UNIT -III

MODELLING FACILITIES DESIRED

The total modelling facilities that one would look for in any system can be broadly categorized as follows:

The geometric modelling features.

- ✓ The editing or manipulation features.
- ✓ The display control facilities.
- ✓ The drafting features.
- ✓ The programming facility.
- ✓ The analysis features.
- ✓ The connecting features

A. Geometric Modelling Features

The various geometric modelling and construction facilities that one should expect to have in any good system are as follows:

1. Various features to aid geometric construction methods, such as Cartesian and polar coordinates, absolute and incremental dimensions, various types of units, grid, snap, object snap, layer, etc.
2. All 2D analytical features, such as points, lines, arcs, circles, conics, splines, fillets, chamfers, etc. In each of these features, various constructional features including interactive and dynamic dragging facilities.
3. Majority of the 3D wireframe modelling facilities including 3D lines, 3D faces, ruled surfaces, linear sweep from 2D topology with any sweep direction, rotational sweep, and tapered sweep. General sweep with twist. Revolving about an axis with axis or radial offset for generating helical or spiral shapes.
4. Solid modelling with various basic primitives such as block, cylinder, sphere, cone, prism, torus, pyramid, quadrilateral, along with the ability to apply the boolean operators on any solid that can be constructed using the other techniques available in the modeler.
5. Skinning around regular and arbitrary surfaces. Profiles (cross-sections),both analytical and arbitrary placed across any 3D curve.
6. Sculptured surfaces of the various types like Bezier, Coons and other free form

surfaces.

7. Comprehensive range of transformation facilities for interactively assembling the various solid models generated by the modeler with features such as surface filleting and trimming.

B. Editing or Manipulation Features

1. Transformations such as move, copy, rotate, scale, elongate or compress, b mirror or to any arbitrary coordinate frame.
2. The editing features used to alter the already drawn geometric entities, such as stretching, trimming or trimming to any intersection, delete or erase, undo or redo.
3. Symbols in drawing refer to often-repeated set in a number of drawings, which may consist of a number of geometric entities that are grouped together and stored as a symbol. This symbol can be recalled at any scale, at any angle or exploded if necessary to treat all of them as separate entities.

C. Display Control Facilities

In this range of features are all the facilities needed for interacting with the modelling system so as toobtain the necessary feedback at the right time during the niodelling stage. The facilities required are:

1. **Window** - to identify a set of entities for any possible display or editingfunction.
2. **Zoom** - to change the scale of display of the image selected on the screen.
3. **Pan** - to move the image on screen without changing the scale at which the drawing is displayed on the screen.
4. **Hidden** - to remove hidden lines or hidden surfaces for viewing the geometry inproper form
5. **Shading** - to show the 3D view of the image on screen complete with the lightsource location and the resultant light and shade as it appears on the image.

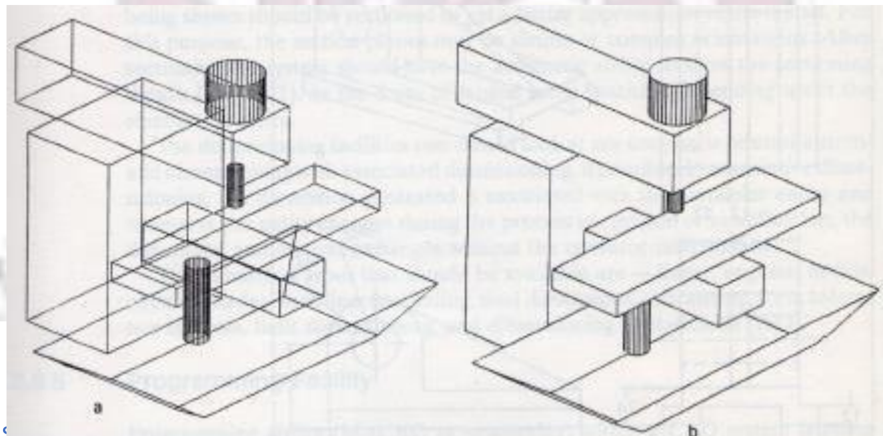
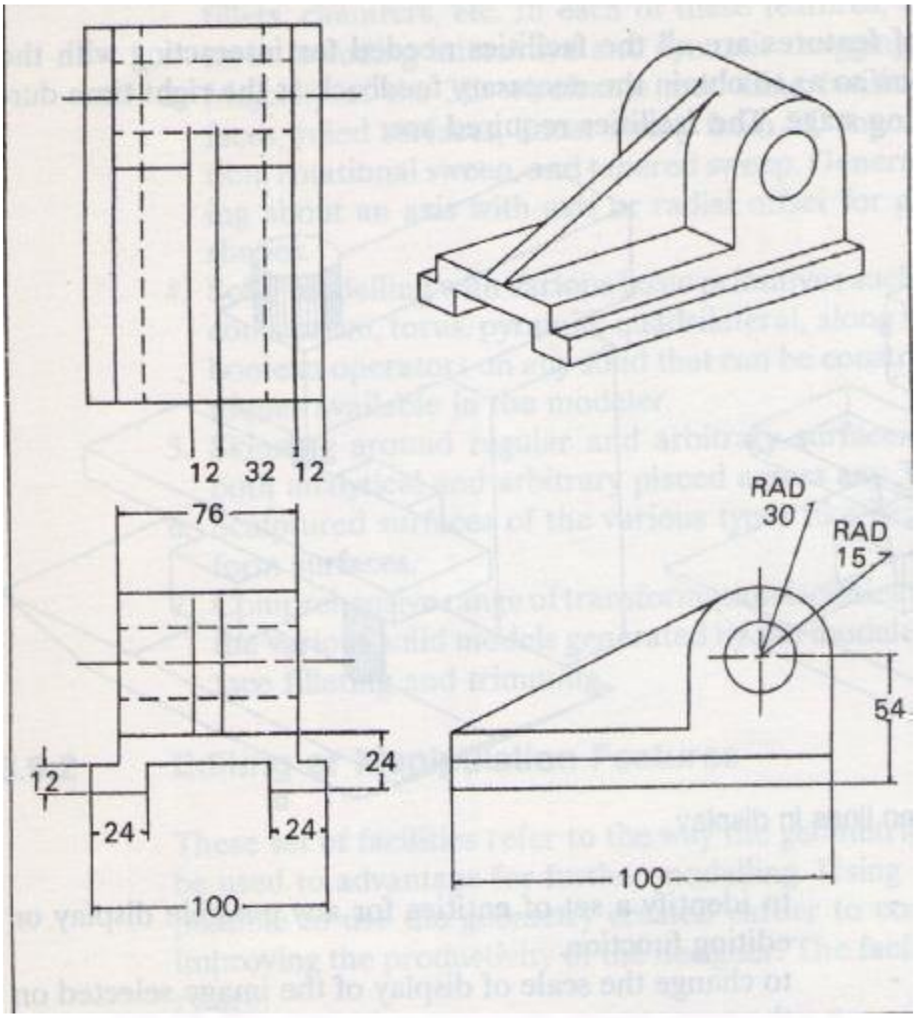


Fig. 3.29 Elimination of hidden lines in display

D. Drafting Features

These facilities refer to the way the model developed can be utilized for the purpose of transmitting the information in hard copy form for other applications, such as part prints onto the shop floor, or maintenance manuals for the equipment. A really large range of facilities are required in this particular category, and it is sometimes treated as a separate module in the modeling system.

A large number of types of views should be obtained from the solid model of the geometry stored in the data base. The types of views required may be , as for display functions, such as perspective views, orthographic views (Fig.), isometric views, and axonometric views.



E. Programming Facility

Programming ability (MACRO programming) within a CAD system is going to be a very useful feature. It is well-known that not all kinds of facilities would be available in any general purpose CAD system. Therefore, it is necessary that the CAD system would have to be customised for a given range of application processes specific to the company. For this purpose, if a programming facility exists in a CAD system, it is possible to program specifically for an application, making use of all the features available in the system for either modelling or for any specific application based on the information generated during the modelling. Some such examples are, the GRIP in Unigraphics and GLUE in CAM-X. The availability of such a program helps the user to input the least amount of information for any required design, if the application programs are written well using the programming language.

F. Analysis Features

In this range, the kind of analysis facilities that are required to be carried on the product models being generated should be considered. The simplest kind to the most sophisticated features may be available under this category. The simplest facilities may be calculating perimeter, area, volume, mass, centre gravity, moment of inertia, radius of gyration, etc.

G. Connecting Features

Modelling is only the start of the complete process of a product evolution, and as such the data generated is used directly by the other systems. It is, therefore, necessary that the internal data format in which the data is stored by the modelling system should be well documented and should also have very good connectivity (data interfacing) with other allied modules. Ideally, an integrated data base structure would be useful wherein all the various modules share the common data base. However, this would only be possible if all the modules are developed at a single developer as in the case of Computer vision or Unigraphics for CAD/CAM integration.

Numerical Control

Many of the achievements in computer-aided design and manufacturing have a common origin in numerical control (abbreviated NC). The conceptual framework established during the development of numerical control is still undergoing further refinement and enhancement in today's CAD/CAM technology.

Numerical control can be defined as a form of programmable automation in which the process is controlled by numbers, letters, and symbols. In NC, the numbers form a program of instructions designed for a particular work part or job. When the job changes, the program of instructions is changed. This capability to change the program for each new job is what gives NC its flexibility. It is much easier to write new programs than to make major changes in the production equipment.

NC technology has been applied to a wide variety of operations, including drafting, assembly, inspection, sheet metal press working, and spot welding. However, numerical control finds its principal applications in metal machining processes. The machined work parts are designed in various sizes and shapes, and most machined parts produced in industry today are made in small to medium-size batches. To produce each part, a sequence of drilling operations may be required, or a series of turning or milling operations. The suitability of NC for these kinds of jobs is the reason for the tremendous growth of numerical control in the metalworking industry over the last 25 years.

BASIC COMPONENTS OF AN NC SYSTEM

An operational numerical control system consists of the following three basic components:

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

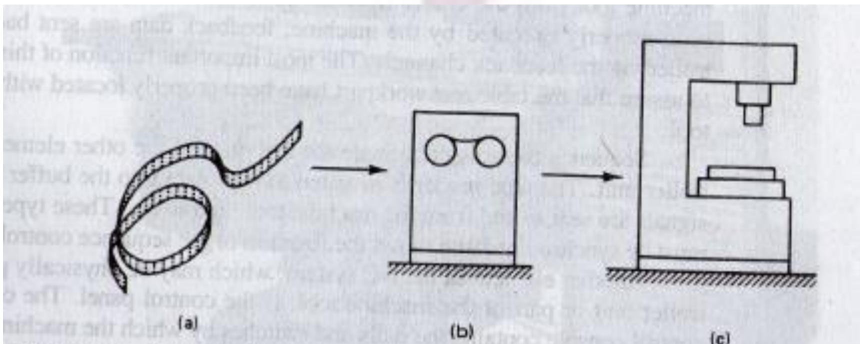


FIGURE 7.1 Three basic components of a numerical control system: (a) program of instruction; (b) controller unit; (c) machine tool.

Program of instructions

The program of instructions is the detailed step-by-step set of directions which tell themachine tool what to do. It is coded in numerical or symbolic form on some type of input medium that can be interpreted by the controller unit. The most common input medium today is l-in.-wide punched tape. Over the years, other forms of input media have been used, including punched cards, magnetic tape, and even 35-mm motion picture film.

There are two other methods of input to the NC system which should be mentioned. The first is by manual entry of instructional data to the controller unit. This method is called manual data input, abbreviated MDI, and is appropriate only or relatively simple jobs where the order will not be repeated.

The second other method of input is by means of a direct link with a computer. This is called direct numerical control, or DNC.

The program of instructions is prepared by someone called a part programmer. The programmer's job is to provide a set of detailed instructions by which the sequence of processing steps is to be performed. For a machining operation, the processing steps involve the relative movement between the cutting tool and the workpiece.

Controller unit

The second basic component of the NC system is the controller unit. This consist of the electronics and hardware that read and interpret the program of instruction and convert it into mechanical actions of the machine tool. The typical elements of a conventional NC controller unit include the tape reader, a data buffer, signal output channels to the machine tool, feedback channels from the machine tool, and the sequence controls to coordinate the overall operation of the foregoing elements. It should be noted that nearly all modern NC systems today are sold with microcomputer as the controller unit. This type of NC is called computer numerical control (CNC

The tape reader is an electromechanical device for winding and reading the punched tape containing the program of instructions. The data contained on the tape are read into the data buffer. The purpose of this device is to store the input instructions in logical blocks of information. A block of information usually represents one complete step in the sequence of processing elements. Another element of the NC system, which may be physically part of the controller unit or part of the machine tool, is the control panel. The control panel or control console contains the dials and switches by which the machine operator runs the NC system.

Machine tool or other controlled process

The third basic component of an NC system is the machine tool or other controlled process. It is the part of the NC system which performs useful work. In the most common example of an NC system, one designed to perform machining operations, the machine tool consists of the worktable and spindle as well as the motors and controls necessary to drive them. It also includes the cutting tools, work fixtures, and other auxiliary equipment needed in the machining operation.

THE NC PROCEDURE

To utilize numerical control in manufacturing, the following steps must be accomplished;

1. Process planning.

The engineering drawing of the work part must be interpreted in terms of the manufacturing processes to be used. This step is referred to as process planning and it is concerned with the preparation of a route sheet. The route sheet is a listing of the sequence of operations which must be performed on the work part. It is called a route sheet because it also lists the machines through which the part must be routed in order to accomplish the sequence of operations.

2. Part programming.

A part programmer plans the process for the portions of the job to be accomplished by NC. Part programmers are knowledgeable about the machining process and they have been trained to program for numerical control. They are responsible for planning the sequence of machining steps to be performed by NC and to document these in a special format. There are two ways to program for NC:

- **Manual part programming**
- **Computer-assisted part programming**

In **manual part programming**, the machining instructions are prepared on a form called a part program manuscript. The manuscript is a listing of the relative cutter/workpiece positions which must be followed to machine the part. In **computer-assisted part programming**, much of the tedious computational work required in manual part programming is transferred to the computer. This is especially appropriate for complex work- piece geometries and jobs with many machining steps

3. Tape preparation.

A punched tape is prepared from the part programmer's NC process plan. In **manual part**

programming, the punched tape is prepared directly from the part program manuscript on a typewriter like device equipped with tape punching capability. In **computer-assisted part programming**, the computer interprets the list of part programming instructions, performs the necessary calculations to convert this into a detailed set of machine tool motion commands, and then controls a tape punch device to prepare the tape for the specific NC machine.

4. Tape verification.

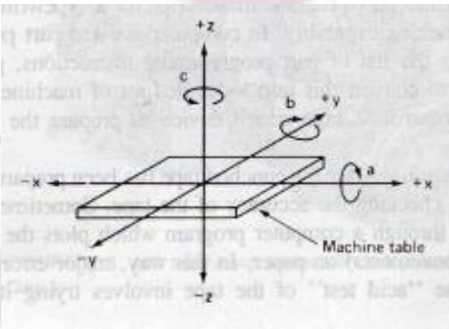
After the punched tape has been prepared, a method is usually provided for checking the accuracy of the tape. Sometimes the tape is checked by running it through a computer program which plots the various tool movements (or table movements) on paper..In this way, major errors in the tape can be discovered. The **"acid test"** of the tape involves trying it out on the machine tool to make the part. A foam or plastic material is sometimes uses this tryout

5. Production

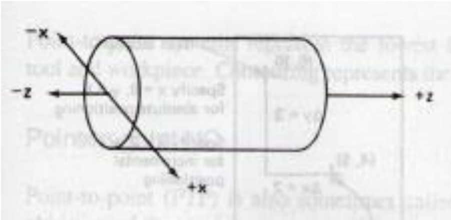
The final step in the NC procedure is to use the NC in production. This involves ordering the raw work parts, specifying and preparing tooling and any special fixturing that may be required, and setting up the machine tool for the job. The machine tool operator's function during product to load the raw work part in the machine and establish the starting position of cutting tool relative to the workpiece. The NC system then takes over machines the part according to the instructions on tape.

NC COORDINATE SYSTEMS

Two axes, x and y, are defined in the plane of the table, as shown in Fig. The z axis is perpendicular to this plane and movement in the z direction is controlled by the vertical motion of the spindle. The positive and negative directions of motion of tool relative to table along these axes. NC drill presses are classified as either two-axis or three-axis machine depending on whether or not they have the capability to control the z axis.



For turning operations, two axes are normally all that are required to command the movement of the tool relative to the rotating workpiece. The z axis is the axis of rotation of the workpart, and x axis defines the radial location of the cutting tool.



The purpose of the coordinate system is to provide a means of locating the tool in relation to the workpiece. Depending on the NC machine, the part programmer may have several different options available for specifying this location.

Fixed zero and floating zero

NC machines have either of two methods for specifying the zero point-

The first possibility is for the machine to have a **fixed zero**. In this case, the origin is always located at the same position on the machine table. Usually, that position is the southwest corner (lower left-hand corner) of the table and all tool locations will be defined by positive x and y coordinates.

The second and more common feature on modern NC machines allows the machine operator to set the zero point at any position on the machine table. This feature is called **floating zero**. The part programmer is the one who decides where the zero point should be located. The decision is based on part programming convenience.

Absolute positioning and incremental positioning

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

If a hole is to be drilled at a spot that is 8 in. above the x axis and 6 in. to the right of the y axis, the coordinate location of the hole Would be specified as x = +6.000 and y = +8.000.

By contrast, incremental positioning means that the next tool location must be defined with reference to the previous tool location. If in our driling example, suppose that the previous hole had been drilled at an absolute position of x = +4.000 and y = +5.000. Accordingly, the incremental position instructions would be specified as x = +2.000 and y = +3.000 in order to move the drill to the desired spot.

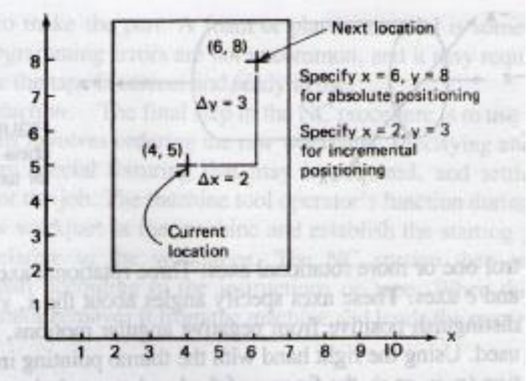


FIGURE 7.6 Absolute versus incremental positioning.

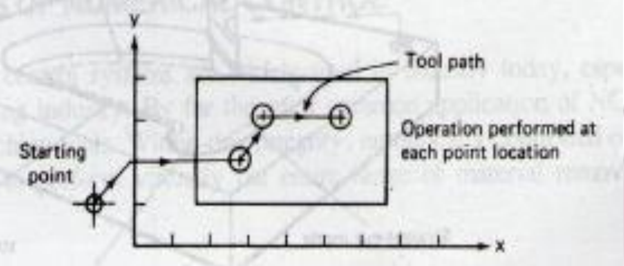
NC MOTION CONTROL SYSTEMS

In order to accomplish the machining process, the cutting tool and workpiece must be movedrelative to each other. In NC, there are three basic types of motion control systems:

1. Point-to-point
2. Straight cut
3. Contouring

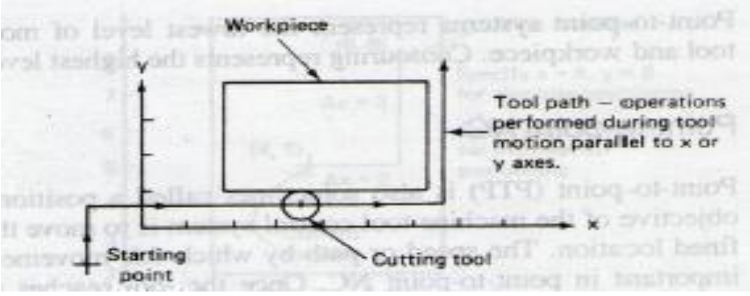
1. Point-to-point NC

Point-to-point (PTP) is also sometimes called a **positioning system**. In PTP, the objective of the machine tool control system is to move the cutting tool to a predefined location. The speed or path by which this movement is accomplished is not important in point-to-point NC. Once the tool reaches the desired location, the machining operation is performed at that position.



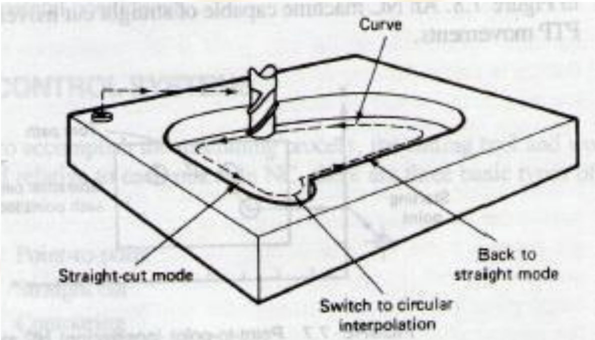
2. Straight-cut NC

Straight-cut control systems are capable of moving the cutting tool parallel to one of the major axes at a controlled rate suitable for machining. It is therefore appropriate for performing milling operations to fabricate workpieces of rectangular configurations. With this type of NC system it is not possible to combine movements in more than a single axis direction.

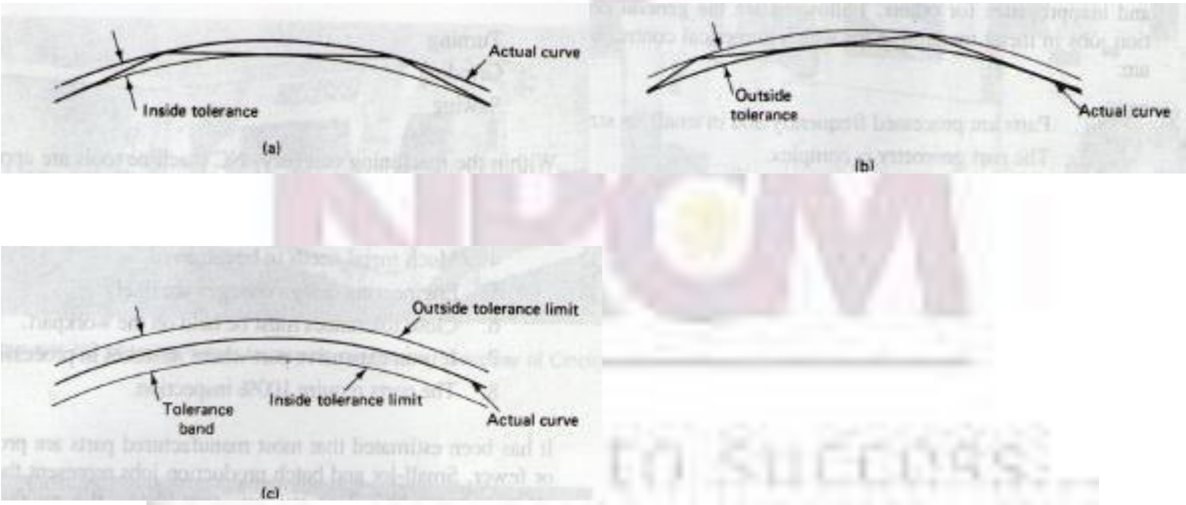


3. Contouring NC

Contouring is the most complex, the most flexible, and the most expensive types machine tool control. It is capable of performing both PTP and straight-cut operations. In addition, the distinguishing feature of contouring NC systems is their capacity for simultaneous control of more than one axis movement of the machine tool. The path of the cutter is continuously controlled to generate the desired geometry of the workpiece. For this reason, contouring systems are also called **continuous-path NC systems**.



In order to machine a curved path in a numerical control contouring system the direction of the feed rate must continuously be changed so as to follow the path. This is accomplished by breaking the curved path into very short straight-line segments that approximate the curve.



APPLICATIONS OF NUMERICAL CONTROL:

Numerical control systems are widely used in industry today, especially in the metalworking industry. By far the most common application of NC is for metal cutting machine tools. Within this category, numerically controlled equipment has been built to perform virtually the entire range of material removal processes, including:

- Milling
- Drilling and related processes
- Boring
- Turning
- Grinding

NC machine tools are appropriate for certain jobs and inappropriate for others. Following are the general **characteristics of production jobs in metal machining** for which numerical control would be most appropriate;

1. Parts are processed frequently and in small lot sizes.
2. The part geometry is complex.
3. Many operations must be performed on the part in its processing
4. Much metal needs to be removed.
5. Engineering design changes are likely.
6. Close tolerances must be held on the workpart.
7. It is an expensive part where mistakes in processing would be costly.
8. The parts require 100% inspection.

Advantages of NC

1. Reduced nonproductive time. Numerical control has little or no effect on the basic metal cutting (or other manufacturing) process. However, NC can increase the proportion of time the machine is engaged in the acutal process. It accomplishes this by means of fewer setups, less time in setting up, reduced workpiece handling time, automatic tool changes on some machines, and so on.
2. Reduced fixturing. NC requires fixtures which are simpler and less costly to fabricate because the positioning is done by the NC tape rather than the
3. Reduced manufacturing lead time. Because jobs can be set up more quickly with NC and fewer setups are generally required with NC, the lead time to deliver a job to the customer is reduced.

4. Greater manufacturing flexibility. With numerical control it is less difficult to adapt to engineering design changes, alterations of the production schedule changeovers in jobs for rush orders, and so on.
5. Improved quality control. NC is ideal for complicated workparte where the chances of human mistakes are high. Numerical control produces parts with greater accuracy, reduced scrap, and lower inspection requirements.
6. Reduced inventory. Owing to fewer setups and shorter lead times with numerical control, the amount of inventory carried by the company is reduced.
7. Reduced floor space requirements. Since one NC machining center can often accomplish the production of several conventional machines, the amount of floor space required in an NC shop is usually less than in a conventional shop.

Disadvantages of NC

1. Higher investment cost. Numerical control machine tools represent a more sophisticated and complex technology. This technology costs more to buy than its non-NC counterpart.
2. Higher maintenance cost. Because NC is a more complex technology and because NC machines are used harder, the maintenance problem becomes more acute.
3. Finding and/or training NC personnel. Certain aspects of numerical control shop operations require a higher skill level than conventional operations Part programmers and NC maintenance personnel are two skill areas where available personnel are in short supply. The problems of finding, hiring, and training these people must be considered a disadvantage to the NC shop.

NC part programming

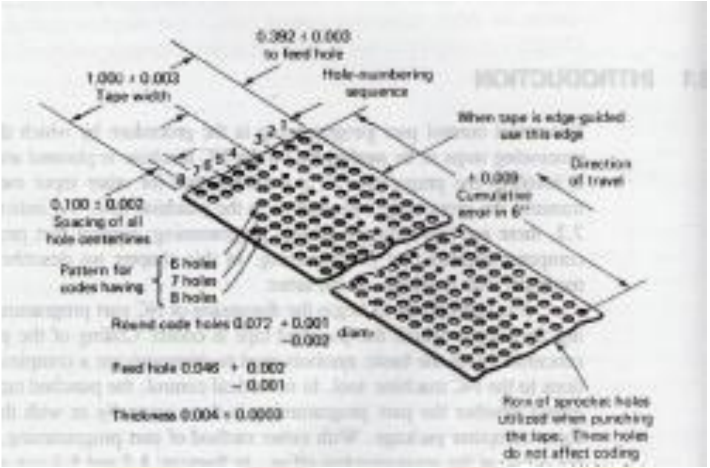
Numerical control part programming is the procedure by which the sequence of processing steps to be performed on the NC machine is planned and documented. It involves the preparation of a punched tape (or other input medium) used to transmit the processinginstructions to the machine tool.

It is appropriate to begin the discussion of NC part programming by examining the way in which the punched tape is coded. Coding of the punched tape is concerned with the basic symbols used to communicate a complex set of instructions to the NC machine tool. In numerical control, the punched tape must be generated whether the part programming is done manually or with the assistance of some computer package. With either method of part programming, the tape is the net result of the programming effort.

THE PUNCHED TAPE IN NC

The part program is converted into a sequence of machine tool actions by means of, the input medium, which contains the program, and the controller unit, which interprets the input medium. The controller unit and the input medium must be compatible. That is, the input medium uses coded symbols which represent the part program, and the controller unit mustbe capable of reading those symbols.

The most common input medium is punched tape. The tape has been standardized so that tape punchers are manufactured to prepare the NC tapes, and tape readers (part of the controller unit) can be



manufactured to read the tapes. The punched tape used for NC is 1 in. wide. there are eight regular columns of holes running in the lengthwise direction of the tape. There is also a ninth column of holes between the third and fourth regular columns. However, these are smaller and are used as sprocket holes for feeding the tape.

There are two basic methods of preparing the punched tape;

- The first method is associated with manual part programming and involves the use of a typewriter like device. The operator types directly from the part programmer's handwritten list of coded instructions. This produces a typed copy of the program as well as the punched tape.

- The second method is used with computer-assisted part programming. By this approach, the tape is prepared directly by the computer using a device called a tape punch.

By either method of preparation, the punched tape is ready for use. During production on a conventional NC machine, the tape is fed through the tape reader once for each workpiece. It is advanced through the tape reader one instruction at a time. While the machine tool is performing one instruction, the next instruction is being read into the controller unit's data buffer. This makes the operation of the NC system more efficient. After the last instruction ha$ been read into the controller, the tape is rewound back to the start of the program to be ready for the next workpart.

A binary digit is called a bit. It has a value of 0 or 1 depending on the absence or presence of a hole in a certain row and column position on the tape. (Columns of hole positions run lengthwise along the tape. Row position run across the tape.) Out of a row of bits, a character is made. A character is a combination of bits which represents a letter, number, or other symbol. A word is a collection of characters used to form part of an instruction. Typical NC words are x position, y position, cutting speed, and so on.

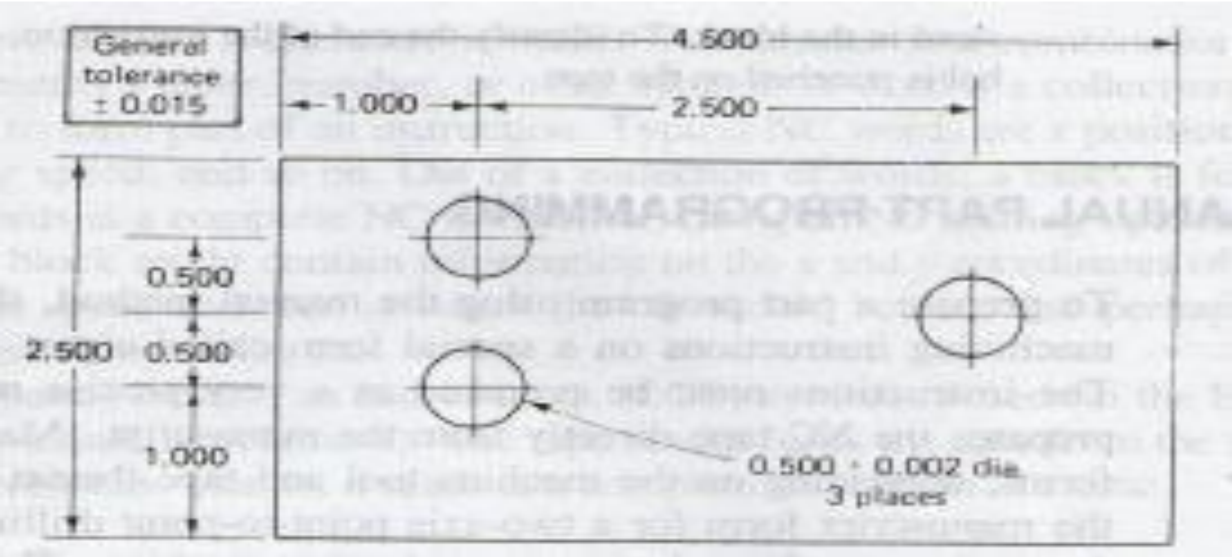
NC words

Following is a list of the different types of words in the formation of a block.

- SEQUENCE NUMBER (n-words): This is used to identify the block.
- PREPARATORY WORD (g-words): This word is used to prepare the controller for instructions that are to follow.
- COORDINATES (x-, y-, and z-words): These give the coordinate positions of the tool.
- FEED RATE (f-word): This specifies the feed in a machining operation.
- CUTTING SPEED (s-word): This specifies the cutting speed of the process, the rate& which the spindle rotates.
- TOOL SELECTION (t-word): This word would be needed only for machines with a too turret or automatic tool changer.
- MISCELLANEOUS FUNCTION (m-word): The m-word is used to specify certain miscellaneous or auxiliary functions which may be available on the machine tool.

MANUAL PART PROGRAMMING

To prepare a part program using the manual method, the programmer writes the machining instructions on a special form called a part programming manuscript. The instructions must be prepared in a very precise manner because the typist prepares the NC tape directly from the manuscript. Manuscripts come in various forms, depending on the machine tool and tape format to be used. Manual programming jobs can be divided into two categories: point-to- point jobs and contouring jobs. Except for complex workparts with many holes to be drilled, manual programming is ideally suited for point-to-point applications. On the other hand, except for the simplest milling and turning jobs, manual programming can. Become quite time consuming for applications requiring continuous-path control of the tool.



NC part programming manuscript
Two axis point-to-point drill press
tab sequential format

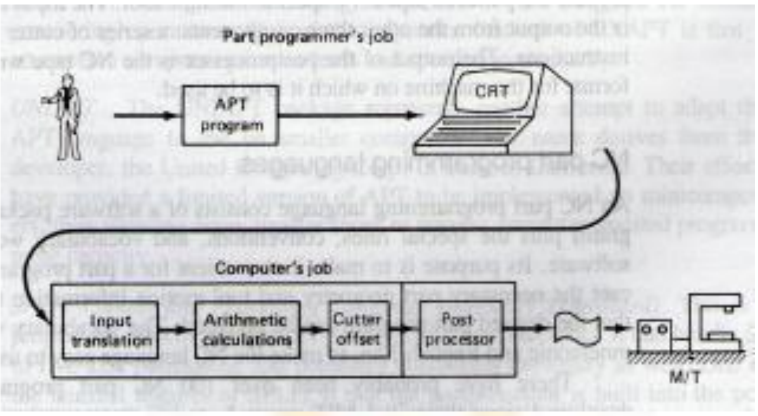
Part No. EXAMPLE 8.2 Date 4/4/79

Part Name HOLE PLATE Prepared by MPG

Seq. No.	Tab EOB	x-COORD	Tab EOB	y-COORD	Tab EOB	Feed	Tab EOB	Speed	Tab EOB	m-WORD	Tab EOB	Comments
00	TAB	0.0	TAB	0.0	EOB							ZERO
RWS												
01	TAB	1.0	TAB	2.0	TAB	3.55	TAB	592	TAB	13	EOB	DRILL 1
02	TAB		TAB	1.0	EOB							DRILL 2
03	TAB	3.5	TAB	1.5	EOB							DRILL 3
04	TAB	-1.0	TAB	3.0	TAB		TAB		TAB	06	EOB	TOOL CHG
05	TAB	3.5	TAB	1.5	TAB	3.82	TAB	382	TAB	13	EOB	REAM 3
06	TAB	1.0	TAB	1.0	EOB							REAM 2
07	TAB		TAB	2.0	EOB							REAM 1
08	TAB	-1.0	TAB	3.0	TAB		TAB		TAB	06	EOB	TOOL CHG
09	TAB		TAB		TAB		TAB		TAB	30	EOB	REWIND
												CHG PART

COMPUTER-ASSISTED PART PROGRAMMING

Many part programming language systems have been developed to perform automatically most of the calculations which the programmer would otherwise be forced todo. This saves time and result in a more accurate and more efficient part program.



Part programmer's job

The computer's job in computer-assisted part programming consists of the following steps:

1. **Input translation**- The part programmer enters the program written in the APT or other language. The input translation component converts the coded instructions contained in the program into computer-usable form, preparatory to further processing.
2. **Arithmetic Calculations**- The arithmetic calculations unit of the system consists of a comprehensive set of subroutines for solving the mathematics required to generate the part surface. These subroutines are called by the various part programming language statements. The arithmetic unit is really the fundamental element in the part programming package. This unit frees the programmer from the time-consuming geometry and trigonometry calculations, to concentrate on the workpart processing.
3. **Cutter offset** - The purpose of the cutter offset computations is to offset the tool path from the desired part surface by the radius of the cutter. This means that the part programmer can define the exact part outline in the geometry statements. Thanks to the cutter offset calculation provided by the programming system, the programmer need not be concerned with this task.
4. **Post processor**- The postprocessor is a separate computer program that has been written to prepare the punched tape for a specific machine tool. The input to the postprocessor is the output from the other three components: a series of cutter locations and other instructions. The output of the postprocessor is the NC tape

written in the correct format for the machine on which it is to be used.

NC part programming languages

An NC part programming language consists of a software package (computer program) plus the special rules, conventions, and vocabulary words for using that software. Its purpose is to make it convenient for a part programmer to communicate the necessary part geometry and tool motion information to the computer so that the desired part program can be prepared.

The following NC languages in current use;

- APT (Automatically Programmed Tools)
- ADAPT (Adaptation of APT)
- EXAPT (Extended subset of APT)
- UNIAPT
- SPLIT (Sundstrand Processing Language Internally Translated)

THE APT LANGUAGE

APT is not only an NC language; it is also the computer program that per forms the calculations to generate cutter positions based on APT statements. APT is a three- dimensional system that can be used to control up to five axes. APT can be used to control a variety of different machining operations. There are over 400 words in the APT vocabulary.

There are four types of statements in the APT language:

1. **Geometry statements.** These define the geometric elements that comprise the workpart. They are also sometimes called definition statements.
2. **Motion statements**. These are used to describe the path taken by the cutting tool.
3. **Postprocessor statements.** These apply to the specific machine tool and control system.They are used to specify feeds and speeds and to actuate other features of the machine.
4. **Auxiliary statements**. These are miscellaneous statements used to identify the part, tool,tolerances, and so on.

Geometry statements

To program in APT, the workpart geometry must first be defined. The tool is subsequently directed to move to the various point locations and along surfaces of the workpart which have been defined by these geometry statements. The definition of the workpart elements must precede the motion statements.

The general form of an APT geometry statement is this: symbol =

geometry type/descriptive data

Example:-

For point	P1 =POINT/5.0,4.0,0.0
Line	L3 = LINE/P3, P4
	Or
	L4 = LINE/P5, PARLEL, L3
Plane	PL1 = PLANE/PI, P4, P5OR
	PL2 = PLANE/P2, PARLEL, PL1
Circle	CI = CIRCLE/CENTER, PI, RADIUS, 5.0

Example:

PO = POINT/0, -1.0,0

P1 = POINT/6.0, 1.125,0P2

= POINT/0, 0, 0

P3 = POINT/6.0, 0, 0 P4

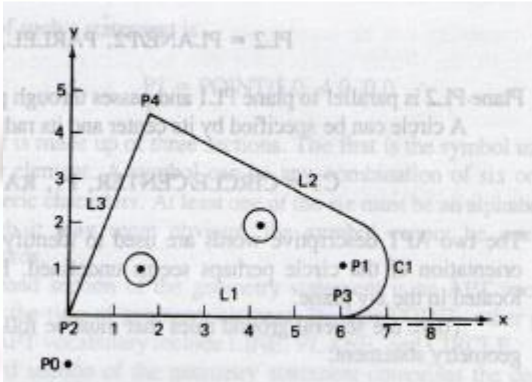
= POINT/1.75, 4.5,0LI

= LINE/P2, P3

C1 = CIRCLE/CENTER, P1, RADIUS, 1.125L2 = LINE/P4, LEFT, TANTO, C1

L3 = LINE/P2, P4

PL1 = PLANE/P2, P3, P4



Motion statements

APT motion statements have a general format, just as the geometry statements do. Thegeneral form of a motion statement is

motion command/descriptive data

Example of a motion statement is

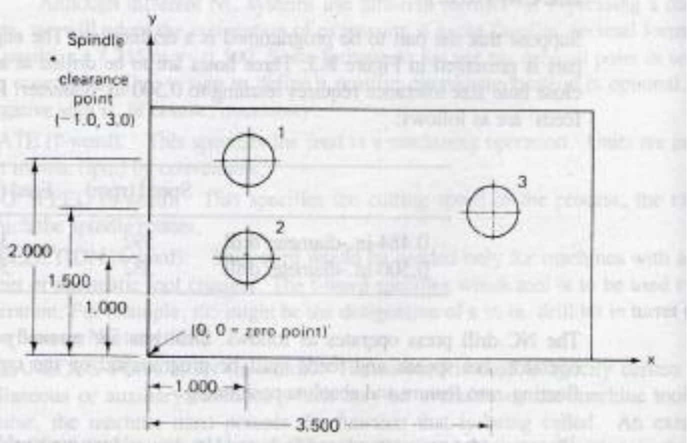
GOTO/P1 FROM/TARG

FROM/-2.0, -2.0, 0.0

The GODLTA command specifies an incremental move for the tool. For example, the statement

GODLTA/2.0, 7.0, 0.0

Example:



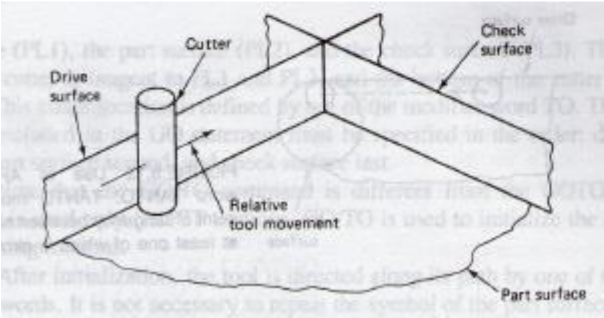
P1 = POINT/1.0.2.0,0 P2 =
POINT/1.0. 1.0,0 P3 =
POINT/3.5, 1.5, 0
P0 =POINT/- 1.0, 3.0, 2.0
FROM/P0
GOTO/P1
GODLTA/0,0,-1.0
GODLTA/0,0,+1.0
GOTO/P2
GODLTA/0,0,-1.0
GODLTA/0,0,+1.0
GOTO/P3
GODLTA/0,0,-1.0
GODLTA/0,0,+1.0
GOTO/P0

This is not a complete APT program because it does not contain the necessary auxiliary and postprocessor statements. However, the statement sequence demonstrates how geometry and motion statements can be combined to command the tool through a series of machining steps.

CONTOURING MOTIONS.

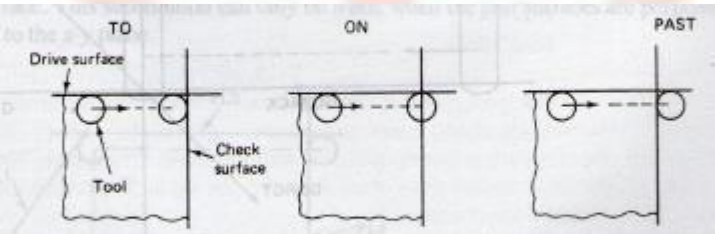
Contouring commands are somewhat more complicated because the tool's position must be continuously controlled throughout the To accomplish this control, the tool is directed along two intersecting surfaces. These surfaces have very specific names in APT ;

1. **Drive surface.** This is the surface that guides the side of the cutter.
2. **Part surface.** This is the surface on which the bottom of the cutter rides. The "partsurface" may or may not be an actual surface of the workpar.
3. **Check surface.** This is the surface that stops the movement of the tool in its currentdirection. In a sense, it checks the forward movement of the tool.



The APT contour motion statement commands the cutter to move along the drive and part surfaces and the movement ends when the tool is at the check surface. There are six motion command words:

GOLFT	GOFWD	GOUP
GORGT	GOBACK	GODOWN



Postprocessor statements

To write a complete part program, statements must be written that control fie operation of the spindle, the feed, and other features of the machine tool. These are called postprocessor statements. Some of the common postprocessor statements are:

COOLNT/
END
FEDRAT7
MACHIN/
RAPID
SPINDL/
TURRET/

Auxiliary statements

The complete APT program must also contain various other statements, called auxiliary statements. These are used for cutter size definition, part identification, and so on. The following APT words used in auxiliary statements are:

CLPRNT	INTOL/
CUTTER	OUTTOL/
FINI	PARTNO

Advantages of CAD/CAM in NC Programming:

1. **Savings in geometry definition**- Since the part geometry data have already been created during design using the CAD/CAM graphics system, the part programmer is not required to redefine the geometry of the part. This can be a time-consuming procedure in conventional APT programming.
2. **Immediate visual verification**- The graphics terminal provides a display of the tool path for immediate verification by the part programmer. Most programming errors can be detected by the user and corrected at the time the error is made. With conventional APT or other NC language, there is a delay between writing the programand the verification/correction process.
3. **Use of automatic programming routines**- For common part programming situations such as profiling and pocketing, the use of automatic MACRO-type routines yields a significant reduction in part programming time.
4. **One-of-a-kind jobs**- Because the part programming time is significantly reduced when using a CAD/CAM system, numerical control becomes an economically attractive method for producing one-of-a-kind jobs. Without CAD/CAM, the time required to prepare the part program represents a significant obstacle which often precludes the use of NC for one/off production.
5. **Integration with other related functions**- There is the obvious opportunity to integrate the product design function with part programming. Other opportunities for functional integration within manufacturing also exist. These include tool design, process planning, preparation of operator and setup instructions, grouping of partsinto families for programming convenience, and so on.

MANUAL DATA INPUT

Manual data input (MDI) involves the entry of part programming data through a CRT display at the machine site; hence the use of punched tape is avoided. The programming process is usually carried out by the machine operator. NC systems equipped with MDI capability possess a computer (microcomputer) as the control unit.

The great advantage of MDI is its simplicity. It represents a relatively easy way for a small shop to make the transition to numerical control. There is a minimum of change in normal shop procedures needed to use NC systems featuring manual data input. Since no punched tape is employed with MDI, the shop is spared the expense of tape punching equipment normally associated with NC.

one of the biggest disadvantages of manual data input is that the machine tool itself is not productive while programming is being accomplished. In essence, a very expensive piece of equipment is being utilized to prepare the part program. The more complicated the program, the more time is taken when the machine is not cutting metal.

Computer Controls in NC

Use of the digital computer has also permitted substantial improvement to be made in the controls for NC. There are three NC-related control:

1. **Computer numerical control (CNC**) - Computer NC involves the replacement of the conventional hard-wired NC controller unit by a small computer (minicomputer or microcomputer). The small computer is used to perform some or all of the basic ,NC functions by programs stored in its read/write memory. One of the distinguishing features of CNC is that one computer is used to control one machine tool.
2. **Direct numerical control (DNC)** - DNC involves the use of a larger computer to control a number of separate NC machine tools.
3. **Adaptive control** - It does not require a digital computer for implementation. Adaptive control machining denotes a control system that measures one or more process variables (such as cutting force, temperature, horse power, etc.) and manipulates feed and/or speed in order to compensate for undesirable changes in the process variables. Its objective is to optimize the initial adaptive control projects relied on analog controls rather than digital computers, Today, these systems employ microprocessor technology to implement the adaptive control strategy.

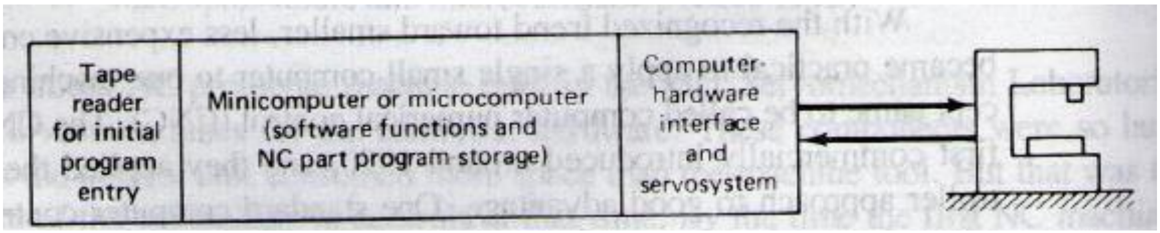
PROBLEMS WITH CONVENTIONAL NC

1. **Part programming mistakes** - In preparing the punched tape, part programming mistakes are common. The mistakes can be either syntax or numerical errors, and it is not uncommon for three or more passes to be required before the NC tape is correct. Another related problem in part programming is to achieve the best sequence of processing steps. This is mainly a problem in manual part programming.
2. **Nonoptimal speeds and feeds** - n conventional numerical control, the control system does not provide the opportunity to make changes in speeds and feeds during the cutting process.
3. **Punched tape** - Paper tape is especially fragile, and its susceptibility to wear and tear causes it to be unreliable NC component for repeated use in the shop.
4. **Tape Reader** – The tape reader tht interprets the punched tape is generally acknowledge among NC users to be the least reliable hardware component of the machine.
5. **Controller** – The conventional NC controller unit is hard-wired. This means that its control features cannot be easily altered to incorporate improvements into the unit.
6. **Management Information** – the conventional NC system is not equipped to provide timely information on operational performance to management.

COMPUTER NUMERICAL CONTROL

Computer numerical control is an NC system that utilizes a dedicated, stored program computer to perform some or all of the basic numerical control function. Because of the trend toward downsizing in computers, most of the CNC system sold today use a microcomputer- based controller unit

The external appearance of a CNC machine is very similar to that of a conventional NC machine. Part programs are initially entered in a similar manner. Punched tape readers are still the common device to input the part program into the system. However, with conventional numerical control, the punched tape is cycled through the reader for every workpiece in the batch. With CNC, the program is entered once and then stored in the computer memory. Thus the tape reader used only for the original loading of the part program and data. Compared to regular NC, CNC offers additional flexibility and computational capability. New system options can be incorporated into the CNC controller simply by reprogramming the unit. Because of this reprogramming capacity, both in terms of part programs and system control options, CNC is often referred to by the term "soft-wire‖ NC.



Functions of CNC

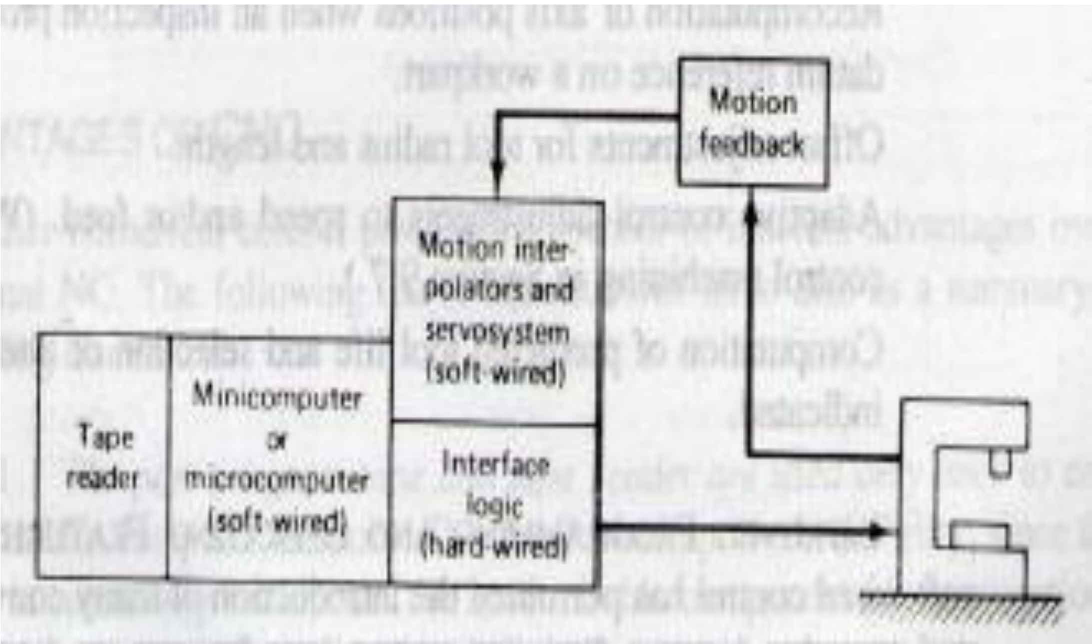
The principal functions of CNC are:

1. Machine tool control
2. In-process compensation
3. Improved programming and operating features
4. Diagnostics

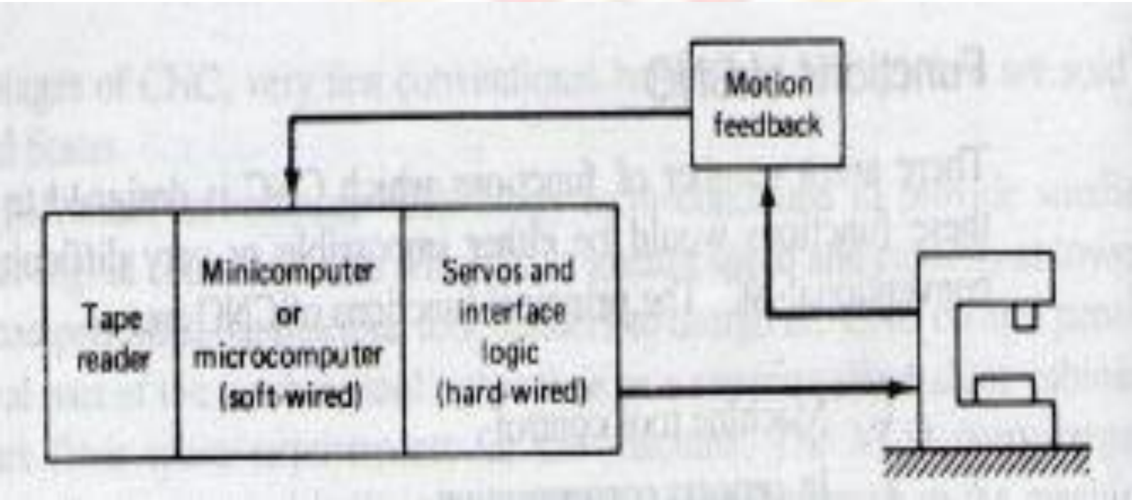
➤ **MACHINE TOOL CONROL-** The primary function of the CNC system is control of the machine tool. This involves conversion of the part program instructions into machine tool motions through the computer interface and servosystem. Some of the control functions, such as circular interpolation, can be accomplished more efficiently with hard wired circuits than with the computer. This fact has lead to the development of two alternative controller designs in CNC:

1. Hybrid CNC
2. Straight CNC

In the **hybrid CNC system**, the controller consists of the soft-wired computer plus hard-wired logic circuits. The hard-wired components perform those functions which they do best, such as feed rate generation and circular interpolation. The computer performs the remaining control functions plus other duties not normally associated with a conventional hard-wired controller. There are several reasons for the popularity of the hybrid CNC configuration. As mentioned previously, certain NC functions can be performed more efficiently with the hard-wired circuits.



The **straight CNC system** uses a computer to perform all the NC functions. The only hard-wired elements are those required to interface the computer with the machine tool and the operator's console. Interpolation, tool position feedback, and all other functions are performed by computer software.



- **IN-PROCES COMPENSATION**- A function closely related to machine tool control is in-process compensation. This involves the dynamic correction of the machine tool motions for changes or errors which occur during processing.
- **IMPROVED PROGRAMING and OPERATING FEATURE** - The flexibility of soft-wired control has permitted the introduction of many convenient programming and operating features.
- **DIAGNOSTICS**- NC machine tools are complex and expensive systems. The complexity increases the risk of component failures which lead to system down time.

It also requires that the maintenance personnel be trained to a higher level of proficiency in order to make repairs. The higher cost of NC provides a motivation to avoid downtime as much as possible. CNC machines are often equipped with a diagnostics capability to assist in maintaining and repairing the system.

Advantages of CNC

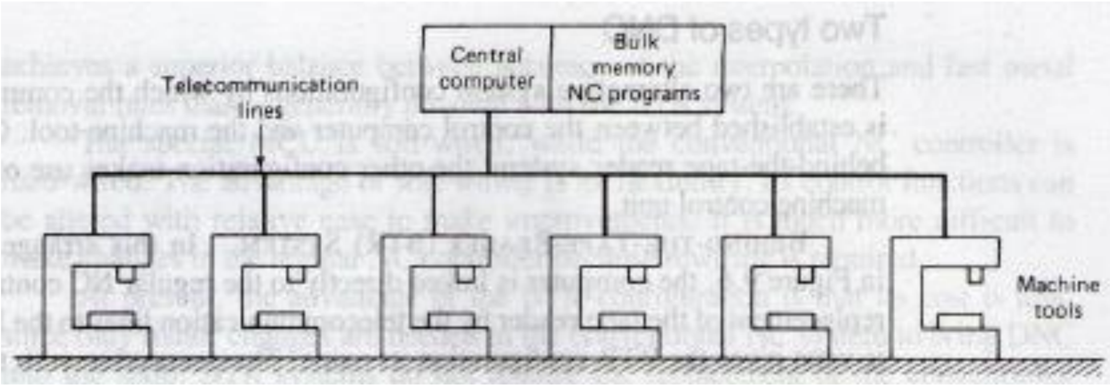
1. The part program tape and tape reader are used only once to enter the program intocomputer memory.
2. Tape editing at the machine site. The NC tape can be corrected and even optimized(e.g., tool path, speeds, and feeds) during tape tryout at the site of the machine tool.
3. Metric conversion- CNC can accommodate conversion of tape prepared in Units ofinches into the International System of units.
4. Greater flexibility- One of the more significant advantages of CNC over conventional NC is its flexibility.
5. User-written programs - One of the possibilities not originally anticipated for CNC was the generation of specialized programs by the user.
6. Total manufacturing system- CNC is more compatible with the use of a computerizedfactory-wide manufacturing system.

DIRECT NUMERICAL CONTROL

Direct numerical control can be defined as a manufacturing system in which number of machines are controlled by a computer through direct connection and in real time. The tape reader is omitted in DNC, thus relieving the system of its least reliable component. Instead of using the tape reader, the part program is transmitted to the machine tool directly from the computer memory. In principle, one larger computer can be used to control more than 100 separate machines. The DNC computer is designed to provide instructions to each machine, tool on demand. When the machine needs control commands, they are communicated to it immediately.

Components of a DNC system

1. Central computer
2. Bulk memory, which stores the NC part programs
3. Telecommunication lines
4. Machine tools



Functions of DNC

The principal functions of DNC are:

1. NC without punched tape
2. NC part program storage
3. Data collection, processing, and reporting
4. Communications

- NC WITHOUT PUNCHED Tape. One of the original objectives in direct mmerical control was to eliminate the use of punched tape. Several of the problems. with conventional NC discussed in Section 9.2 are related to the use of mnched tape (the relatively unreliable tape reader, the fragile nature of paper tape, the difficulties in making corrections and changes in the program contained on punched tape, etc.). There is also the expense associated with the equipment that produces the punched tape. All of these costs and inconveniences can be eliminated with the DNC approach.
- NC Part Program Storage. A second important function of the DNC system is concerned with storing the part programs. The program storage subsystem must be structured to satisfy several purposes. First, the programs must be made available for downloading to the NC machine tools. Second, the subsystem must allow for new programs to be entered, old programs to be deleted, and existing programs to be edited as the need arises. Third, the DNC software must accomplish the postprocessing function.
- Data Collection, Processing, and Reporting. The two previous functions for DNC both concerned the direct link from the central computer to the machine tools in the factory. Another important function of DNC involves the opposite link,, the transfer

of data from the machine tools back to the central computer. DNC involves a two-waytransfer of data.

- COMMUNICATIONS. A communications network is required to accomplish the previous three functions of DNC. Communication among the various subsysterns is a function that is central to the operation of any DNC system.

Advantages of DNC

1. Elimination of punched tapes and tape readers
2. Greater computational capability and flexibility
3. Convenient storage of NC part programs in computer files
4. Programs stored as CLFILE
5. Reporting of shop performance
6. Establishes the framework for the evolution of the future computer automated factory

ADAPTIVE CONTROL MACHINING SYSTEMS

For a machining operation, the term adaptive control denotes a control system that measures certain output process variables and uses these to control speed and/or feed. Some of the process variables that have been used in adaptive control machining systems include spindle deflection or force, torque, cutting temperature, vibration amplitude, and horsepower. In otherwords, nearly all the metal cutting variables that can be measured have been tried in experimental adaptive control systems. The motivation for developing an adaptive machining system lies in trying to operate the process more efficiently. The typical measures of performance in machining have been metal removal rate and cost per volume of metalremoved.

Two types of adaptive control

1. Adaptive control optimization (ACO)- In this form of adaptive control, an index of performance is specified for the system. This performance index is a measure of overall process performance, such as production rate or cost per volume of metal removed. The objective of the adaptive controller is to optimize the index of performance by manipulating speed and/or feed in the operation.

   Most adaptive control optimization systems attempt to maximize the ratio of work material removal rate to tool wear rate.

   IP = a function of (MRP/TWR)where

   MRR = material removal rate

   TWR = tool wear rate

2. Adaptive control constraint (ACC) - The production AC systems utilize constraint limits imposed on certain measured process variables. Accordingly, these are called adaptive control constraint (ACC) systems. The objective in these systems is to manipulate feed and/oi speed so that these measured process variables are maintained at or below their constraint limit values.

Benefits of adaptive control machining

There are obviously many machining situations for which it cannot be justified. Adaptive control has been successfully applied in such machining processes as milling, drilling, tapping, grinding, and boring. It has also been applied in turning, but with only limited success. Following are some of the benefits gained from adaptive control in the successful applications.

1. Increased production rates
2. Increased tool life
3. Greater part protection
4. Less operator intervention
5. Easier part programming