7.1 Introduction

The subjects of CNC and Motion Control are among the more technical areas of machine tools. To those familiar with CNC, this section may seem long and elementary, but to those new to CNC it may appear brief and complex. However, the primary goal is to provide a comprehensive educational overview of CNC and Motion Control. A secondary goal is to assist you in preparing for the certification exam. If you already have extensive CNC knowledge, then concentrate on the unfamiliar areas.

This section covers: 1) Computer Numerical Control Basics, 2) Machine Tools And Their Controls, 3) CNC Part Programming, 4) Microprocessor and Memory Technology, 5) Motion Control Technology, 6) PLC Technology and 7) Sensor Technology. Additional, independent study in each area is recommended for proper certification exam preparation.
A major resource in developing this chapter was the Modern Machine Shop, Guidebook to CNC Technology and Manufacturing Software. Our thanks to Gardner Publications for their permission to use and adapt these materials. (Modern Machine Shop Magazine, copyright 1992, Gardner Publications, Inc., Cincinnati, Ohio).

**STUDY GUIDE FOR THESE TOPICS**

**Computer Numerical Control Basics**
Background material for those with little or no exposure to CNC controls in their area of machine tool sales. It is elementary in nature.

**Machine Tools and Their Controls**
This is an excellent primer on coordinate systems, axis nomenclature, basic and advanced features, feedback devices and CNC operation.

**CNC Part Programming**
The sales engineer should be familiar with basic techniques and terms because they directly relate to the proper utilization and application of the CNC machine.

**Motion Control Technology**
A brief review of a machine’s motors and axis drive systems.

**PLC and Sensor Technology**
Programmable Logic Controllers handle a great deal of the machine’s control and logic functions. PLC use and design are covered in this section. Also, sensors are critical to the control and operation of today’s machines. Here, the common sensors are explained.
7.2 Glossary

3 + 2 5-axis machining

5-axis simultaneous machining

Absolute coordinates - The axes coordinates with respect to the coordinate system zero.

Absolute dimensioning - Dimensioning all part features from a single origin reference point.

Absolute encoder - An absolute encoder is an encoder that maintains position information when power is removed from the system. This eliminates the need to reference the machine on power up. The absolute encoder typically has a battery to provide power to the position detection circuits when the main power is removed.

Acceptance test - A test to determine the performance, capability, and conformity of software or hardware to design specifications.

Accuracy - Conformity of an indicated value to a value accepted as a standard. The accuracy of a control system is determined by the difference between the actual position taken by a machine slide and the ideal position. The observer, apparatus, environment, and method of determination influence the degree of accuracy obtained.

Adaptive control (AC) - A method incorporating automatic means to change the type and/or influence of control parameters to achieve near optimum processing performance. Adaptive control is used to optimize independent parameters such as speeds and feeds to be consistent with processing constraints such as quality of surface finish and cutter life. For instance, by altering the speed and feeds of cutting tools according to sensory feedback, variables such as heat and torque in machining can be regulated.

Address - A name, number, or label identifying a computer register, memory location, storage device or any other computer data source or destination.

Algorithm - A finite set of rules or procedures for accomplishing a given result by proceeding on a logical step-by-step basis. CNC and computer programs are developed by this method.

Ambient temperature - A temperature within a given volume, e.g., a room or a building.

American Standard Code for Information Interchange (ASCII) - Pronounced “askey.” The standard 7 or 8-bit code used for exchanging alphanumeric information between intelligent devices and associated equipment. It represents 128 data and control characters. The eighth bit, when used, is a parity bit for error testing.

Analog - Analog data imply continuity of information such as would be developed directly from a rheostat or speedometer. An analog device is generally dependent for
operation on a continuous signal that varies in magnitude with the variation in current or voltage as determined by the sensing unit (often contrasted with digital).

**Application Programmable Interface (API)**

**Architecture** - The physical and logical arrangement of a computer.

**Artificial intelligence (AI)** - The capability of a device to perform functions, such as reasoning, learning, self-improvement, and self-correction, that are normally associated with or require human intelligence.

**Automatic programming** - An area of artificial intelligence research that involves creating software that will allow the computer to perform certain stages of the generation or alteration of software programs.

**Automatic tool change (ATC)** - The process of changing machining tools such as drills, grinders, and saws in an automatic and unmanned fashion, the use of computer control to change tooling on a machine tool.

**Background processing** - The processing of lower-priority tasks whenever high-priority tasks or real-time operations are not occurring. Programming, editing, updating the display and housekeeping functions can be treated as background processing on some CNC and PLC units.

**Backlash** - The clearance, slack, or play between adjacent movable parts, such as in a leadscrew and a nut, is called backlash. Backlash may be the result of wear, clearance or both between mating parts.

**Backplane** - A printed circuit board located on the inside back panel or a rack or chassis housing designed for inserting various types of printed circuit boards, such as input/output modules. The backplane includes a data bus, power bus, and mating connectors for accepting various types of plug-in type modules.

**Bandwidth** - How much information you can send through a connection. Usually measured in bits-per-second or mega-bits-per-second. A full page of English text is about 16,000 bits. A fast modem can move about 57,000 bits in one second. Full-motion full-screen video would require roughly 10,000,000 bits-per-second, depending on compression. Ethernet communications are performed at 100-mega-bits-per-second.

**Baud** – Formally, it is defined as the reciprocal of the shortest pulse width in a data communication stream, but usually used to refer to the number of binary bits transmitted per second during a serial transmission. Baud rate is the number of transmission elements per second in a communications line.

**B-axis** - B is an angle defining rotary motion around the y-axis. The positive direction of the b-axis is in the direction of an advancing right-handed screw in the + y direction.

**Binary** - 1) A numbering system based on the number 2, and which uses only the digits 0 and 1 to express numeric values.
**Binary circuit** - A circuit operating as a switch, indicating either of two modes: on or off. Also known as direct circuit and digital circuit.

**Binary digit (BIT)** - 1) Abbreviation for binary digit. 2) A numeral, either 1 or 0, in the binary scale of notation.

**Block** – Part program information that composes one working unit of several working units of an operation.

**Bus** - A conductor, or group of conductors considered as a single entity, which transfers signals or power between elements.

**Byte** - A sequence of binary digits operated upon as a single unit. A byte may be comprised of 8, 16, 32, 64 or more binary digits (bits), depending upon the computer system.

**Canned cycle** - A preset sequence of events (hardware or software) initiated by a single command (e.g. G84 or Cycle/Tap for a CNC tapping cycle).

**Canned software** - Generalized programs, also called software packages, that are prewritten and debugged and are designed to perform one or more general functions.

**Cartesian-coordinate system** - A coordinate system with axes of dimensions that is intersecting and perpendicular (orthogonal). The origin is the intersection of the three coordinates - x, y, and z-axes - that locate a point in space and measure its distance from any of three intersecting coordinate planes. The coordinates are used to identify points for positioning the machine tool axis.

**C-axis** - C is an angle defining rotary motion around the z-axis. The positive direction of the c-axis is in the direction of an advancing right-handed screw in the + z direction.

**Central Processing Unit (CPU)** - The portion of a computer that is the basic memory or logic. It includes the circuits controlling the interpretation and execution of instructions.

**Chip** - Small piece of semiconductor material (typically silicon) upon which one or more integrated circuits are formed.

**Circular interpolation** - A mode of contouring control that uses the information contained in a single block to produce an arc of a circle.

**Clock rate** - The speed or frequency at which a controller transfers words or bits through internal logic sequences.

**Closed-loop system** - A system in which a reference signal from a controller is compared with a position signal generated by a monitoring unit on the machine tool (feedback) - the difference is used to adjust the machine tool to reduce the difference to zero.
**Computer Integrated Manufacturing System (CIM/CIMS)** - A multi-machine manufacturing complex linked by a material handling system and including features such as tool changers and load/unload stations. Under the control of a computer, various workpieces are introduced into the system, and then randomly and simultaneously transported to the CNC machine tools and other processing stations.

**Computer-Aided Design (CAD)** - The use of computers to aid in designing products.

**Computer-Aided Manufacturing (CAM)** - The use of computers to aid in the various phases of manufacturing. A common CAM system is the program used to convert CAD drawings into CNC programs. The CAM system may include methods to select tools, set the sequence of operations and test the program using graphic simulation. A post processor converts the CAM programming into a part program for a particular machine and CNC combination.

**Computer Numerical Control**

**Contouring control system** - A CNC system that generates a contour by controlling a machine or cutting tool in a path resulting from the coordinated, simultaneous motion of two or more axes.

**Constant Surface Speed**

**Cutter radius compensation** - A feature on CNC machines to compensate the programmed cutting tool path for the radius of the cutter used. The operator enters the nominal tool radius into the CNCs tool geometry radius offsets. Any slight variation or wear can be entered in the tool wear radius offset.

**Cutter path** - The cutting path described by the center of the cutting tool.

**Digital data** - Data supplied in discrete, discontinuous form as digits, quantized pulses, or other coding elements.

**Direct Numerical Control (DNC)** - The use of a computer to feed a part program one block at a time. DNC is typically used for part program that are too large to load into a CNC. This is often the case for the complex surfaces using in the die mold and aerospace industries. DNC is also called “drip feed”.

**Encoder** - An electro-mechanical transducer used to produce digital data (code) indicating angular or linear position. Also see absolute encoder.

**Ethernet** - A standardized method of networking computers.

**Feedback** - The signal or data sent back to a commanding unit from a controlled machine or process for use as input in subsequent operations.

**Feedback device** - An element of a control system that converts linear or rotary motion to an electrical signal for comparison to the input signal (e.g., encoder or linear scales).
Feedback resolution - The smallest increment of dimension distinguished by the feedback device and reproduced as an electrical output.

Feedrate - The rate of movement between a machine element and a workpiece in the direction of cutting. Feedrates are typically specified in feed per minute, feed per revolution (of the spindle) and sometimes inverse time.

Feedrate number - A coded number read from the tape that describes the feedrate function, usually denoted as the “F” word.

Feedrate override - A variable manual control function used to reduce or increase the programmed feedrate.

Fiber-optic communications - Communication system and components in which optical fibers are used to carry signals from point to point. Fiber optics may be used to carry signals between CNC components such as a CNC control and its drive system because the technology provides a high bandwidth and noise immunity.

File transfer - A function that moves an entire file and its associated attributes between systems. Also see file transfer protocol.

File transfer protocol – A standard network protocol used to transfer files from one host to another host over a TCP-based network, such as the internet. Can be used to transfer part programs and other manufacturing files from a host computer to a CNC or visa versa.

Firewall - A combination of hardware and software that separates a Network into two or more parts for security purposes.

Flexible Manufacturing System (FMS) - Computer-controlled configuration consisting of one or more modules and a materials handling system designed to machine more than one part number at low to medium volume levels.

Gigabyte

Hardware - The physical equipment of a system, as opposed to software; the mechanical, electrical, magnetic features of a system that are permanent components.

Hardwired logic - Logic control functions that are determined by the way that devices are connected. Hardwired interconnections are wired for a specific purpose and, contrasted to programmable logic or software solutions, are relatively unalterable.

Hexadecimal numbering system - A numbering system having a base of 16. There are sixteen hexadecimal symbols: 0, 1, 2, 3, 4, 5, 6, 7, 8, 9, A, B, C, D, E, and F.

HTML (Hyper Text Markup Language) - The coding language used to create Hypertext documents for use on the World Wide Web. The code is device independent and the local device can determine much of the look and feel based on user preference. The "hyper" in Hypertext comes from the fact that in HTML you can specify that a block of text, or an
image, be linked to another file on the Internet. HTML files are meant to be viewed using a "Web Browser".

**Human Machine Interface (HMI)**

**IEEE** - Institute of Electrical and Electronics Engineers.

**Incremental programming** - A coordinate system where positions or distances are specified relative to the current position.

**Input/output (I/O)** - The transfer of information into or out of a computer.

**Internet** - The vast collection of inter-connected networks that are connected using the *TCP/IP* protocols and that evolved from the *ARPANET* of the late 60's and early 70's. The Internet connects tens of thousands of independent networks into a vast global Internet and is the largest Wide Area Network (WAN) in the world.

**Intranet** - A private network inside a company or organization that uses the same kinds of software that you would find on the public Internet, but is only for internal use.

**Interpolation (Circular)** - The control of a cutting tool in a complete circle or arc by a machine control unit, which has been given basic statements such as coordinates of center point, radius, direction of travel, and coordinate locations of arc end points.

**Interpolation (Helical)** – A third axes (or forth) moves linearly as two other axes (X/Y) describe a circle. The result is the tool moves in a helical motion. Helical interpolation is often used as a more efficient method to plunge feed when machining pockets.

**Interpolation (Linear)** - A function of control enabling data points to be generated between specific coordinate positions to allow simultaneous movement of two or more axes of motion in a defined geometric pattern. For example, in CNC, curved sections can be approximated by a series of straight lines or parabolic segments.

**Interpolation (NURBS)** - A function of controlling axes to move the two along a complex curve defined by a Non-Uniform Rational B-Spline. NURBS curves are common in CAD systems but are typically converted to short linear line segments by the CAM systems. NURBS interpolation increases processing speed and machined surface quality. Many of the leading CNC control systems can convert the linear line segments created by CAM systems back into NURBS curves for the CNC to process.

**Kilobyte (KB)** - One thousand bytes.

**Ladder diagram** - A diagram that expresses the machine-tool-builder-programmed logic of the controller in relay-equivalent symbology.

**LAN (Local Area Network)** - A computer network limited to the immediate area, usually the same building or floor of a building.
Large Scale Integration (LSI) - Any integrated circuits having more than 100 interconnected individual devices, such as gates and transistors, manufactured into a single semiconductor chip.

Liquid crystal display (LCD) - A low power segment display that aligns material in liquid suspension under the influence of a voltage applied such that it reflects ambient light and displays alphanumeric characters.

Local area network (LAN) - A data communications network that spans a physically limited area (generally less than 5 miles), provides high-bandwidth communication over inexpensive media (generally coaxial cable or twisted pair), provides a switching capability and is usually owned by the user (i.e., not provided by a common carrier).

Logic diagram - A diagram that uses logic symbols to represent logic operation in a digital system.

MCU -Machine Control Unit (MCU). A generic reference to the control unit that converts a part program into machine motion. Most often referring to the Computer Numerical Control (CNC) unit.

Megabyte - One million bytes or one thousand kilobytes.

Memory - A device into which information can be stored, and later retrieved. There are a variety of memory types, but in general memory is either “volatile”, meaning it retains its contents only while power is applied to it, and “non-volatile,” meaning it retains its contents even after power is removed. Types of memory include EPROM, EEPROM, EAROM, PROM, ROM, RAM, DRAM, SDRAM, and DDR SDRAM. (Synonymous with storage).

Microprocessor - A basic element of a central processing unit manufactured on relatively few integrated-circuit chips. It has an instruction set that is expandable by means of microprogramming.

Modem (MOdulator, DEModulator) - A device that connects a computer to a phone line - a telephone for a computer. A modem allows a computer to talk to other computers through the phone system.

Nanosecond - One-billionth of a second.

Network - A collection of logic elements connected to perform a specific function.

Node - Any single computer connected to a network.

Open-loop system - A control system that is incapable of comparing output with input for control purposes; that is, no feedback is obtainable. Stepping motors servo systems are open loop.
Parity - A means of testing the accuracy of binary numbers used in transmitted, recorded, or received data. A self-checking code is used in which the total number of 1's or 0's is always even or odd.

Part program - A complete set of data and instructions written in source languages for computer processing or written in machine language for manual programming for the manufacturing of parts on a CNC machine.

Postprocessor - A computer program which converts generalized or centerline output, obtained from the general purpose processor and all other programming instructions for a machine and control, into a form that can be correctly interpreted by a particular machine/control system.

Programmable Logic Controller (PLC) - A solid-state or computerized industrial control system with a memory which can be set to operate in a specified manner to store instructions that implement functions such as I/O control logic, timing, counting, arithmetic, and data transmission.

Programmable Read-Only Memory (PROM) - A memory that is programmed only by special routines. Once programmed with permanent data, such as a mathematical formula, it becomes a Read-Only Memory (ROM).

Random Access Memory (RAM) - A type of memory that can be accessed (read from) independent of the time of the last access or the location of the most recently accessed data.

Read-Only Memory (ROM) - Digital storage device that can be read from but cannot be written into by the computer.

Repeatability - Closeness of agreement among repeated measurements of the same characteristics by the same method under identical conditions. Also known as reproducibility.

Resolution - The smallest increment of distance that can be read and acted upon by a CNC system. The resolution of the control system may be smaller than the smallest programmable increment of the machine tool. For example, the resolution of a control system may be 0.00001 inch, but the smallest programmable increment may be 0.0001 inch. The higher resolution controls system will make smoother surfaces when contouring.

Root-Mean-Square (RMS) - The alternating value, which corresponds to the direct current value that produces the same heating effect.

Serial operation - Type of information transfer within a controller whereby the bits are handled sequentially rather than simultaneously, as they are in parallel operation.

Software - All programs, routines, and documents associated with a computer.
Solenoid - An electromagnet with a movable core which, when energized, can move a small mechanical part a short distance.

TCP/IP (Transmission Control Protocol/Internet Protocol) - This is the suite of protocols that defines the Internet. Originally designed for the UNIX operating system, TCP/IP software is now included with every major kind of computer operating system. To be truly on the Internet, your computer must have TCP/IP software.

Terabyte - 1024 gigabytes.

Word - An ordered set of characters, which may be used to cause a specific action of a machine tool. (e.g., X, Y, and Z together with the numerical values are dimension words; G and F words describe prepatory and feedrate functions, respectively.) A word may consist of either two (2) or four (4) bytes of information.

WWW (World Wide Web) - Frequently used (incorrectly) when referring to "The Internet", WWW has two major meanings - First, loosely used: the whole constellation of resources that can be accessed using Gopher, FTP, HTTP, telnet, USENET, WAIS and other tools. Second, the universe of hypertext servers (HTTP servers), which are the servers that allow text, graphics, sound files, etc. to be mixed together.

XML - Extensible Markup Language (XML) is a markup language that defines a set of rules for encoding documents in a format that is both human-readable and machine-readable. Is increasingly used for configuration files on computers.
7.3 Computer Numerical Control Basics

(Portions adapted from and used with permission of Modern Machine Shop Magazine)

WHAT IS COMPUTER NUMERICAL CONTROL?

Computer numerical control is the operation of a machine by a series of coded instructions comprised of numbers, letters of the alphabet, and other symbols. These are translated into pulses of electrical current or other output signals that activate motors and other devices to run the machine.

On machine tools, CNC commands may range from positive positioning of the spindle in relation to the workpiece, to auxiliary functions such as selecting a tool station on a turret or controlling the speed and direction of spindle rotation. The commands, gathered together and logically organized so that they will direct a machine tool in a specific task (usually the complete machining of a workpiece), comprise a CNC part program. Such a program, which contains the intelligence of the programmer, may be stored and utilized at some later date to obtain exactly the same results as when first utilized.

In traditional manual machine tool operation, a machinist or operator studies a workpiece drawing and then directs the machining sequence. His decisions and actions are based largely on intuition and skills. These skills are learned through long training and practice and, at their highest level. The results, however, are never fully consistent. They change from operator to operator or from day to day and even hour to hour because of fatigue, boredom, and even daily attitude.

Numerical control of a machine tool is depicted in Figure 2. With CNC, the control of a machine has moved away from operator skill and intuition to an entirely conceptual documentation of all machine motions and functions needed to machine a workpiece. Thus, a part program document is a clearly defined process sheet for machining a workpiece. That process documentation is then converted into coded instructions in a form that can be acted upon by a machine control unit, which converts the coded instructions into electric signals that activate machine tool functions and cutting tool movements. It is thus very important to realize that CNC is not a machining method. It is a concept of machine control.
HOW THE CNC CONTROLS THE MACHINE

The operation of the CNC in controlling the machine can be easily understood when simplified to its most basic function; moving a machine slide to a defined position.

**Fig. 1--**A conventional machine’s slide is moved by an operator turning the handwheel. Accurate positioning is accomplished by the operator counting the number of revolutions made on the handwheel plus the graduations on the dial. (Courtesy Gardner Publications)

**Fig. 2--**A CNC machine takes the commanded position from a part program--any difference between the commanded position and the feedback signal reading will generate a signal from the CNC to run the drive motor in the proper direction to cancel any errors. (Courtesy Gardner Publications)

Consider first the typical manual machine with a handwheel connected to a leadscrew, Figure 1, a concept that has been with us for more than a century. Each revolution of the screw advances the table by the amount equal to the lead of the screw, say 0.100 inch. If
the dial on the handwheel is divided into 100 equal parts, an operator can accurately position the table to an accuracy of 0.001 inch by adding together the number of turns of the handwheel and the number of graduations on the dial. In this example, the operator is serving a visual feedback function. In order to perform the same function without the operator, a considerable amount of electronics in the CNC are required, and a drive motor and a feedback device must be added to the machine as shown in Figure 2.

The CNC will compare the input dimension with the current slide position. If any difference exists, a signal will be sent to the drive motor to rotate it at a controlled speed in the direction required to cancel the difference. The feedback device provides information on the progress of the move. As the two signals come closer together, the drive motor signal is steadily reduced until the two signals match and no further motor drive signal is generated. When this null condition exists, all slide motion is stopped.

If the cutting forces force the slide out of position, the feedback device immediately sends a signal to the CNC, which recognizes an out-of-null condition and sends a signal to the drive motor to overcome the cutting forces and move the slide back into position. This explanation ignores many aspects of machine control such as feed rate control, interpolation, coordination of axes, and so on--all of which help make CNC work.

Standard Computer Numerical Controls have the capability of fully controlling a number of axes simultaneously: two, three, four, five, or more. A particular control may be able to control the primary or additional axes through programmable controllers.

More information and pictures are available in Section 7.7 Motion Control Technology.

**MACHINE AXES OF MOTION**

Every machine tool is characterized by some combination of sliding linear motion and rotating motion. The linear motions are known as longitudinal, transverse, and vertical. The rectangular Cartesian system of coordinates fills the need of CNC. With all the machine tools and equipment that are numerically controlled, the diversity and complexity of configurations necessitate standardization of all the axes designations.
STANDARDS FOR AXES DESIGNATION

Both the Electronic Industries Association and the Aerospace Industries of America have addressed this need with standards. The EIA 267-B and the NAS 938 standards are virtually the same in meaning and both standards follow the right-hand rule of coordinates, as illustrated in Figure 3.

The rule governs how the primary axes of a machine should be designated. As shown at upper left in Figure 3, hold the right-hand with thumb (X), forefinger (Y), and middle finger (Z) in three directions perpendicular to each other. This provides a fixed relationship of the three primary axes: X, Y and Z. The axis of the main spindle travel is always Z, that of the longest travel perpendicular to Z is X. By rotating the hand, and looking along the spindle or Z axis, it can be seen that on a vertical spindle machining center, for example, the Y axis would be the transverse motion or cross-travel. In the case of a horizontal machining spindle, the Y-axis would be the vertical travel.

Further, it is the purpose of the right-hand rule of coordinates to determine the positive (+) direction of each axis. The positive direction is from the base of the finger to the tip. Consider the base of all three fingers to be the same point, O, in the palm of your hand. So the positive direction of each axis on a machine is outward from the point at which the three axes (extended) would intersect. As shown at lower left in Figure 3 (and at upper right), the positive rotary motion is in the right or clockwise direction looking along the finger (or axis) in its positive direction.

The EIA standard 267-B lists 14 different designations of axes of motion. No one machine will have all 14. A quick glance at Figure 4 will show the application of axes designations to various machines, which combine motions in different ways.
The three primary linear motions are, as already mentioned, X, Y and Z. The primary Z motion is always coincident with or parallel to the axial centerline of the main spindle, and the Z motion is that movement which advances or retracts the main spindle of the machine. The primary X motion is the longest travel perpendicular to Z, and is normally horizontal. The primary Y motion is also horizontal on a vertical-spindle machine and is the shortest travel perpendicular to Z. The Y motion is vertical on a horizontal spindle machine. Note again Figure 4.

Characters A, B and C designate angular or rotary motions around the X, Y and Z axes, respectively, or axes parallel to them. Thus, since B designates a rotary motion around Y, a rotary indexing table on a horizontal machining center is referred to as having a rotary B (or Beta) motion, which is around a centerline parallel to the vertical Y travel of the machine's spindle carrier.

Characters U, V and W are secondary linear motions parallel to X, Y and Z, respectively, while characters P, Q and R are third or tertiary linear motions parallel to X, Y and Z respectively. Secondary motions are those next nearest to the respective parallel principal axis; tertiary motions are those farthest away. If characters U, V, W, P, Q and R are not used as just stated, they may be used to designate special axes not parallel to X, Y and Z.

Characters D and E are normally used to designate secondary angular or rotary motion either parallel to A, B or C or around special axes. D may also be used to indicate a tertiary feed function, and E to indicate a secondary feed function.

In assigning designations, it is assumed that the cutting tool moves in relation to the workpiece. However, on some machine tools it is the table that moves relative to the spindle (and cutting tool). This means the workpiece moves in a direction opposite to that in which the tool would move. There are certain axes designations, which neither standard has yet incorporated. For example, there are motions or axes that are parallel to and "slaved" to a principal motion or axis. If, for instance, there are three parallel spindles mounted on the gantry of a large profile milling machine, and all spindles are to work in unison, it is common practice to designate one as the principal spindle or Z1 and to designate the other "slave" spindles as Z2 and Z3.
Fig. 4--These diagrams indicating the axes of machine motion are taken from EIA Standard RS-267-B with the exception of the vertical spindle machining center and the horizontal spindle machining center. (Courtesy Gardner Publications)
7.4 Machine Tools
And Their Controls

(Portions adapted from and used with permission of Modern Machine Shop Magazine)

COMPUTER NUMERICAL CONTROL

Whatever the CNC technology, they share many common features.

Nearly all CNCs have some means for entering data manually right at the control—relatively simple machine instructions, a full part program, or updating or editing a part program. Program edit is a control feature that permits changes to be made in the part program. Programs can be entered or edited while another program is running using a “background editor”.

CNCs may have a full alphanumeric QUERTY keyboard while others have a limited keyboard arrangement optimized for a particular machine type.

The keyboard is also to set tool and workpiece coordinate system offsets and select the current part program.

Features in the CNC can make the programmer's job easier and quicker, and reduces the length of the part program. For example: (1) canned cycles and subroutines permanently stored in CNC memory, and (2) conversational programming languages.

Canned cycles provide common subroutines such as hole-making cycles, bolt-hole circles, and pocketing sequences. Such capabilities are a definite aid when manual programming is utilized.

Conversational programming is an application on the CNC that simplifies at the machine programming. Typically, the operator/programmer is prompted to fill in the blanks in response to a selected operation such as facing, turning, pocket milling. A new part program can often be created while another program is running.

Developments in computer graphics are having a major impact on the very nature of the CNC. Some CNCs display a two or three-dimensional graphic view of the part and tooling as it is being machined.

The physical hardware components for a CNC are virtually the same whether they are controlling a lathe, a milling machine or a mill-turn machine. In fact, the mill-turn or hybrid machine is blurring the lines between milling and turning. It is not the hardware elements, but rather the CNC manufacturer’s software executive program that makes a
control unit "think" like a milling machine or a lathe. The user does not typically tamper with the executive program.

A portion of the computer memory is usually reserved for the machine tool builder to use for interface logic, control of tool-changer mechanism, or other miscellaneous functions that help adapt the control to a particular machine. It is in essence, a second-tier executive program.

High-speed Ethernet communication is increasingly being adopted for part program transfers and data collection. Basic Ethernet connectivity and file transfer protocols are available on most current technology CNCs.

Specialized hardware (custom microprocessors and circuitry) may be used to optimize the systems performance for intense functions such as high-speed interpolation. In particular, axis control is often performed using specialized microprocessors called digital signal processors (DSPs), which are not part of the typical PC architecture.

CNCs may have proprietary tools that allow the machine tool builder (and end-user) to create custom screens and applications and feature advanced communication capabilities.

**PC-Integration**

Today’s CNC can be seamlessly integrated with PC-technology. While many designs are possible, three common variations are:

1. A CNC system that links to an external PC computer.
2. A CNC system with a built-in PC computer/processor.
3. A PC computer with a CNC system card installed.

Incorporating a PC with a CNC provides a great deal of design flexibility and power. In some cases, these systems have their own proprietary hardwire and software designs but the inclusion of the PC allows for extensive customization by the machine tool builder to provide unique solutions to their customers. In some cases the customization can be extended to third-party developers and the end-user.

The open architecture CNC typically accepts third-party software programs. Most systems, but not all, operate under an operating system such as Windows®.

A few challenges PC integrated with CNC are: (1) the unreliability of hard disk technology in the vibration and dusty environment of the shop floor. (2) the lengthy time to startup and shut down the control with a Windows operating system. (3) the vulnerability to a computer virus. The first two concerns are now being addressed with solid state disk drives and lighter weight embedded windows architectures.
DIGITAL INPUT/OUTPUT INTERFACE (I/O)

In addition to controlling the tool cutter path, CNC systems must also control other devices on the machine tool such as automatic tool changers, turret tool changers, coolant systems, lube systems, chip conveyors and door interlocks through switches, sensors, solenoids, valves and motor starters. In complex applications there can be hundreds of input/output connections.

A separate I/O interface is typically inserted between the external device and CNC control. It converts low-level voltage signals from the CNC to higher-voltage, higher power signals needed by external devices such as relay and solenoids. It also converts higher-voltage signals from switches to the logic level signals required by the CNC. In conventional I/O systems, all inputs and outputs are routed to the electrical cabinet.

Some high-volume machine tool builders may build custom printed circuit boards with customized I/O and interconnections to reduce the manufacturing time of interconnecting discrete devices, reduce cost and increase reliability.

For large machines and for some machine auxiliary devices that require I/O points a long way from the main electrical cabinet, distributed I/O blocks are available. An I/O block can be mounted in a remote location, for example, on a tool changer or material handling unit. The I/O block then communicates with the main CNC/PLC via a distributed I/O
protocol such as Ethernet/IP, Profibus®, DeviceNet™, ControlNet™, Interbus or other proprietary protocols.

In some cases, a separate PLC (Programmable Logic Controller) is used for the I/O interface, however, most CNC designs incorporate PLC functionality within the CNC system as a single package.

Refer to Section 7.8 “PLC & Sensor Technology”.

**SERVO CONTROL**

The primary function of a CNC system is to control the machine tool’s axis servomotors and spindle drives. The CNC sends signals to the servo and spindle drive amplifiers using a proprietary or industry standard protocol. Some systems use fiber optic cables that provide a way to handle large amounts of high-speed data (high bandwidth) and minimize the effects of the electrical noise found in some manufacturing facilities. Additional information on servo control can be found in section 7.7 Motion Control Technology.

**CNC MACHINE APPLICATIONS**

CNC positions the work or the tool on virtually every type of machine tool; turret drills, boring machines, milling machines, machining centers and turning centers, lathes, grinders, routers and water jet. In sheet metal shops, CNC is found on turret punch presses, brakes, shears and flame cutters. CNC is used on electrical discharge machines (EDM), riveting machines, welders, and inspection machines. In electronic plants, it is used on printed circuit board drilling machines and on wire wrapping machines.

CNC technology has successful applications in virtually every class of machining, including turning, milling, grinding, pressworking and numerous other manufacturing processes and types of equipment. These include resistance welders, punch presses, shears, flame cutters, riveting machines, fabric cutters and many others.

For brevity, continuity and speedy comprehension, this text in presenting the fundamentals of CNC leans heavily on the specifics of CNC applied to the milling processes, where it started, and to the multi-function machining center. But, every process or machine type imposes its own unique requirements, as indicated by the following examples:

1. Turning operations have traditionally been two-axis work although many CNC turning machines offer additional axes to accommodate secondary operations. Today, many turning machines have three, four, five, six, seven or more axis of CNC controlled movement. These multi-operation machines can perform ID, OD, drilling, ID and OD, threading, milling, ID tapping along with regular
turning. These CNC lathes are truly machining centers for small and medium sized components. Dual turret and dual spindle lathes are also common.

2. CNC control of two-axis (X and Y) table movement is common on TW/ EDM (traveling-wire/electrical discharge machines). Additionally, U and V positioning of the subtable that carries the upper wire-guide may be required. This permits taper relief for cutting stamping and extrusion dies, or the cutting of different top and bottom profiles needed for twisted airfoil shapes.

3. Machines for laying composite tape (for example, graphite impregnated epoxy) on sculptured molds for complex aerospace parts require up to nine axes of control. In addition to the X, Y, Z basic moves on the machine, angular and rotating moves of the tape head and angular and traverse moves of each of the dual cutter blades must be provided for by the CNC programmer.

4. CNC is also being applied effectively to all four major gear-production methods: shaping, hobbing, milling and broaching. Gear-tooth profiles can be especially complex to machine. Modern machines must be able to process a broad range of gear geometries, sizes and materials. As many as eight servo-driven axes must be independently controlled, with multiple-axis interpolation.

5. More grinding machines are also being equipped with CNC. With CNC, it is possible to program the machine to do the dressing cycles to automatically compensate for the amount of wheel diameter lost during the dressing cycle. Using CNC controlled dressing wheels it is possible to create simple and complex wheel shapes.

Grinding is often a high-precision operation. Therefore, automatic gaging may be a part of the overall production cycle with automatic feedback and automatic compensation being included to achieve the required precision. CNC grinding machines may be universal, surface, centerless, creep feed, and tool grinding. Some machines in the latter category control eight or more axes of motion to generate complex tool geometry that would be difficult to achieve in any other manner.

CNC BENEFITS

CNC machine tools offer the following benefits over manual machine tools:

More Machine Utilization & Productivity – When a CNC machine is efficiently programmed, it spends a much higher percentage of its time with the spindle actually removing metal; the nonproductive time is greatly reduced. While a conventional machine tool may be operating 20 percent of the time, it is not unusual to find CNC machines in operation over 75 percent of the shift with two and three shifts a day common. This translates into shorter lead times for finished components.
Greater Accuracy & Repeatability - Most machine tools equipped with CNC systems are designed with more rigidity than conventional machines to accommodate the higher acceleration forces of servo drive systems. Also, the servo drives and feedback systems control the programmed position or tool path with more precision than manually operated hand wheels. This translates into more accuracy on the workpiece. Once the first workpiece is machined, the machine will repeat the programmed path on part after part. This means that workpiece consistency or repeatability is excellent. In some cases this could reduce or eliminate hand fitting of workpieces in final assembly.

Lower Operator Skill & Costs - The CNC has drastically altered the role and function of the operator. The traditional machine tool frequently required a skilled machinist. A CNC can usually be operated with less skilled employees. Once the workpiece is setup in the fixture or workholding device, operators may only be required to load and unload parts and monitor the machine for obvious problems. Setting up the initial fixtures and workholding devices may require a semi-skilled or skilled worker.

Better Production Scheduling – The programmed, repetitive nature of the CNC machine means that production scheduling can be based on known, standardized machine cycle times. This allows the production department additional management control and scheduling accuracy.

Fewer Machine Setups – Fewer workpiece setups are the general rule on CNC machines. More cutting operations can be performed on a workpiece before moving the workpiece into a new orientation. Many CNC machines now perform a wider array of machining operations. An example is a CNC turning center, which performs both turning operations, and a wide range of milling, slotting, threading operations. Fewer setups mean less changeover time, better part quality and lower manufacturing costs.

Lower Tooling Costs – Conventional machine tools rely heavily on expensive, complex fixturing and tooling. Previously, precision jigs, guides and bushings provided the basis for workpiece accuracy. CNC machines rely on the inherent accuracy of the machine tool and control system. Most shops program their CNC machine to standard tool sizes. This also reduces costs.

Higher Flexibility – CNC programs can easily be modified to allow for workpiece changes or family of parts variations. Once a part program is developed it can be modified to work on similar machines in the shop or machines in other locations. This allows manufacturers to shift production between plants or suppliers while maintaining workpiece dimensional integrity.

More Safety – Once the part program and tooling are setup and proved-out, the machine will repeat its operation the same with every workpiece. This consistency will increase operator safety.
**Expandable** - Additional features and upgrades are easily made through software changes rather than hardware changes.

**HMI Integration** – Human Machine Interfaces can enhance the usability and functionality of the CNC. Some large automotive end-users have standard HMI requirements so that all machine tools have a similar “look and feel” regardless of the CNC or machine tool manufacturer. They do this to simplify training and improve ease of use. Machine tool builders can also develop their own HMIs to provide value added features to basic the CNC. In the case that a machine tool builder uses more than one manufacturer’s CNC, a custom HMI allows them to deliver the same “look and feel” across their product lines.

**Software Solutions** – A larger number of manufacturing and business software applications can run on the open architecture CNC system. These software packages provide solutions for performance, functionality, reliability, training, communication, process management and production.

CNC systems play an important role in the lean manufacturing environment. Proper utilization will reduce: 1) defects, 2) direct labor, 3) support labor, 4) capital equipment expenditures, 5) space and 6) inventory. Refer to Section 6 for additional information.

**MACHINING CENTERS**

The machining center will do a number of different operations such as milling, drilling, boring, facing, spotting, counterboring, threading and tapping, frequently on four or more faces of the workpiece, in a single setup. It is designed to make a complete, or nearly complete, part without having to be transported from machine to machine.

A powerful feature of the machining center is the automatic tool-changing capability. Tool changer capacities continue to grow with some models holding over 300 cutting tools in a storage device (magazine, matrix, carousel or movable rack) ready for quick access by the tool changing mechanism. The part program running in the CNC directs the tool changer to select a specific tool for the next operation and to move it into a ready position. When cutting process with the current tool is completed, the part program directs the tool changer to interchanged the tool in the ready position with the tool in the machine spindle. The tool taken from the spindle is then returned to storage. In some cases, tools have a fixed pocket location, while in other cases tools can be programmed to random pocket locations.

Some machining centers, however, have a turret tool changer similar to a lathe. This kind of tool changer has a very fast tool change time but a limited number of tools. It is ideal for very small parts when the cutting cycles are short so tool change time becomes a significant factor.

Tool exchange times are quoted as either “tool to tool” or “chip to chip”. Tool to tool time only includes the time to exchange the tool after the spindle has arrived at the tool.
exchange park position. This time can be less than one second on high performance machines. Chip to chip time is longer than tool to tool time and includes the time the tool stops cutting the workpiece until it returns to cutting after the exchange.

Machining centers are generally classed as vertical or horizontal, according to the attitude of the spindle. Vertical machining centers are usually selected to machine flat, plate-like or beam-like parts. Horizontal machining centers are usually best suited to the machining of parts that are essentially box-like, cubic or prismatic in nature. However, clever tooling, fixtures and parts processing tend to broaden the applications for each type.

On many machining centers, the workpiece (or fixture) is bolted directly onto the worktable. Some machines provide for rotary indexing of the workpiece. These rotary index tables vary in the number of programmable positions; 60, 360 or 720, for example. Full contouring capability is often available with a 360,000-position rotary table. Five-axis machining centers allow the part to be machined from practically any orientation. This may be achieved by rotating the spindle head in 2-axis, rotating the table in 2-additional axis or a mix of rotating head and rotating table.

![Fig. 6—5-axis machine configuration]( Courtesy FANUC America Incorporated)

Machining centers can also be classed as being traveling column or traveling table (sometimes just called table type), according to whether the column (with the tool) travels longitudinally relative to the table or the table travels longitudinally relative to the column (and tool). If the traveling column type is equipped with two post-type workstations, unloading and loading of workpieces can be done at one station while machining is progressing at the other. Very often, a machine with a single workstation is equipped with some kind of pallet shuttle arrangement to eliminate any lost machining time for the unloading and loading of workpieces. More complex machining centers may be capable of other programmable movements, such as tilt and swivel of the spindle head and column, and secondary or tertiary motions parallel to the principal axes of motion.

**CNC FEATURE CONTENT**

There are hundreds of possible CNC features available. CNC people often speak of the feature content of a particular control unit. A few of these features have already been mentioned. Some are standard for certain types of machine applications while others are
optional. On a modern CNC unit, most of the features are wholly or partially software and must be included in the executive program by the control builder with some features being co-developed with the machine builder. Such features reside in memory.

A particular feature, standard or optional, offered by two different control builders may not operate exactly the same or have the same name. It's possible that neither would exactly fit the brief description given here for a feature by the same name.

Some of the more common features include:

**Program Edit with Program Copy:** This feature permits part programs to be corrected or modified at the CNC. "Program copy" permits the stored part program and the edit data to be copied as a corrected program into another storage area of memory. The data is then cleared from edit memory, freeing it for further corrections.
**Part Program Memory:** The modern CNC has several choices for part program memory. The traditional “internal” memory is built-in to the CNC and is typically high-grade and fast. It is also typically expensive to upgrade and has a limited maximum size (for example 8-Mbytes). Part programs of contoured parts generated with a CAM system can become very large and those created for aerospace and die mold applications are even larger. If a PC is available, programs can also be stored on the hard disk or solid state disk drive and only the active program is loaded into the internal memory. CNCs can also access external memory devices such as flash memory and USB memory sticks. For the largest part programs the CNC can run programs stored on remote networked computers. If a part program cannot fit in the internal memory, it can be downloaded and executed in chunks, spooling from the PC hard disk, external memory card or network. This typically requires some buffer memory to manage the spooling and make sure the CNC does not run out of blocks to execute.

**Canned or Fixed Cycles:** Basically, this is a storage capability built into the control unit for a particular cycle that is used over and over. It saves the programmer the time required to rewrite or redevelop the cycle; he just inserts a code to indicate the cycle is to be executed.

**Parametric Subroutines:** Permits programmer to store frequently used sequences of program data in the CNC memory, to be called into action by code in the main part program. This feature is a major benefit to manufacturers who are machining similar or families of parts.

**Macros:** Most CNC units have permanent memory storage available for certain part program macros such as bolthole patterns, pocket-cleanout routines, drilling and tapping cycles, and other frequently utilized routines. End-users and machine tool builders can develop custom macros for their specific machine or application.

**Automatic Acceleration and Deceleration (ACC/DEC or Acc and Dec):**
Provides smooth, automatic execution of programmed velocity changes that occur from one data block to the next. This is especially important at start and stop of a cut in maintaining good surfaces and accuracies when milling through a tight corner.

**Tool Retract:** Permits retracting the tool from the cut, should it become necessary to interrupt the machining cycle. In some cases, this feature is coupled with provision for automatic return to point of interruption.

**Interpolation:** With the advance in control unit design, more flexibility has been added to control units, including the development of new interpolation methods, such as helical and cubic beyond the normal linear, circular, parabolic or NURBS. The user will need a clear understanding of the interpolation method in his control unit because of its effect upon programming. Interpolation types are described in more detail elsewhere in this chapter.

**Range Checking or Collision Zones:** CNC automatically monitors axes instructions to ensure that no slide movements are attempted beyond specified ranges. Minimum and maximum permissible dimensions for each machine axis
can be stored in the CNC memory. Other safety zone may be established around other machine elements such as a tailstock on a lathe. Within those limits, the part programmer may, when necessary, establish a machining envelope that will take into consideration such factors as fixture details that would lie in a cutter path if any overshoot occurred. If there is a programming error that would take a tool outside of the defined range, the CNC will not permit the move to start, thus eliminating scrapped parts and damaged machines.

**Dry Run:** Allows the part program to be cycled through a non-cutting tryout cycle at a specified override of the programmed feed rate for program debugging.

**Feedrate Override:** Permits the operator or setup man to manually override the programmed feed rate under certain conditions. Normally, it functions as a safety device or a method to fine tune the cutting rate. If a tool is perceived to be feeding too fast and may encounter trouble, the operator can slow down the rate of feed. Or the part programmer can program conservative feed rates, allowing the operator to increase the rate to what appears to be a more acceptable level. On some controls, an override can be established per tool and be in effect each time that tool is used. Many shop managers are trained to observe the Feedrate Override dials when walking past CNC machines. Ideally, it would be set at 100% if the operator and programmer were involved in program optimization. This is especially true of repetitive jobs. However, there are exceptions to the 100% rule such as when workpiece materials change (i.e. hardness of a casting or uneven castings) and the operator must manually compensate. In a worst-case scenario, it may indicate that the operator is purposely running a job too slow to increase overtime hours.

**Jog:** Permits the operator to jog or power-feed the slides without having to create a part program or keying in a block of data to define a move. Two forms of jog are standard on most CNC controls today: continuous (power-feed) and incremental. In continuous, the slide will feed at the rate set by the operator as long as he holds in the jog button. In incremental, the operator sets the desired distance to be traveled each time he pushes the jog button: typically 0.0001, 0.001, 0.01, 0.1 and 1.0 inch. Some controls provide separate buttons for incremental jog and power-feed jog. Usually, slides are jogged individually.

**Position Set:** Eliminates the need to precisely position the fixture. Operator moves machine to align spindle with datum point on fixture (or workpiece) and enters programmed datum point coordinates. The CNC establishes and stores a zero shift dimension.

**Zero Shift:** Permits operator to shift the zero or base data point to any place within the working area or within a specified range. Thus, the part program can be aligned with the part. This assists the operator in setting up.

**Fixture Offset:** Provides capability to automatically compensate for fixture or workpiece misalignment.
Feedhold, program stop or slide hold: Permits the operator to stop the machine motion to inspect a cut and then proceed without losing his reference to the part program.

In-Process or On-Line Diagnostics: Most CNCs now offer some degree of in-process machine and/or control diagnostic capability. The CNC performs a constant monitoring of various conditions and devices. Typically, if an error or malfunction is detected, or a changing condition nears a critical point, an alert signal and appropriate message are displayed on the control's display.

Communications Interface: In today’s just-in-time manufacturing environment, communications plays an increasing valuable role to improve scheduling, reduce setup time, provide real time status information and provide data for process improvement. Much of the communications technology is based on Ethernet technology. Standard protocols such as Ethernet/FTP (file transfer protocol) for file transfer and Ethernet/IP for device interconnectivity. Currently, many CNC manufacturers have proprietary standards for data collection, but there are standard interfaces such as OPC and MTConnect that can be used to connect these CNCs with off-the-shelf SCADA software.

RS232C Serial Interface: This serial data port and interface allows input and output of program data. The operator, through the control panel, controls the interface. Today, the serial interface is being replaced by Ethernet communications.

Direct Numerical Control (DNC): Direct numerical control is a feature that allows a control to “drip-feed” a part program from a storage location other than the CNCs main memory. DNC was traditional used in industries that machine large complex surfaces, like those used in the die mold and aerospace industries. However, parts in every industry are becoming more complex in order to reduce setup time, improve quality and minimize lead times. Part programs are therefore becoming larger. DNC may be used today to drip-feed part programs from a remote host computer using Ethernet, the hard disk of an integrated PC or a large internal or external memory card.

Conversational Programming: This feature allows the operator to program parts at the CNC panel using only the workpiece prints. Conversational programming is typically used in tool rooms and some job shops. Most systems navigate the operator through a variety of machining operations by displaying a series of questions and fill-in-the-blanks graphical screens. The display graphics are an important part of conversational programming. A view of the workpiece and toolpath is normally displayed along with estimated cycle time. Programs can be saved and used again at a later date.

Tool Length Offsets: Refer to Section 9.9 “CNC Tool Management”

Cutter Diameter Compensation: Refer to Section 9.9 “CNC Tool Management”
**Tool Nose Radius Compensation:** Refer to Section 9.9 “CNC Tool Management”

**Look Ahead:** When machining complex, 3-dimensional contours, the control uses many points that are close together. Generally, this type of path would be programmed with a constant program path velocity. Since the servo drives cannot reach the required final velocity in short path distances, the machine never obtains optimal cutting. For example, if a single increment were .004” in length, a cutter traveling at 400ipm might require .4” to stop and/or change direction. Therefore, machine feedrates must be programmed to accommodate the worst case encountered along the path. This might be 15ipm. Without doing so would yield undesirable results such as gouging or corner rounding can occur. The look ahead feature will read ahead a number of blocks of data (the actual number of blocks will vary during the operation) to look for problems. It will then automatically assign individual feedrates to each small incremental move to maximize cutting conditions. This feature is popular in mold and die cutting operations as well as complex aerospace surfaces. Machines capable of 100ipm feedrate and Look Ahead could easily outperform a machine rated at 400ipm not supplied with Look Ahead. This feature depends heavily on a machine having properly optimized acceleration and deceleration, which comes through servo optimization and inherent machine rigidity.

**Block Processing Speed:** A key measurement of CNC performance is the speed that it processes blocks of program data. While this feature or specification is important, there are several factors that cannot be ignored. First, CNC manufacturers define the term “block” differently. Some only count it as a single “word” such as “X15” while others defines it as the entire line or block of program data. This must be considered in determining or comparing the CNC’s processing speed capability. Second, is that this rating only applies to the control’s capability, not the machine tool’s capability. It is entirely possible that the machine tool cannot keep up with the CNC’s processing capability due to lack of rigidity or poor servo response.

**Interpolation Rate:** CNC systems have a cyclic period of time called an interpolation period. During this time period the CNC instructs all axes to move a certain number of counts. Since the velocity is constant during this time period the movement is a linear segment. For example, if the interpolation period is 1 millisecond, the control instructs the axes how many counts to move during that period. The faster the interpolation rate, the smaller the linear segments. Higher interpolation rates may mean greater workpiece accuracy, better surface finish and lower machining cycle time.

**Machine Utilization Monitoring & Management:** Some CNC controls have the ability to monitor and collect data throughout the manufacturing work shift. Monitoring of data such as machine up time, part counting, program processing times, feedrate and spindle speed override and tool in cut time are are valuable for
management evaluation of machine utilization. Normally, this data can be printed or sent to a remote computer via a network.

**Leadscrew Pitch Error Compensation:** This feature compensates for ballscrew errors. Many CNC machines obtain a feedback signal from a rotary encoder attached to the ball screw. This type of feedback measuring system assumes that the lead of the ballscrew is constant at every point within the ballscrew’s traversing range. However, ballscrews are not manufactured to zero tolerance standards. The resulting manufacturing error is called “leadscrew error”. Additional machine assembly tolerances can add even more error to the measuring system. Therefore, some controls are equipped with a Leadscrew Pitch Error Compensation feature that allows entry of compensation values for measured point along the traverse range. Normally, each machine axis is calibrated with a freestanding laser-measuring instrument and corrections are then entered into a data table in the CNC system.

**Electronic Handwheels:** The machinist turning hand wheels attached to axis leadscrews controls manual machine tools. With a few exceptions, CNC machine tools do not have mechanical handwheels since the CNC control and servos position the machine automatically. Many machines have a single electronic handwheel that can be switched to any axis. The rate at which the axis moves in proportion to the handwheel can be set. The handwheel is useful for precise manual positioning during setup. Some machines have multiple handwheels that allow actual machining for low-volume or prototype work. These handwheels are not mechanically connected to the ballscrew. Instead, they send electronic pulses to the servo system, which then move the axis.

**Constant Surface Speed (CSS):**
When machining a cylindrical workpiece on a lathe, the diameter of the workpiece is constantly changing when performing a facing operation and when machining at different diameters. Surface speed is defined as the speed at the tool tip. If the RPM remains constant while the tool feeds across the workpiece surface (decreasing diameter), then the surface speed will be changing during the cut. The cutting characteristics

---

Fig. 7 – Facing Operation with Constant Surface Speed
(Courtesy Decision Technology, Inc.)
will be changing across the cut and give non-uniform results (surface finish, accuracy, etc.). However, with CSS the CNC control calculates and adjusts the rpm to maintain a constant surface speed at the tool tip. In the example at the right the spindle RPM will gradually increase as the tool traverses (x axis) the workpiece face while keeping the surface speed constant. This same feature can be employed to keep the surface speed constant with successive cuts along the diameter in the z-axis.

Adaptive Control: Refer to Section 7.8 “PLC & Sensor Technology”.

5-AXIS CNC FEATURE CONTENT

There are two types of 5-axis machining:

3 + 2 machining: With 3 + 2 machining, the part is rotated to a specific orientation, prior to machining. The orientation of the part does not change during machining. The 2 rotary axes may be indexed, that is they can only move to specific position and then they are locked during machining. Typically the part is tilted to the required plane or orientation and then the standard 3-axes (X/Y/Z) machine the part just like any other 3-axis part.

Simultaneous 5-axis machining: The axis of a simultaneous 5-axis machine can move during machining. Some parts like complex molds, aerospace parts and compressor impellors can only be machined with the simultaneous 5-axis machines. Additional features are required to simplify simultaneous 5-axes machining.

3 + 2 5-AXIS CNC FEATURES

Since, 3 + 2 machining actually performed 3-axis machining when the part has been oriented, most standard 3-axis features work the same including tool length compensation, cutter radius compensation and workpiece coordinate system offsets.

Tilted working plane: Simplifies the programming and setup required for the rotary axis and tilting axis, so that the workpiece is indexed to the orientation suitable for the machining with a simultaneous 3-axis machining program. The tilted plane may be established manually through operator inputs, or automatically using a touch probe.

SIMULTANEOUS 5-AXIS CNC FEATURES

Historically, the workflow for programming and setup was entirely dependent on the CAD/CAM system. Any change in machine, workholding, workpiece or tooling would
require a new part program to be posted with the new parameters. The modern simultaneous 5-axis workflow is similar to the standard 3-axis workflow. The part program only contains information about the geometry of the part and the machining process. The machine specifics, workpiece orientation and tooling details are all handled by the CNC and using input provided by the operator during setup.

The standard 3-axis or 3 + 2 CNC only has to work in the fixed planes (G17/G18/G19). However, a simultaneous 5-axis machine must handle functions like tool length compensation, cutter radius compensation and contouring in any plane and regardless of the machine configuration (head rotation, table rotation or mixed rotation).

**Tool center point control:** The tool tip position is automatically controlled so that it moves along the programmed path at the specified speed even if the tool direction varies. It allows the part program to only consider the surface of the part when generating tool paths. The adjustments for tool length geometry and wear are compensated for by the CNC in the tool axis direction, which may require the movement of up to all five axes. The programmer can also specify an inclination angle to be maintained between the tool and the workpiece cutting path to maximize the life of ballnose tools.

**3D cutter compensation:** Fundamentally, 3D cutter compensation achieves the same result as standard 3-axis cutter compensation, only it has to work at any tool orientation. Along with tool center point control, it allows the tooling to be independent of the part program.

**3D workpiece setting error compensation:** Using 3D workpiece setting error compensation the CNC can adjust the part program to machine a part at any orientation, simplifying setup. Typically, the actual orientation of the part will be measured using a touch probe.

**3D interference check:** Since the part programmer cannot be sure of the final machining parameters (tooling, fixtures, part orientation), it is useful if the CNC can perform interference check in real time.

**Manual feed in the tool direction:** Typically, axes are moved in manual on a per axis basis. It can be challenging for an operator to retract or return a tool inside a feature that is on an arbitrary plane. Providing the ability to jog the tool in the tool axis simplifies manual interventions.
7.5 CNC Part Programming

(Adapted from and used with permission of Modern Machine Shop Magazine)

INCH OR METRIC

Most CNC units have both inch and metric capability. The choice may be implemented either by a switch, setting or by a specific code entered in the part program.

ABSOLUTE OR INCREMENTAL MODE

There are two basic modes to specify axis positions; one is "absolute" and the other is "incremental." Absolute positions are dimensioned from the same program zero. Increment positions are dimensioned from the current axis position.

It is possible to switch back and forth from one mode to the other within the same part program. Doing this, however, demands extreme caution by the CNC programmers to insure their own reference basis.

On a machining center, a preparatory code command (G90=absolute, G91=incremental) is used to specify the mode. On a lathe, X and Z letter addresses are used to specify absolute moves and U and W letter addresses are used to specify incremental moves. The programmer generating a part program manually can work from the drawing and use the most convenient mode - absolute, incremental, or a combination of the two.

If a programmer is preparing a manuscript with absolute dimensioning, positions will be stated as measurements from a single fixed zero or origin point. The point may be fixed on the machine itself such as a corner of a worktable or, more likely, some specific point on the part. Most CNCs are equipped with a "floating zero" or "zero shift" which enables the programmer to establish the zero point for a particular part in the most convenient location. When preparing incremental input, all program locations are given in terms of distance and direction from the current position of each axis.

POSITIONING OR CONTOURING

CNC programming falls into two conceptual realms; positioning, also known as point-to-point for processes like hole drilling and punching, and contouring, or continuous path for cutting complex paths. Today’s CNC are built with a mixture of positioning and contouring capabilities.
A simple comparison between positioning and contouring is illustrated in Figure 8. If a drilling tool is positioned at X0, Y0, and then at X3, Y4, there will be two holes drilled, one at each location. The coordinate positions are the hole centers; the hole sizes are determined by the drill diameter. If, however, a milling cutter is advanced into the work at X0, Y0 and then sent in a direct line to X3, Y4 without being retracted, the result will be a slot machined between the two coordinate points; the width will be the diameter of the milling cutter. There would have to be a direct ratio of three units of X movement for each four units of movement taking place simultaneously in Y.

Fig. 8-- If a hole is to be drilled at the X3, Y4 location, the centerpoint of the drill is directed to the programmed point. If a slot represented by the broken line is to be milled from the origin point to X3, Y4, the centerpoint of the cutting tool is sent from the first to the second position. Slot width is determined by the cutting tool diameter. To get this straight-line configuration, the table drivescrews or spindle-moving mechanisms are controlled by the CNC so that three units of X motion occur simultaneously with four units of Y movement. (Courtesy Gardner Publications)

POSITIONING

Positioning is the type of work typified by drilling and punching, where the drill or punch moves over and across the workpiece to a specific location and there performs the programmed event or sequence of events. It may be drilling, tapping, boring, punching, spot welding, riveting or any operation completed at a fixed workpiece location in terms of a two-axis coordinate position. The process is repeated until all machining is completed at the programmed positions on that workpiece.

While moving from one coordinate position to the next, the cutting tool is not in contact with the workpiece. The tool may move above the fixed workpiece, or the table may move under the tool. It is only after the tool and workpiece attain the programmed relationship to each other that the tool advances into the workpiece.

Today’s CNCs provide the ability to interpolate the positioning move so the axis arrive at the hole at the same time, providing a predictable path to allow enhance interference checks to be performed.

A good programmer will keep the moves as short as possible to minimize non-machining travel time. Where several different cutting tools are used in a program, he will weigh tool-changing time against travel time to achieve the optimum combination. In most instances, it will pay to remain with one tool until all operations requiring its use are finished.
ROTARY TABLE POSITIONING

Many machining centers have a rotary index table; indexable usually to 360 positions (Figure 9). On most of today's controls, the index command position is programmed directly in degrees and usually in absolute terms even if the incremental mode is in effect. The actual direction of rotation is by shortest path from one position to the next. Some tables with a small number of index positions assign a single digit identity to each position for programming purposes.

![Diagram of a rotary table and spindle](image)

**Fig. 9—On a horizontal machining center, the polar axis of a rotary positioning table is aligned parallel with the machine's Z-axis. Moving the index table 30 degrees clockwise (CW) from the polar axis (zero degrees) puts the cutter 30 degrees into the first quadrant. A hole can be drilled along that line or paralleling it by programmed linear movement of the Z-axis. (Courtesy Gardner Publications)**

CONTOURING

In contouring or continuous-path operations the cutting tool is in constant contact with the workpiece as coordinate movements take place. Thus, a path is literally machined as programmed movements occur. The most prevalent contouring operations use milling cutters and lathe tools to profile and sculpture workpieces. Contouring may also apply to flame cutting, laser cutting, sawing, seam welding, and grinding.

The path that the cutting tool maintains with the workpiece is all-important and must be provided for at all times in the program. The major difference between a positioning and a contouring control is the latter's interpolation features, which coordinates the axes movements and permits the programmer to specify a vector feed rate.
PROGRAMMING FOR CONTOURING

There are both similarities and differences in programming for contouring (or continuous path) and programming for positioning. Interpolation is the CNCs ability to precisely control each axis velocity so that the tool tip moves along a defined path at a programmed feedrate. The control breaks the complete move up into tiny segments and determines the velocity that each axis must move in that time so that the tool will stay on the programmed path. At the end of each interpolation period, the axes feedback devices allow any error to be corrected in the next set of velocities.

The contouring principle can be illustrated with a simple circle, defined as an infinite number of lines joined end-to-end to form a closed plane, with each line equidistant from a center point. Because the number of lines is infinite, the circle outline is analog in nature. Computer numerical control is not analog. It is digital and finite. Thus, to approximate a curve of any kind, a sufficient number of points must be defined within the working framework of rectangular coordinates to describe a circle as shown in Figure 10.

*Fig. 10--Configurations obtained from continuous-path machining are the result of a series of straight-line, parabolic span or higher-order curves. The degree to which curved surfaces correspond to their design depends on how many lines or spans are used. It can be seen that four equal chords in a circle describe a square. Six make a hexagon, and as the number increases the lines themselves come closer to a perfect circle. Number of lines needed is determined by a maximum tolerance allowed between the design of the curved section and the actual chord programmed. The program for a parabolic span type control unit requires enough spans for any deviation to stay within an acceptable tolerance. If, for example, the tolerance indicated by T is 0.001 inch, 30-degree spans are sufficient to machine a 10-inch arc. Cubic interpolation is very similar in principle to the parabolic. (Courtesy Gardner Publications)*

The circle is machined by connecting the points with a series of tool movements. This holds true for circles, arcs, curves or shapes that are entirely free-form and the result of an artistic design. It would be possible to program the most intricate facial sculpture if enough coordinate points could be defined that would describe the surface. From figure 11 it can be concluded that the high the resolution of interpolation or the faster the interpolation rate, the more smooth the surface can be machined.
The modern CNC has many different interpolation paths that can be programmed. The basic ones are linear interpolation (along a straight line), circular interpolation (clockwise or counter clockwise around a radius or complete circle) and helical interpolation (moving the third axis perpendicular to two axes moving along a circular interpolation path). More complex interpolations include cubic interpolation and NURBS interpolation, which are described later in this document.

**LINEAR INTERPOLATION**

Linear interpolation (G01) is precisely that: programmed points connected by straight lines. Linear interpolation is used for straight-line moves, whether the span length between two points is very short or very long.

In programming a contouring cut with linear interpolation, the X and Z in lathe, the X and Y coordinate positions in machining center two-axis work (Figure 11), or X, Y and Z positions in three-axis work (Figure 12) are given for the beginning and end point of each line segment. The end point of line one becomes the beginning point of line two and so on throughout the program. These coordinate positions have to appear in the program to define each line segment.

---

**Fig. 11--Two-axis linear interpolation requires programming start-point and end-point coordinates, and a vector feed rate. Linear interpolation uses any two of the machine's linear axes simultaneously. (Courtesy Gardner Publications)**

**Fig. 12--Three-axis linear interpolation uses the three linear axes (X, Y and Z) of the machine simultaneously. Again, programming requires start point, end point and a vector feed rate. (Courtesy Gardner Publications)**
Linear interpolation is often used to produce complex curves by programming numerous and tiny linear motions that approximate the path of the curve (figure 13).

![Fig. 13--Linear interpolation is commonly used to approximate a curve. The more program points are added, the less the programmed path will deviate from the desired path. (Courtesy Gardner Publications)](image)

There are several challenges with this kind of programming:

**Large programs:** Even relatively small parts can generate a huge number of moves for a complex three-dimensional part. Large part program memory, drip-feed with buffers from a networked drive, and integrated drive or external memory devices are some of the solutions on a modern CNC.

**Block processing time:** A better surface is produce with ever smaller linear segments, except now the CNC has more blocks to read and interpret. The modern CNC uses a buffer to prevent “block starvation” when drip-feed part programs and the CNC processor are ever faster at processing numerous blocks.

**Surface finish problems:** The modern CNC servo system on a high-performance machine has a very high gain and can replicate the programmed surface exactly. Unfortunately, this means you can see the transition points where linear segments meet in the part surface. These sharp transitions also create minute machine vibrations that cause feedrates to be limited. In these applications the CNC can convert the linear segments in the part program back into the original complex curve defined in the CAD system and interpolate it natively using NURBS interpolation. This technology both increases the surface quality and the potential feedrate. It also allows the programmer/CAM system to use less points to define the curve, reducing the size of the part program and reducing the issues of block processing time.

**CIRCULAR INTERPOLATION**

Circular interpolation moves a tool in an arc or complete circle, either clockwises (G02) or counterclockwise (G03). The programmer must provide the coordinate location of the end point of the arc and either the radius of the arc or the coordinate location of the center of the arc or circle. A feedrate must also be specified.
The CNCs circular interpolator breaks up the circular span into a series of the smallest increment of movement possible. Modern CNCs are capable of nanometer resolution, 0.000001mm or 0.000254-inches. (figure 14) The interpolator automatically computes the moves required to describe the circular cut and generates the servo signals that will move the cutting tool to machine the cut. The tool path produced with most modern controls will be within plus or minus one pulse.

On a 3-axis or 3 + 2 5-axis machine, circular interpolation is limited to a two-axis plane (G17/G18/G19); it cannot interpolate circular movements in a 3D plane. A true simultaneous 5-axis machine can interpolate a circle in any arbitrary plane.

**Fig. 14--Two-dimensional circular interpolation uses any two linear axes of the machine simultaneously. It requires programming a start point, end point, radius, and cutter feed rate. Some controls can generate a 360-degree circle; others can only generate one quadrant at a time. (Courtesy Gardner Publications)**

**HELICAL INTERPOLATION**

The third most common form of interpolation is helical. As shown in figure 15, this internally combines use of two-axis circular interpolation with a linear movement of the third axis. All three axes move simultaneously to create a helical spiral cutter path. This method has its most common application in milling large internal diameter threads and plunge feeding.

**Fig. 15--Helical interpolation is three-dimensional, utilizing circular interpolation in two axes with a simultaneous linear movement occurring in the third axis. All three axes move simultaneously to generate a helical spiral cutter path. While the illustration shows circular in the X-Y plane with a linear move in Z to define the lead, most controls provide full plane switching capability. (Courtesy Gardner Publications)**
NURBS INTERPOLATION

NURBS is short for Non-Uniform Rational B-Spline, which is a mathematical representation of a 3-dimensional object. Most CAD/CAM applications support NURBS, which can be used to represent analytic shapes, such as cones, as well as free-form shapes, such as car bodies. While most other forms of interpolation use straight lines and chords to define a curved surface, NURBS can define a complex surface by using “control points”, “weights” and “knots”. In Circular Interpolation, a few pieces of data can be used to define a curve (i.e. start point, end point, center of the arc). In NURBS tool paths, data relates to the locations and “weights” of control points, each of which exercises a “pull” to give the curve its trajectory or path. This process is similar to taking a straight metal thread and pulling it into a curve by using a magnet at different points along the thread. Just a few numbers can define a complex curve. The benefits of using NURBS are:

- Greater workpiece accuracy.
- Shorter cycle times.
- Better surface finish.
- Smaller part programs. A typical NURBS block can replace 5 to 10 blocks of straight-line moves.
- Reduced risk of data starvation and potentially faster block processing speed.

This technology is used extensively by mold and die machining shops that require complex non-uniform curves and long part programs.

Many of today’s CNCs can interpolate NURBS curves. They can also recreate a curve from a short line segment program created by a CAM system and reconstruct the NURBS curve that was originally created in the CAD system. This results is higher feedrates, better surface finish and a part that more accurately represents the designer’s intent.
MANUAL OR COMPUTER PROGRAMMING

Manual Programming
The manual creation of contouring programs is in general only feasible in those situations where there is a low degree of complexity. Such cuts as straight lines; tapers, arcs or slopes may be programmed manually with a reasonable degree of success. Some high volume parts used in automotive may be manually programmed to achieve the highest level of optimization for the shortest cycle time. Conversational programming adds the ability to develop more sophisticated programs on the CNC. However, there is a point where manual programming becomes totally impractical.

CAD/CAM Programming
Today, most parts are designed by computer-aided design (CAD). The part is designed visually on a computer screen, often as a 3D solid model. The CAD file includes all the appropriate dimensions. These CAD files with the designed part characteristics can be quickly converted to a tool path suitable for the CNC system using a computer-aided manufacturing system (CAM). The CAM programmer selects part features and applies tooling and processes to machine that feature. CAM systems typically include graphical verification and some provide full 3D animation of the part program displaying the part, tooling and sometimes the machine.

A CAM system delivers time savings and productivity increases in the manufacturing of today’s products.

Conversational Programming
Conversational programming is a middle ground between manual programming and CAD/CAM programming. Some smaller job shop customers may not provide electronic CAD files. There are also applications in tool rooms where conversational programming is used as semi-automatic manual machining. Creating low volume parts from simple sketches or paper drawings may be achieved efficiently using conversational programming.

Similar to CAM programming, conversational programming is operations based. The programmer/operator selects an operation such as facing, roughing, drilling or tapping, and then the CNC application steps the operator through entering the information required to generate the desired part feature. Conversation programming systems often include graphical animation with representations of the part and tooling so that the operator can verify the part program before machining a part.

PROGRAMMING CODES

Programming Codes and Functions
A CNC needs more information than axis positional data to manufacture a part. Additional instructions are required to define how the positional data is to be processed, to control the machines auxiliary functions, set spindle speeds and to select tools. This data is provided in the part program in the form of G-codes (preparatory), M-codes
(auxiliary functions), S-codes (spindle), T-codes (tool change), and other codes. The following typical codes are given for reference only. Consult with current ISO, DIN and RS274 standards, which define G and M codes as well as the machine tool builders and CNC manufacturer’s programming manual.

**G-codes** (preparatory codes) changes the way the part program positional data is interpreted and processed. The programmer or CAM post will include the appropriate G-codes for: rapid positioning (G01), linear (G01), circular or helical interpolation (G02/G03), inch (G20) or metric (G21) dimensional input, absolute (G90) or incremental (G91) data input, feed per minute (G94) or feed per revolution (G95), and so on. Typical codes are shown below. Be cautioned that individual CNCs may have slightly differing codes.

<table>
<thead>
<tr>
<th>Code</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>G00</td>
<td>Point to point, positioning – Rapid traverse</td>
</tr>
<tr>
<td>G01</td>
<td>Linear interpolation</td>
</tr>
<tr>
<td>G02</td>
<td>Circular or helical interpolation arc CW</td>
</tr>
<tr>
<td>G03</td>
<td>Circular or helical interpolation arc CCW</td>
</tr>
<tr>
<td>G04</td>
<td>Dwell</td>
</tr>
<tr>
<td>G17</td>
<td>XY plane selection</td>
</tr>
<tr>
<td>G18</td>
<td>ZX plane selection</td>
</tr>
<tr>
<td>G19</td>
<td>YZ plane selection</td>
</tr>
<tr>
<td>G20</td>
<td>Inch programming</td>
</tr>
<tr>
<td>G21</td>
<td>Circular interpolation arc CW (short dimensions)</td>
</tr>
<tr>
<td>G33</td>
<td>Thread cutting, constant lead</td>
</tr>
<tr>
<td>G34</td>
<td>Thread cutting, increasing lead</td>
</tr>
<tr>
<td>G40</td>
<td>Cutter compensation cancel</td>
</tr>
<tr>
<td>G41</td>
<td>Cutter compensation-left</td>
</tr>
<tr>
<td>G42</td>
<td>Cutter compensation-right</td>
</tr>
<tr>
<td>G54-G59</td>
<td>Work offset selection</td>
</tr>
<tr>
<td>G80</td>
<td>Fixed cycle cancel</td>
</tr>
<tr>
<td>G81</td>
<td>Fixed cycle 1</td>
</tr>
<tr>
<td>G82</td>
<td>Fixed cycle 2</td>
</tr>
<tr>
<td>G83</td>
<td>Fixed cycle 3</td>
</tr>
<tr>
<td>G84</td>
<td>Fixed cycle 4</td>
</tr>
<tr>
<td>G85</td>
<td>Fixed cycle 5</td>
</tr>
<tr>
<td>G86</td>
<td>Fixed cycle 6</td>
</tr>
<tr>
<td>G87</td>
<td>Fixed cycle 7</td>
</tr>
<tr>
<td>G88</td>
<td>Fixed cycle 8</td>
</tr>
<tr>
<td>G89</td>
<td>Fixed cycle 9</td>
</tr>
<tr>
<td>G90</td>
<td>Absolute programming</td>
</tr>
<tr>
<td>G91</td>
<td>Incremental programming</td>
</tr>
<tr>
<td>G94</td>
<td>Feed per minute programming</td>
</tr>
<tr>
<td>G95</td>
<td>Feed per revolution programming</td>
</tr>
</tbody>
</table>
M-codes are used by the programmer to define a machine or control auxiliary function such as stopping the cycle, defining the end of a part program, actuating tool change mechanism, and any other control or machine single-command function.

<table>
<thead>
<tr>
<th>Code</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>M00</td>
<td>Program stop</td>
</tr>
<tr>
<td>M01</td>
<td>Optional (planned) stop</td>
</tr>
<tr>
<td>M02</td>
<td>End of program</td>
</tr>
<tr>
<td>M03</td>
<td>Spindle On CW</td>
</tr>
<tr>
<td>M04</td>
<td>Spindle On CCW</td>
</tr>
<tr>
<td>M05</td>
<td>Spindle Stop</td>
</tr>
<tr>
<td>M06</td>
<td>Tool change</td>
</tr>
<tr>
<td>M07</td>
<td>Coolant On - Mist</td>
</tr>
<tr>
<td>M08</td>
<td>Coolant On - Flood</td>
</tr>
<tr>
<td>M09</td>
<td>Coolant OFF</td>
</tr>
<tr>
<td>M10</td>
<td>Clamp</td>
</tr>
<tr>
<td>M11</td>
<td>Unclamp</td>
</tr>
<tr>
<td>M12</td>
<td>Unassigned</td>
</tr>
<tr>
<td>M13</td>
<td>Spindle On CW &amp; coolant On</td>
</tr>
<tr>
<td>M14</td>
<td>Spindle On CCW &amp; coolant On</td>
</tr>
<tr>
<td>M15</td>
<td>Spindle Stop with Coolant On</td>
</tr>
<tr>
<td>M16-M18</td>
<td>Unassigned</td>
</tr>
<tr>
<td>M19</td>
<td>Spindle Orientation On</td>
</tr>
<tr>
<td>M20</td>
<td>Spindle Orientation Off</td>
</tr>
<tr>
<td>M21</td>
<td>Table Rotate CW / Tool Magazine Right</td>
</tr>
<tr>
<td>M22</td>
<td>Table Rotate CCW / Tool Magazine Left</td>
</tr>
<tr>
<td>M23</td>
<td>C-Axis Enable / Tool Magazine Up</td>
</tr>
<tr>
<td>M24</td>
<td>C-Axis Disable / Tool Magazine Down</td>
</tr>
<tr>
<td>M25</td>
<td>Tailstock Engaged / Tool Clamp</td>
</tr>
<tr>
<td>M26</td>
<td>Tailstock Retracted / Tool Unclamp</td>
</tr>
<tr>
<td>M27</td>
<td>Clutch Neutral On</td>
</tr>
<tr>
<td>M28</td>
<td>Clutch Neutral Off</td>
</tr>
<tr>
<td>M30</td>
<td>End of Program</td>
</tr>
<tr>
<td>M31</td>
<td>Interlock bypass</td>
</tr>
<tr>
<td>M40-M45</td>
<td>Gear changes if used; otherwise unassigned</td>
</tr>
<tr>
<td>M98</td>
<td>Call Subprogram</td>
</tr>
<tr>
<td>M99</td>
<td>End Subprogram</td>
</tr>
</tbody>
</table>

Other Code Letters used by the programmer are shown below. These may vary by CNC type and manufacturer:

- D Tool radius
- F Feedrate
- H Tool length
- I Arc Center in X
- J Arc Center in Y
- K Arc Center in Z
- N Line Number
- R Radius Code
- S Spindle Speed
- T Tool Selection
- U Incremental X value
- V Incremental Y value
- W Incremental Z value
PROGRAM STRUCTURE

A CNC program consists of one or more "blocks" and each block contains one or more "words". A word is generally a "letter address " followed by a value. An example of a block is:

```
N350 G02 X-25.0 Y52.0 R4.0 M08
```

The machine control executes or processes one block at a time.

Control systems differ in the format they accept data. For instance, some control manufacturers must have leading zeros on values and codes, for example G02 not G2. Blocks may start with a block number (N350 in above example). If used, it is normal practice to go up in tens to allow blocks to be inserted when changes are required.

Normally, only program changes in axis positions must be programmed. Most codes are modal and stay active until another code in the same group is programmed.
G-CODE PROGRAMMING EXAMPLE

The following simple part program demonstrates three types of G-code controlled motion: 1) rapid, 2) straight and 3) circular. It also uses tool length compensation and cutter radius compensation to adjust the path for the 0.75 end mill used to machine the part.

Follow the program example one step at a time. If you have trouble, find an application engineer or programmer and ask them to explain each block of data.

Fig. 18—Part Example. Top and front views.
Typical G-Code Part Program (See above)
The following is the part program for the workpiece drawing shown above figure 18. The positions P1 through P10 in figure 19 below are the programmed positions based on the drawing. The tool outlines are the actual tool positions the CNC used after compensating for the 0.75 diameter tool.

O0002 (EXAMPLE PROGRAM);
N010 G54 G90 (SELECT COORDINATE SYSTEM, ABSOLUTE MODE);
N020 M06 T01 (SELECT TOOL #1 – 0.75 DIA END MILL);
N030 S350. M03 (START SPINDLE 350 RPM CW);
N040 G00 X-1.0 Y-0.125 (RAPID TO START PSN POINT 1);
N050 G43 H01 Z-0.25 (INSTATE TOOL LENGTH COMPENSATION, RAPID TO WORK);
N060 G41 G01 Y0.25 D01 F3.5 M08 (TURN ON CUTTER RADIUS COMPENSATION);
N070 G01 X5.25 (MACHINE IN STRAIGHT MOTION TO POINT 2);
N080 G03 X5.75 Y0.75 R0.5 (MACHINE CCW CIRCULAR MOTION TO POINT 3);
N090 G01 Y3.25 (MACHINE IN STRAIGHT MOTION TO POINT 4);
N100 G03 X5.25 Y3.75 R0.5 (MACHINE CCW CIRCULAR MOTION TO POINT 5);
N110 G01 X0.75 (MACHINE IN STRAIGHT MOTION TO POINT 6);
N120 G03 X0.25 Y3.25 R0.5 (MACHINE CCW CIRCULAR MOTION TO POINT 7);
N130 G01 Y0.75 (MACHINE IN STRAIGHT MOTION TO POINT 8);
N140 G03 X0.75 Y0.25 R0.5 (MACHINE CCW CIRCULAR MOTION TO POINT 9);
N150 G02 X2.25 Y-1.25 R1.5 (MACHINE CW CIRCULAR MOTION TO POINT 10);
N160 G91 G28 Z0.0 M05 (RETURN Z HOME AND STOP SPIDLE);
N170 G91 G28 X0.0 Y0.0 M09 (RETURN X AND Y HOME, STOP COOLANT);
N180 M30 (END OF PROGRAM);
7.6 Motion Control Technology

The term "motion control" refers to the controls and devices on the machine tool that provide motion to the machine axes and spindles. The control of the machine's axes (and some spindle positions) is called a "servo" system. While motion control is a complex topic, this chapter will limit its discussion to simplified theory and practices.

The vast majority of CNC machine tools use closed-loop systems. An axis position command is sent from the CNC, to the servo amplifier, to drive the machine’s axes. The CNC constantly compares the command signals to the feedback signals mounted to the machine elements and corrects for any error. This assures that the programmed command is being acted upon and properly executed.

Feedback signals are critical to closed-loop systems. They are generated by one of several devices:

**Rotary Encoders** are an electromechanical device that converts the angular position or motion of a motor or ballscrew to a digital code. The encoder's rotor shaft is coupled directly with the motor shaft or lead screw. This is the most common method of feedback in CNC applications.

![Fig. 20 - Machine Control System Servo Loop](image)  
*(Courtesy Decision Technology, Inc.)*

![Fig. 21 – Rotary Encoder](image)  
*(Courtesy Heidenhain)*
common feedback system found on machine tools.

In most machining situations rotary encoders deliver reliable, high accuracy feedback. However, any mechanical backlash (i.e. ball nuts and couplings) or drive train inaccuracies (i.e. ballscrew inaccuracy, sag or heat growth) are not taken into account as they occur “beyond” what the rotary encoder is measuring. The CNC can compensate for some of this error with features such as backlash and leadscrew pitch error compensation. The errors must be measured and input into the control.

Angle encoders are available for rotary axis or rotary table applications.

Many CNC manufacturers are now using absolute serial encodes. The serial interface provides higher operating speeds and noise suppression than the traditional digital interface.

Several rotary encoder technologies are available:

- **Optical Encoder**: An optical encoder has a disk having equally spaced slots cut into its periphery. As the screw is turned, the slots in the circular disk pass a light source (LED) and photodiode sensor, creating a stream of pulses that are counted by the CNC.

- **Serial Encoder**: Serial encoders are used for high-speed, high-resolution applications, where the rate of pulses generated by conventional encoders is limited by the bandwidth of the encoder to CNC cable. There are a number of standard and proprietary interfaces and protocols for serial encoders.

- **Absolute Encoder**: An "absolute" encoder maintains position information when power is removed from the system. The position of the encoder is available immediately on applying power and the axes do not have to be referenced or homed before running a part program. Absolute encoders have a battery which maintains the detection circuitry even when the main CNC power is turned off.

Many machine tools today use absolute serial optical encoders.

**Linear Scales**: A linear scale feedback device consists of parts, a scale and a slider. The scale is normally mounted to a fixed-position machine member, and the slider is mounted to the associated moving member. As the carrier travels along the column, the slider correspondingly moves along the scale. Scales are a cost effective way to obtain...
superior positioning accuracy. With the measuring scales and/or reading heads mounted directly on the moving machine member they can take into account such issues as backlash, ballscrew inaccuracies and machine heat growth.

There are several linear scale technologies and each have their strengths and weaknesses:

**Glass Scales:** A glass scale (also called a linear encoder) is manufactured with photo-etched graduations. The reading head passes over and photoelectrically scans the graduations on the stationary part using a light beam. As the light passes through the scale (transmitted light method) or is reflected back from the scale (reflected-light method) it is diffracted and read by photovoltaic cells. These cells sense small changes in the light intensity, which can be interpolated into a position signal. Normal wear on drive train components will not reduce the machine’s performance. Circular scales are available for rotary axis or rotary table applications. Scales are standard equipment on some machine tools and optional on others.

**Absolute Scales:** Absolute scales have two sets of graduations. One is the normal high-resolution scale and the other is a low resolution scale that reports the absolute position. Like absolute encoders, a machine with absolute scales does not need to be referenced on power up.

**Serial interfaces:** The same restrictions and benefits apply as for serial encoders discussed earlier.

**Laser Beam** generators and reflectors are affixed to a moving and stationary machine member. The laser beam system is probably the most accurate feedback system. It is an expensive option and is therefore only found on machine tools requiring ultra high precision.

**SERVO DRIVES**

The servo drives in today’s CNC systems are very sophisticated. They use microprocessor and/or digital signal processors to optimize performance in conjunction with the CNC. The connectivity between the CNC and the servo drive is a high speed two-way digital signal. The communication medium may be electrical or use fiber optic cables to increase bandwidth and noise suppression. SECOS

The CNC receives more intelligent information about the servo motor and machine than the simple position feedback. With more information about motor current, motor position, heat and other parameters, the CNC and servo system can implement higher order dynamic algorithms to compensate for undesirable characteristics in the machine tool or application. Artificial intelligence technologies can be used to record dynamic response and improve performance on the next part.
Advanced servo drive systems are energy efficient, converting kinetic energy from the machine during deceleration back into electricity and pumping back into the supply rather than burning it as heat in regenerative resistors.

**SERVO MOTORS**

The motors used in today are either rotary AC servos motors or linear motors.

**AC servos motors** have permanent magnets in the rotor and coils in the stator that create a magnetic field, producing mechanical rotation. The armatures in servo motors are designed to be lightweight, low-inertia components so that they can respond quickly to excitation-voltage changes.

Torque and acceleration are the key specifications when evaluating servo motor selection and application. Higher torque is needed in machining operation requiring axis thrust such as large hole drilling and heavy milling. A machine with insufficient thrust could limit metal removal rates. If the part program requires more thrust than the machine’s rating the machine axis could stall. This would be a dangerous situation. Typically, servo motor torque would be within a range of 0.5 to 90 lb-ft. Acceleration is important for machine cycle times because many machine tools are constantly accelerating and decelerating.

**Linear Motors** are the newest advancement in machine tool servo motor design and application providing very high accelerations and speed. They are simple in design – take an AC motor with a cylindrical rotor and stator and stretch them out flat. They have several advantages and a few limitations. First, they are highly responsive with a high acceleration rate because the high-rotary inertia of the rotary motor and ballscrew are eliminated, which results in a high performance machine. Second, they allow very fast machine movement. It is not unusual to see linear motor machines rated at over 3,000 IPM (75 meters per minute) traverse rates. Third, since they are mounted directly to the moving and stationary components of a machine (i.e. bed and saddle) they simplify machine design by eliminating all of the mechanical components in the drive train such as ballscrew, pillow blocks, bearings, couplings, etc. Fourth, they have no contacting components, which translates into zero motor wear. There are a few disadvantages. First is higher cost, but that is somewhat offset
with the simpler machine design. Second, careful consideration should be made before placing a linear motor machine in a cast iron machining environment. The magnetic attraction of cast iron dust to the motor may be problematic for some machine designs.

**SPINDLE DRIVES**

CNC machine spindles are generally driven by AC or DC variable-speed drives. These drives are well suited for machinery that experience frequent starts, stops, and reversals in harsh manufacturing environments. AC drives are excellent for precise speed matching where several motors (multi-spindle applications) are required. AC and DC variable-speed motors can deliver considerable levels of torque at relatively high rotational speeds.

The CNC system sends a command signal to the drive amplifier, which controls the motor. Modern amplifiers are digital, microprocessor-based controllers that can increase motor performance through the use of real-time monitoring and control.

The same technology discussed for axis servo systems have been applied to spindle drives. Drive capabilities now allow full servo control of spindle rotation such that a 20,000-RPM spindle can also be used as a low-rpm, high-resolution rotary axis. This enables contouring using the spindle as a full contouring rotary axis (i.e. c-axis). This extends the machining capability to include benefits such as:

- Rigid tapping.
- Spindle contouring.
- Part exchange (i.e. robot or lathe sub-spindle).
- Complex milling on a lathe with motorized tools.

**SPINDLE MOTORS**

Spindle motors can be of two types:
1) Integral and
2) External (reference Figure 25).
Integral (Configuration #1) where the rotor of the electric motor becomes the main spindle shaft. The integral design allows for reduced vibration and noise while increasing top rpm and efficiency. Integral designs do not have gear trains. Therefore, the design is limited to a single speed range and horsepower/torque characteristics. The major application limitation is that these units do not develop full horsepower until higher rpm levels. Some designs use dual winding motors to increase the speed range, thus creating the effect of an “electronic gearbox”. This design can be seen in configuration #1 in Figure 28.

a. Benefits – Eliminates gears, belts and pulleys and the associated power loss and vibration. Smaller machine footprint. Higher RPM’s are normally possible.

b. Limitations - Heat buildup in spindlehead. Repair or replacement of spindle motor can be harder. With no gear ranges, lack of full HP rating at lower RPM’s.
• **External – Direct In-Line Coupling** (Configuration #2) where the motor drives the main spindle shaft by means of a directly-coupled external motor.
  a. **Benefits** – Eliminates gears, belts and pulleys and the associated power loss and vibration. Motor repair or replacement is easy.
  b. **Limitations** – Coupling misalignment. With no gear ranges, lack of full HP rating at lower RPM’s.

• **External – Belt System** (Configuration #3) where the motor drives the main spindle shaft by means of a belt and pulley to an external motor.
  a. **Benefits** – Eliminates gears and the associated power loss and gear vibration. Motor repair or replacement is easy.
  b. **Limitations** – Belt stretch, wear and vibration. Some systems are a single gear range while others offer multiple gear ranges through a variable pulley scheme.

• **External – Gear System** (Configuration #4) where the motor drives the main spindle shaft by means of gears and shafts to an external motor.
  a. **Benefits** – Provides a wider speed range and full horsepower at lower RPM’s. Gear ranges can accommodate a wide range of machining applications.
  b. **Limitations** – Some power loss and vibration.

**SPINDLE PERFORMANCE CURVES**

Matching the correct spindle performance with a specific application is very important. Speed, torque and power curves should be reviewed to verify that the necessary torque (pound-feet or Newton-meter) and power (horsepower or kilowatts) are suitable for the intended machining operations.

Over the machine’s total rated speed range the spindle has two distinct performance characteristics. From “minimum speed” to “base speed” most machine tools operate in a “constant torque” range. From “base speed” to “maximum rated speed” they operate in a “constant power” range.

• **Minimum speed** – the lowest controllable spindle speed in rpm. While the spindle can be programmed to go slower than this rating, it is unpredictable in its actual speed. Some machines have the capability to control minimum speed all the way to zero.

• **Base speed** – This is the speed at which the spindle will begin to output at rated power (horsepower or kilowatts).
- **Maximum Rated Speed** – Generally, this is the maximum spindle speed shown in the machine’s specifications. While the spindle may be able to run at speeds higher than the top rated speed, the power will begin to drop.

Different machining operations have different torque and power needs. For instance, a large diameter spade drill or milling cutter would require high torque values. If a customer buys the machine and later finds that they cannot obtain the needed torque value at the speed (rpm) best suited for their workpiece and tooling they are going to be very upset. Have your application engineer or the customer’s technical staff review the machine’s performance curves prior to the final machine purchase.

Another example is when a machining center is sold with a single gear range, typical of an integral spindle. It may not reach rated horsepower until 1,000 rpm or higher. If the customer needs to cut cast iron at rated horsepower at less than 1,000 rpm they are going to be unsatisfied with the machine tool’s performance. This machine may be better suited for machining light metals such as aluminum. Light metals are normally machined at higher speeds where full rated horsepower will be available.

Another potential misapplication deals with machine tools featuring gear ranges. Some machines are designed with gear ranges spread out over a wide speed range. As seen in Figure 30, if the ranges are spread too far the gear ranges will not overlap and “power notches” will be present. If the customer’s workpiece has a critical machining operation requiring specific speed and power values that occur in a “power notch” the
machine will not be able to perform adequately. Speeds and feeds will have to be adjusted to non-optimal settings and could substantially lower productivity. However, if the gear ranges do overlap with little or no power notching, the geared machine may be a good bet for heavy milling such as steel or cast iron that require full power at low speeds.

Overall, the ideal general-purpose machine would give maximum torque and power over the widest speed range. Additional information on spindle drive systems can be found in Section 10.5 and 10.6.

NOTES
7.7 PLC & Sensor Technology

PLC TECHNOLOGY

A Programmable Logic Controller (PLC) is used in most machine tools to customize the CNC for a specific machine application. In particular it implements the auxiliary functions on the machine like the automatic tool changer, coolant and operators panel. In the basic form, the PLC reads inputs from the CNC, switches and sensors, and writes to output inside the CNC, output devices such as relays, solenoids and lamps.

Output Devices:
1. Motor starters
2. Solenoids
3. Relays
4. Indicators lights & displays
5. Alarms
6. Simple motion control

Input Devices:
1. Limit switches
2. Push buttons & operator switches
3. Counters
4. Relays
5. Simple motion control

The PLC logic engine may be integrated within the CNC, either with a dedicated processor or using the main CNC CPU. Some machine tools use a commercially available external PLC.

A PLC includes a central processing unit (CPU), memory and an input/output modules. In general PLC use a CPU optimized to make binary decisions and communicate with common input/output devices found on machine tools. Most PLCs do not need to make the complex contouring and mathematics functions of a CNC. PLC can be programmed using traditional relay ladder diagrams or simple symbolic languages. However, advanced CNC/PLC interfaces may include window functions to initiate complex procedures such as move axes or initiate the execution of a CNC subroutine.

A PLC may serve as the main controller on simple production machines, transfer lines or those not requiring complex interpolated axis positioning. High-end PLC technology is powerful with the incorporation of state-of-the-art microprocessors and the ability to manage complex multi-cycle feedrate motion control typical of that found on transfer type machines.
PLC’s can usually be interfaced into a host computer for higher-level factory automation control. Also like the open architecture, PC-based CNC, there are soft PLC’s where all of the logic is programmed and carried out within a standard personal computer. Plug in input/output boards and the soft PLC is complete.

**SENSOR TECHNOLOGY**

Advanced machine tool automation would not be possible without sensors. A sensor is a device that responds to a physical stimulus (pressure, heat, motion, magnetism, sound, etc.) and transmits the information back to a controlling device. On machine tools, sensors and probes monitor workpiece, tooling, process, or machine conditions. This information is used to make adjusts and corrections that optimize the machining process.

**Workpiece monitoring** includes 1) part counting, 2) part or pallet identification, 3) setup alignment with respect to part datums, 4) evaluating stock conditions (excess or insufficient) on the rough part prior to machining, 5) depth-of-cut calculations, 6) machining time, 7) in-process gauging and 8) post-machining gauging. **Machine monitoring** includes 1) feed-force data from spindle and axis motors, 2) adaptive control, 3) collision protection, 4) thermal expansion detection and 5) machine deflection. **Tool monitoring** includes 1) tool identification, 2) tool length and diameter offsets, 3) tool wear offsets and, 4) tool breakage detection.

Sensor and probes vary widely in their design and application. One common type is the **Touch-trigger probe** that detects dimensional data. They can be mounted to the part, the pallet, the machine, or held in a tool holder. The tool holder approach is convenient on machining centers as it allows the machine access to the probe through the tool changer. Touch probes are used to detect tool length and diameter offsets, part and pallet dimensions, stock conditions, part identification through probing dimensions and features and machine dimensional changes (ex: spindle growth). The signal can be sent via hardwire, radio frequency, or laser communication to the control pickup device. This type of probe is extremely versatile and reliable.

**Piezoelectric strain sensors** use the electro-mechanical properties of a quartz crystal to generate an electrical signal whenever the crystal is physically distorted. This type of sensor is ideal for feed-force and deflection detection. For example, mounted to a bearing housing it will generate an electrical signal when the housing experiences the normal distortion of machine forces. This signal can be converted to indicate the force experienced at the housing. These are used in adaptive control systems, machining force calculations, and tool wear schemes.
**Vision sensors** use sophisticated imaging processors to detect shape and size characteristics. They are used in barcode detection as well as high-end part identification using feature and image comparison routines.

Many other sensor types are used on machine tools. Acoustic sensors can detect vibrations and sounds (ex: tool breakage) while electrical resistance sensors can measure current flow (ex: torque) in a motor circuit. All of these sensors and probes assist in making the automated machine tool more intelligent and productive.

**ADAPTIVE CONTROL**

Adaptive control represents a specialized application involving sensors. By monitoring feed-force levels of both the spindle and axis servo drives an adaptive control unit can optimize the machining parameters, such as spindle speeds and feedrates, while in production (real-time mode). By measuring motor torque and making immediate adjustments, the following benefits are obtained: 1) motor overload protection, 2) tool life optimization, 3) maximum stock removal rates, 4) machine collision protection, 5) compensation for operator inexperience and 6) compensation for workpiece variances. The system reacts so quickly to the feedback from the sensors that the machine, in some conditions, can rapid traverse into a workpiece and automatically change to machining feedrates without damage to the workpiece or tooling.

**NOTES**
7.8 Review Questions

These review questions are provided for study purposes only and are not on the CMTSE certification exam. Correctly answering these questions does not guarantee a passing exam grade.

1. To mill threads in a large hole using a 3-axis CNC machining center, what type of interpolation would be best?
   1. Circular
   2. Helical
   3. Parabolic
   4. Linear

2. The primary function of a linear encoder is to:
   1. assist in accurate part inspection.
   2. register the orientation of a rotary table.
   3. provide position feedback of the machine slide.
   4. record and identify tools in the tool changer.

3. A closed loop system on a CNC machine refers primarily to the:
   1. microprocessor.
   2. graphic display system.
   3. punched tape or floppy disk input.
   4. position feedback.

4. CNC controls are an improvement over "NC" controls primarily because CNC has:
   1. the ability to communicate with programmable logic controls (PLC).
   2. memory and programmable computing power.
   3. laptop interface and networking capability.
   4. much faster tape reading capability.

5. To machine a part designed using computer-aided design (CAD) the next step is to:
   1. generate tool path data in CAM based on the CAD file.
   2. transfer the CAD file to a laptop.
   3. reproduce the drawing and add dimension data for the machine operator.
   4. provide the shop supervisor with the drawing and await approval.
6. Which type of interpolation is used for a diagonal straight-line motion?
   1. Linear interpolation
   2. Helical interpolation
   3. Circular interpolation
   4. Cubic interpolation

7. When programming a CNC, "G" codes normally handle which one of the following?
   1. Tool changes
   2. Macros
   3. Preparatory instructions
   4. Slide position data

8. How many axes of simultaneous CNC control are required for circular interpolation on a 3-axis machine?
   1. 1
   2. 1 ½
   3. 2
   4. 3

9. The use of conversational programming is clearly evident when the operator:
   1. uses a microphone to vocally program the control.
   2. recalls previous program data stored in the control memory.
   3. downloads program information from another control.
   4. responds to questions and prompts on the screen of the control.

10. How many simultaneous axes of control are required for milling a thread using helical interpolation?
    1. 3
    2. 4
    3. 5
    4. 6
ANSWERS TO REVIEW QUESTIONS

1. (2)
2. (3)
3. (4)
4. (2)
5. (1)
6. (1)
7. (3)
8. (3)
9. (4)
10. (1)