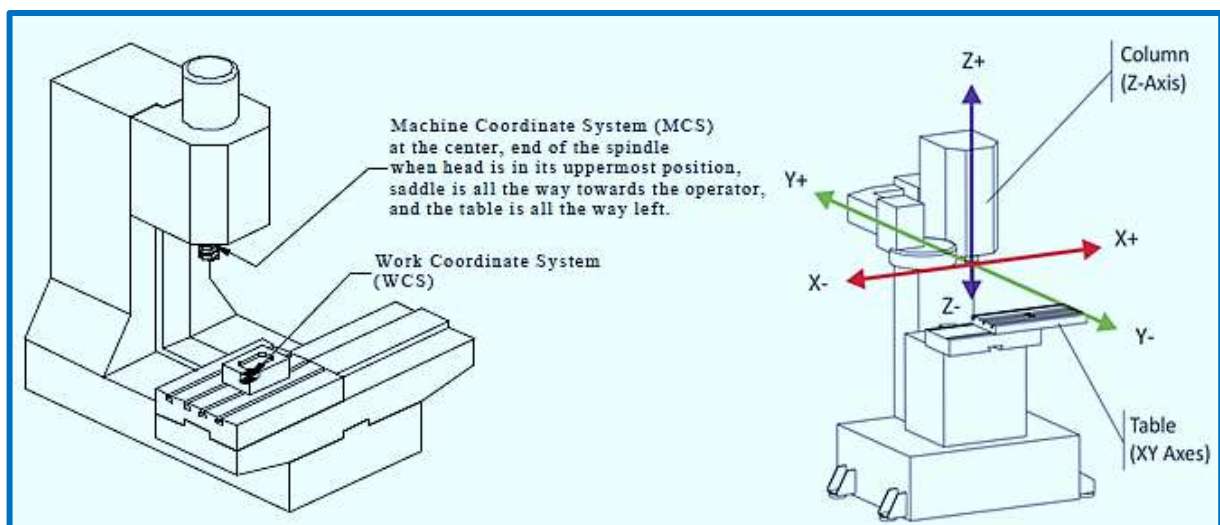

CNC MACHINING CENTERS:

A machining center is simply a CNC milling machine with an automatic tool changer including a tool magazine, a work part positioner, a pallet changer and a safety enclosure. There are a number of different types

of Machining Centers differentiated by the number of programmable axis. All CNC Machining Centers are usually designed with features, to reduce non-productive time. The main features are:

- ✓ **Automatic tool changer:** The tools are contained in a storage unit that is integrated with the machine tool. When a cutter needs to be changed, the tool drum rotates to the proper position and an automatic tool changer (ATC) operating under program control, exchanges the tool in the spindle for the tool in the tool storage unit. Capacities of **tool storage** unit commonly range from **16 to 80** cutting tools.

- ✓ **Automatic work part positioner:** Many horizontal and vertical machining centers have the capability to orient the work part relative to the spindle. This is accomplished by means of a rotary table on which work part is fixed. The table can be oriented at any angle about a vertical axis to permit the cutting tool to access almost the entire surface of the part in a single setup.

- ✓ **Automatic pallet changer:** Machining centers are often equipped with two (or more) separate pallets that can be presented to the cutting tool using an automatic pallet changer. While machining is performed with one pallet in position at the machine, the other pallet is in a safe location away from the spindle. In this location, the operator can unload the finished part and then fixture the raw work part for next cycle.

Coordinate Systems:

Regardless of the machine design, you always program as if you are moving the tool. On most machining centers, the head moves the cutter up and down, so a move in the **+Z** direction moves the cutter and head up and **–Z** direction the cutter is moved down. Most machines move the table only in the **XY** directions.


Machining centers save production time and cost by reducing the need for moving a part from one machine to another. Turning centers with increased capacity tool changers are also making a strong appearance in modern production shops. The CNC machines are also capable of executing many different types of lathe cutting operations, and can be programmed with absolute or incremental coordinates.

Thus, the a **+X** move actually moves the table to the left, but the cutter moves in the **+X** direction relative to the part, and as turning centers, most machining center programming are done in absolute mode.

(a) Axis Designation for Horizontal Z

(b) Axis Designation for Vertical Z
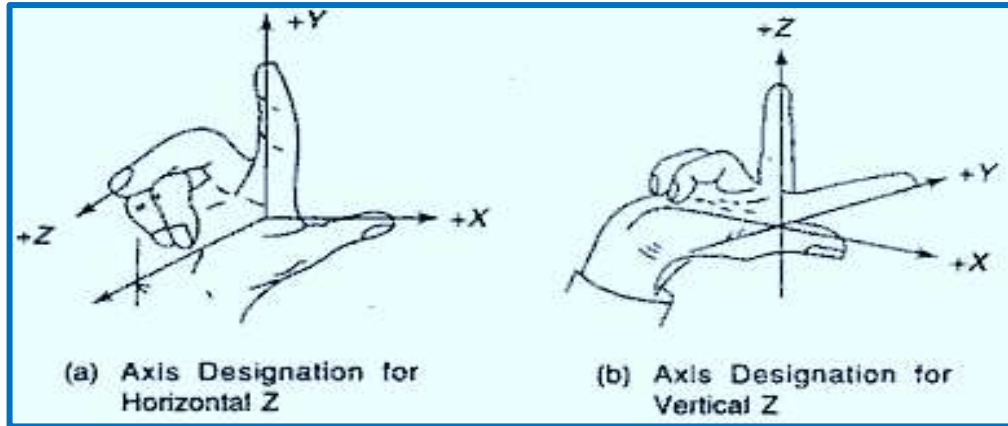
The right-hand rule for linear operation.
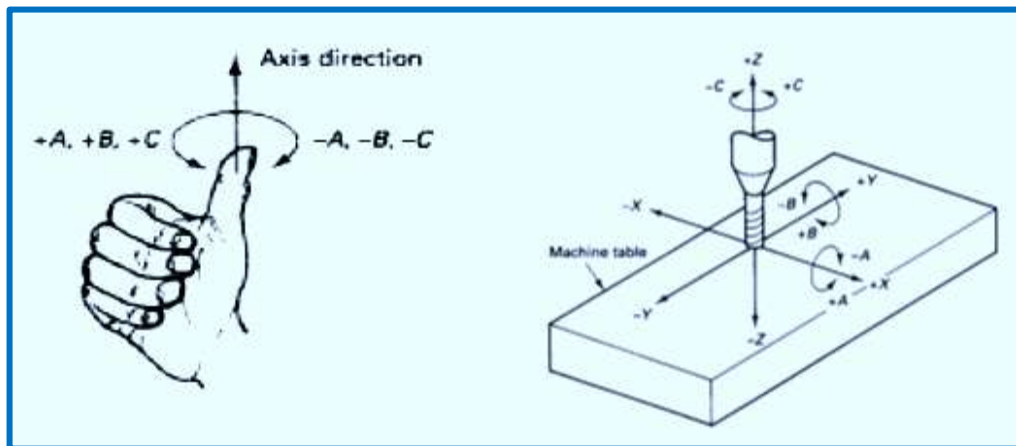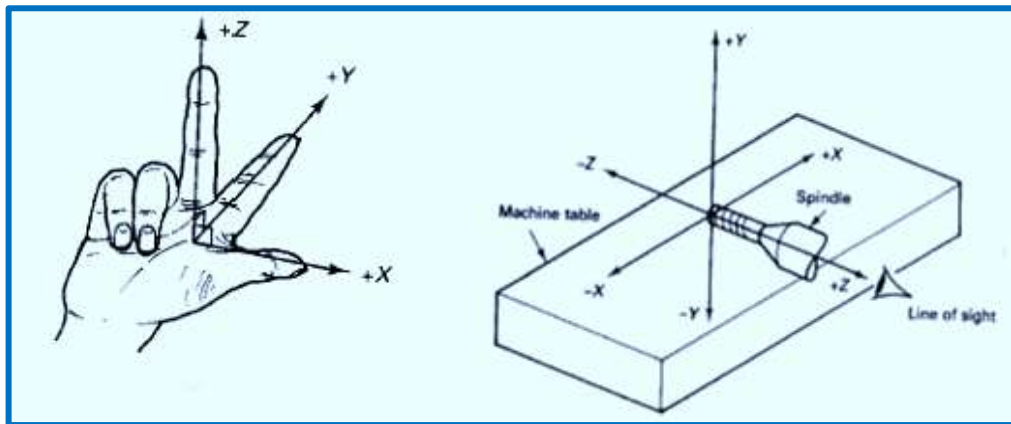
The CNC machining centers come equipped with automatic tool changers capable of **changing 90** or more tools. Many are also fitted with movable rectangular worktables called pallets. The pallets are used to automatically load and unload workpieces. At a single setup, machining centers can perform many opera-tions as **turning, drilling, milling, grinding, gear cutting, balancing, gauging**, **tapping, boring,**

counterboring, and so on. Additionally, by utilizing indexing heads, some centers are capable of executing these tasks on many different faces of a part and at specified angles.

Turning Centers		Machining Centers	
Absolute 0,0 start	Incremental 0,0 start	Absolute 0,0,0 start	Incremental 0,0,0 start
G0 X3. Z2. X3.5	G0 U3. W2. U0.5	G90 G0 X3. Y2.0 X3.5	G91 G0 X3. Y2. X0.5
Notice the use of X,Z for absolute and U,W for incremental.		Notice that G90 signifies absolute coordinates and G91 signifies incremental, and both use X,Y,Z.	

A gantry type CNC machining center

Types of Machining Centers:

Machining centers are the latest development in CNC technology, classified as **vertical, horizontal, or universal,** described below. The designation refers to the orientation of the machine spindle.

1. **Vertical Machining Center (VMC)**: Has its spindle on a vertical axis relative to the work table, typically used for flat work that requires tool access from top. Example, mold and die cavities, large components of aircraft.
2. **Horizontal Machining Center (HMC)**: Is used for cube shaped parts where tool access can be best achieved on the sides of the cube.
3. **Universal Machining Center (UMC)**: Has a work head that swivels its spindle axis to any angle, between horizontal and vertical. Example, airfoil shapes, curvilinear geometries.

Vertical Machining Centre   Horizontal Machining Centre
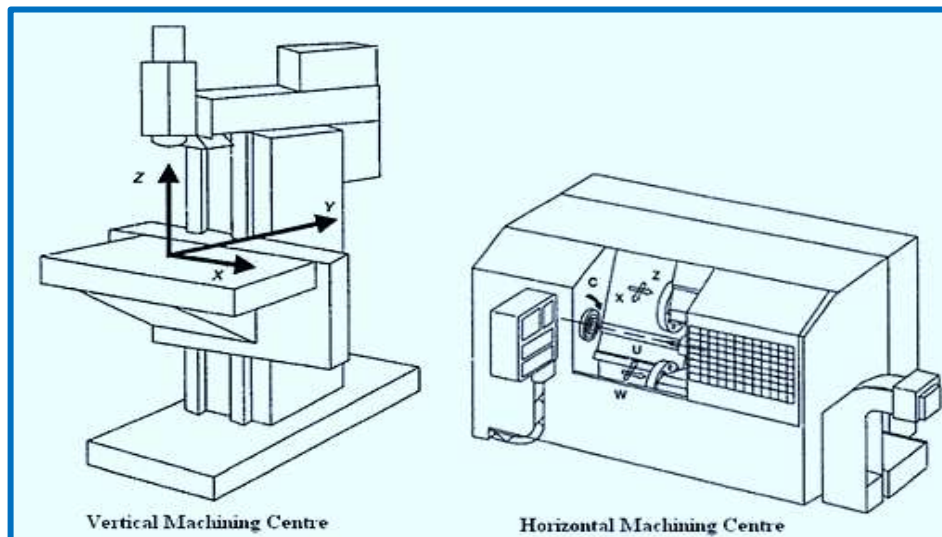
The term "multi-tasking machine" is used to include all of these machine tools that accomplish multiple and often quite different types of operations. The processes that might be available on a **single multi-tasking machine include milling, drilling, tapping, grinding and welding**. The main advantages of this new class of highly versatile machine compared to more conventional CNC machine tools include:

- ✓ Reduced part handling;
- ✓ Increased accuracy and repeatability;
- ✓ The parts utilize the same fixture throughout their processing;
- ✓ Faster delivery of parts in small lot sizes.

(a) Three Axis Boring Mill   (b) Four Axis Boring Mill

CNC Horizontal Boring Mills in 3 and 4 axes
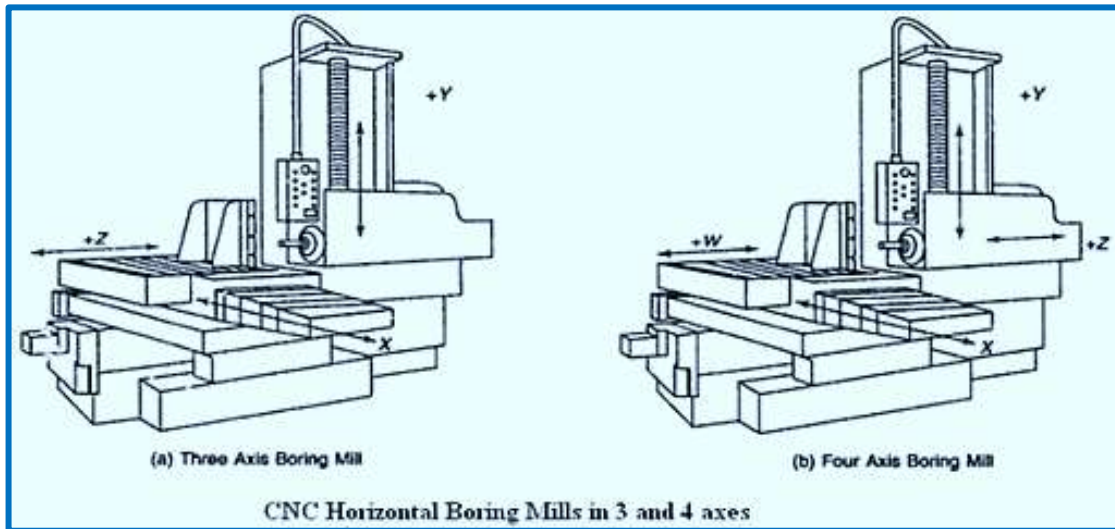
Three Axis Machining Center: has programmable X and Y axes in the plane of the table and a Z axis in the spindle's direction. With three axes, we can machine one surface of a cube with the end of the cutter and four additional surfaces with the side of the cutter. The illustration aside depicts a three axis Horizontal Machining Center. Note the different orientation of the X, Y, and Z axis. The surface of a cube can be machined with the end of the cutter and four additional surfaces with the side of the cutter.
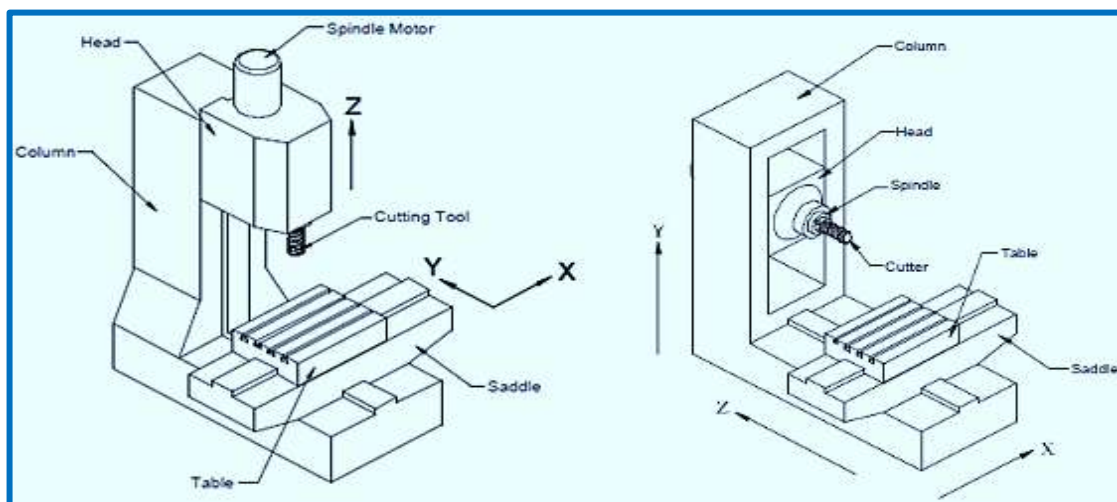
Three-axis machine centers

Four Axis Machining Center: are generally horizontal, and the table rotates to create the four axis. Frequently, a fixture called a tombstone (see sketch) is mounted on the table and many small parts machined at once on a large machine. Besides complex geometry, can be often machined in only one setup on a four or five axis machine that could require two or more setups in a simpler machine.
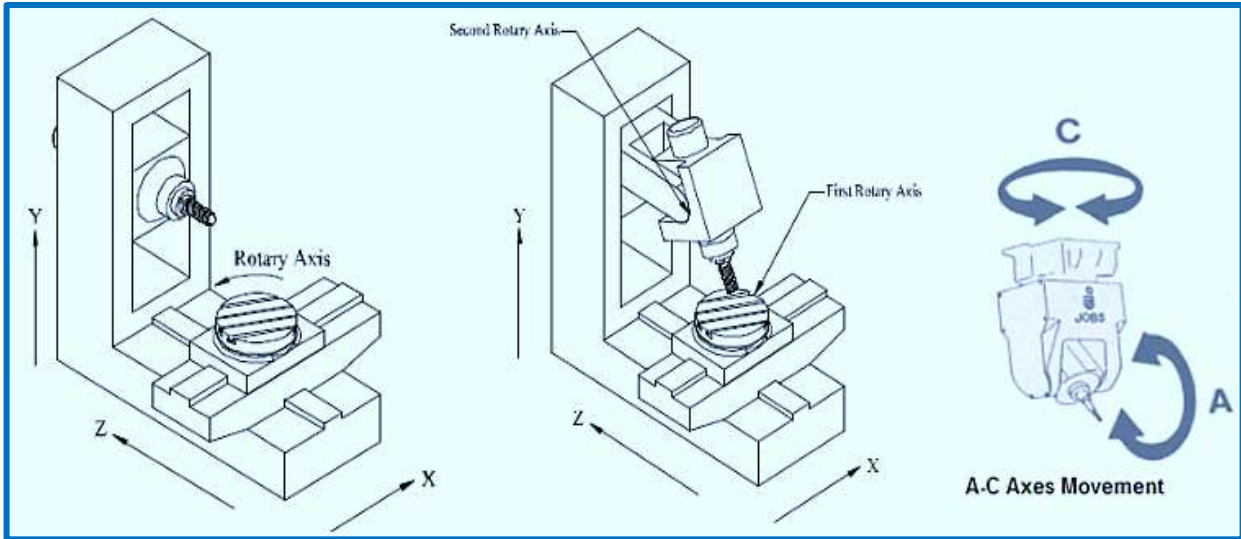
Five Axis Machining Center: is very similar to the four axis machine except the spindle rotates from horizontal to vertical. There are now many CAM software systems available to support multiaxis machining including software that can automatically **convert** 3-axis tool paths into 5-axis tool paths. These machines are used to machine complex parts and molds in the aerospace and automotive industries.

Five Axes CNC Vertical Axis Machining Centre
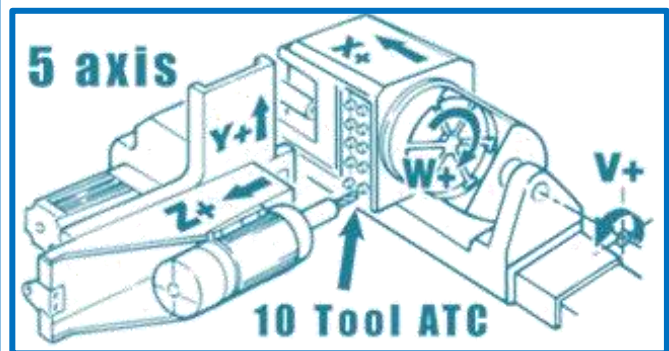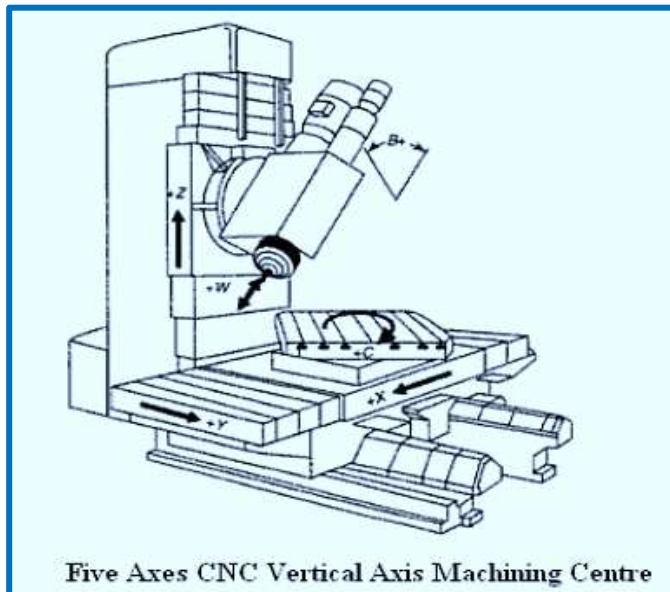
Most machining centers can be fitted with a pallet changer to increase productivity. On a plain machine, it sits idle while the operator removes the completed parts and loads the fixture with new ones. With a pallet changer, the operator unloads and reloads one pallet while the machine works on the other. This way, the machine can continuously cut the work parts.

Typical Shuttle Pallet with six Pallet Carousel

Other types of CNC Equipment:

In addition to machining centers and turning centers, CNC technology has also been applied to many other types of manufacturing equipment. Among these are wire Electrical Discharge Machines (wire EDM) and Laser Cutting Machines. The work piece material is removed **due to the erosion** caused by sparks that move horizontally with the wire.

Wire EDM machines utilize a very **thin wire** (0.0008 to 0.012 in.) as an electrode, the wire is stretched between diamond guides, and carbide that conducts current to the wire, which cuts the part like a band-saw. CNC is used to control horizontal table movements. Wire EDM machines are very useful for producing mold inserts, extrusion and trim dies, as well as form tools.

CNC Wire EDM System

Laser cutting CNC machines utilize an intense beam of focused laser light to cut the part. Material under the laser beam undergoes a rapid rise in temperature and is vaporized. If the beam power is high enough, it will penetrate through the material. Because no mechanical cutting forces are involved, lasers cut parts with minimal distortion. They have been very effective in machining slots and drilling holes.

- ✓ The machine utilizes an intense beam of focused laser light to cut the part.
- ✓ Material under the beam experiences a rapid rise in temperature and is vaporized.
- ✓ Laser cuts with a minimum of distortion, no mechanical cutting forces.

Machining Center Components:

- ✓ Main parts;
- ✓ Spindle and collets;
- ✓ Pallets;
- ✓ Automatic tool changer (ATC);
- ✓ Touch probes.

The main parts of vertical mills are: base, column, knee, saddle, table, ram, and tool head. The spindle and headstock components are rigidly mounted as an X, Y-axis traveling column assembly. The Z & B-axis table motion is a separate casting that moves forward and backwards for the Z-axis and pallet change positions. The spindle head assembly is pneumatically counter balanced.

- ➢ **Motor**: The motor supplies the power to the spindle.
- ➢ **Toolhead**: The toolhead houses the spindle, located at the end of the Ram and also contains the motor.
- ➢ **Column**: The column of the milling machine, along with the base, is the major structural components. They hold, align, and support the rest of the machine.
- ➢ **Table**: Holds and secures the workpiece for machining.
- ➢ **Saddle**: The saddle it attached to the knee. The saddle provides the in and out, or Y axis travel of the table.
- ➢ **Knee**: The knee supports the saddle and the table. The knee can be moved up and down for workpiece positioning.
- ➢ **Ram**: The ram allows the Toolhead to slide in and out. The ram gives the machine greater capacity and flexibility. It is recommended that the tool head be kept as close to the column as possible during heavy milling work.
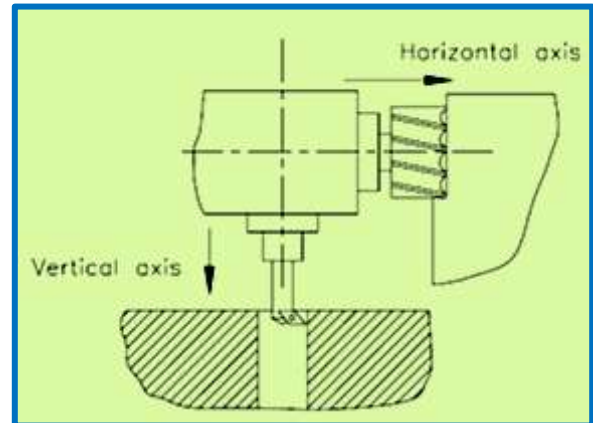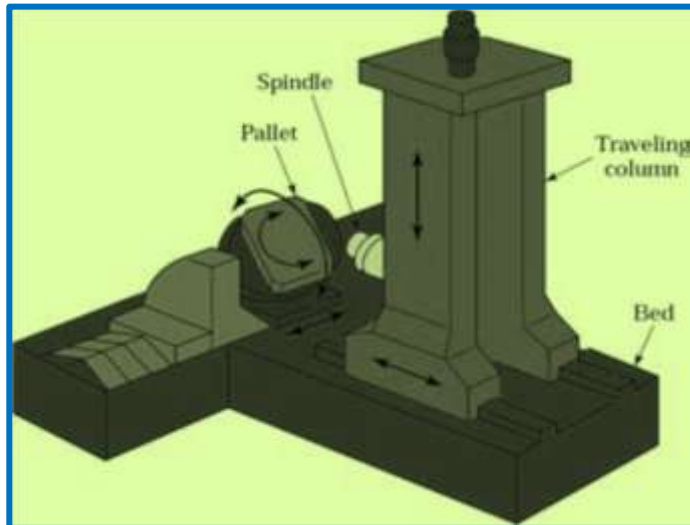
Machine tool spindle and collets:

In machine tools, a spindle is a rotating axis of the machine, which often has a shaft as its heart. Some machine tools that specialize in high-volume mass production have a group of 4, 6, or even more main spindles. These are called multispindle machines. For example, gang drills and many screw machines are multispindle machines. A machine tool may have several spindles, such as the headstock and tailstock spindles on a bench lathe. Although a bench lathe has more than one spindle (counting the tailstock), it is not called a multispindle machine; it has only one main spindle.

Spindle types: The spindle is nothing more than a motor that lets a tool be attached to it and takes commands from a servo; the tool cuts the metal, the table moves, the motion control system controls the precision and movement, and the software does the rest. The basic spindle specifications in a CNC machining center are: max spindle speed, peak spindle motor horsepower and max spindle motor torque.

Collets types: A collet is a holding device that forms a collar around the object to be held, exerting a strong clamping force when it is tightened, used to hold a workpiece or a tool. An external collet is a sleeve with a cylindrical inner surface and a conical outer surface. The collet can be squeezed against a matching taper, the way its inner surface contracts to a slightly smaller diameter, squeezing the tool or workpiece, to a secure holding.

ER collets: Is the most widely used clamping system in the world. The standard sizes are ER-8, ER-11, ER-16, ER-20, ER-25, ER-32, ER-40 and ER-50. The "ER" comes from an existent "E" and appended the "R" for "Rego-Fix", the first manufacturer. The number is the outside collet diameter in millimeters. ER collets can hold any circular shank, metric or imperial and may also be used on a lathe to hold work pieces.

R8 collets: R8 collets were developed by Bridgeport Machines, Inc. for use in milling machines. Unusually, R8 collets fit into the machine taper itself (i.e. there is no separate chuck) and tools with integral R8 taper can also be directly fitted. R8 was developed to allow rapid tool changes and requires an exact match between collet and tool shank diameter.

Collets are most often is made of spring steel, with one or more kerfs along its length to allow it to expand and contract, and are mounted directly in a machine spindle (single or multiple), in an adaptation chuck, or in a collet block. The collet pads can be changed much quicker than a standard collet, the cost is lower, and the pads require much less storage space.
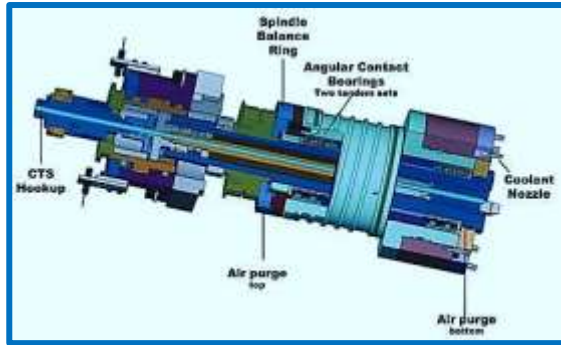
Pallets: An automatic moveable table that supports a workpiece and slides or pivots into and out of the machining center. Multiple pallets allow an operator to set up a part while another is being machined. The workpiece is attached to a pallet prior to entering a machining center.
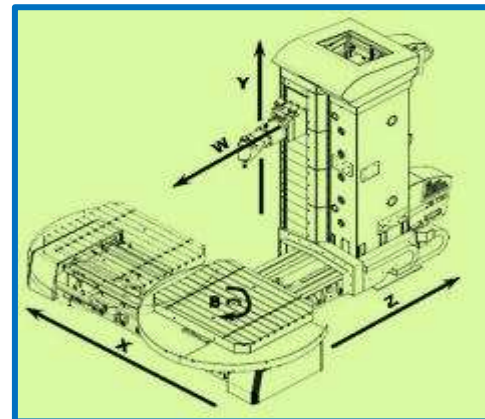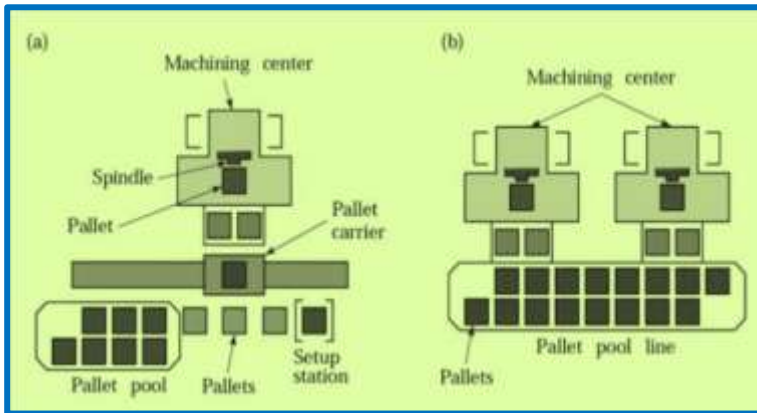

Operators can load parts on one pallet while other is in the machine. The automated pallet changing system enables the operator to keep unloading parts and setting up new jobs while the machine runs. The system automatically **exchanges pallets in less than 30 seconds** with ± 0.0001-inch repeatability and amplifies the time savings.
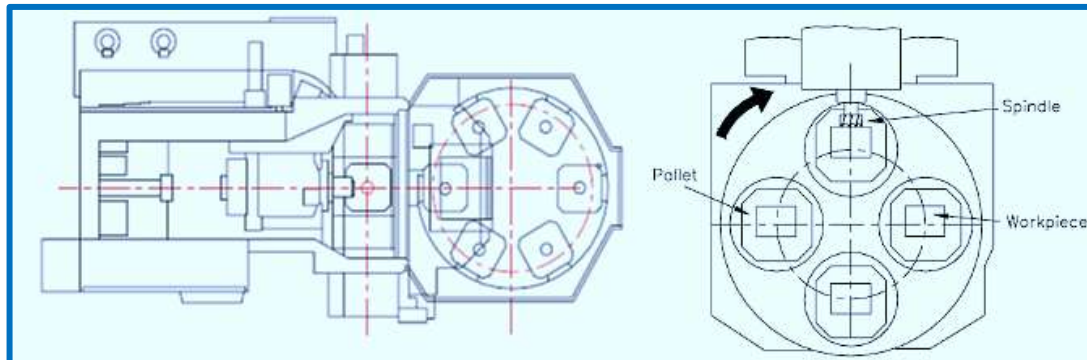
ATC (Automatic Tool Changing):

In the tool magazine the tool change takes place using a gripping system also called tool changer. The change takes place with a double arm gripping device after a new tool has been called in the NC program as follows:

- Positioning the desired tool in magazine into tool changing position;
- Taking the work spindle into changing position;
- Revolving the tool gripping device to the old tool in the spindle and to the new tool in the magazine;
- Taking the tools into the spindle and magazine and revolving the tool gripping device;
- Placing the tools into the spindle sleeve or magazine;
- Returning the tool gripping device into home position.
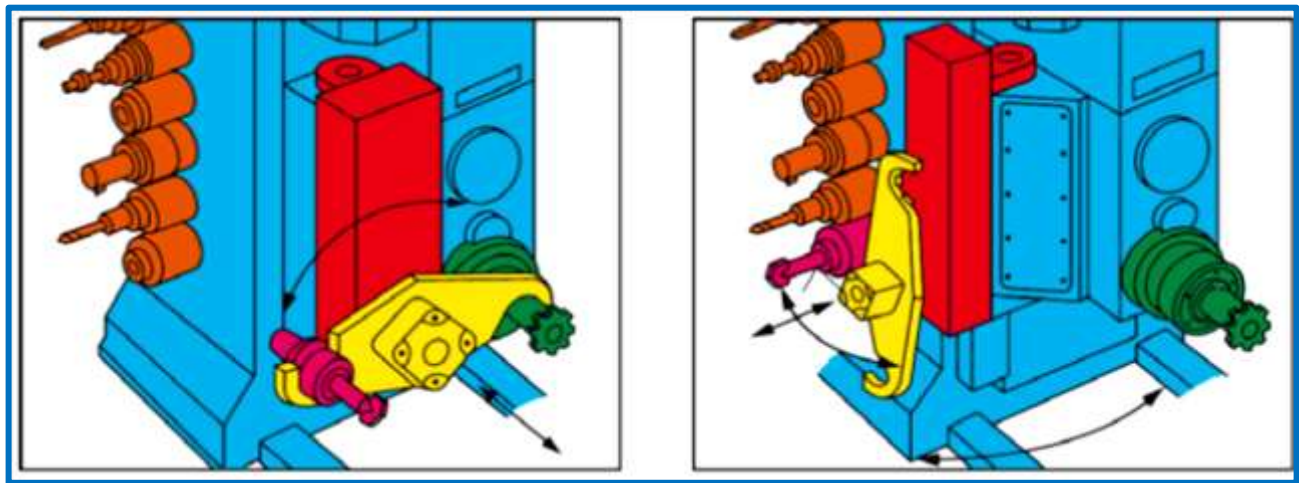
The tool change procedure takes between 6 to 15 seconds, whereby the quickest tool changers are able to make the tool change in merely one second.


| • Automatic Tool Changing;	• Tool storage: up to 200 cutting tools can be stored;
– Saves time (takes only 5 – 10 sec typically).	– Magazine; Drum or Chain.

Depending on the type and application area these tool change facilities can simultaneously take various quantities of tools and set the tool called by the NC program into working position. The most common types are: Tool Turrets and Tool Magazines.

Tool Turrets: Are mostly used for lathes and the tool magazine for milling machines. If a new tool is called by the NC program the turret rotates as long as the required tool achieves working position. Presently such a tool change only takes fractions of seconds.
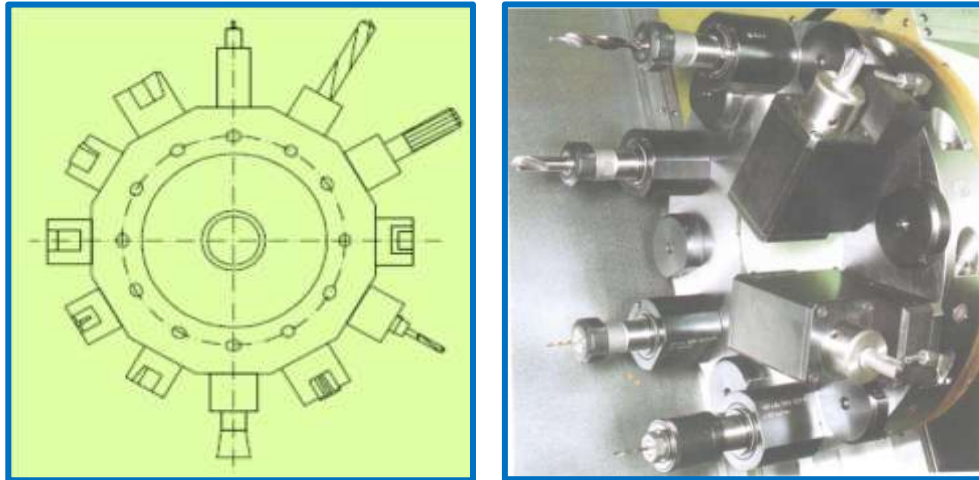
A tool turret with driven tooling

Tool Magazines: There are **chain magazines**, **ring magazines**, **umbrella magazines** and **flat magazines,** as shown below, consecutively. Depending on the type and size, the turrets of the CNC machines have 8 to 16 tool places. In large milling centers up to 3 turrets can be used simultaneously. If more than 48 tools are used tool magazines of different types are used in such machining centers allowing a charge of up to 100 and even more tools.
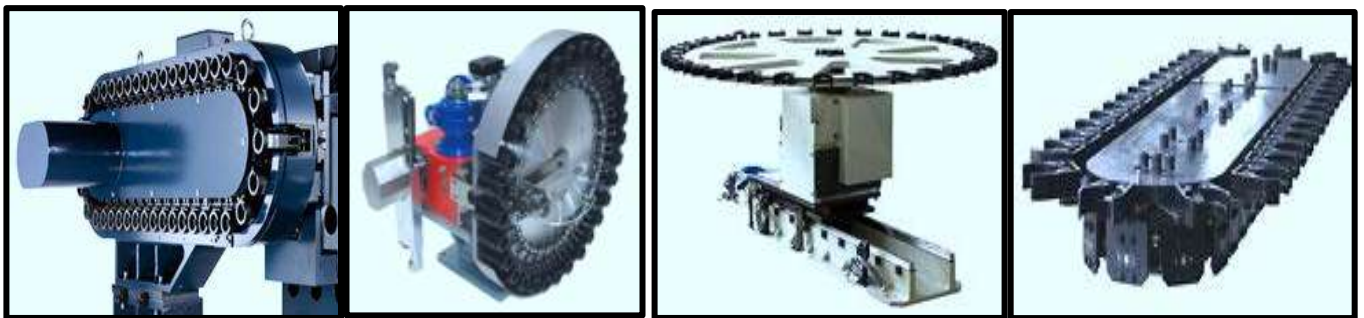
Chain tool magazine:

1. Tool holder high strength, deterioration and corrosion resistant.
2. Chains are supported on both sides; movement is highly stable.
3. Tool reverse mechanism uses curve cam design ensures smooth tool reverse movement.
4. Tool magazine mechanism is stiff. It can handle high speed tool changes and heavy tools.
5. Tool tray movement utilizes barrel cam design; movement is smooth, quiet, and precise.

Umbrella tool magazine:

1. Stable clamp strength and excellent contact with tools, protects the main spindle and tool tray during improper tool change.
2. The tool magazine can be mounted stationary style or mobile style, driven by precision motor, sequential tool change, movement is smooth, quiet, and precise.
3. Tool magazine drive source can be the servo motor and electric motor.

Flat tool magazine:

1. Stable clamp strength and excellent contact with tools, also protects the main spindle and tool tray during improper tool change.
2. Chain transaxle has rail guide as support on both sides, movement is smooth and positioning is precise.
3. Special tool clamp design allows tool magazine to be used in horizontal tool changes.
4. Tool holder can be used with pneumatic tool selection systems, hydraulic rotational systems or many tool change cams that are available on the market.
5. Tool magazine drive source can be the servo motor and electric motor.

Touch Probes:

Touch probes determine workpiece dimensions, tool dimensions and wear, but can also be used for a number of applications.

- Datuming of the workpiece
- Tool offset measurement
- Tool breakage monitoring
- Digitizing
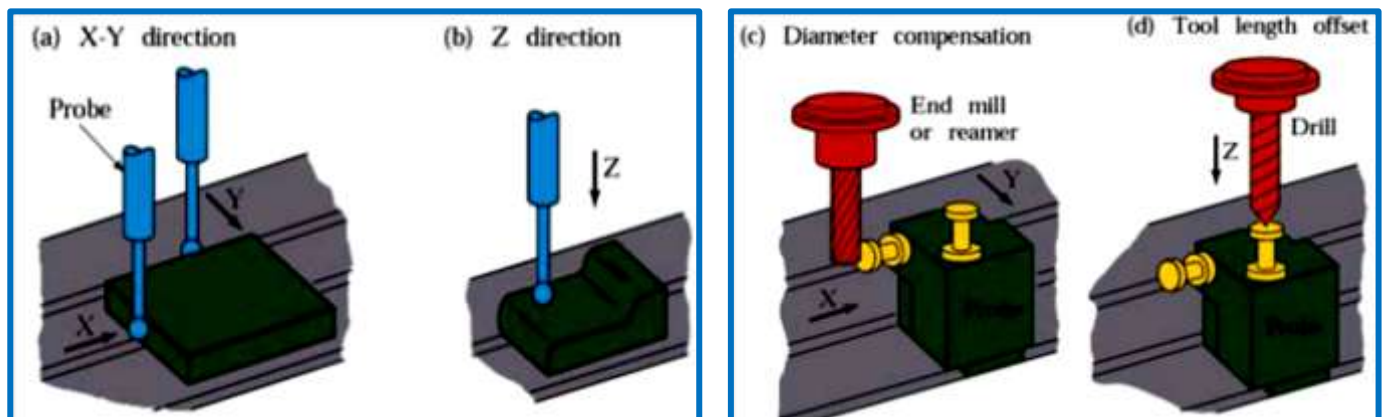
The touch probes are used for a fast, precise and automatic determination of workpiece position and workpiece dimensions on machine tools, enhance the machining accuracy and increases the productivity of a machining process.
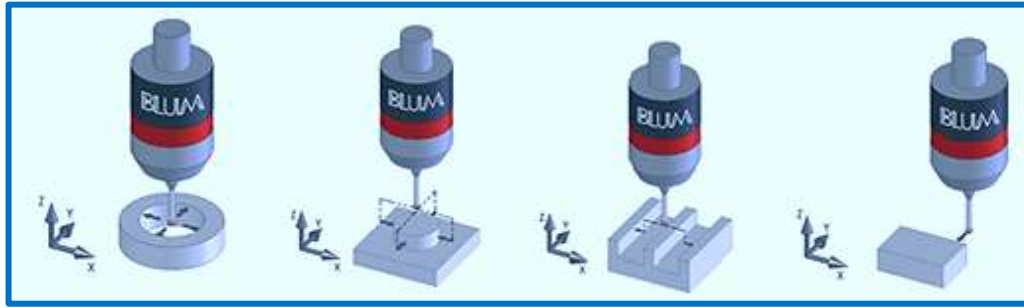
Chip Removal:

Chips and fines from the machining process, fall directly to the bottom of the conveyor or brushed-off for the top of the filter cell. The chips are then, scraped from the bottom and discharged out of the conveyor into a chip waste receptacle. Filtration in the range of 150 µm to 400 µm is also typically employed.
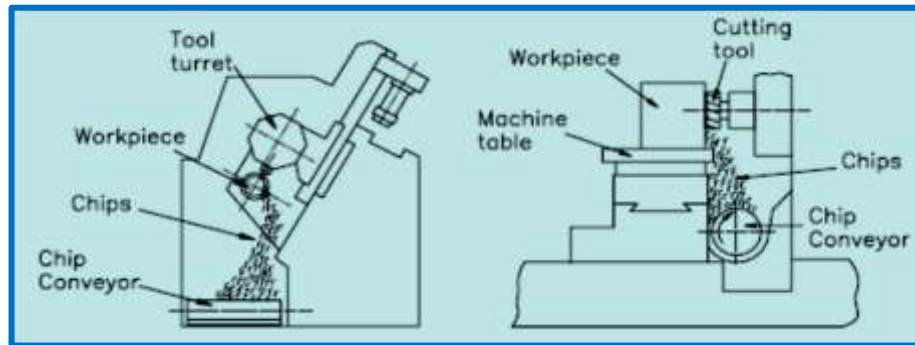
Thus, the filtered coolant, downstream consumables, bag filters, and cutting tools, lives are also extended. These systems can be provided in a wide variety of widths, lengths, inclines and belt pans to meet any chip removal application requirements.
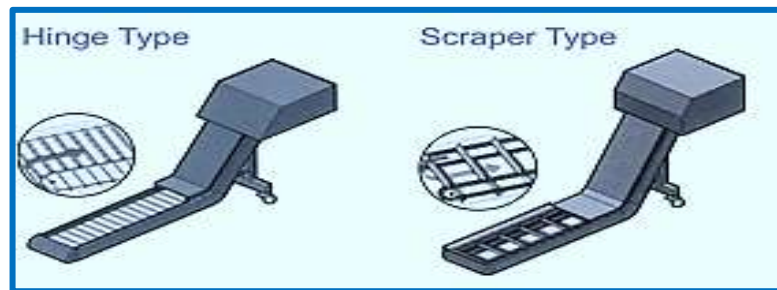
Chip conveyor types:

Comparison of Conventional and CNC machining parameters:

Parameters	Conventional	High speed machining
Spindle speeds, rpm	4,000	8,000 ~ 50,000
Axis feed rates, mm/min	10,000	2,500 ~ 60,000
Rapid feed rates, mm/min	20,000	20,000 ~ 60,000
Accelerations, g	--	0.5 ~ 2.0

TYPES OF CNC MACHINES:

CNC Milling Machines:

Milling is a machining process of using rotary cutters to remove materials from a workpiece, in a direction and an angle with the axis of the tool. It covers a wide variety of different operations and machines, on scales from small individual parts to large, heavy-duty gang milling operations. It is one of the most commonly used processes in industry and machine shops today for machining parts to precise sizes and shapes.

For simple machining controllable, hydraulic chuck jaws are sufficient. For milling on all sides the complete machining should be possible with as few re-clamping as possible. For complicated milling parts milling fixtures, also with integrated automatic rotation, are being manufactured or built out of available modular systems to allow, as far as possible, complete machining without reclamping. Work part pallets, which are loaded with the next work part by the operator outside the work room and then automatically taken into the right machining position, are increasingly being used.

There are two types of CNC Milling machines, one is horizontal and the other is vertical, classified on the basis of the positioning of the Spindle Motor. CNC Milling machines are also available in different sizes that help fabricate parts like the 3, 4, ad 5 axis CNC milling devices. Larger axis means, companies can produce comparatively faster and better results. Besides rendering better productivity and perfections, the CNC Milling machines provide a wide range of other benefits with their out-of-the-box features. Here's a list of the top 6 advantages of CNC Milling Machines.
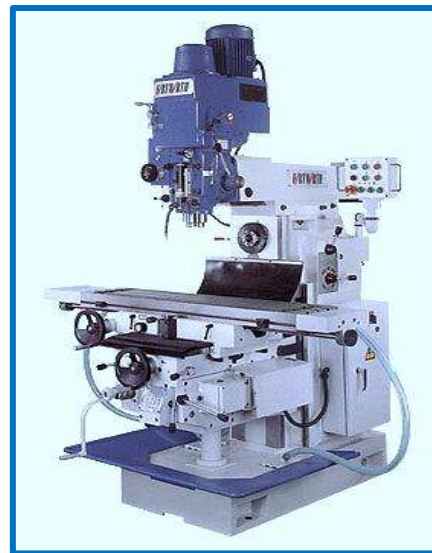
There is a high degree of standardization of the tooling used with CNC milling machines, and a lesser degree with manual milling machines. To ease up the organization of the tooling in CNC production many companies use a tool management solution. CNC milling machines nearly always use SK (or ISO), CAT, BT or HSK tooling. SK tooling is the most common in Europe, while CAT tooling, sometimes called V-Flange Tooling, is the oldest and probably most common type in the USA.

The CAT tooling was invented by **Caterpillar**, in order to standardize the tooling used on their machinery. CAT tooling comes in a range of sizes designated as CAT-30, CAT-40, CAT-50, etc. The number refers to the Association for Manufacturing Technology (formerly the National Machine Tool Builders Association (NMTB)) Taper size of the tool.

The greater plurality of standards also may apply (e.g., Morse, Jarno, Brown & Sharpe, Van Norman, and other less common builder-specific tapers). However, two standards that have seen especially wide usage are the Morse #2 and the R8, whose prevalence was driven by the popularity of the mills built by Bridgeport Machines of Bridgeport, Connecticut. These mills so dominated the market for such a long time that "Bridgeport" is virtually synonymous with "manual milling machine".

A CAT-40 toolholder　　　　　HSK-63A toolholder

- ✓ These machining centers use computer controls to cut different materials.
- ✓ They are able to translate programs consisting of specific number and letters to move the spindle to various locations and depths.
- ✓ Used to make 3D prototypes, molds, cutting dies, printing plates and sights.

CNC Lathes and Turning Centers:

The design of a CNC lathe varies with different manufacturers, but they all have some common elements. The turret clamps the tool holders, and indexes them as needed, the spindle holds the workpiece and there are slides that let the turret move in multiple axis simultaneously. With rapid growth in this industry,

different CNC lathe manufacturers use different user interfaces which sometimes make it difficult for operators as they have to be acquainted with them.

The CNC lathes are able to make fast, precision cuts using indexable tools and drills with complicated programs. Normally, they cannot be cut on manual lathes. Often include 12 tool holders and coolant pumps to cut down on tool wear. Lathe cutting operations simultaneously on a rotating part are capable of executing many different types of. The major categories of CNC turning centers can be classified as under:

a. Turn mill centers (X, Z, C);
b. Multiple axis turning centers (X, Z, C, Y);
c. Vertical turning centers;
d. Twin turret turning centers;
e. Multiple spindle turning centers;
f. Integrated material handling.

A combination lathe, often known as a 3-in-1 machine, introduces drilling or milling operations into the design of the lathe. These machines have a milling column rising up above the lathe bed, and they utilize the carriage and top slide as the X and Y axes for the milling column. The 3-in-1 name comes from the idea of having a lathe, milling machine, and drill press all in one affordable machine tool. They may be found in smaller, non-machine-oriented businesses where the occasional small part must be machined.

CNC Grinders

A tool and cutter grinder is used to sharpen milling cutters and tool bits, with other cutting tools. It is an extremely versatile machine used to perform a variety of grinding operations: **surface, cylindrical or complex shapes**. Highly automated Computer Numerical Control (CNC) machines are becoming increasingly common due to the complexities involved in the process. Modern CNC tool and cutter grinders enhance productivity by typically offering features such as automatic tool loading as well as the ability to support multiple grinding wheels.

High levels of automation, as well as, automatic in-machine tool measurement and compensation, allow extended periods of unmanned production. With careful process configuration and appropriate tool support, tolerances less than **5 μm (0.0002")** can be consistently achieved even on the most complex parts.

During normal use, cutting edges either wear or chip. The geometric features of cutting tools can be auto-matically measured within the CNC tool grinder and the tool ground to return cutting surfaces to optimal condition. A **3D simulation** of the entire grinding process and the finished part is possible, and detection of any potential mechanical collisions and calculation of production time. Such features allow parts to be de-signed, as well as, the production process optimized, entirely within the software environment.

Tool and **cutter grinders** can be adapted to manufacturing precision machine components, and when used for these purposes are called as CNC Grinding System, widely used to produce parts for aerospace, medical, automotive, and other industries. Extremely hard and exotic materials are generally no problem for today's grinding systems and the multi-axis machines are capable of generating complex geometries.

- ✓ Grinding metal process uses a coated wheel that slowly removes metal to create a part;
- ✓ Surface finish, size and roundness requirements: A grinding process can achieve a 6Ra, hold less than a 0.0002" size tolerance and grind parts to within 30 millionths roundness;
- ✓ It's a more cost effective and stable process to grind ceramics and carbides than it is to turn them;
- ✓ Automotive, aerospace, and medical require parts to be ground rather than hard turned.
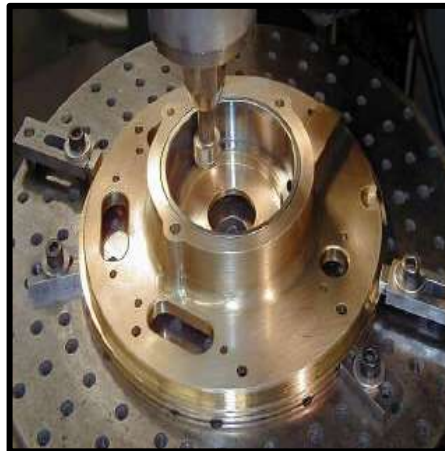
CNC Drilling:

Computer Numerical Control (CNC) Drilling is commonly implemented for mass production, however, may be multi-function machining centers which can also mill and/or turning. In order to be cost-effective, the appropriate type of CNC drilling machine needs to be applied to any particular part geometry. For hole patterns with large differences in sizes and high volume, a geared head is most appropriate. If holes are close to each other and high throughput is desired, a gearless head can locate spindles close together so that the hole pattern can be completed in one pass.

- ▪ **The bench drill:** For drilling holes through raw materials, such as wood, plastic and metal.
- ▪ **The pillar drill:** A larger drilling machine version that stands upright on the floor. As the bench drill, it can be used to drill larger pieces of materials and produce bigger holes.

The type of chips formed can be an indicator of the machinability of the material, with long gummy chips may reduce machinability. Drilled holes should be located perpendicular to the workpiece surface, when possible, to minimize the drill bit's tendency to be deflected, which causes the hole to be misplaced. The drilling types include:

- ✓ Center punching;
- ✓ Spot drilling (i.e., center drilling);
- ✓ Spot facing, which is facing a certain area on a rough casting or forging to establish, essentially, an island of precisely known surface in a sea of imprecisely known surface;
- ✓ Constraining the position of the drill bit using a drill jig with drill bushings.

Surface **finish in drilling** may range from **32 to 500 microinches**. Finish cuts will generate surfaces near 32 microinches, and roughing will be near 500 microinches. Cutting fluid is commonly used to cool the drill bit, increase tool life, increase speeds and feeds, increase the surface finish, and aid in ejecting chips.

Application of **fluids** is usually done by **flooding the workpiece or by applying a spray mist**. In deciding which drill(s) to use it is important to consider the task at hand and evaluate which drill would best accomplish the task.

There are a variety of drill styles that each serves a different purpose, as described below. The **subland drill** is capable of drilling more than one diameter. The **spade drill** is used to drill larger hole sizes. The **indexable drill** is useful in managing chips.

Gun drilling: Another type of drilling operation is called gun drilling, commonly used for drilling smaller diameter and deep holes. The gun drilling uses self-centering bits, what allows deep accurate holes, similar to a twist drill, however, the bits are designed with bearing pads that slide along the surface of the hole keeping the drill bit on center. Gun drilling is usually done at high speeds and low feed rates.

Trepanning: Is commonly used for creating larger diameter holes (up to 915 mm (36.0 in)) where a standard drill bit is not feasible or economical. Trepanning removes the desired diameter by cutting out a solid

disk similar to the workings of a drafting compass. Trepanning is performed on flat products such as sheet metal, granite (curling stone), plates, or structural members like I-beams. Trepanning can also be useful to make grooves for inserting seals, such as O-rings.

Microdrilling: Refers to the drilling of holes less than 0.5 mm (0.020 in). Drilling of holes at this small diameter presents greater problems since coolant fed drills cannot be used and high spindle speeds are required. High spindle speeds that exceed 10,000 RPM also require the use of balanced tool holders.

CNC Boring:

In machining, **boring** is the process of enlarging a hole that has already been drilled (or cast), by means of a single-point cutting tool (or of a boring head containing several such tools), for example as in boring a gun barrel or an engine cylinder. Boring is used to achieve greater accuracy of the diameter of a hole, and can be used to cut a tapered hole. Boring can be viewed as the internal-diameter counterpart to turning, which cuts external diameters. There are various types of boring.

Lineboring: The boring bar may be supported on both ends (which only works if the existing hole is a through hole), or it may be supported at one end (which works for both through holes and blind holes).

Backboring: The process of reaching through an existing hole and then boring on the "back" side of the workpiece (relative to the machine headstock).

Boring operations on small workpieces can be carried out on a lathe while larger workpieces are machined on boring mills. Workpieces are commonly 1 to 4 meters (3 ft 3 in to 13 ft 1 in) in diameter, but can be as large as 20 m (66 ft). Power requirements can be as much as 200 horsepower (150 kW). Cooling of the bores is done through a hollow passageway through the boring bar where coolant can flow freely. Tungsten-alloy disks are sealed in the bar to counteract vibration and chatter during boring. The control systems can be computer-based, allowing for automation and increased consistency.

In CNC boring machines, the positional measurement at the coordinates X, Y, Z and W is carried out by an encoder (optionally X, Y and Z by a linear measuring). The angle positioning of the spindle in the axis C is controlled by a rotational sensor and by the mutual positional coupling between the spindle and the feeding screw also thread cutting is possible. Measurement of the position B is controlled by an angle sensor, which is fastened to the fixed pin in the table center. Spindle speed can be continuously changed within the whole range of working revolutions. CNC Boring machines come in a large variety of sizes and styles, such as the examples, shown below:

CNC Horizontal Boring Mill represents the newest technology and concept of floor-type horizontal boring mills that are currently on the global marketplace. Its powerful headstock consists of a movable slide ram (Z-axis) and a moveable live spindle (W-axis). The machine column adopts a crosswise movement (along the X-axis) while rotary table moves longitudinally (V-axis/B-axis).

CNC Table Type Horizontal Boring and Milling Machine is a new type of the machine, which is built on a sliding guiding. The frame of this machine is seated on a firm stand and linked with a bed in the axis X.

The frame is immovable. Workpiece motion is performed by a special cross-type machine execution, where the plate of the rotary table is moved along the axes X and Z.

Note: The process of enlarging an existing hole or internal cylindrical surface can be accomplished on a lathe or a machine tool specifically designed for the process, such as a horizontal boring machine.

TOOL COLLISION DETECTION:

Many CNC programmers don't know anything about their working environment. Machines may have load sensing systems on spindle and axis drives, but some do not. They blindly follow the machining code provided and it is up to an operator to detect if a crash is either occurring or about to occur, and for the operator to manually abort the cutting process. Machines equipped with **load sensors can stop** axis or spindle movement in response to an overload condition, but this does not prevent a crash from occurring.

It may only limit the damage resulting from the crash. Some crashes may not ever overload any axis or spindle drives. Thus, the CNC tools must be manually "homed" or "zeroed" to have any reference to work from, and these limits are just for figuring out the location of the part to work with it, and aren't really any sort of hard motion limit on the mechanism.

It is often possible to drive the machine outside the physical bounds of its drive mechanism, resulting in a collision with itself or damage to the drive mechanism. Many machines implement control parameters limiting axis motion past a certain limit in addition to physical limit switches. However, these parameters can often be changed by the operator.

Collision detection and avoidance is possible, through the use of absolute position sensors (optical encoder strips or disks) to verify that motion occurred, or torque sensors or power-draw sensors on the drive system to detect abnormal strain when the machine should just be moving and not cutting, but these are not a common component of most hobby CNC tools.

OPERATION PROCEDURES:

Step-by-step procedures can keep you from having to memorize every function that you must perform on your CNC machining center. You will soon memorize procedures for task that you perform on a regular basis – but written procedures will always help you perform lesser used tasks.

While you have been introduced to the various buttons and switches on a typical machining center, it is unlikely that you have memorized them all. It is also unlikely that you know the appropriate order by which you should press buttons and activate switches to do anything on the machine. For example, think of what it takes to perform one of the most basic tasks a setup person or operator must do – to power up the machine. Just because you know the related buttons and switches, does not mean you know the step-by-step procedure needed to turn on a CNC machining center. The procedure for a given machining center might look something like:

Step 1: Turn on the main breaker (in back of the machine).
Step 2: Press the power on button on the control panel.
Step 3: Press the hydraulic system on button on the machine panel.
Step 4: Follow the procedure to do a manual zero return.

With such a procedure, turning on the machine is not at all difficult. While you should still know the function of the related buttons and switches, this procedure provides you with sequential order (steps) you need. And similar procedures for other important tasks will be just as easy to follow (if you have them). Admittedly, the most often used procedures will be soon memorized – and because most experienced setup people and operators have them memorized, they haven't written them down.

Most companies don't have a written set of procedures for their CNC machines. Though this may be the case, nothing stops you from developing a written set of procedures for yourself. While you'll need the help of an experienced person, you can easily develop a set of procedures that will help you (and others) remember the steps necessary to do just about anything on the machine. We divide the **procedures** needed to run a CNC machining center into **five categories**:

1. Manual procedures:
- Power up the machine;
- Do a manual zero return;
- Manually start the spindle;
- Manually jog the axes;
- Use the handwheel;
- Manually load and remove cutting tools in the spindle.

2. Setup procedures.
3. Manual data input (MDI) procedures.
4. Program editing procedures.
5. Program operation procedures.

BASIC PROGRAMMING CONCEPTS:

In order for a CPU to accomplish meaningful work, it must have two inputs: **instructions and data**. Instructions tell the CPU what actions need to be performed on the data. We have already seen how data is represented in the computer, but how do we represent the instructions? The answer is that we represent instructions with **binary codes** just like data.

The program concepts are the standard approach used today. To apply it, a machine is designed to recognize certain **bit patterns as representing certain instructions**. This collection of instructions along with the coding system is called the **machine-language** because it defines the means by which we communicate **algorithms** to the machine. Thus, both inputs to the CPU are stored in a memory, and the CPU functions by following a cycle of fetching an instruction, decoding it, and executing it.

This process is known as the **fetch-decode-execute cycle**. The cycle begins when an instruction is transferred from memory to the IR (Instruction Register) along the data bus. Then, the unique bit patterns are extracted and sent to the Decoder, what make up the machine-language. This component is responsible for the **second step** of the cycle, which is, recognizing which operation the bit pattern represents and activating the correct circuitry to perform the operation.

Sometimes this involves reading data from memory, storing data in memory, or activating the ALU to perform a mathematical operation. Once the operation is performed, the cycle begins again with the next instruction. The CPU always knows where to find the **next instruction** because the Program Counter holds the address of the current instruction. Each time an instruction is completed, the program counter is advanced by one memory location.

Each machine instruction is composed of two parts: the **op-code and the operand**. The bit pattern appearing in the op-code field indicates which of the elementary operations, such as STORE or JUMP, requested by the instruction. For example, in the case of a STORE operation, the information in the operand field indicates which register contains the data to be stored and which memory cell is to receive the data.
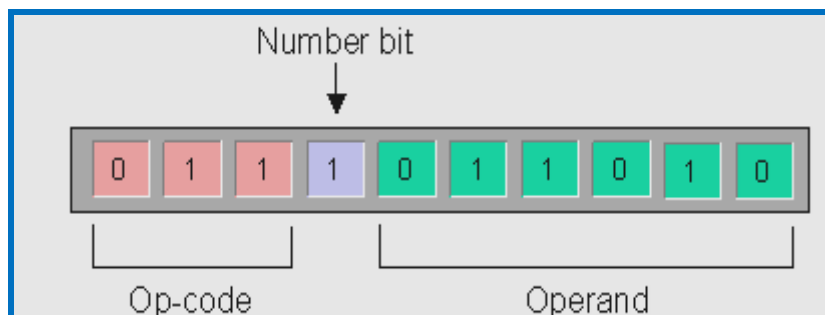
The **first three bits** represent the op-code and the **final six bits** represent the operand. The **middle bit** distinguishes between operands that are memory addresses and operands that are numbers. When the bit is **set to '1'**, the operand represents a number. A simple set of machine instructions for our CPU are listed in the table below.

Notice that all the op-codes are given an English mnemonic to simplify programming, and together, these mnemonics are called an **assembly language**. Programs written in assembly language must be converted to their **binary** representation before the CPU can understand them. This usually done by another program called an **assembler**, hence the name. See the example below:

Op-code	Mnemonic	Function	Example
001	LOAD	Load the value of the operand into the Accumulator	LOAD 10
010	STORE	Store the value of the Accumulator at the address specified by the operand	STORE 8
011	ADD	Add the value of the operand to the Accumulator	ADD #5
100	SUB	Subtract the value of the operand from the Accumulator	SUB #1
101	EQUAL	If the value of the operand equals the value of the Accumulator, skip the next instruction	EQUAL #20
110	JUMP	Jump to a specified instruction by setting the Program Counter to the value of the operand	JUMP 6
111	HALT	Stop execution	HALT

A simple machine language

In the machine language above, notice that some of the operands include the **# symbol**. This symbol tells the CPU that the operand represents a number rather than a memory address. Thus, when the assembler translates an instruction with a **# symbol**, the resulting machine code will have a '1' in the position of the number bit.

Also notice the central role that the Accumulator register plays. Nearly all the operations affect the value of this register since the Accumulator acts as a temporary memory location for storing calculations in progress. With our machine language defined, we are ready to take a look at some simple programs.

The first program is called Sum. This program adds the numbers stored in two memory locations. Mathematically, this program represents the formulas x = 2, y = 5, x + y = z where the variables x, y, and z correspond with the memory locations 13, 14, and 15 respectively. The instructions for the program are listed below.

| #	Machine code	Assembly code	Description
0	001 1 000010	LOAD #2	Load the value 2 into the Accumulator
1	010 0 001101	STORE 13	Store the value of the Accumulator in memory location 13
2	001 1 000101	LOAD #5	Load the value 5 into the Accumulator
3	010 0 001110	STORE 14	Store the value of the Accumulator in memory location 14
4	001 0 001101	LOAD 13	Load the value of memory location 13 into the Accumulator
5	011 0 001110	ADD 14	Add the value of memory location 14 to the Accumulator
6	010 0 001111	STORE 15	Store the value of the Accumulator in memory location 15
7	111 0 000000	HALT	Stop execution

Sum program

The second program is called Count. This program counts up to a number specified by the programmer in the first instruction. Notice that this program incorporates a loop construction by using the JUMP and EQUAL instructions. Every time the value in the Accumulator is incremented, the count is tested to see if it has reached the specified amount.

#	Machine code	Assembly code	Description
0	001 1 000101	LOAD #5	These two operations set the count value to five
1	010 0 001111	STORE 15	
2	001 1 000000	LOAD #0	Initialize the count to zero
3	101 0 001111	EQUAL 15	Test to see if count is complete; if yes, skip next instruction and go to instruction 5; if no, go to next instruction
4	110 1 000110	JUMP #6	Set Program Counter to 6
5	111 0 000000	HALT	Stop execution
6	011 1 000001	ADD #1	Increment the count in the Accumulator
7	110 1 000011	JUMP #3	Set Program Count to 3

Every **CNC program** is a sequence of program blocks, written together to form a complete tool-path for one or many tools. This **tool-path** tells to a CNC machine how the operator wants his component be machined. Every single CNC program block adds/alters/modifies some useful information to a program.

CNC MACHINE PROGRAMMING:

The CNC machine programming contains all geometric data about the part and motion information to move the cutting tool with respect to the work piece. Basically, the machine receives instructions as a sequence of blocks, containing commands to set machine parameters; speed, feed and other relevant information. A block is equivalent to a line of codes in a part program.
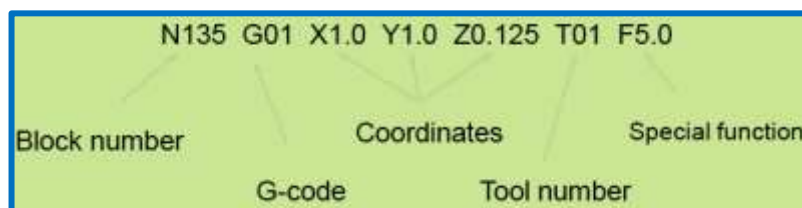
The transfer of an engineering blueprint of a product to a part program can be performed manually using a calculator or with the assistance of a computer language. A part programmer must have an extensive knowledge of the machining processes and the capabilities of the machine tools. In this section, we describe how the part programmers execute manually the part programs.

In CNC program block there are six words separated by spaces. Every word is a **group of alphanumeric characters**, led by a character as above **N, G, X, Y, F, M** and consist of some signed/unsigned numeric value as above give **50, 01, 100, –50.02, 100, 08**. Thus, a CNC program block is normally written as:

N50 G01 X100 Y-50.02 F100 M0

Alphabetic & Special Address Codes:

Every letter of the alphabet is used as a machine address code. Codes are either modal, which means they remain in effect until cancelled or changed, or non-modal, which means they are effective only in the current block. The table below lists the most common address codes.

A	Rotation about X-axis.
B	Rotation about Y-axis.
C	Rotation about Z-axis.
D	Cutter diameter compensation (CDC) offset address.
F	Feed rate.
G	G-Code (preparatory block code).
H	Tool length offset (TLO).
I	Arc center X-vector, also used in drill cycles.
J	Arc center Y-vector, also used in drill cycles.
K	Arc center Z-vector, also used in drill cycles.
M	M-Code (miscellaneous code).
N	Block Number.
O	Program Number.
P	Dwell time.
Q	Used in drill cycles.
R	Arc radius, also used in drill cycles.
S	Spindle speed in RPM.
T	Tool number.
X	X-coordinate.
Y	Y-coordinate.
Z	Z-coordinate.

Alphabetic Address Code Definitions:

A, B, C - 4th/5th Axis Rotary Motion: Rotation about the X, Y or Z-axis respectively. The angle is in degrees and up to three decimal places precision.

Example: G1 A30.513 B90.

D - Tool Diameter Register: D is accompanied by an integer that is the same as the tool number (T1 uses D1, etc.). It is always used in conjunction with G41 or G42 and a XY move (never an arc). When called, the control reads the register and offsets the tool path left (G41) or right (G42) by the value in the register.

Example: G1 G41 X1. D1.

F - Feed Rate: Sets the feed rate when machining lines, arcs or drill cycles. Feed rate can be in Inches per Minute (G94 mode) or Inverse Time (G93 mode). Feed rates can be up to three decimal places accuracy (for tap cycles) and require a decimal point.

Example: G1 X1. Y0. F18.

G - Preparatory Code: Always accompanied by an integer that determines its meaning. Most G-codes are modal. Expanded definitions of G-codes appear in the next section of this chapter.

Example: G2 X1. Y1. I.25 J0.

H - Tool Length Compensation Register: This code calls a tool length offset (TLO) register on the control. The control combines the TLO and Fixture Offset Z values to know where the tool is in relation to the part datum. It is always accompanied by an integer (H1, H2, etc.), G43, and Z coordinate.

Example: G43 H1 Z1.

I - Arc Center or Drill Cycle Data: For arc moves (G2/G3), this is the incremental X-distance from the arc start point to the arc center. Certain drill cycles also use I as an optional parameter.

Example: G2 X.1 Y2.025 I0. J0.125.

J - Arc Center or Drill Cycle Data: For arc moves (G2/G3), this is the incremental Y-distance from the arc start point to the arc center. Certain drill cycles also use J as an optional parameter.

Example: G2 X.1 Y2.025 I0. J0.125.

K - Arc Center or Drill Cycle Data: For an arc move (G2/G3) this is the incremental Z-distance from the arc start point to the arc center. In the G17 plane, this is the incremental Z-distance for helical moves. Certain drill cycles also use J as an optional parameter.

Example: G18 G3 X.1 Z2.025 I0. K0.125.

M - Preparatory Code: Only one M-code is allowed in each block of code. Block numbers can make the CNC program easier to read, required for CAD/CAM programs with no subprograms. Block numbers are integers up to five characters long with no decimal point. M8, N - Block Number.

Example: N100 T2 M6

O - Program Number: Programs are stored on the control by their program number. This is an integer that is preceded by the letter O and has no decimal places.

Example: O0002 (PROJECT 1).

P – Delay: Dwell (delay) in seconds, accompanied by G4 unless used within certain drill cycles.

Example: G4 P.1.

Q - Drill Cycle Optional Data: The incremental feed distance per pass in a peck drill cycle.

Example: G83 X1. Y1. Z-.5 F12. R.1 Q.1 P5.

R - Arc Radius or Drill Cycle Optional Data: Arcs can be defined using the arc radius R or I, J, K vectors are also used by drill cycles as the return plane Z value.

Example: G83 Z-.5 F12. R.1 Q.1 P5.

S - Spindle Speed: Spindle speed in revolutions per minute (RPM). It is an integer value with no decimal, and always used in conjunction with M3 (Spindle on CW) or M4 (Spindle on CCW).

Example: S3820 M3.

T - Tool number: Selects tool. It is an integer value always accompanied by M6 (tool change code).

Example: T1 M6.

X – X - Coordinate: Coordinates are modal, so there is no need to repeat them in subsequent blocks if they do not change.

Example: G1 X1.1252.

Y – Y - Coordinate: Coordinate data for the Y-axis.

Example: G1 Y1.

Z – Z - Coordinate: Coordinate data for the Z-axis.

Example: G1 Z-.125

Special Character Code Definitions: The following is a list of commonly used special characters, their meaning, use, and restrictions.

% - Program Start or End: All programs begin and end with % on a block by itself. This code is called tape rewind character (a holdover from the days when programs were loaded using paper tapes).

( ) – Comments: Comments to the operator must be all caps and enclosed within brackets. The maximum length of a comment is 40 characters and all characters are capitalized.

Example: (T2: .375 END MILL).

/ - Block Delete: Codes after this character are ignored if the Block Delete switch on the control is on.

Example: / M0.

; - End of Block: This character is not visible when the CNC program is read in a text editor (carriage return), but does appear at the end of every block of code when the program is displayed on the machine control.

Example: N8 Z0.1;

Application of G & M - Codes:

The G & M Codes make up the most of the contents of the CNC program. Codes that begin with G are called preparatory words because they prepare the machine for a certain type of motion. The most common G-codes are shown in this table below:

G0	Rapid motion, used to position the machine for non-milling moves.
G1	Line motion at a specified feed rate.
G2	Clockwise arc.
G3	Counterclockwise arc.
G4	Dwell.
G28	Return to machine home position.
G40	Cutter Diameter Compensation (CDC) off.
G41	Cutter Diameter Compensation (CDC) left.
G42	Cutter Diameter Compensation (CDC) right.
G43	Tool length offset (TLO).
G54	Fixture Offset #1.
G55	Fixture Offset #2.
G56	Fixture Offset #3.

G57	Fixture Offset #4.
G58	Fixture Offset #5.
G59	Fixture Offset #6.
G80	Cancel drill cycle.
G81	Simple drill cycle.
G82	Simple drill cycle with dwell.
G83	Peck drill cycle.
G84	Tap cycle.
G90	Absolute coordinate programming mode.
G91	Incremental coordinate programming mode.
G98	Drill cycle return to Initial point (R).
G99	Drill cycle return to Reference plane (last Z Height)

Application of M - Codes:

M0	Program stop. Press Cycle Start button to continue.
M1	Optional stop. Only executed if Op Stop switch on the CNC control is turned ON.
M2	End of program.
M3	Spindle on Clockwise.
M4	Spindle on Counterclockwise.
M5	Spindle stop.
M6	Change tool.
M8	Coolant on.
M9	Coolant off.
M30	End program and press Cycle Start to run it again.

G - Codes References:

G00 - Rapid Positioning Motion (X, Y, Z, A, B)
G01 - Linear Interpolation Motion (X, Y, Z, A, B, F)
G02 - Circular Interpolation Motion Clockwise-CW (X, Y, Z, A, I, J, K, R, F)
G03 - Circular Interpolation Motion Counter Clockwise-CCW (X, Y, Z, A, I, J, K, R, F)
G04 - Dwell (P) (P=Seconds)

G09 - Exact Stop, Non-Modal

G10 – Data Setting

G17 - Circular Motion XY Plane Selection (G02 or G03)

G18 - Circular Motion ZX Plane Selection (G02 or G03)

G19 - Circular Motion YZ Plane Selection (G02 or G03)

G20 - Inch Coordinate Positioning

G21 - Metric Coordinate Positioning

G28 - Machine Zero Return Thru Ref. Point (X, Y, Z, A, B)

G29 - Move to Location through G28 Ref. Point (X, Y, Z, A, B)

G30 – Return to 2nd Position (Pallet/Tool Change)

G40 - Cutter Compensation Cancel

G41 - 2D Cutter Compensation, Left (X, Y, D)

G42 - 2D Cutter Compensation, Right (X, Y, D)

G43 - Tool Length Compensation + (H, Z)

G49 - Tool Length Compensation Cancel G43/G44/G43

G52 - Work Offset Positioning Coordinate

G53 - Machine Positioning Coordinate, Non-Modal (X, Y, Z, A, B)

G54 - Work Offset Positioning Coordinate #1

G55 - Work Offset Positioning Coordinate #2

G56 - Work Offset Positioning Coordinate #3

G57 - Work Offset Positioning Coordinate #4

G58 - Work Offset Positioning Coordinate #5

G59 - Work Offset Positioning Coordinate #6

G73 - HS Peck Drilling Canned Cycle (X,Y,A,B,Z,I,J,K,Q,P,R,L,F)

G74 - Reverse Tapping Canned Cycle (X, Y, A, B, Z, J, R, L, F)

G76 - Fine Boring Canned Cycle (X,Y,A,B,Z,I,J,P,Q,R,L,F)

G77 - Black Bore Canned Cycle (X,Y,A,B,Z,I,J,Q,R,L,F)

G80 - Cancel Canned Cycle

G81 - Drill Canned Cycle (X, Y, A, B, Z, R, L, F)

G82 - Spot Drill / Counterbore Canned Cycle (X,Y,A,B,Z,P,R,L,F)

G83 - Peck Drill Deep Hole Canned Cycle (X,Y,A,B,Z,I,J,K,Q,P,R,L,F)

G84 - Tapping Canned Cycle (X,Y,A,B,Z,J,R,L,F)

G85 - Bore In ~ Bore Out Canned Cycle (X, Y, A, B, Z, R, L, F)

G86 - Bore In ~ Stop ~ Rapid Out Canned Cycle (X, Y, A, B, Z, R, L, F)

G87 - Bore In ~ Manual Retract Canned Cycle (X,Y,A,B,Z,R,L,F)

G88 - Bore In ~ Dwell ~ Manual Retract Canned Cycle (X,Y,A,B,Z,P,R,L,F)

G89 - Bore In ~ Dwell ~ Bore Out Canned Cycle (X,Y,A,B,Z,P,R,L,F)

G90 - Absolute Positioning Command

G91 - Incremental Positioning Command

G92 - Global Work Coordinate System

G93 - Inverse Time Feed Mode, ON

G94 - Inverse Time Feed OFF / Feed per Minute, ON

G98 - Canned Cycle Initial Point Return

G99 - Canned Cycle R Plane Return

M – Codes References:

M00 - Program Stop
M01 - Optional Program Stop
M02 - Program End
M03 – Spindle, ON Clockwise (S)
M04 – Spindle, ON Counterclockwise (S)
M05 - Spindle Stop
M06 - Tool Change (T)
M08 – Coolant, ON
M09 – Coolant, OFF
M30 - Program End and Reset
M31 - Chip Auger Forward
M33 - Chip Auger Stop
M34 - Coolant Spigot Position Down, Increment
M35 - Coolant Spigot Position Up, Decrement
M36 - Pallet Part Ready
M41 - Spindle Low Gear Override
M42 - Spindle High Gear Override
M50 - Execute Pallet Change
M83 - Auto Air Jet ON
M84 - Auto Air Jet OFF
M88 - Coolant through Spindle, ON
M97 - Local Sub-Program Call (P, L)
M98 - Sub-Program Call (P, L)
M99 - Sub-Program / Routine Return of Loop (P)

Lathe G - Codes:

G00 - Rapid Positioning Motion
G01 - Linear Interpolation Motion
G02 - Circular Interpolation Motion CW
G03 - Circular Interpolation Motion CCW
G04 - Dwell (P) (P=Seconds)
G09 - Exact Stop, Non-Modal
G18 - Circular Motion ZX Plane Selection (G02 or G03)
G20 - Inch Coordinate Positioning
G21 - Metric Coordinate Positioning
G28 - Machine Zero Return Thru Ref. Point
G29 - Move to Location through G28 Ref. Point
G32 - Threading
G40 - Tool Nose Compensation Cancel
G41 - Tool Nose Compensation, Left
G42 - Tool Nose Compensation, Right

G43 - Tool Length Compensation

G49 - Tool Length Compensation Cancel G43/G44/G43

G50 - Spindle Speed Clamp/Set Global Coordinate, Offset

G51 - Cancel Offset (Yasnac)

G52 - Set Local Coordinate System (Fanuc)

G53 - Machine Coordinate Selection

G54 - Work Offset Positioning Coordinate #1

G55 - Work Offset Positioning Coordinate #2

G56 - Work Offset Positioning Coordinate #3

G57 - Work Offset Positioning Coordinate #4

G58 - Work Offset Positioning Coordinate #5

G59 - Work Offset Positioning Coordinate #6

G61 - Exact Stop Modal

G64 - G61 Cancel

G70 - Finishing Cycle

G71 - OD/ID Stock Removal Cycle

G72 - Face Stock Removal Cycle

G73 - Irregular Path Stock Removal Cycle

G74 - Face Grooving Cycle, Peck Drilling

G75 - OD/ID Grooving Cycle, Peck Drilling

G76 - Threading Cycle, Multiple Pass

G80 - Cancel Canned Cycle

G81 - Drill Canned Cycle

G82 - Spot Drill / Counterbore Canned Cycle

G83 - Peck Drill Deep Hole Canned Cycle

G84 - Tapping Canned Cycle

G85 - Bore In ~ Bore Out Canned Cycle

G86 - Bore In ~ Stop ~ Rapid Out Canned Cycle

G87 - Bore In ~ Manual Retract Canned Cycle

G88 - Bore In ~ Dwell ~ Manual Retract Canned Cycle

G89 - Bore In ~ Dwell ~ Bore Out Canned Cycle

G90 - OD/ID Turning Cycle, Modal

G92 - Threading Cycle, Modal

G94 - End Facing Cycle, Modal

G95 - Sub spindle Rigid Tap

G96 - Constant Surface Speed (CSS) On

G97 - Constant Surface Speed Cancel

G98 - Feed Per Minute

G99 - Feed Per Revolution

Lathe M - Codes:

M00 - Program Stop

M01 - Optional Program Stop

M02 - Program End
M03 - Spindle ON Clockwise (Forward)
M04 - Spindle ON Counterclockwise (Reverse)
M05 - Spindle Stop
M08 - Coolant ON
M09 - Coolant OFF
M10 - Clamp Chuck
M11 - Unclamp Chuck
M12 - Auto Air Jet On
M13 - Auto Air Jet Off
M14 - Clamp Main Spindle
M15 - Unclamp Main Spindle
M19 - Orient Spindle with P value
M21 - Tailstock Forward
M22 - Tailstock Reverse
M23 - Thread Chamfer On
M24 - Thread Chamfer Off
M30 - Program End and Reset
M31 - Chip Auger Forward
M33 - Chip Auger Stop
M36 - Parts Catcher Up
M37 - Parts Catcher Down
M41 - Low Gear
M42 - High Gear
M88 - High Pressure Coolant On
M89 - High Pressure Coolant Off
M133 - Live Tool Drive Forward
M134 - Live Tool Drive Reverse
M135 - Live Tool Drive Stop

Note: Only one M-Code may appear in each block of code.

PROGRAM BLOCKS & SEQUENCE NUMBERS:

First, the machining parameters are determined. **Second**, the optimal sequence of operations is evaluated. **Third**, the tool path is calculated. **Fourth**, a program is written. Each line of the program, referred to as a block, contains the required data for transfer from one point to the next. A **typical line** is given below.

N100 G91 X -5.0 Y7.0 F100 S200 T01 M03 (EOB);

Sequence Number, N: Typically **three digits**, its purpose is to identify the **specific machining operation** through the block number particularly when testing a part program. The CNC program block sequence **N Numbers,** make the logic easier, as while you will start the CNC cycle programming.

G-Code: Every CNC program starting with G is called a **G-code**. G01 is the G-code for Linear - Interpolation meaning that, you want the CNC machine tool to go to a specific position, with a controlled **feed** (read below Tool Feed), giving the specific position and feed amount. The G-codes prepares the MCU circuits to perform a specific operation. The sequence number, as the above example, G91, may be a 91 mm linear position.

Positioning X, Y: The CNC lathe machine normally has **two axis X and Z**. Thus the programming X -5.0, Y -7.0 (as shown above after G91) denotes the axis positioning X and Y for this CNC lathe machine. In the above block, **X moves** a distance of **5 in.** in the **negative** direction while **Y moves** a distance of **7 in.** in the **positive** direction.

Feedrate, F: Every motion command requires the feed to be given, thus, the tool can be moved to a calculated given position. It is expressed in **in/min** or **mm/min** and, is used in contouring or point-to-point or straight-cut systems. For example, a feedrate of F100 implies **100 in/min** or **100 mm/min**.

Spindle speed, S: Programmed in Revolution per Minute, is expressed as RPM, or by a **three-digit code** number, which is related to the RPM. The S200, on the example above, may stand for 200 RPM.
Tool Number, T: Each cutting tool has a different code number consisting of a maximum of **five digits**. The tool is automatically selected by the Automatic Tool Changer (ATC) when the code number is programmed in a block.

Miscellaneous Function, M: The *M* miscellaneous code may order the CNC machine to open the coolant (coolant pumping, ON), to spin the spindle or to stop spindle. The M03, on the example above, may stand for any miscellaneous operation.

EOB: The EOB character is used at the end of each block to complete a line.

As mentioned, a part program is a set of instructions often referred to as blocks, each of which refers to a segment of the machining operation performed by the machine tool. Each block may contain several code words in sequence, that is:

- ✓ Coordinate values (X, Y, Z, etc.) to specify the desired motion of a tool relative to a work piece. The coordinate values are specified within motion code words and related interpolation parameters to indicate the type of motion required (e.g. point-to-point, or continuous straight or continuous circular) between the start and end coordinates.
- ✓ Machining parameters such as, feed rate, spindle speed, tool number, tool offset compensation parameters etc.
- ✓ Codes for initiating machine tool functions like starting and stopping of the spindle, on/off control of coolant flow and optional stop. In addition to these coded functions, spindle speeds, feeds and the required tool numbers to perform machining in a desired sequence are also given.
- ✓ Program execution control codes, such as block skip or end of block codes, block number etc.
- ✓ Statements for configuring the subsystems on the machine tool such as programming the axes, configuring the data acquisition system, etc.

A typical block of a part program is shown below. Another way of writing a block is explained below showing the variety of the code words. Each of these code words configures a particular aspect to be used during the machining of the particular segment.
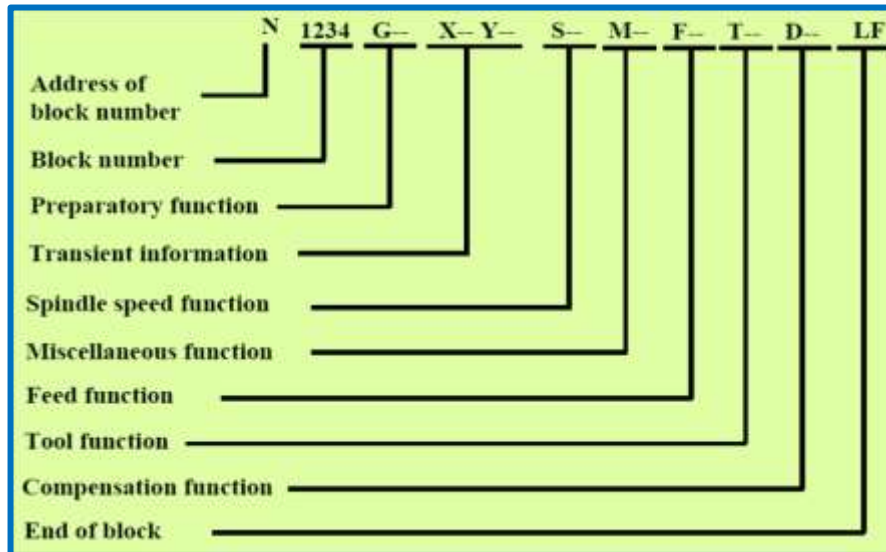

CNC AXIS POSITIONING:

1. **X, Y, Z**: CNC Machine absolute coordinates;
2. **I, J, K**: Circular dimension coordinates, for distances to the arc center;
3. **A, B, C**: Angular coordinates.

For example, a CNC lathe axis **X100, Y-50.02**, means that the tool should move 100 (mm or inch set by the G-code) in positive X-axis direction, and move the tool to **Y–50.02** (mm or inch) in negative Y-axis direction, as shown graphically below:
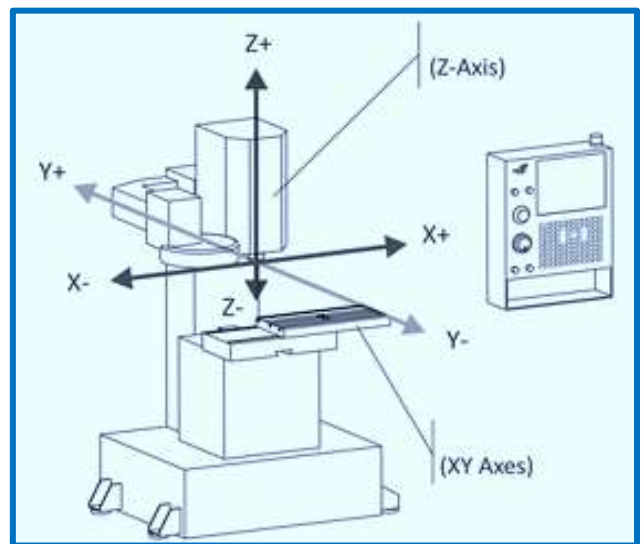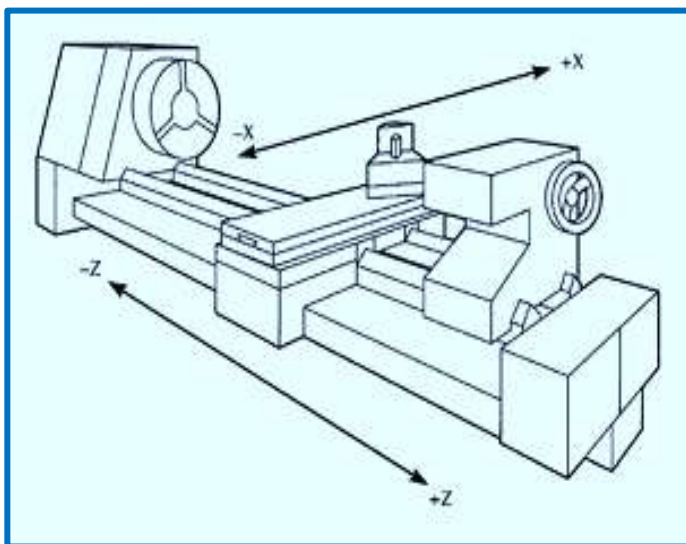
CNC Lathe Machine normally has two axis X, Z - CNC milling machine normally has three axis X, Y, Z.

Note: All **angular dimensions** are specified in **revolutions or degrees**. In some systems, common coordinate distances are used, in others, the axis positioning are programmed in BLUs (Basic Length Unit). For example, **1 BLU = 0.0001"** means that the axis will move 0.0001", for every one electrical pulse received by the servo motor.
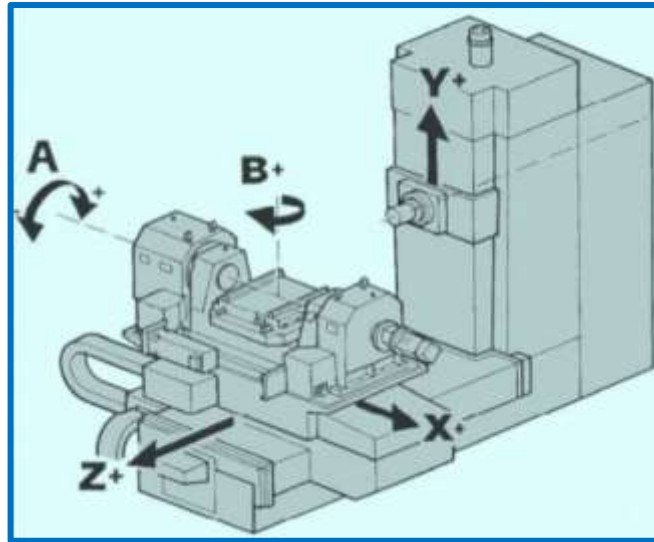
The **CNC turning centers** can be classified as:

a. Turn mill centers (X, Z, C);
b. Multiple axis turning centers (X, Z, C, Y);
c. Vertical turning centers;
d. Twin turret turning centers;
e. Multiple spindle turning centers;
f. Integrated material handling.

The typical shapes of component that can be machined by the combination of **X, Z and C-axis** movements, are shown below:
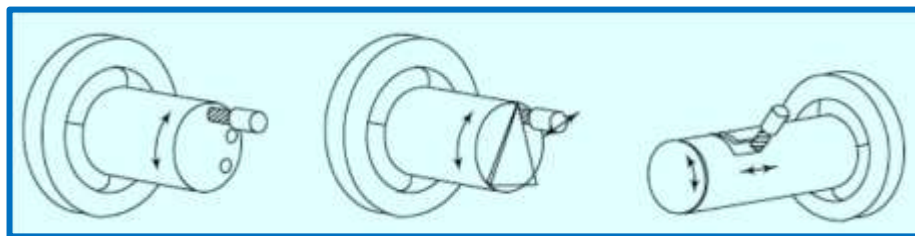
Circular Motion Feedrate, F:

In **linear** motions, the feedrate of the cutting tool **is not corrected** for the cutter radius. But in **circular** motions, the feedrate should be corrected. For cutting around the **outside** of a circle, the plus sign (+) is used, and the feedrate is **increased**. For cutting around the **inside** of a circle, the minus sign (-) is used, and the feedrate is **decreased**, as follows:

F = [(part contour radius ± tool radius)/part contour radius] (required feedrate).

Example:

If the required feedrate is **6 in/min**, part contour radius is **1 .5 in**, and the cutter diameter is **1.0 in**, what is the feedrate at the **top** and **bottom** circles?

Answer: Feedrate at the top is to be **8 in/min** and at the bottom is to be **4 in/min**.

Spindle control:

As described above, the spindle speed is programmed by the letter **'S'** followed by four digit number, such as **S1000**. The spindle speed in **revolutions per minute** is also known as **constant rpm or direct rpm**. The change in tool position does not affect the rpm commanded. It means that the spindle RPM will remain constant until another RPM is programmed.

Constant surface speed is almost exclusively used on lathes. The RPM changes according to diameter being cut. The smaller the diameter, the more RPM is achieved; the bigger the diameter, the less RPM is commanded. This is changed automatically by the machine speed control unit while the tool is changing positions. This is the reason that, this spindle speed mode is known as diameter speed. There are two ways to define speed: Revolutions per minute (RPM, and Constant surface speed;

Tool selection:

Tool selection is accomplished using **'T'** function followed by a four digit number where, first two digits are used to call the particular tool and last two digits are used to represent tool offset in the program. The tool offset is used to correct the values entered in the coordinate system preset block. This can be done quickly on the machine without actually changing the values in the program. Using the tool offsets, it is easy to set up the tools and to make adjustments.

PROGRAM SUBROUTINES:

Programmed order of instruction or unchanging sequences can become a subprogram, as shown:

N10; N20; N30;….; N70 G22 N5; N80; N90....

- Repetitive machining motions;
- Functions relating to tool change;
- Hole patterns, grooves and threads;
- Machine warm-up routines;
- Pallet changing;

TOOL LENGTH COMPENSATION & INTERPOLATION:

The programmed point on the part is the command point. It is the destination point of the tool. The point on the tool that is used for programming is the tool reference point. These points may or may not coincide,

depending on the type of tool used and machining operation being performed. When drilling, tapping, reaming, countersinking or boring on the machining center, the tool is programmed to the position of the hole or bore center - this is the command point.

Tool Length Compensation: There are a variety of cutting tool types that are used on machining centers, including, center drills, spot drills, drills, taps, reamers, boring bars, end mills, and face mills. Each type of cutting tool requires its own type of tool holder. Most straight-shank tools use a collet holder. Others (like end mills) use a tool holder having a set-screw to clamp the cutting tool in place.

Yet others (like face mills and taps) require a very special style of tool holder – designed specifically for the cutting tool type. Then, as the program is written, the **programmer may not know** the **precise length** of any given tool. Figure below illustrates this point with five different tools.
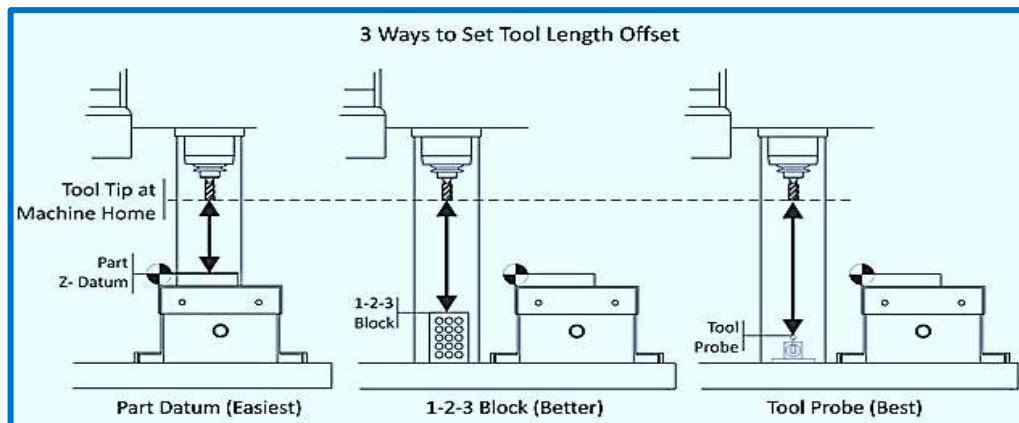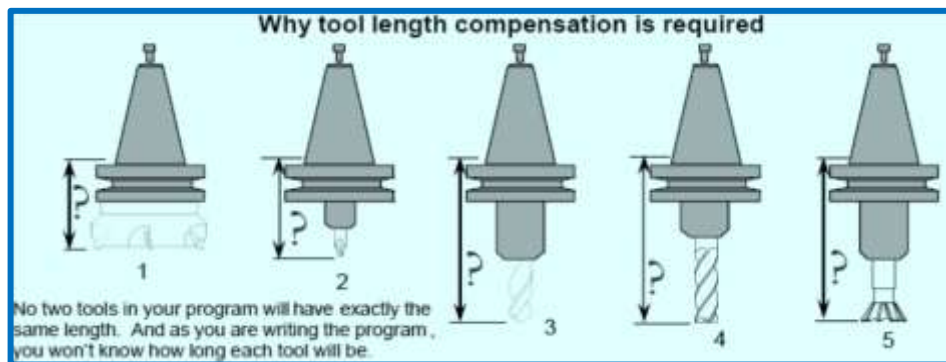
Interpolation: Interpolation consists in the calculation of the coordinated movement of several axes using the programmed parameters, in order to obtain a resulting trajectory, which can be of various types, as:

- Straight line;
- Circular;
- Helicoidal.

The interpolation module computes instant by instant position commands for the servo module, which in turn, drives the motors. There are two types of interpolators, as follows: Process interpolator (for continuous axes) and Point-to-point interpolator (for point-to-point axes).
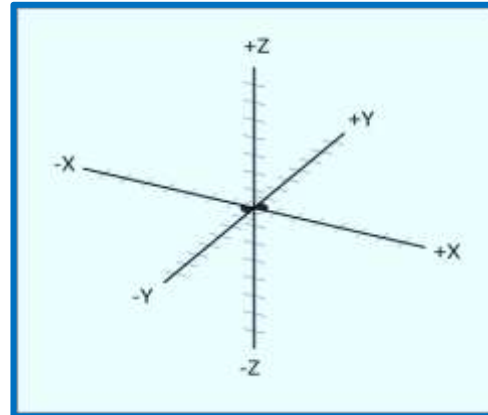
THE CARTESIAN COORDINATE SYSTEM:

The CNC motion is based on a 3D Cartesian coordinate system. The basis of this system is the number line marked at equal intervals and the axis is labeled (X, Y or Z). One point on the line is designated as the origin, and numbers at right side are marked as positive and on the left side are negative.
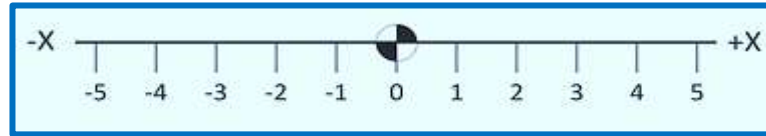

X-Axis Number Line

The Cartesian coordinate system consists of three number lines, labeled X, Y and Z, set at 90° to each other as shown below. The origin, or datum, is where the three axes cross each other. The labels, orientations, and directions of the Cartesian coordinate system are typical of most Vertical Machining Center.

3D Cartesian Coordinate System

Quadrants:

Any two axes form a plane. Planes are named by the axes that define them. For example, Figure 4.3 shows the XY plane, which is the primary work plane for machining on a VMC. A plane can be divided into four quadrants, labeled I, II, III and IV with axes designations as shown in the illustration below.
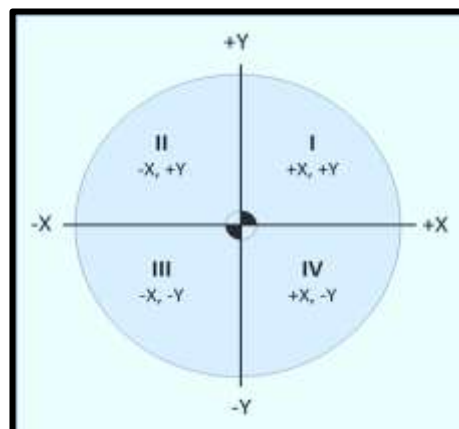
Quadrants

Units:

The CNC programs can be written in either **Inch** or **Metric units**. The machine can be switched with a single code to accept either. When a piece is designed in metric, it is usually converted to inch units for machining, and metric tools are used only when no inch equivalent is available (for example when creating metric tapped holes).

THE CNC COORDINATE SYSTEM:

Machine coordinate system: The **machine coordinate** system of the CNC machine tool is commonly **defined by the manufacturer** and **cannot** be changed. The point of origin for this machine coordinate system, also called machine zero point M, cannot be shifted in its location (see figure A).

Work part coordinate system: The **work coordinate** system is usually **defined by the programmer** and **can** be changed. The location of the point of origin for the work part coordinate system, also called work part zero point W, can be specified as desired (see figure B).

M   Machine zero point   W   Work part zero point
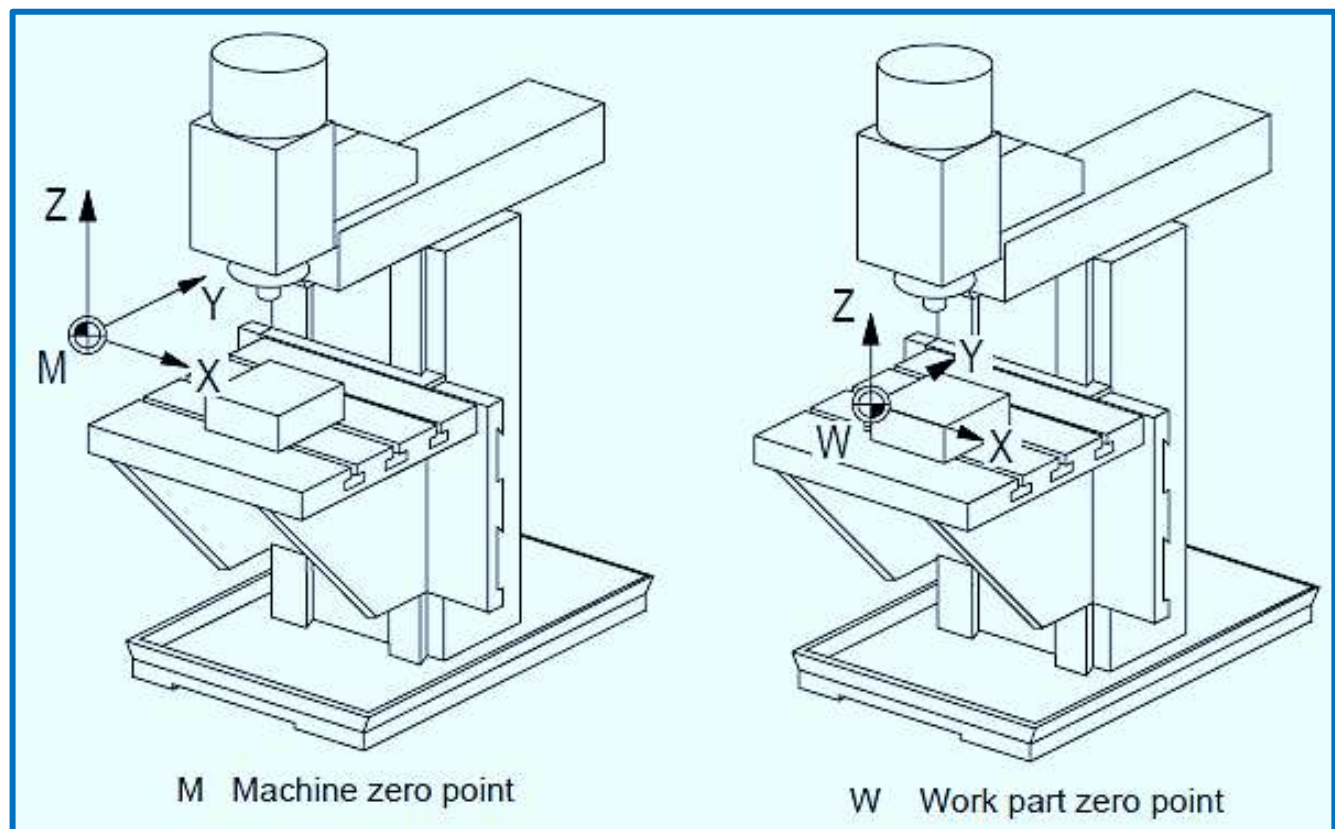
Figure A: Machine coordinate system   Figure B: Work coordinate system

In order for the part programmer to plan the sequence of position and movements of the cutting tool relative to the workpiece, it is necessary to establish a standard axis system, by which the relative position can be specified. Using an **NC drill press,** as **an example**, the drill spindle is fixed in vertical position and the table is moved and controlled relative to the spindle.

However to make things easier for the programmer we adopt the viewpoint that the workpiece is stationary while the drill is moved relative to it. Accordingly, the coordinate system of axes is established with respect to the machine table. Two axis are defined as shown below:
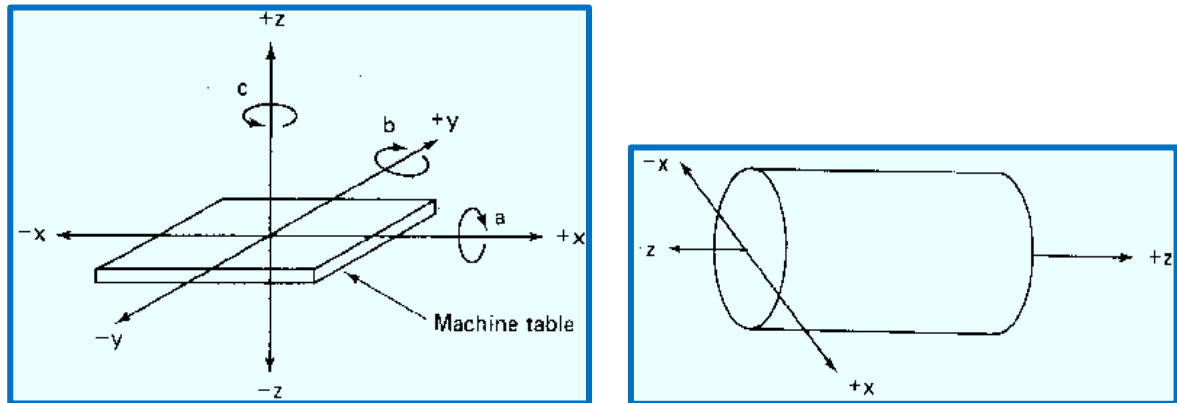

Two axes X and Y are defined in the plane of the table. The Z axis is defined in the plane perpendicular to the table and the movement in the z direction is controlled by the vertical motion of the spindle. The positive and negative directions of motion of the cutting tools are relative to the table along these axes. The NC **drill presses** are classified as either two axis or the three axis machines, depending on whether or not they have the capability to control the z-axis.

Absolute & Incremental Positioning:

Another option sometimes available to the part programmer is to use either an absolute system of **tool positioning** or an incremental system. **Absolute** positioning means that the tool locations are always defined in relation to **the zero point**.

If a hole is to drilled at the spot which is **8** inches **above the X-axis** and **6** inches to the **right of the y-axis** the coordinate location of the hole would be specified as **x = +6.000** and **y = +8.000**. By contrast incremental positioning means that the next tool locations must be defined with reference to the previous tool location. Each machine will have its own specific axis orientation as shown below:
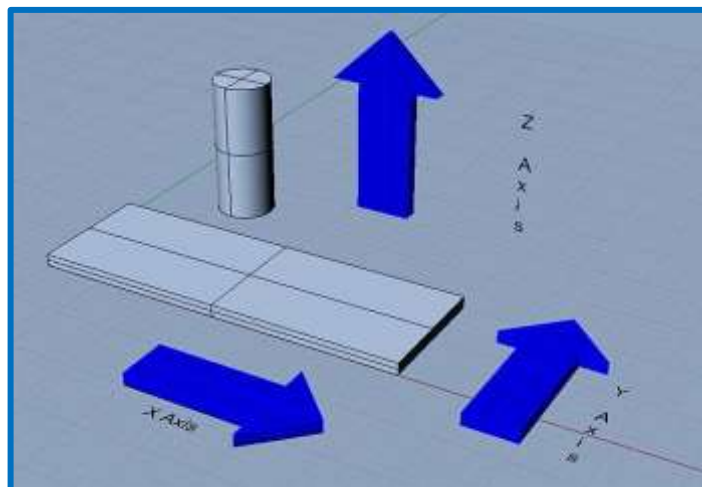
The cylinders in each drawing represent the spindle of the machine. Be sure to have a look at exactly how the axes are laid out on your machine. For example, horizontal mills are turned around considerably from this drawing. Lathes can get a lot more complicated than the simple 2-axis version shown below:
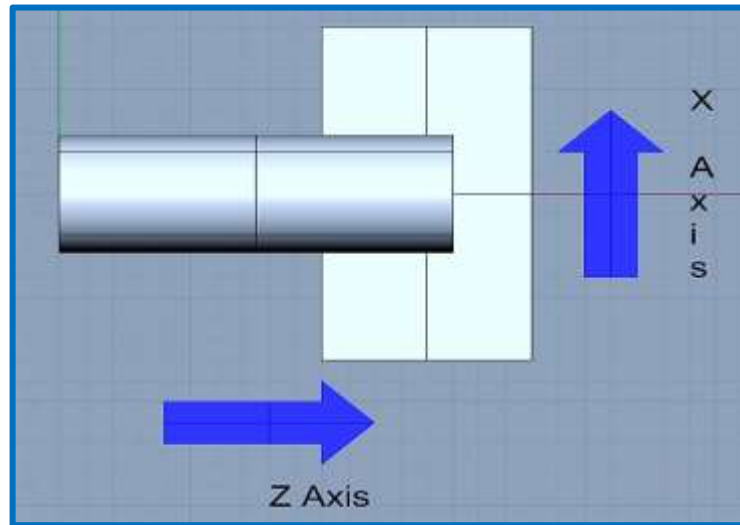
A typical 2-Axis lathe

The 4-Axis, 5-Axis, have much more complex configurations. Note that we have added two rotational axes to the basic mill diagram to provide an A-Axis and a B-Axis. In general, A, B, and C are rotational axes that rotate around axes formed by the X, Y, and Z respectively, as the example with a 5-axis setup, below:
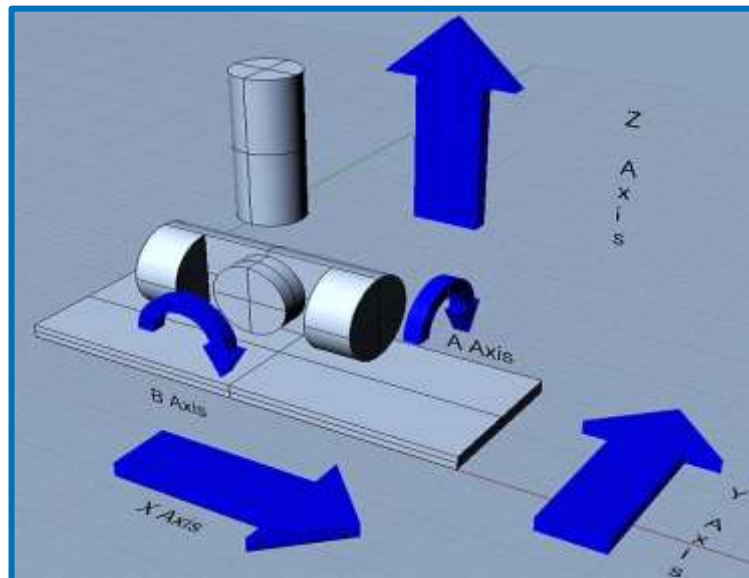
A 5-Axis Mill with trunnion table

Other features of the location system:

The purpose of the coordinate system is to provide a means of locating the tool in relation to the workpiece. Depending on the NC machine the part programmer may have several different options available for specifying this location. A Numerical Control Machine and similar machine tools use the axis sys-

tem similar to the drill system. However in addition to the three linear axes, these machines may possess the capacity to control one or more rotational axes. These axes are used to specify the angle about the x, y and z axes respectively.

To distinguish **positive from negative** angular motions the **right hand rules** can be used. Using the right hand with thumb pointing in the direction of positive linear axis direction the fingers of the hand are curled to point the positive rotational direction. For **turning operations two axes** are required to command the movement of the tool relative to the rotational workpiece as shown in the illustrations above.

Function	Address	Meaning of address
Program number	O(EIA)/(ISO)	Program number
Block sequence number	N	Sequence number
Preparatory function	G	Sercifies a motion mode (Linear, arc, etc)
Dimension word	X, Z	Command of moving position(absolute type) of each axis
	U, W	Instruction of moving distance and direction(incremental type)
	I, K	Ingredient of each axis and chamfering volume of circulat center
	R	Radius of circle, corner R, edge R
Feed function	F, E	Designation of feedrate and thread lead
Auxiliary function	M	Command of ON/OFF for operating parts of machine
Spindle speed function	S	Designation of speed of main spindle or rotation time of main spindle
Function (Tool)	T	Designation of tool number and tool compensation number
Dwell	P, U, X	Designation of dwell time
Dewignation of program number	P	Designation of calling number of auxiliary program
Designation of sequence No	P, Q	Callling of compound repeat cycle, end number
Number of repetitions	L	Repeat time of auxiliary program
Parameters	A, D, I, K	Parameter at fixed cycle

EXPRESSING COORDINATES IN G-CODE:

Take the axis letter and add the value. Spaces between the letter and its value are optional. For example, a position that is **1 inch** from **0 along X**, **2 inches along Y**, and **3 inches along Z** is written as:

X1 Y2 Z3 or X 1 Y 2 Z 3;

Sometimes, it is very convenient to refer to Relative instead of Absolute coordinates. Suppose a cutter is positioned at some point and an operator needs to cut a piece **1" square** with the corner aligned to that point. This is easily done with relative moves:

X1; Y1; X-1; Y-1;

In essence, it is moved 1" right in X, 1" up in Y, 1" left in X (negative dimension), and then 1" down in Y (negative dimension). Now the operator has a 1" square piece, whose lower left corner is the initial point. There are lots of cases where relative moves are switched back and forth.

Sometimes, relative coordinates are referred with special axis letters. For example, IJK may be a relative XYZ when defining arc centers, or UVW may be used alongside a XYZ referring to relative coordinates without the need of changing back and forth between the relative and absolute modes. In other words, **XYZ** is used always as **absolute** and **UVW** is always **relative**.

Zero Point Determination:

It is the programmer who determines the position of the zero point – using a logical method for deciding where it is placed. Consider the drawing shown below that shows an example of how program zero is determined in the X and Y axes.
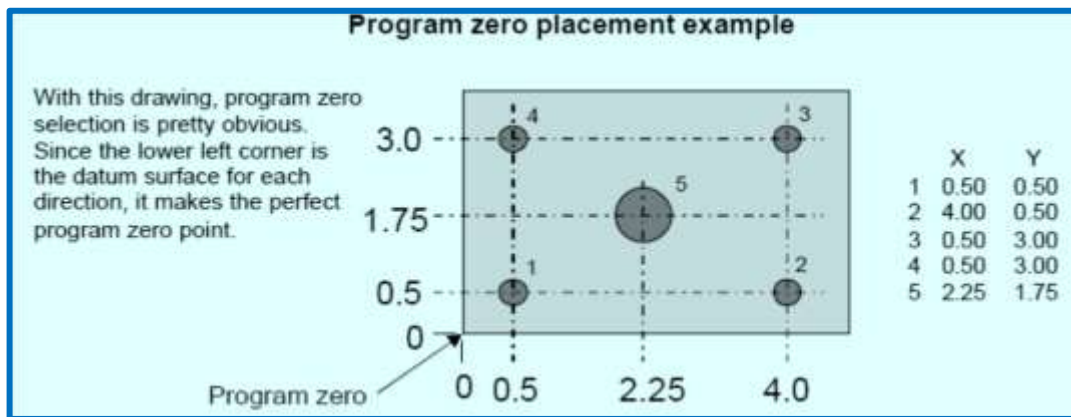
Relative and Absolute Coordinates:

To switch between Relative and Absolute Coordinates the usual way is to use G90 for Absolute Coordinates and G91 for Relative Coordinates, for example:

G90: (Switch to absolute coordinates);
G0 X0Y0Z0: (Move to program origin at 0, 0, 0);
G91: (Switch to relative coordinates);
G0 X1Y1: (Move one unit right in X and one unit right in Y);
G90: (Switch back to absolute coordinates).

Most controls startup in relative/incremental mode - G91. This is done because it is thought to be safer to start in incremental than start in absolute. The truth is, to make sure when doing a program setting to either G90 or G91, to make whatever the operator expects.

Some controllers allow both styles to be active at once using UVW. The UVW format is also convenient and a little shorter. As already mentioned, I and J are relative coordinates in a similar case to UVW, intended for arc center location. Then, the previous program could be written this way:

G90: (Switch to absolute coordinates)
G0 X0Y0Z0: (Move to program origin at 0, 0, 0)
**G0 U1V1**: (Move one unit right in X and one unit right in Y)

Circular Motion G02 and G03:

Drawing a complete circle involves not just a coordinated motion but the reversal of direction at each of the 4 quadrant points, where the points are corresponding to 0, 90, 180, and 270 degrees. Like the **linear motion** (initiated by G00 and G01), the command G02 establishes a mode for **clockwise** circular arcs and G03 establishes a mode for **counter-clockwise** circular arcs. Once G02 or G03 mode is established, the arcs can be defined in **G-Code** by identifying two endpoints and the center, which must be equidistant from each endpoint, or an alarm may occur.

The current **control point** or location when the block is begun establishes **one endpoint**. The **other** may be established by **XYZ** coordinates. The center is a bit more complex, commonly identified by using **I, J, or K** to establish the *relative offsets* from the starting point of the arc to the center. Here is a typical clockwise arc. The arc below starts at **X0Y2** and finishes at **X2Y0**. Its center is at **X0Y0** and could be specified in **G-code** like this: G02 (This command set up the clockwise arc mode);

Command: G02 X2Y0 I0J-2.0;
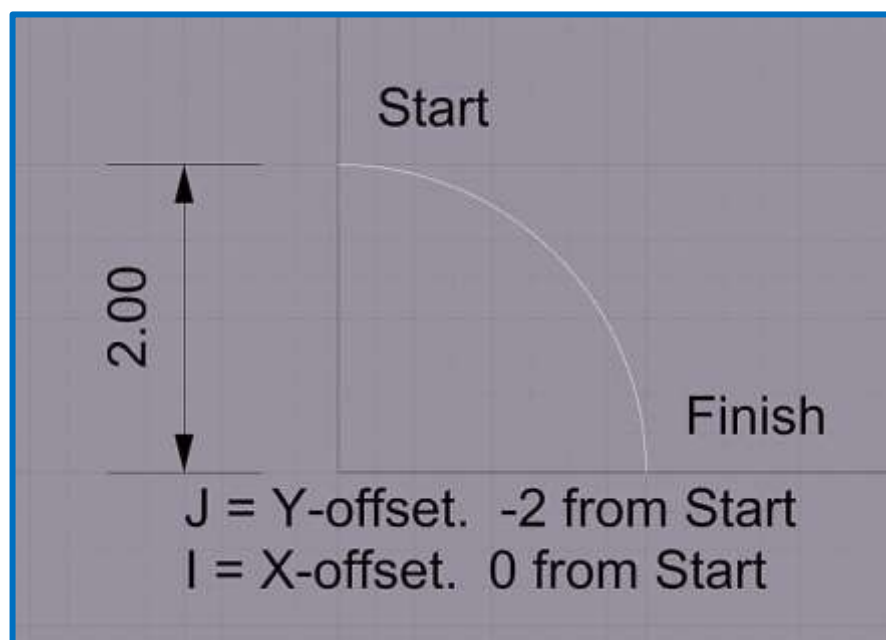
Defining an arc's center with IJK...

In above illustration, **I** and **J** specify relative coordinates from the start point to the center. In other words, if the **I** value is the starting point **X**, and the **J** value is the starting point **Y**, the **X** and **Y** for this center can be found. The center can be also defined, by specifying the radius of the circle. In this case, the above circle program has a radius of 2, so the **G-code** may be also written as:

G02;
X2Y0 R2;

Full Circles With XYZ:

Full circles are also defined when the start and endpoints are identical and the center is specified via IJK (R leads to an infinite number of circles). Here's a simple G-code program **that produces 3 circles**:

N45 G0 X-2. Y. 75
N46 G1 Z-. 5 F10.
N47 Y. 5 F30. S2000
N48 G2 J-1. 1
N49 G1 Y. 75
N50 Z. 2
N51 G0 X. 75 Y-3. 4
N52 G1 Z-. 5 F10.
N53 X. 5 F30.
N54 G2 I-1. 1
N55 X. 75
N56 Z. 2
N57 G0 X-4. 75 Y-3. 4
N58 G1 Z-. 5 F10.
N59 X-4. 5 F30.
N60 G2 I1. 1
N61 G1 X-4. 75
N62 Z. 2
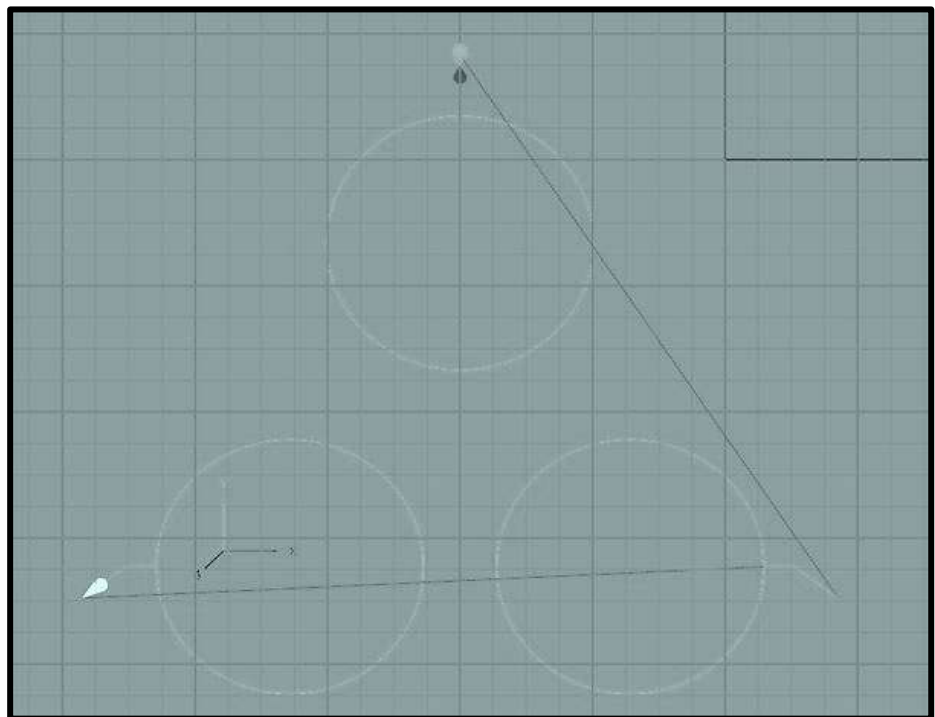
M-Code Utilization:

The **M-code** should not be programmed when the command paragraph contains **S-code** or **T-code**. It is favorable for M-code to program a paragraph independently.

M00: Use this command for main spindle stop, cutting oil, motor stop and tape reading stop.

M01: This function is the **same as M00**; it is effective when the optional stop switch of console is **ON**. This command shall be overridden if the optional stop switch is **OFF**.

M02: Indicates the end of a main program.

M30: This is the **same as M02** and it returns to the starting position of the program when the memory and the tape are running.

M76, M77: These codes are effective when thread process is programmed by G92, and are used for **ON** and **OFF** of thread beveling.

T Function:

T function is a tool selection code made of 4 digits, used for designation of tool numbers and tool compensation. Example: **T 0 2 0 1**. First two digits (0 2) calls the tool number and last two digits (01) the tool compensation value when it is memorized in the storage.

## G-CODE BASIC PROGRAMMING EXAMPLES:

There are multiple CNC machine controls such as Fanuc, Haas CNC, Siemens, Sinumerik, etc. However, for a beginner's level, it is better to understand and start learning a CNC machine controls which is most widely used and understood, and almost every other CNC control has some similarities. To completely understand the working of G-codes and tool paths will take some time, and the **scope of this course** is not specific in programming. Nevertheless, below can be seen some basic examples.

G00 (Positioning): Each axes moves as much as commanded data in rapid feedrate. Example:

G00 X150.0 Z100.0
X200.0 Z200.0
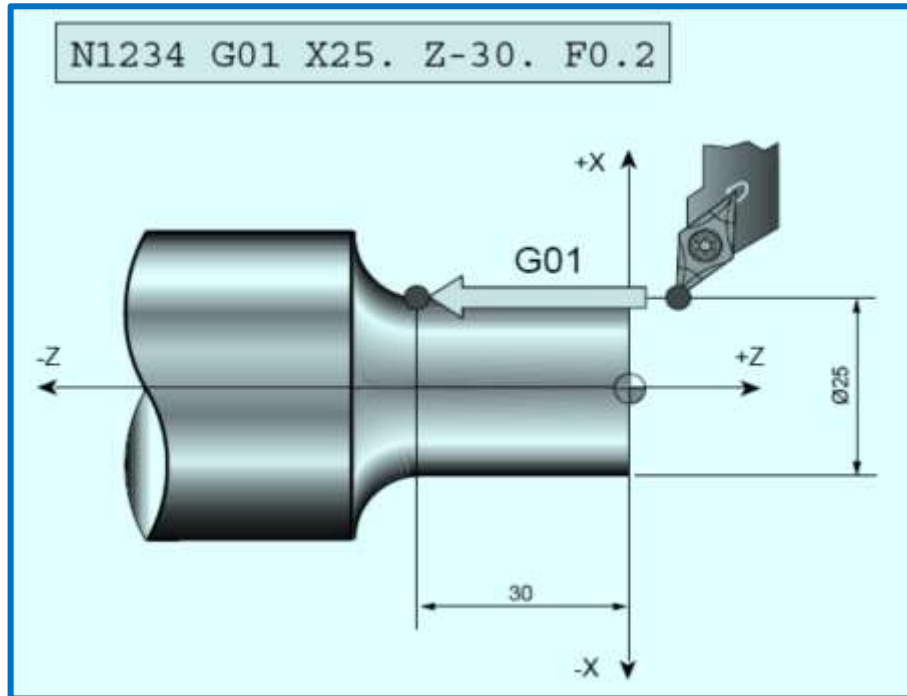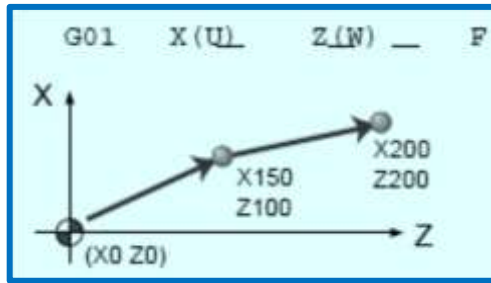
Or,

G00 U150.0 W100.0
U50.0 W100.0
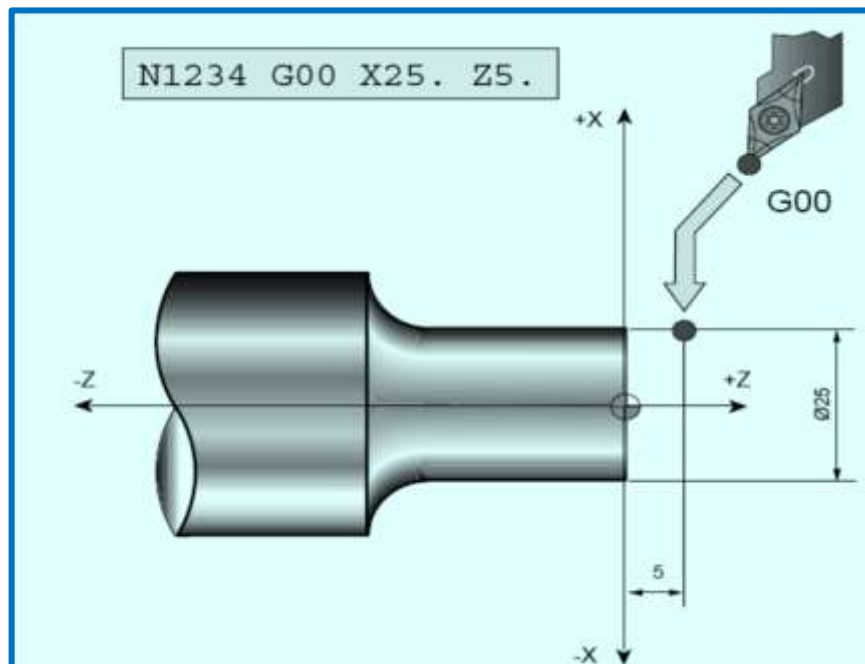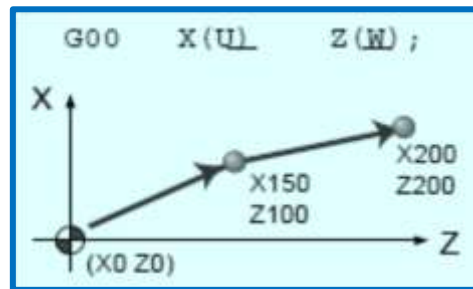
G01 (Linear interpolation): Each axes moves straightly as much as commanded data in commanded rate. Example:
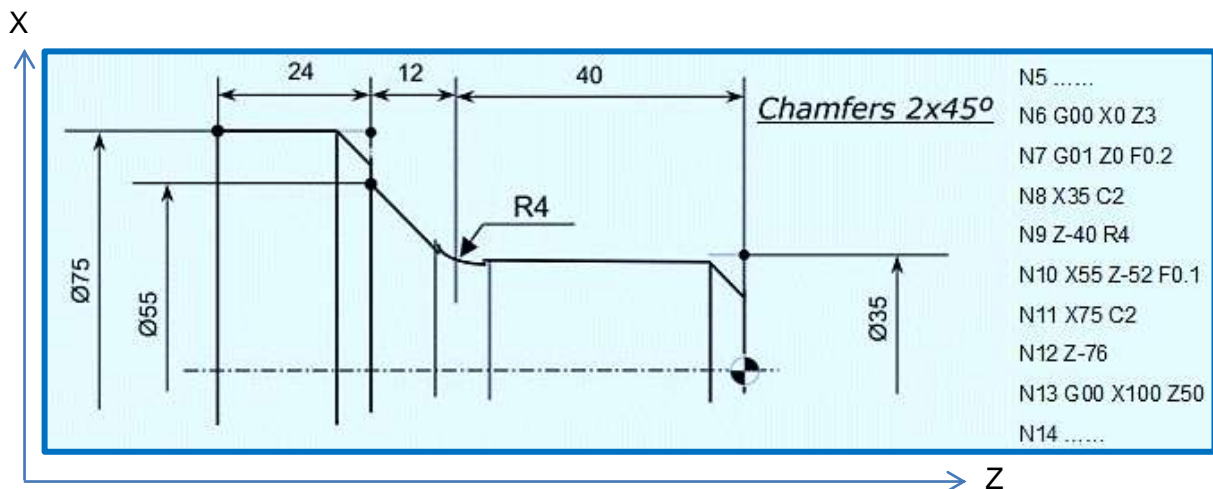
G01 X150.0 Z100.0 F0.2
X200.0 Z200.0

Or,

G01 U150.0 W100.0 F0.2
U50.0 W100.0

Chamfering "C" and Corner "R": In the following cases, **errors** may occur. (G01 Mode). When instruction one of **I, K, R** and **X** and **Z** are at the same time; when programming two of **I, K, R** in the same block; when programming **X** and **I** or **Z** and **K**; and when the moving distance is less than the next command are not right angled.


```
N5 ......
N6 G00 X0 Z3
N7 G01 Z0 F0.2
N8 X35 C2
N9 Z-40 R4
N10 X55 Z-52 F0.1
N11 X75 C2
N12 Z-76
N13 G00 X100 Z50
N14 ......
```

N1 G01 Z30.0 R6.0 F0.2:
N2 X100.0 K-3.0:
N3 Z0:
(N2 X100.0 C3.0:)

G02, G03 (Circular interpolation): G02 **(**Right hand coordinate – CW). G03 Left hand coordinate – CCW)
Each axis interpolates circularly to the commanded coordinate in instructed speed. Example:

G02 X(u) Z(w) R_ F_ :

G01 X30.0 Z60.0 F0.3 :
Z35.0 :
G02 X40.0 Z30.0 I5.0 :
(G02 U10.0 W-5.0 I5.0)
G01 X50.0 :
Z0 :

G03 X(u) Z(w) R_ F_ :

G01 X40.0 Z60.0 F0.3 :
G03 X50.0 Z55.0 K-5.0 :

Below, is a very simple CNC milling machine programming for a beginner level:

N05 G0 G90 X40 Y48 Z2 S500 M3
N10 G1 Z-12 F100
N15 X20 Y18 Z-10
N20 G0 Z100
N25 X-20 Y80
N30 M2

N05: The tool traverses in rapid traverse on P1, three axes concurrently, spindle speed = 500 rpm, clockwise;
N10: Infeed on Z-12, feed 100 mm/min;
N15: Tool travels on a straight line in space on P2;
N20: Retraction in rapid traverse;
N30: End of program.

The same way the CNC machine gives us the utmost flexibility, any program can be in metric system or imperial system. The codes **G20** and **G21** are used to change between **inches** and **metric** system. The example below uses the **G20-code**:

The example below uses the **G21-code**:

The **G20-code** turning cycle is used for simple turning however multiple passes are possible by specifying the X-axis location of additional passes:

G20 X… Z… F   or   G20 U… W… F…

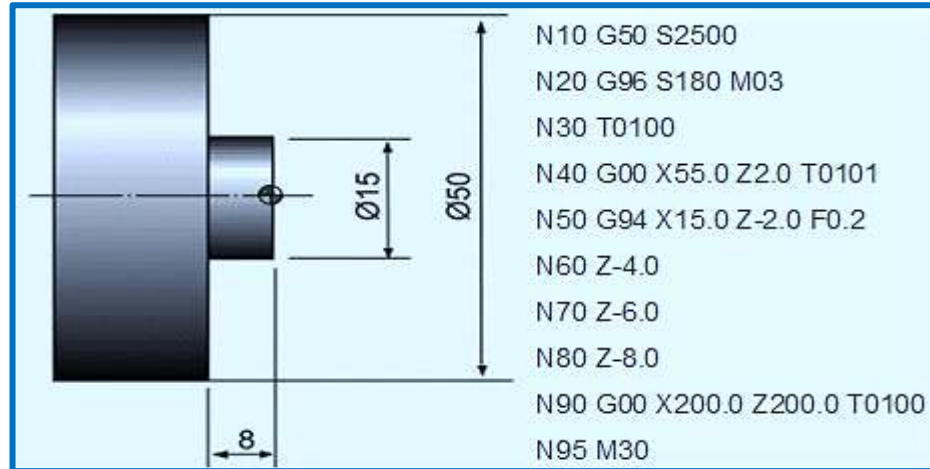**X** – Diameter to be cut (absolute);
Z – End point in z-axis (absolute);
F – Feed-rate;
U – Diameter to be cut (incremental);
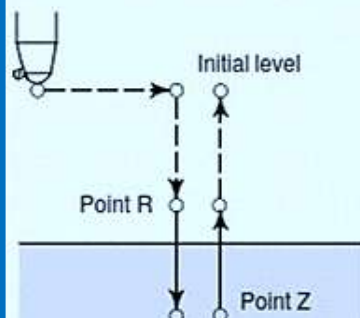**W** – End point in z-axis (incremental).
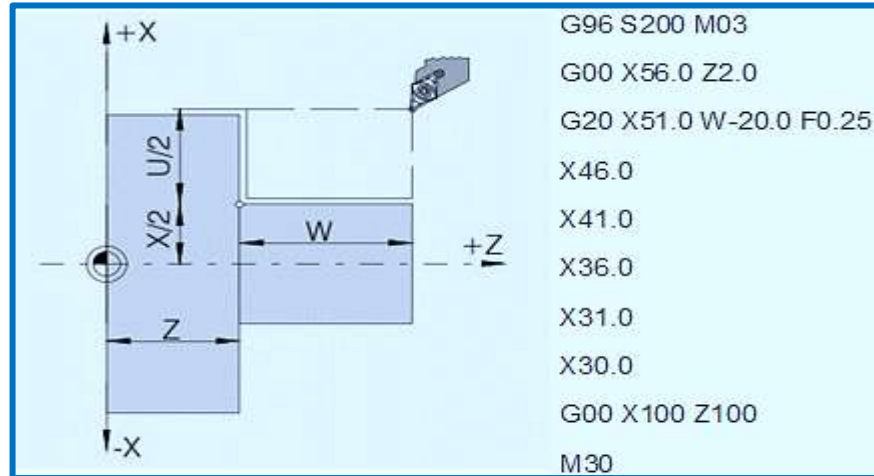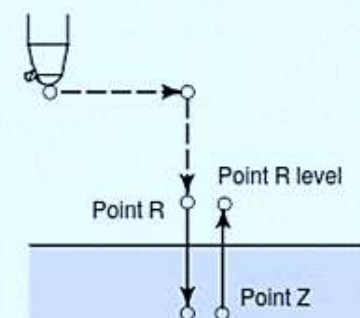
The **G94-code** is used for rough facing or simple facing (one-pass facing), however, multiple passes are possible by specifying the Z-axis location of additional passes.
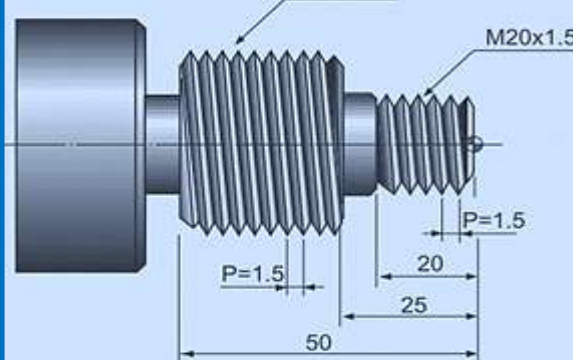
The **G85-code**, boring cycle operation can be performed:

1 – After positioning along X and Y axis, rapid traverse is performed to point R.
2 – Boring/Reaming is performed from point R to end-depth-point Z with specified feed F.
3 – After completing depth Z with feed F, Tool returns with the same feed F.
4 – Return plane is dependent on G98, G99 G-codes.

Note: If G98 is specified with G85 boring cycle the tool returns to Initial-level. If G99 is specified then tool will return to R level.
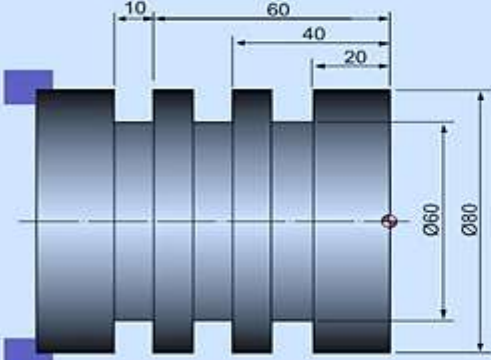
G85 (G98)	G85 (G99)	M3 S100

```
M3 S100
G90 G99 G85 X300. Y−250. Z−150. R−120. F120.
Y−550.
Y−750.
X1000.
Y−550.
G98 Y−750.
G80 G28 G91 X0 Y0 Z0
M5
```

The G76-code threading cycle can be used for internal threading on CNC lathe machines. This G76 threading example actually cuts external threads on two different diameters.


```
N10 T3
N20 G97 S800 M03
N30 G00 X30 Z5 T0303
N40 G76 P021060 QI00 R100
N50 G76 X18.2 Z-20 P900 Q200 FI.5
N60 G00 X50 Z-20
N70 G76 P021060 QI00 R100
N80 G76 X38.2 Z-52 P900 Q200 FI .5
N90 G00 X200 Z200
N100 M30
```

The G75-cod canned cycle grooving can be performed:

X = Groove Depth;
Z = Last groove position in z-axis;
P = Peck increment in x-axis;
Q = Stepping in z-axis;
R = Relief amount at end of the cut.


```
N10 G50 S500 T0100
N20 G97 S400 M03
N30 G00 X90.0 Z1.0 T0101
N40 X82.0 Z-60.0
N50 G75 R1.0
N60 G75 X60.0 Z-20.0 P3000 Q20000 F0.1
N70 G00 X90.0
N80 X200.0 Z200.0 T0100
N90 M30
```

USING MDI TO MOVE THE AXIS:

MDI means **"Manual Data Input"**. To use MDI to move the axis of a machine is necessary to know how to use the G-Codes. Example, the G00-code (some machines will accept G0 too) tells the machine to "rapid" motion tools. "Rapids" are when a machine moves absolutely as fast as it can to the place the command was given to move.

When a G01-code is performed, this command tells the CNC to switch from rapids to linear interpolation, but gives a fine ability to control the speed of the machine also using the "Feedrate" command, which is just an F. For example, the machine may be started with "G01 F20" in the MDI, which is, to limit the machine's speed to 20 mm per minute, or 20 inches per minute.

SPINDLE AND COOLANT USING M-CODES:

Using CNC machines is necessary to fire up the spindle and the coolant, using the "M-codes". The M-Codes are considered as "Miscellaneous". Fanuc refers to M-Codes as "Auxiliary" functions.

Then, to start with the spindle going, the code is "S" word (S for speed). Example, to go for 4000 rpm is necessary entering S4000 in the MDI. However, the spindle won't actually start to rotate, because this command must also show the "M03" for clockwise rotation. Nevertheless, on some machines, typically lathes, the M13 and M14 codes can also be found. The M13 starts the spindle running clockwise and M14 runs counterclockwise.

Coolant is switched on using the M08-code for flood, and the M07-code for mist or just air, depending on how the machine is set up. To stop the spindle the operator may type the M05-code, and to shut off all coolant with the M09-code.
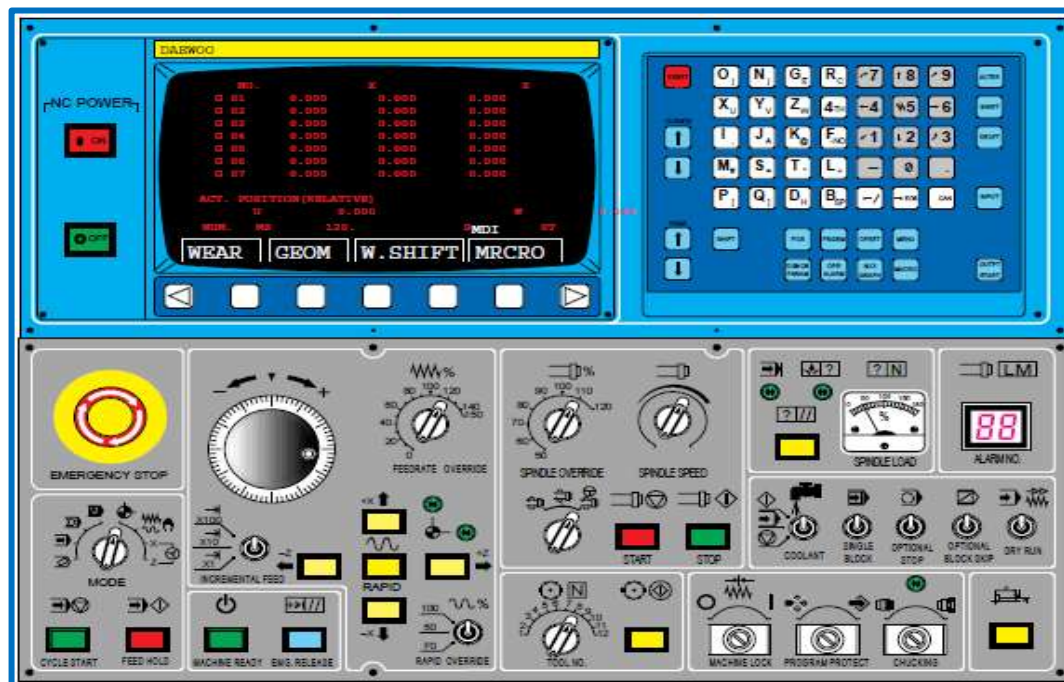
TOOLS & TOOL HOLDERS:

Lathes also apply some of the same tools that mills use, including spot drills, drills, and taps, most turning is done using carbide inserts. Inserts are gripped in holders, which in turn are bolted to the lathe turret. The figure below is a left-handed holder, as the tool cutting edge points to the left, viewing from the top.
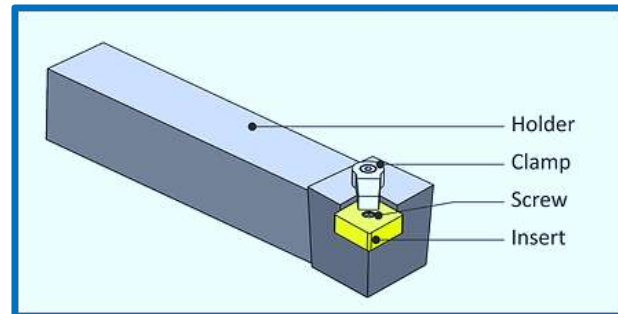
Typical Lathe Tool Holder

Carbide insert also called "widia", is a hard material used in machining tough materials such as carbon steel or stainless steel, as well as in situations where other tools would wear away, such as high-quantity production runs. Carbide inserts are made of highly engineered composite structures, coatings, and geometry features to achieve great accuracy and high material removal rates.
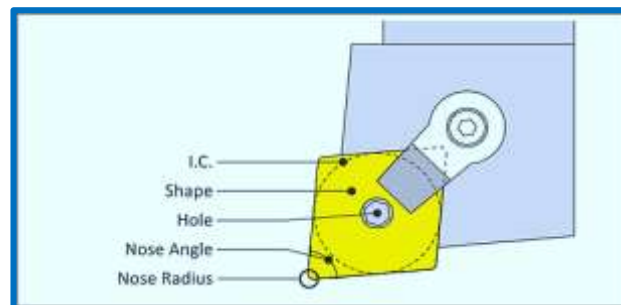
Insert Terms

Chip Breaker:

A chip breaker is a feature in the face of the insert that disrupts the flow of chips, in such way that the workpiece break into short segments, rather than forming a long, stringy chip.
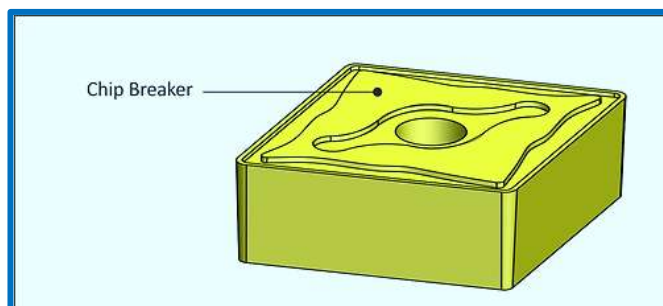
Chip Breaker

Relief Angle:

Most inserts have drafted faces on the walls. This is called Relief Angle. Relief prevents the walls of the insert from rubbing against the part.
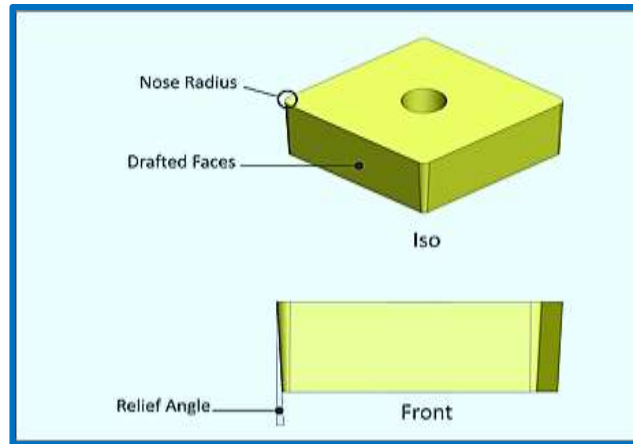
Relief Angle

Tool Cutting Angles:

The edge of the tool in the cut direction forms an angle with a line perpendicular to the cut direction. The angle formed by the trailing edge and parallel to the cut direction is called the End Cutting Angle. The purpose of these angles is to provide proper clearance between the tool and work piece. For example, the **80 degree insert shown** is rigid and has a cutting angle for facing and rough turning operations. However, complex contours may require a **55 or 30 degree** insert to provide end clearance for the tool and holder.
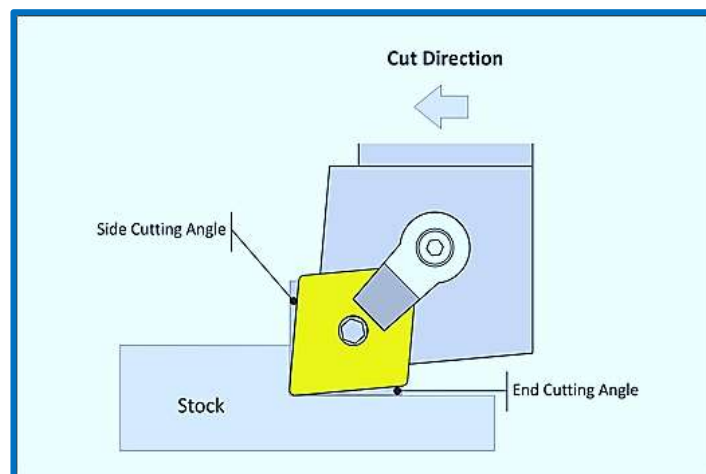
Cutting Angles

Rake Angle:

Rake angle is set by the tool holder. Rake angle helps control the direction of the chip and cutting pressure. Angle is measured from face of the insert to the Z-X plane of the machine.
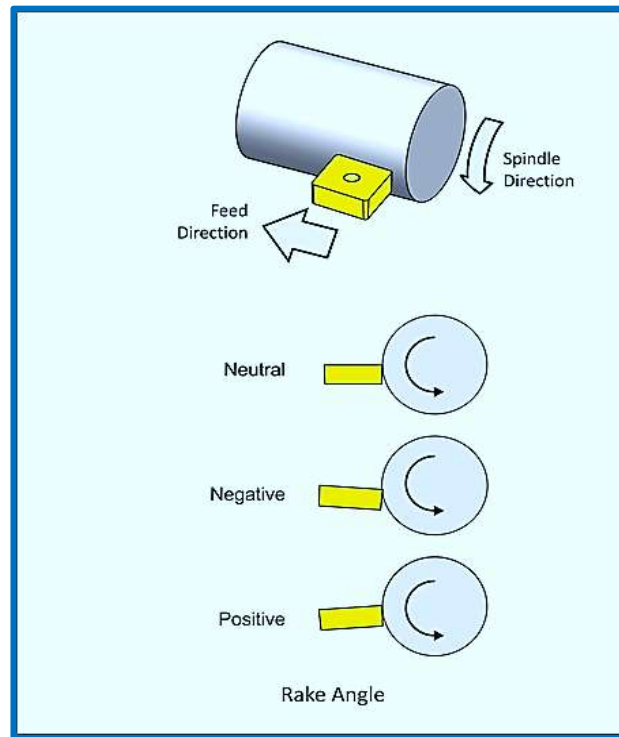
Rake Angle

CUTTING SPEEDS AND FEEDS:

Speeds and feeds are calculated based on the maximum work diameter for the tool. Use the same speed formulas as for mills.

$$Speed\left(\frac{rev}{min}\right) = \frac{SFM\left(\frac{ft}{min}\right) \times 3.82}{Dia(in)}$$

Speed is the machine spindle RPM (how fast the part turns).

SFM is the Surface Feet per Minute based on the insert manufacturers' recommendation for the material and cutting conditions.

3.82 is a constant that relates the surface feet per minute to work piece diameter.

Dia, is the diameter of the tool when it first engages the material.
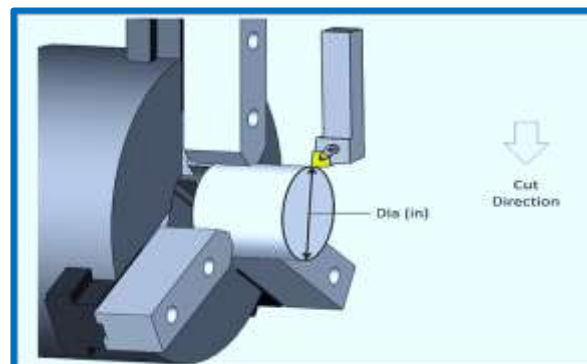
Facing Tool Engaging Material

Example:

Calculate the cutting speed for a facing operation given the following values:

Part Diameter	2.0 in
SFM	500 ft/min

Solution:

$$Speed\left(\frac{rev}{min}\right) = \frac{SFM\left(\frac{ft}{min}\right) \times 3.82}{Dia(in)}$$

$$Speed\left(\frac{rev}{min}\right) = \frac{500\frac{ft}{min} \times 3.82}{2.0in}$$

$$Speed\left(\frac{rev}{min}\right) = 955\frac{rev}{min}$$

Constant Surface Speed (CSS):

To maintain a constant rate of material removal as the cutting diameter decreases, most CNC machines automatically speed up the spindle, based on how far the tool moves towards center. This constantly variable spindle control is called **Constant Surface Speed** (CSS) mode. It is commanded on most machines using **G96-code** to activate, and **G97-code** to de-activate, as shown in the following example:
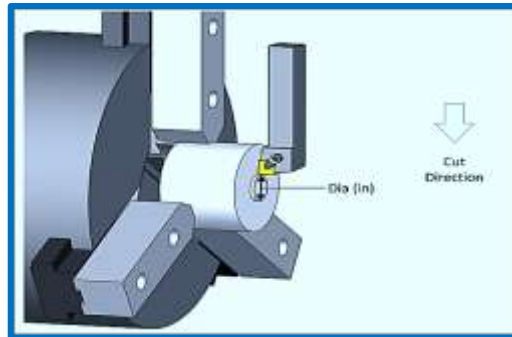
Decreasing Cut Diameter As Tool Nears Center

Lathe Program Using CSS:	
G50 S2000	G50 sets the maximum spindle speed for this tool.
G97 S1200 T101 M03	G97 cancels CSS so the spindle speed does not change as the turret rapids to position.
G00 X2.1 Z0.	Tool moves to cutting start.
M08	Coolant ON.
G96 S1200	CSS mode on, Spindle begins at 1200 RPM.
G01 X-.05	Face part. As tool moves from a diameter of 2.1 inches to X-.05, the spindle will increase speed up to the limit of S2000 RPM.

The **G50** command is important because it keeps the spindle from over-speeding. CSS does not apply where the tool does not change its position along X. For example, don't use CSS mode for drilling or tapping on part centerline. Feed is based on the speed and manufacturer' recommended Feed per Revolution (FPR) of the tool.

Example:

Calculate the cutting feed for tool with the following parameters.

Speed	955 rpm
IPR	0.015 in/rev

Solution:

$$Feed\left(\frac{in}{min}\right) = Speed\left(\frac{rev}{min}\right) \times IPR\left(\frac{in}{rev}\right)$$

$$Feed\left(\frac{in}{min}\right) = 955\frac{rev}{min} \times .015\frac{in}{rev}$$

$$Feed\left(\frac{in}{min}\right) = 14.3\frac{in}{min}$$

TOOL OFFSET AND DATUM:

The Tool Length Offset (TLO) value is the distance from the tip of the tool with the spindle in the home position and must be set for each tool used in the current job. They can be set using a height gage, reference tool, fixture location, or on the stock to be machined. Once the values are determined, they are stored in the controller for use during program operation.

Lathe tools have **two tool offsets: Z and X**, which is the distance from the imaginary tool tip at home position to the part datum. Many CNC machines now include a tool probe as standard or optional equipment. The part datum is usually set as the center-face of the finished part, as shown below.
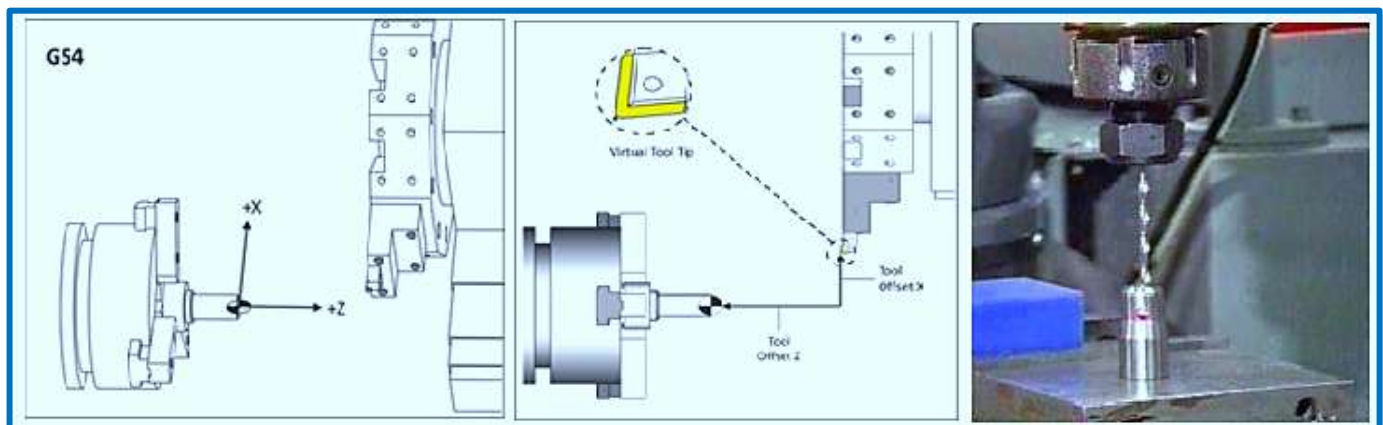
Part Datum and Tool Offsets

CAD/CAM BASIC PROGRAMMING:

CAD means Computer-Aided Design and CAM stands for Computer Aided Manufacturing. The CAD-CAM-system makes the most of the preparing work such as; directly takes the geometries, uses a cutting value table cutting speed (set automatically), the input and coordinates are automatically defined before machining. For automatic programming the sequence of operations in generating an NC-program is:

1. The CAD-CAM system is a complete machining software based on CAD technical drawings. Advantages lie in the possibility of making changes easily, copying and printing all generated CAD drawings several times.

2. CAM is also considered a numerical control (NC) programming tool, wherein two-dimensional (2-D) or three-dimensional (3-D) models of components generated in CAD software are used to generate the G-code to drive computer numerically controlled (CNC) machine tools. Simple designs such as bolt circles or basic contours do not necessitate importing a CAD file.

3. In the CAD-system only the drawing itself was generated, other information like measures or tolerances had to be set manually. CAM may also refer to the use of a computer to assist in all operations of a manufacturing plant, including planning, management, transportation and storage.

4. The CAD-CAM-system has a much wider range of tasks. The system should generates directly from the drawing NC-blocks for the production of the part. This means all points and contour elements have to be in the system with required tolerance.

5. First the work part must be geometrically defined. A representation of the finished part as well as the blank is necessary.

6. Thus, the individual machining operation is specified. The programming system assists the programmer in selecting the appropriate tool and automatically calculates the necessary cutting data.

7. Finally a NC-program for a specific CNC-machine tool with a specific CNC-control is generated and can then be transferred to the machine.

8. Contours can be used in the CAD-CAM-system for automatically generation in NC-programs, when the construction has been done appropriate. The system supplies the function "contour string" and a "contour tracing" function.

9. CAD-CAM software systems provide a wide range of capabilities in 3, 4 and 5 axis machining ranging from manual tool-path creation and editing to fully automated feature based machining.

10. The system includes powerful turn-milling methods to multi axial machining including the CNC Swiss-Type Lathe, with ability to program roughing, finishing, grooving, threading and many parts of machining operations.

CAD/CAM Drawing Example:

CAD-system milling by programming the NC-program for the following plate illustrated below. It is to be manufactured as individual workpart on a CNC milling machine:

Blank: 100 x 100 x 30 mm
Clamping: Height 70 mm
Material: AlMgSi1

WORK PLAN			
contour milling	T01	shell end mill	MW-040/032 HSS ISO 2586
pocket milling	T11	slot milling tool	MS-18.0/063L HSS ISO 1641
core hole drilling ⌀ 6.8	T12	drill	DR-06.80/069 HSS ISO 235
threading M8	T02	tap	TA-M08.0/1.25 HSS ISO 2857
predrilling ⌀ 19.8	T04	step drill	DS-19.8/11.5-118 HSS ISO 3439
reaming 20H7	T08	reamer	RE-D20.0/H7 HSS ISO 521

Starting NC mill programming: Before starting the mill program you should choose a zoom window, so that only necessary elements are visible;

Selecting a CNC machine: Starting the mill program some steps have to be done (only once) as a necessary basis, and cannot be changed during generation of the NC-program;

Defining the location of workpart zero point: Every NC-program requires a workpart zero point, to which measures relate. You choose this point before the programming, if necessary, it can be changed later by using zero point shifts.

CALCULATION FORMULAE:

♠ Process time(sec/ea) $= \dfrac{\pi \cdot D. L \times 60}{100V \times F} = \dfrac{\text{Cutting length} \times 60}{\text{Average of rotating time}} = \sec$

♠ Output(8Hrs/day) $= \dfrac{8\text{Hrs} \times 60 \times 60}{\text{Required time per unit}} = ea$

♠ Required day for process $= \dfrac{\text{Object time} \times \text{Quantity to be processed}}{\dfrac{8 \times 60}{60}} = Day$

♠ Surface roughress $= \dfrac{\text{Feed volume}^2}{8 \times NOSER} \times 1000 = R.t\ \mu m$

♠ Cutting volume $= cm^3/min$

    $V. F.D = LT$

    $\dfrac{ft \times W \times D}{1000} = ML$

V = Cutting speed
F = Feed volume(mm/rev)
D = Depth of cutting
ft = Feedrate(mm/min)
W= Width of cutting

♠ Cutting condition(Material : AL)

    * EXTREME – FINISHING   V = 870
                                        F = 0.05~0.15
                                        t = 0.025~2.0

        ∠ FINISHING   V = 720
                                        F = 0.1~0.3
                                        t = 0.5~2.0

        ∠ LIGHT   V = 600
          ROUGHING   F = 0.2~0.5
                                          t = 2.20~4.0

CUTTING DATA RECOMENDATION:

Material	Classification	Depth of cutting d(mm)	Cutting speed v (m/min)	Feedrate F (mm/rev.)	Material of tool
Carbon steel 60kg/mm (Tensile strength)	Stock removal	3 ~ 5	180 ~ 200	0.3 ~ 0.4	P 10 ~ 20
		2 ~ 3	200 ~ 250	0.3 ~ 0.4	P 10 ~ 20
	Finishing	0.2 ~ 0.5	250 ~ 280	0.1 ~ 0.2	P 01 ~ 10
	Thread		124 ~ 125		P 10 ~ 20
	Grooving		90 ~ 110	0.08 ~ 0.2	P 10 ~ 20
	Center drill		1000 ~ 1600 rpm	0.08 ~ 0.15	SKH 2
	Drill		~ 25	0.08 ~ 0.2	SKH9
Alloy steel 140kg/mm²	Stock removal	3 ~ 4	150 ~ 180	0.3 ~ 0.4	P10 ~ 20
	Finishing	0.2 ~ 0.5	200 ~ 250	0.1 ~ 0.2	P 10 ~ 20
	Grooving		70 ~ 100	0.08 ~ 0.2	P 10 ~ 20
Cast Iron HB 150	Stock removal	3 ~ 4	200 ~ 250	0.3 ~ 0.5	K 10 ~ 20
	Finishing	0.2 ~ 0.5	250 ~ 280	0.1 ~ 0.2	K 10 ~ 20
	Grooving		100 ~ 125	0.08 ~ 0.2	K 10 ~ 20
Aluminum	Stock removal	2 ~ 4	400 ~ 1000	0.3 ~ 0.5	K 10
	Finishing	0.2 ~ 0.5	700 ~ 1600	0.1 ~ 0.2	K 10
	Grooving		350 ~ 1000	0.1 ~ 0.2	K 10
Bronge Brass	Stock removal	3 ~ 5	150 ~ 300	0.2 ~ 0.4	K 10
	Finishing	0.2 ~ 0.5	200 ~ 500	0.1 ~ 0.2	K 10
	Grooving		150 ~ 200	0.1 ~ 0.2	K 10
Stainless steel	Stock removal	2 ~ 3	150 ~ 180	0.2 ~ 0.35	P 10 ~ 20
	Finishing	0.2 ~ 0.5	180 ~ 200	0.1 ~ 0.2	P 01 ~ 10
	Grooving		60 ~ 90	~ 0.15	P 10 ~ 20

GLOSSARY:

1-2-3 Block - A precision ground block that is 1x2x3 inches. Useful for helping set up the machine.

2D - A prismatic part of constant thickness.

2-½ D - A prismatic part with multiple thicknesses. All planes are parallel to each other.

3D - A part that is not completely prismatic, including molds and organically shaped parts.

Burr - A sharp edge or flashing.

Block - A line of code in a CNC program.

CAD - Computer Aided Design

CAM - Computer Aided Manufacturing.

CDC - Cutter Diameter Compensation.

CNC - Computer Numerical Control.

Chip Load - The amount of material removed with each pass of a tool cutting edge.

Datum - Point on the part from which all coordinates are referenced.

Drill Chart - A table that shows drill sizes, decimal equivalent, and the size drill to use for a specified tap.

Feed - The linear speed at which a tool advances through material.

First Article - The first part machined with a new program and setup.

G-M Code - The CNC machine tool language.

Gouge - An error that causes an overcut on the part.

High Speed Machining (HSM) - Toolpath strategies that minimize machining time and tool breaking.

Home - Location of machine after it is turned on and moved to its start position.

ID - Inside diameter or contour, like a pocket.

Insert - Carbide composite cutting tool used extensively for turning operations.

Job - A sequential list of one or more machining operations that share the same setup and fixture offset.

Machine Coordinate System - Coordinates in reference to the machine Home position.

Mill - A machine tool that removes material by spinning a tool and moving it in relation to the part.

OD - Outside diameter or contour.

Overcut - Describes a machined feature is too big because not enough material was removed during machining.

Post Processor - The part of a CAD/CAM system that actually creates and formats the CNC program for a specific machine and control.

Rapid - The fastest linear feed rate at which a machine tool can move.

Reference Point - Some point on the part that can be found reliably by mechanical means.

Rigid Tapping - The capability of a CNC machine to control a tap without the use of a special tapping attachment.

Scallop - Ridges left on the part, usually from a ball or bull nose end mill on a 3D sculpted surface.

Setup- A complete set of work holding to machine one side of a part.

Speed - How fast a tool turns in RPM.

Spindle - Part of CNC machine that grips and spins the cutting tool.

Stepdown - Distance the tool moves down in Z between machining passes.

Stepover - Distance the tool moves over in the XY plane between machining passes.

Stock Allowance - Amount of material left after a machining operation or to be removed by a finish pass.

Spring Pass - Two or more machining passes that follows the exact same path. Used to straighten walls and improve surface finish.

TLO - Tool Length Offset. Used to tell CNC machine the length of a tool.

Tool Changer - A device on a CNC machine that automatically takes tools from a carousel and places them in the machine spindle.

T-Slot - A slot in the machine table into which a special T-nut can be inserted. Used to fasten clamps or parts to the machine table.

Turret - Lathe assembly that attaches the lathe tool holder to the machine.

Undercut - Describes a machined feature is too small because too much material was removed during machining.

REFERENCES:

CNC Machine Tools and Control Systems; Yamazaki Mazak Corp., Japan;

G and M Programming for CNC Milling Machines; Denford CNC; Fanuc; England;

Teachware CNC Technology; MTS Mathematisch Technische Software; Entwicklung GmbH; Germany;

CNC Programming: Workbook; Michael J Peterson;

CNC Program; Daewoo Heavy Industries; Japan;

Hacettepe University; Ankara, Turkey;

Indian Institute of Technology Kharagpur;

Comando Numérico Computadorizado; Escola SENAI "Roberto Mange", Campinas, Brazil.

Links:

http://www.cnccookbook.com/CCCNCGCodeCourse.htm

http://www.hsmworks.com/docs/cncbook/en/#start

http://www.helmancnc.com/how-to-make-cnc-program/

http://www.cncci.com/resources/articles/CNC%20basics%201.htm

http://users.bergen.org/jdefalco/CNC/

http://en.wikipedia.org/wiki/Numerical_control

http://en.wikipedia.org/wiki/Machining

http://www.ntma.org/

https://www.nims-skills.org/web/nims/home

http://www.iso.org/iso/products/

http://www.efunda.com/processes/machining/mill_design.cfm

http://www.astm.org/Standards/