LABORATORY MANUAL
MANUFACTURING PROCESSES – 1

TA - 202 LAB
(NECESSARY READING MATERIAL FOR LAB AND PROJECT TURNS FOR ALL LAB USERS)

Department of Mechanical Engineering
INDIAN INSTITUTE OF TECHNOLOGY KANPUR

GENERAL INSTRUCTION

1. Every student should obtain a set of instruction sheets entitled manufacturing processes Laboratory.

2. For reasons of safety, **every student must come to the laboratory in shoes (covering the whole feet)**. It is unsafe for the students to come to the laboratory wearing garments with parts that that hang about loosely and as such the lab users are requested to avoid wearing garments with loose hanging parts. Students should preferably use half-sleeve shirts wherever possible. The Students should also ensure that floor around the machine is clear and dry (not oily) to avoid slipping. Please report immediately to the lab staff on seeing any coolant / oil spillage.

3. An apron will be issued on a returnable basis (returnable after each lab turn) to each student. Students not wearing an apron will not be permitted to the work in the laboratory.

4. Instruments and tools will be issued from the tool room. Every student must produce his identity card for the purpose. Tools, etc. must be returned to the tool room on the same day after work hours.

5. The student should take the permission and guidance of the Lab Staff / Tutors before operating any machine. Unauthorized usage of any machine without prior guidance may lead to fatal accidents and injury.

6. The student will not lean on the machine or take any kind of support of the machine at any point of time. If found leaning on a machine without proper reasons serious action would be taken

7. Power to the machines will be put off 10 minutes before the end of laboratory session to allow the students to return the tools. Please finish up at exactly 4:50pm.

8. **Students are required to clear off the chips from the machine and lubricate the guides etc. at the end of each laboratory session.**

9. Laboratory reports should be submitted on A4 size sheets. The students must fill these reports and hand it over to their group TA after the laboratory session. These have associated some grades.

10. Reports will not be returned to the students. Students may see the graded reports while in the laboratory.

GENERAL INFORMATION

This laboratory is aimed at providing an introduction to the Know-how of common processes used in industries for manufacturing parts by removal of material in a controlled manner. Auxiliary methods for machining to desired accuracy and quality will also be covered. The emphasis throughout the laboratory course will be on understanding the basic features of the processes rather than details of I constructions of machine, or common practices in manufacturing or acquiring skill in the operation of machines. Evidently, acquaintance with the machine is desirable and the laboratory sessions will ' provide adequate opportunity for this.

LABORATORY EXERCISE

I	Turning (T) & Demonstration & Practice	3hrs.
II	Milling & Shaping Demonstration & Practice	3 hrs.
III	Drilling and Fitting (D&F) Demonstration & Practice	3 hrs.
IV	CNC , Demonstration/ Non Traditional Machining processes	3 hrs.

LABORATORY EXERCISES SCHEDULE.

EXERCISE POWER TRANSMITION Video Clip (10 MIN,)			
LAB TURN	LATHE	MILLING & SHAPING	DRILLING & FITTING
I	A	B	C
2	B	C	A
3	C	A	B

On the 4th and 5th Laboratory turns the whole student group would be divided into two groups A & B and would be assigned in the following manner

LAB TURN	CNC Demonstration and programming	Non Traditional Machining processes	Project discussion with guide
4 (First half class)	A	B	
4 (Second half class)	B		4:00-5:00PM
Drawing Submission			
			Grading

NOTES: Each Section will be divided in to three groups (A, B, & C) in the first three lab turns and into two groups (A & B) in the CNC/Non Traditional Machining Processes

The Project consists of conceptualization a device designing, planning of machining and other operations, fabrication of the components and assembly of the device. Before the final evaluation of the completed project a report has to be submitted. This report should contain general description of the completed project, design details, detailed drawings, and fabrication of the components, assembly i and testing, suggestions for improvements.

I	First Turn	:	Project groups should be formed.
II	Second Turn	:	Each group member has to come with an idea of project
III	Third Turn	:	Project discussion with Technical Staff / Guide with Material List.
IV	Fourth Turn	:	CNC demo and on Non Traditional Machining processes at 4i Lab
V	Fifth turn	:	Drawing Submission and Grading on your drawing

 # Number of components in the project should not be more than 25 (Excluding standard parts)
6th to 12th Projects.

MACHINING PROCESSES

Machining is one of the processes of manufacturing in which the specified shape to the work piece is imparted by removing surplus material. Conventionally this surplus material from the work-piece is removed in the form of chips by a mutual interaction between the an appropriate tool and the workpiece. The mechanical generation of these chips can be carried out by a single point or multi point tool or by abrasive operationsas classified below:

Machining Process Classification
|

Single point tool operations	Multi-point tool operations	Abrasive operat:ons
1. Turning	1. Milling	1. Grinding
2. Boring	2. Drilling	2. Lapping
3. Shaping	3. Tapping	3. Honing
4. Planing	4. Reaming	4. Super-finishing
	5. Hobbing	
	6. Broaching	
	7. Sawing	

The process of chip formation in metal cutting is affected by relative motion between the tool and the work-piece achieved with the aid of a device called machine tool. This relative motion can be obtained by a combination of rotary and translatory movements of either the tool or the workpiece or both. The kind of surface that is produced by the operation depends on the shape of the tool and the path it traverses through the materials. When the workpiece is rotated about an axis and the tool is traversed in a definite path relative to the axis, a surface of revolution is generated. When the tool path is parallel to the axis, the surface generated is a cylinder as in straight turning [Figure 1] or boring operations [Figure 2]. Similarly, planes may be generated by a series of straight cuts without rotating the workpiece as in shaping and planing operations [Figure 3]. In shaping the tool reciprocates and the work piece is moved crosswise at the end of each stroke. Planning is done by reciprocating the workpiece and crosswise movement is provided to the tool.

A surface may be machined by the tools having a number of cutting edges that can cut successively through the workpiece materials. In plane milling, the cutter revolves and moves over the work piece as shown Figure 4. The axis of the cutter is parallel to the surface generated. Similarly in drilling, the drill may turn and be fed into the workpiece and the workpiece may revolve while the drill is fed into it [Figure 5]

The machine tools, in general, provide two kinds of relative motions. The primary motion is responsible for the cutting action and absorbs most of the power required to perform the machining action. The secondary motion is that of feed and may proceed in steps or continuously and absorbs only a fraction of the total power required for machining. When the secondary motion is added to the primary motion, machine surfaces of desired geometric characteristics are produced.
.

Consider a situation where both the cutting motions as well as the feed motion (provided at the end of each stroke) are rectilinear and perpendicular to each other simultaneously. Here the machined surface produced is a plane. The line generated by the primary motion (cutting motion) is called the generatrix, while the line representing the secondary motion (feed motion) is called the directrix (Fig. 6a).

Depending upon the shapes of the generatrix and the directrix and their relative orientations. various geometries can be produced on the workpiece. Consider another case when the generatrix is a circle and the dirctrix is a line perpendicular to the plane of the generatrix. It is clear that in this situation the surface produced will be a cylinder (Fig. 6b). A tapered surface can be produced by merely changing the angle that the directrix makes with the plane of the generatrix. When the directrix is in the plane of the circular generatrix (Fig. 6c), lines are generated which results in a plain surface when a number of generatrices and directrices are placed side by side in the direction perpendicular to the plane of the generatrix. In actual practice, the cutting is performed by cutting edge and not a point. Thus.a series of generatrix/directrix combination are involved and the relative motion produces a surface rather than a line.

Basically there are two methods of producing new surfaces, the tracing method and the generation method. In the tracing method the surface is obtained by direct tracing of the generatrices and when the surface produced is the envelope of the generatrix the process is known as generation. Figs. 6(a) & 6{b), the plane and the cylindrical surfaces *are* obtained by direct tracing, while in Fig. 6(c) the final surface geometry is the envelope of the generatrices.
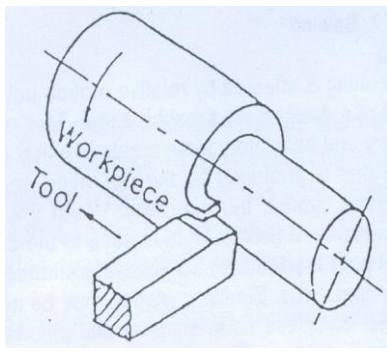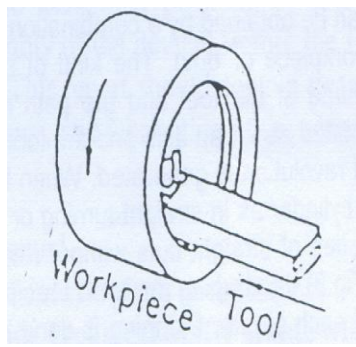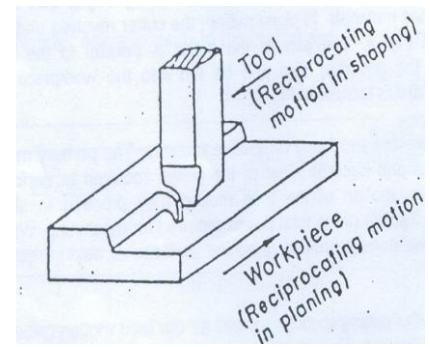
Fig. 1 Straight turning
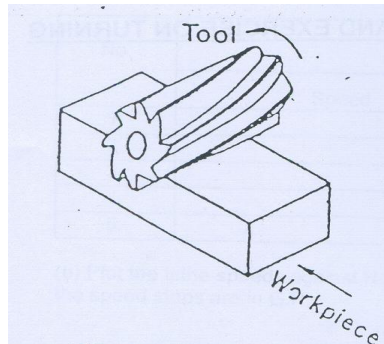

Fig. 2 Straight boring

Fig. 3 Shaping and planning
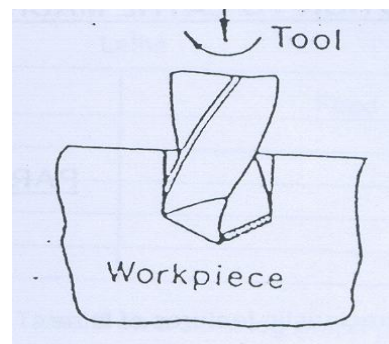

Fig. 4 Plain milling
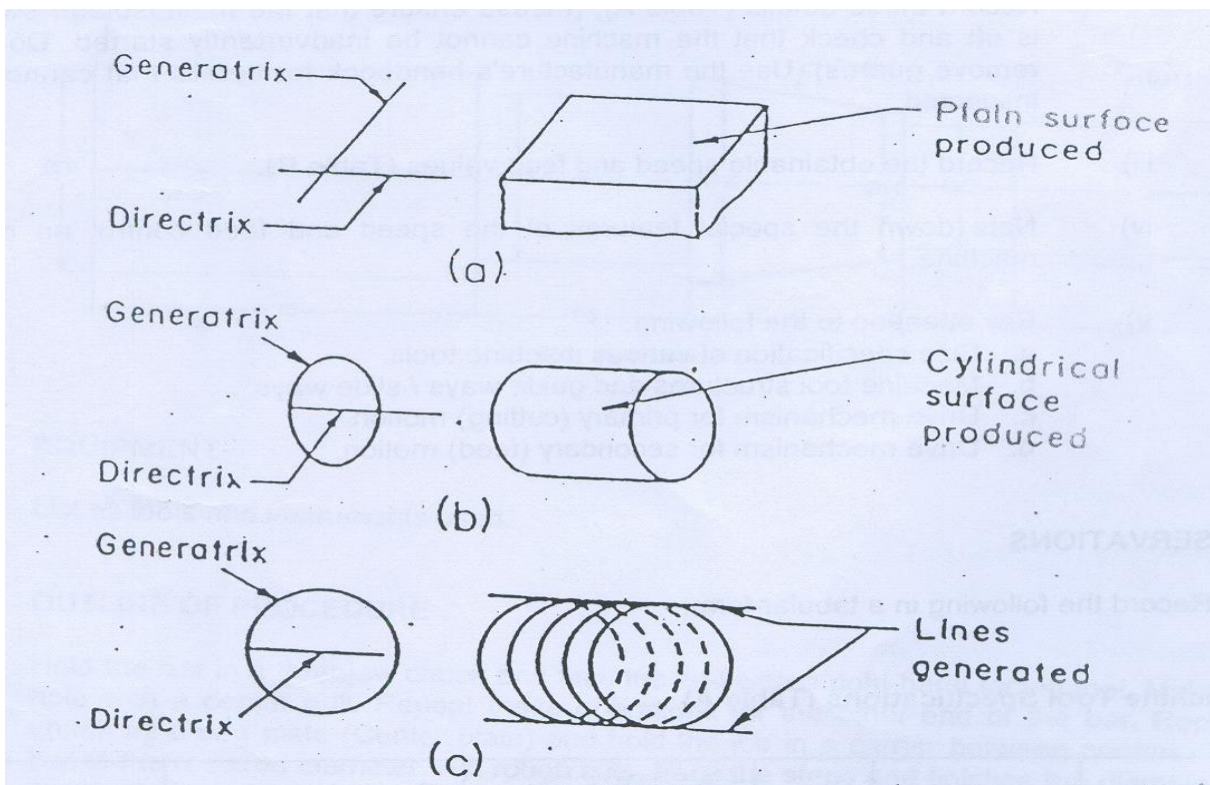

Fig. 5   Drilling

Fig.6   Concept of generatrix and directrix. (a) Rectilinear generatrix and directrix.
(b) Directrix   perpendicular to the plane of generatrix.
(c) Directrix in the plane of generatrix.

INTRODUCTION TO LATHE MACHINE AND EXERCISE ON TURNING

PART (A)

OBJECTIVE: To study the characteristic features of lathe.

OUTLINE OF PROCEDURE

i)   Run the machine at low speed and observe the motions, which control the shapes of the surfaces produced. Note particularly the features, which control the geometrical form of the surface.

ii)   Learn the names of the major units and the components of each machine. Record these details (Table A). (Please ensure that the main isolator switch is off and check that the machine. cannot be inadvertently started. Do not remove guards). Use the manufacture's handbook for details that cannot be inspected.

iii)   Record the obtainable speed and feed values (Table B).

iv)   Note down the special features of the speed and feed control on each machine.

v)   Pay attention to the following :
        a. Size specification of various machine tools.
        b. Machine tool structures and guide ways / slide ways.
        c. Drive mechanism for primary (cutting) motion.
        d. Drive mechanism for secondary (feed) motion.

OBSERVATIONS: Record the following in a tabular form:

Machine Tool Specifications (Table A)

Machine	Type & Make	Size	Speed given to		Feed given to		Type of Surface Produced
			Tool	Work	Tool	Work	
Lathe							

Speed and Feed Data (Table B)

Speed on machine (rpm)	Feed on machine (mm/rev.)

PART (B)

OBJECTIVE: To make the part shown in the sketch from a mild steel rod on a Lathe.

(All dimensions are in MM)

EQUIPMENT: List all tools and instruments used.

OUTLINE OF PROCEDURE

Hold the bar in a three jaw chuck and face the end with a right hand facing tool. Make central hole with a center drill. Repeat these' operations for the other end of the bar. Hold the live centre in three jaw chuck and hold the job in a dog carrier between centers. Turn the bar to the required diameter with rough cuts. Face the steps and finishes the diameters to the required sizes. Machine the roots and the groove with form tools. Machine the taper with the help of the cross-slide swiveling arrangement. Groove the length as per given dimension. Cut the threads.

OBSERVATIONS

    (a)   Measure all dimensions (up to second decimal place) on the specimen turned by your group. Make a neat sketch and indicate all measured dimensions.

    (b)   Discuss briefly how tapered portion was turned.

    (c)   Show the calculation of the required gear ratio for thread cutting.

    (d)   Sketch the main drive unit of 'the- lathe and show how the speed steps are obtained.

MILLING: INTRODUCTION AND PRACTICE

PART (A)

OBJECTIVE: To study the characteristic features of Milling machine.

OUTLINE OF PROCEDURE

i)   Run the machine at low speed and observe the motions, which control the shapes of the surfaces produced. Note particularly the features, which control the geometrical form of the surface.

ji)   Learn the names of the major units and the components of each machine. Record these details (Table A). (Please ensure that the main isolator switch is off and check that the machine cannot be inadvertently started. Do not remove guards). Use the manufacture's handbook for details that cannot be inspected.

jii)   Record the obtainable speed and feed values (Table B).

iv)   Note down the special features of the speed and feed control on each machine.

v)   Pay attention to the following:
        a. Size specification of various machine tools.
        b. Machine tool structures and guide ways / slide ways.
        c. Drive mechanism for primary (cutting) motion.
        d. Drive mechanism for secondary (feed) motion.

OBSERVATION: Record the following in a tabular form:

Machine Tool Specifications (Table A)

Machine	Type & Make	Size	Speed given to		Feed given to		Type of Surface Produced
			Tool	Work	Tool	Work	
Milling							

Speed and Feed Data (Table B)

Speed on machine (rpm)	Feed on machine (mm/rev.)

OBJECTIVE: To machine the hexagonal head and the slot shown in the sketch on the specimen,

HEXAGONAL HEAD

3X3 SLOT

ALL DIMENSIONS IN MM

EQUIPMENT: List all tools / cutters and instruments used.

OUTLINE OF PROCEDURE

Fit the helical cutter on the arbor and hold the specimen between the centers of the dividing head and the tail center with help of dog carrier. Carefully adjust the work piece so that the cutter just touches the top surface of the specimen. Calculate the necessary depth of cut and then mill the six *faces* of the hexagonal head in succession. Change the cutter and mill the rectangular slot.

OBSERVATIONS

(a) Measure all dimensions (up to second decimal place) on the specimen milled by your group. Make a neat sketch and indicate all measured dimensions.

(b) Explain in brief how the required indexing was obtained with the dividing head.

(c) Explain up-milling and down-milling operations. Which one did you use *for* slot milling and why?

(d)   Explain the advantages of using a helical milling cutter.

DRILLING AND FITTING

PART (A)

OBJECTIVE: To study the characteristic features of Drilling machine.

OUTLINE OF PROCEDURE

i) Run the machine at low speed and observe the motions, which control the shapes of the surfaces produced. Note particularly the features, which control the geometrical form of the surface.

ii) Learn the names of the major units and the components of each machine. Record these details (Table A). (Please ensure that the main isolator switch is off and check that the machine cannot be inadvertently started. Do not remove guards). Use the manufacture's handbook for details that cannot be inspected.

iii) Record the obtainable speed and feed values (Table B).

iv) Note down the special features *of* the speed and feed control on each machine,

v) Pay attention to the following:
   a, Size specification *of* various machine tools,
   b, Machine tool structures and guide ways I slide ways.
   c. Drive mechanism for primary (cutting) motion,
   d. Drive mechanism for secondary (feed) motion.

OBSERVATION

Record the following in a tabular form:
Machine Tool Specifications (Table A)

Machine	Type & Make	Size	Speed given to		Feed given to		Type of Surface Produced
			Tool	Work	Tool	Work	
Drilling							

Speed and Feed Data (Table B)

Speed on machine (rpm)

Feed on machine (mm/rev.)

OBJECTIVE: To drill, file, as shown in the sketch, ream and tap holes on the mild steel plate.

(All dimensions are in MM)

EQUIPMENT : List all tools and instruments used.

OUTLINE OF PROCEDURE

File two sides of the mild steel work piece ensuring with a trisquare that the angle are 90°. Mark the object as per dimension with the help of height gauge & supporting "V" block. Punch at the center of the required hole and saw cut lenth . Drill and ream the holes as required, Tap the hole using a set of three taps, and saw cut.

OBSERVATIONS

(a)   Measure all dimensions (up to second decimal place) on the specimen made by your

       group. Make a neat sketch and indicate all measured dimensions.

(b)   Explain how power is transmitted *from* drill spindle to drill shank.

(c)   Sketch a reamer and show its main features.

(d)   Explain why a set of three taps was used.

Overview of Numerical Control

A new technique for controlling the machine / production tools, the Numerical Control (NC) was developed in mid 50's. Prior to this, all the machine / production tools were manually operated and co-ntrolled. Quality of the products produced by manually operated machines is totally dependent on the skills and mind status of the human operator. Numerical control machines are more accurate than manually operated machines, can produce components more uniformly, faster and in the long-run tooling costs are smaller but the initial investment is higher.

Numerical Control (NC) has been defined by the Electronic industries Association (EIA) as 'a system in which actions are controlled by direct insertion of numerical data at some point'. This s -s-tern automatically interprets symbolic instructions (numerical dai3) to control machine tools and other manufacturing systems. Symbolic instructions or the numerical data required to produce a part is called a 'part program'.

Traditionally, in "the NC machining, part drawing of the component to be machined is studied by the NC programmer who translates the information on the drawing to the necessary programme which issues operational instructions to the machine tools. The programme represents the path or action at every momment that the machine tool must take to properly machine the part as described by the engineering drawing.

In the initial stages of NC development the programmed instructions stored on punched tapes where interpreted by electromechanical tape readers connected to the machine tool. The main problem with tapes was that it was very difficult to change the instructions on the tape. Even to make very minor instructions in the programme, new tape had to be made in addition to interrupting machining. Operations. Another draw-back of using the tapes was that they had to be run as many times as the number of components required, which decreased the life of tape.

With rapid developments in computer technology and its capabilities, the problems associated with punched paper/plastic tape were solved. Rapid development in computer technology extended numerical control (NC) to direct numerical control (dNC), computer numerical control (CN C) and distributed numerical control (DNC).
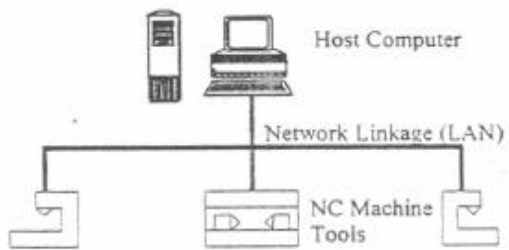

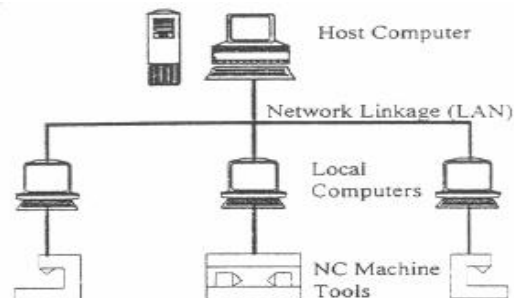Fig. 15.13 Direct numerical control.

Fig. 15.14 Distributed numerical control.

The problems faced by NC lead to the development *of* a concept known as direct numerical control (dNC) which eliminates the use *of* tape as a medium of carrying the programmed instructions. In dNC many machine tools are connected to a host computer through a data transmission link as shown in Fig. 15.13. Here, the NC programmes requirred to operate the machines are stored in the host computer and are fed to the machine tool connected through the data transmission lines. Even though dNC eliminates the use *of* tape, it will suffer whenever the host computer goes down. This shortfall *of* dNC led to the development of computer numerical control (CNC) which allowed NC machines at remote locations to be connected to the host computer.

The development: of programmable logic controllers (PLC) and micro-computers lead to the development of computer numerical control (CNC). In CNC technology, each machine tool has a PLC or a micro-computer which allows the programmes to be input and stored at ea.ch and every machine. In addition. programmes can be developed off-line and down-loaded to the micro-processors/PLCs present at individual machine tools. Even though it rectified the problems associated with the down-time *of* host computer as in dNC, but a new problem known as data management came into picture. For example the same prorarnmme may get loaded on many different microcomputers Witch control the machine tools as there is no communication present

among them. This problem of data rnanagement was solved by the development *of* distributed numerical control (DNC).

Distributed numerical control (Fig. 15.14) \Vas developed by combining the positive point *of* both direct numerical control (dNC) and computer numerical control (CNC). Hence, in distributed numerical control (DNC) both host and local computers are present at individual machine tool. Here host computers are used as main storage devices and programmes are down-loaded to the mini-computers present at various machines where they are stored or transmitted to the NC machines. Mini-computer controllers also serve as back-up memory whenever the host computer is down. Therefore, the NC machines do not have to be down when the host computer is down. An effective data transmission network from the host computer to the micro-computer controls the NC machines and is key to the success *of* distributed numerical control systems. In rest of this book, NC is used as synonym for CNC

Various components present in NC machine tool are:
. Machine tool
. Machine Control Unit (MCU)
. Communication interface and accessories

The machine tool may be any type *of* machine tool used in the manufacturing industry. Machine control unit (MCV) is the control unit that reads and interprets the numerical data/part programme from the tap or any other media and passes on this information in the form of electrical signals to various activators / drive mechanisms *of* the machine to operate the machine tool in the desired way.

Numerical control Machines are classified based on the type of motion control. the presence *of* feed-back loops, the power drives the positioning system used and the number *of* axes *of* motion which can be controlled.

Two types *of* motion controls are used on NC machines. They are point-to-point or continuous-path controlled. As the name implies, the point-to-point (PTP) controlled machines move in a series *of* steps from one point to the other point. A machine with PTP control can perform very limited number *of* machining opera ions. In almost all the cases where PTP control is used,
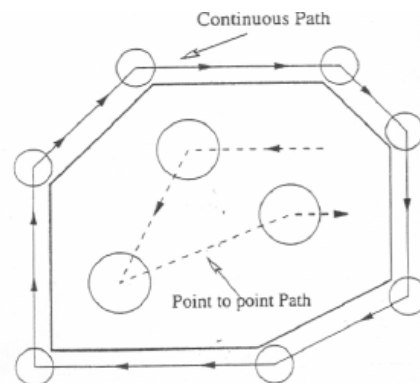

Fig. 15.15 Point-to-point and continuous path.

its function is to move the machine table or the spindle to a desired position to perform the machining operation at that position/point. In general, no machining is performed when the movement from one point to other is taking place, hence movement is made as rapidly as possible.

When there is no need *of* machining between two points, path to be followed between them is not specified. Continuous path machines move uniformly and evenly by controlling the motion of two are more axes simultaneously. Here, the path to be followed along with the destinations have to be mentioned as the machining is invariably done along that path. Figure 15.15 shows the cutting paths followed by PTP and continuous path machines.

Depending on the presence of the feedback, they are classified as open-loop system or closed-loop system (Fig. 15.16 and 15.17). In closed-loop systems, a transducer sends the current table position to the control unit and the driving unit (typically motor) stops running as the table reaches its desired position, hence accurate positioning can be achieved with closed-loop control systems. Closed-loop NC systems use servomotors where as open-loop systems use stepper motors.
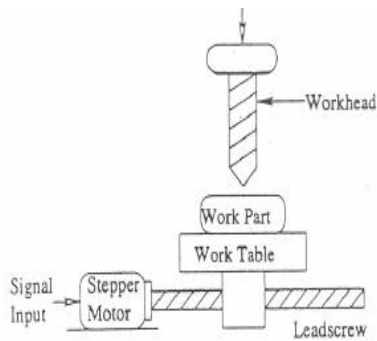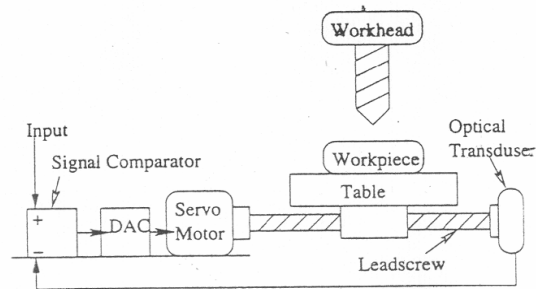
Fig. 15.16 Open-loop system.

Fig. 15.17 Closed-loop system.

NC machines are classified into three types: hydraulic, pneumatic or electric based on the type of power used to drive the axes and spindle. Most of the modern NC machines use electric drives (AC or DC servo motors) to drive its axes and spindle because of its compact size, ease of control and low cost.

Depending on the positioning system used on the machine, they are classified as incremental or absolute positioning machines. Modern NC machine tools allow the programmer to choose either of the above mentioned positioning systems or both of them together through part programme. In absolute positioning system. the end point of the tool for a particular move has to be mentioned in the programme with respect to the origin. But in incremental system, the end point of the tool for a particular move has to be mentioned with respect to the current tool position.

Origin is a reference point used within the NC part program as the basis of defining tool location and other geometric entities. While developing an NC programme the programmer assumes that the tool is located at some Specific point relative to the workpiece to be machined and this point is designated as origin. In practice, during set-up and before starting the programme the operator has to move tile tool to a position which is designated as origin in that particular programme and then presses a button on control panel to select that as the origin. It means that the origin could be any where within the machine work space, hence it is called a floating zero. In the older generation NC machine tools, this facility was not available. The origin was a fixed point and the programmer had to write the programme with respect to that particular point.

The purpose of any production system is to produce a part in the most economic way without compromising on the desired qualities. Many large and small industries have undertaken the conversion from manual machines and processes to NC to get benefits. Some of these are listed below:

. Flexibiiity is high
. Scheduling is easier
. Set-up lead and processing times arc less.
. Greater uniformity and accuracy in cutting.
. Lover overall tooling costs .
. Low inventory
. Inspection cost is less
. Higher degree of interchangeability of parts and tools

In spite of the above mentioned benefits, there are some negative points as well. The most prominent ones are:
. Large initial investments
. Programming costs
. Training and retraining costs for the existing work force

Numerical-Control Programming

10.1 NC PART PROGRAMMING

10.1.1 Coordinate Systems

In an NC system, each axis of motion is equipped with a separate driving source that replaces the hand wheel of the conventional machine. The driving source can be a DC motor, a stepping motor, or a hydraulic actuator. The source selected is determined mainly based on the precision requirements of the machine, as described in Chapter 9.

The relative movement between tools and workpieces is achieved by the motion of the machine tool slides. The three main axes of motion are referred to as the X, Y. and Z axes. The Z axis is perpendicular to both the X and Y axes in order to create a right-hand coordinate system, as shown in Figure 10.1.A positive Motion in the Z direction moves the cutting tool away from the workpiece. This is detailed as follows:

Z AXIS

1. On a workpiece -rotating machine, such as a lathe, the Z axis is parallel In the spindle, and the positive Motion
    moves the tool away from the workpiece (figure 10.2).
2. On a tool-rotating machine, such as a milling or boring machine, the Z axis is parallel to the tool axis, and the
    positive motion moves the tool away from the workpiece (Figures 10.3 and 10.4).
3. On other machines, such as a press, a planing machine, or shearing machine, the Z axis is perpendicular to
    the tool set, and the positive motion increases the distance between the tool and the workpiece.

X AXIS

1. On a lathe, the X axis is the direction of tool movement, and the positive motionmoves the tool away from
    the workpiece
2. On a horizontal milling machine, the X axis is parallel to the table.
3. On a vertical milling machine, the positive X axis points to the right when the prorammer is facing the
    machine.

The **Y axis** is the axis left in a standard Cartesian coordinate system.

Figure 10.1   A right-hand coordinate system. (Y. Koren, *Computer Control of Manufacturing Systems*, McGraw-Hill, 1983.)

Figure 10.3   Coordinate system for a drill. (Y. Koren, *Computer Control of Manufacturing Systems*, McGraw-Hill, 1983.)

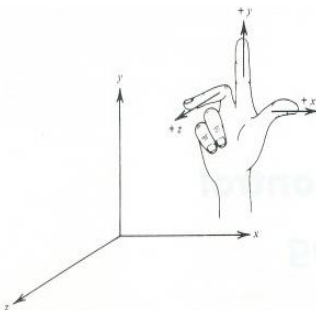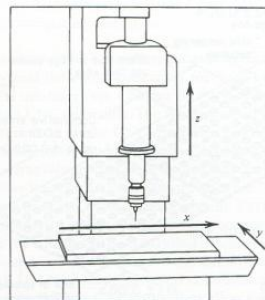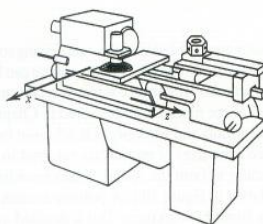Figure 10.2   Coordinate system for a lathe. (Y. Koren, *Computer Control of Manufacturing Systems*, McGraw-Hill, 1983.)
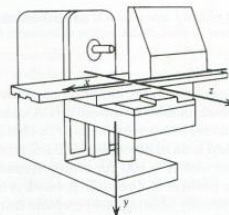
Figure 10.4   Coordinate system for a mill. (Y. Koren, *Computer Control of Manufacturing Systems*, McGraw-Hill, 1983.)

10.1.2 NC Program Storage Media

Modern CNC controllers provide several ways of transferring data. Perhaps the most typical data-communication methods used to transfer part program files is an RS-232C interface (see Chapter 8). An NC part program is stored in a file on a computer or a CNC controller. The file download (or upload) can be initiated by setting up a transfer mode on the CNC controller. On the other side of the communication cable is a computer that sends or receives data byte by byte. The operator must start and end the data-transfer process on both the CNC controller and the computer. Some machines use higher-level protocols to ensure an error-free data transfer. Two of the higher-level protocols used are Kermit and Xmodem. Kermit and Xmodem arc widely accepted in the computer-to-computer telecommunication file-transfer process. These protocols allow the file transfer to be controlled by either the computer or the controller. The computer can send and retrieve data directly.

Some machines also provide local-area network (LAN) instead of serial communication. Ethernet and MAP are two technologies used. Some CNC controllers allow the entire controller function to be initiated from a remote computer through the data-communication network.

10.1.3

A BCD (binary-coded decimal) or ASCII (American Standard Code for Information interchange) code is frequently used in NC applications.

. BCD: An eight-track punched tape is one of the more common input media for NC systems. Hence, all data in the f*orm* of symbols, letters, und numbers must be: represent able by eight binary fields. The BCD code has been devised *to* satisfy this requirement. In a BCD *code,* the numerals 0 through 9 are specified using only the first four tracks, quantities 1,2,4, and 8. Note that the four number: 1.2,4. and 8. added together as needed make all numbers from 1 to 15. Letters, symbols and spccial instructions are indicated by using tracks 5 through 8 in combination with the numeral tracks. A complete BCD character set, based on EIA Standard RS244A, is illustrated in Figure 10.6. Each BCD character must have all odd number of holes. By punching a parity bit along with all even bit strings, all characters have an odd number of holes*.* If an even number of holes is detected, it is by definition an error, and a parity check occurs. This simple method provides some: protection from input errors resulting in part damage.

. ASCII: ASCII was formulated to standardize punched-tape codes regardless of applications (Pressman and Williams, 1979). Hence, ASCII is used in computer and telecommunications as well as in NC applications. ASCII code was devised to support a large character set that includes uppercase and lowercase letters and additional special symbols not used in NC application Figure 10.6 illustrates the ASCII subset applicable to NC. Many new control systems now accept both BCD and ASCII codes. It is likely that the move toward ASCII standardization will progress as older NC equipment is replaced.

10.1.4 Tape Input Formats

The organization of words within blocks is called the tape format (EIA Standard RS274) (Groover, 1980). Four tape formats are used for NC input (Pressman and Williams, 1979):

1. The fixed sequential format requires that each NC block be the same length and contain the same number of characters. This restriction enables the block *to* be divided into substrings corresponding to each of the NC data types. Because block length is invariant, all values must appear even if some types are not required.
2. The block-address format eliminates the need for specifying redundant illformation in subsequent NC blocks through the specification of a change *code.* The change code follows the block sequence number and indicates which values arc to be changed relative to the preceding blocks. All data must contain a predefined number of digits in this form t.
3. The tab sequential format derives its name because words are listed in a fixed sequence and

separated by depressing the tab key (TAB) when typing the manuscript on a Flexowriter. Two *or* more tabs immediately following one another indicate that the data that -would normally occupy the null locations are redundant and have been omitted. An example of tab sequential NC code is

    TOO1 TO1 TO7500 TO6250 T10000 T612 T718 T T EOB
     TO02 T T08725 TO6750 T T T T EOB
    TO03 T T T TO5000 T520 T620 TO1 T EOB

    (T represents a tab character.)

  4. The word-address format places a letter preceding each word and is used to identify the word type and to Address
    the data to a particular location in thecontroller. The X prefix identifies an X-coordinate word, 30 S prefix identifies spindle speed, and so on. The standard sequence of words in 2t block for a three axis NC machine is.

N word   G word   X word   Y word   Z word   F word   S   word
T word   M word   and   EOB

A word-address NC code is
        NOO1 GO1 X07500 Y06250 Z1OOOO F612 5718 EOB NOO2 X08752 Y06750 EOB
        NOO3 Z05000F520 S620 MO1 EOB

A block of NC part program consists of several words. A part program written in this data format is called a G-code program. A G-code program contains the following words:
N. G, X, Y, Z, A, B. C. I, J. K, F, S, T, R. M
Through these words, all NC control functions can be programmed. An EIA standard. RS-273, defines a set of standard codes. However, it also allows for the customizing of certain codes. Even with this standard, there is still
a wide variation of cod format. A program written for one controller often does not run on another. It is, therefore,
essential to refer to the programming manual for the target machine before a program is written. In this section, before the meaning *of* each word is explained, we will first analyze the requirements *of* an NC control.

10.1.5.1 Basic requirement *of* NC machine control.

To control a machine, it is necessary to begin by defining the coordinates *of* the tool motion. It is necessary to specify whether the motion is a positioning motion (rapid traverse) *or* a feed motion (cutting). 'The feed motion includes linear motion and circular motion. Linear motion requires the destination coordinates. When circular interpolation is used, the center of the circle must be given in addition to the destination. Before a cutting motion is called out, the spindle must be turned to the desired rpm and the feed speed must be specified. The spindle can rotate either clockwise *or* counterclockwise. Sometimes coolant is required in machining, and the coolant may be applied in flood *or* mist form. If an automatic tool changer is present, the next tool number has to be known to the controller before a tool can be changed to the machine spindle. The sequence to change the tool also needs to be specified. It is often desirable to aggregate a fixed sequence *of* operations such as drilling holes into a cycle. Using cycle codes can drastically reduce programming effort. Additional information is needed for specific cycle operations. Finally, there are other programming functions, such as units-inch *or* metric-positioning system-absolute *or* incremental, and so on. All *of* these activities can (and in some cases must) be controlled through the NC controller and related part program. These control functions and data requirements are summarized in what follows:

(a) Preparatory functions: the words specify which units, which interpolator, Absolute *or* incremental
    Programming, which circular interpolation plane, cutter compensation, and so on.

(b) Coordinates: define three translational (and three rotational) axes.
(c) Machining parameters: specify feed and speed.
(d) Tool control: specifics tool diameter, next tool number, tool change, and so on.
(e) Cycle functions: specify drill cycle, ream cycle, bore cycle, mill cycle, and clearance plane.
(d) Coolant control: specify the coolant condition, that is, coolant on/off, flood. And mist.

(g) Miscellaneous control: specifies all other control specifics, that is, spindle *on/of,* tape rewind, spindle rotation
    direction, pallet change, clamps control, and so on.
(h) Interpolators: linear, circular interpolation, circle center, and so on.

These control functions are programmed through program words (codes).

1 0.1 5,2 N words

A specific NC function may be programmed using an *NC* word *or* a combination of NC words. All functions can be programmed in one block *of* a program. Many CNC controllers allow several of the same "word" be present in the same block. Thus, several functions can be included in one block, this is normally done by using a word-address format, which is the most popular format used in modern CNC controllers. The sequence *of* the words within one block is usually not important, except for the sequence number that must be the first word in the block. In order to make a program more readable, it is a good practice to follow a fixed sequence. Each word consists *of* a symbol and a numeral. The symbol is either N, G, X, V, and so on. Numerals follow as data in a prespecified format. For example, the format for an *X* word might he ".3.4:' which means three, digits before the decimal point and four digits after the decimal are used. The function of each NC word (code) and their applications are discussed in what follows:

N-code. A part program block usually begins with an "N" word. The N word specifies the sequence number. It is used to identify the block within the program. It is especially useful for program editing. For example when the
format is "4" a proper sequence number would be

<div align="center">N00I0</div>

It is a good practice to program N values in increments of 10 or greater. This allows additional blocks to be inserted between two existing blocks.
G-code. The G-code is also called preparatory code or word. It is used to prepare the MCU for control functions. It indicates that a given control function is requested or that a certain unit or default be taken. There are modal functions and non modal functions. Modal functions are those that do not change after they have been specified once, such as unit selection. Non modal functions are active in the block where they are specified. For example, circular interpolation is a non modal function. Some commonly used G-codes arc listed in the Table 10.1. Some of these functions are explained in what follows.
   G00 is the rapid traverse code that makes the machine move at maximum speed. It is used for positioning motion. When G01. G02, or G03 are specified, the machine moves at the feed speed. G01 is linear interpolation; G02 and G03 are for circular interpolation. For circular interpolation, the tool destination and the circle center are programmed in one block (explained later). G04(dwell) is used to stop the motion for a time specified in the block. G08 and G09 codes specify acceleration and deceleration, respectively. They are used to increase (decrease) the speed of motion (feed speed) exponentially to the desired speed. Before an abrupt turn, decelerate the tool. Rapid acceleration in the new direction may cause a tool to break. The best accuracy can be obtained with acceleration and deceleration codes on and set to lower values. Most NC controllers interpolate circles on only

TABLE 10.1 G CODES

| G00 Rapid traverse | G40 | Cutter compensation cancel |

G01	**Linear interpolation**	**G41**	**Cutter compensation left**
G02	**Circular interpolation CW**	**G42**	**Cutter compensation right**
G03	**Circular interpolation CCW**	**G70**	**Inch format**
G04	**Dwell**	**G71**	**Metric format**
G08	**Acceleration**	**G74**	**Full-circle programming off**
G09	**Deceleration**	**G75**	**Full-circle programming on**
G17	**X-Y plane**	**G80**	**Fixed-cycle cancel**
G18	**Z-X Plane**	**G81-89**	**Fixed cycles**
G19	**Y-Z Plane**	**G90**	**Absolute dimension program**
		G91	**Incremental dimension**

XY. YZ, and XZ planes. The interpolation plane can be selected using G 17, G18 or G 19. When a machine is
equipped with thread-cutting capability, (G33-G35), the part program must specify the proper way to cut the thread. Codes G4O-G43 deal with cutter compensation. They simplify the cutter-center offset calculation. More details of cutter compensation are discussed later in Section 10.2.2. Most canned cycles are manufacturer-defined. They include drilling, peck drilling, spot drilling, milling, and profile turning cycles. The machine-tool manufacturer may assign them to one of the nine G codes reserved for machine manufacturers (GS)-G89). A user also can program the machine using either absolute (G90) or incremental (G91) coordinates. In the same program, the coordinate system can be changed. In order to simplify the presentation, most of the examples given in this chapter use absolute coordinate. Many controllers also allow the user to use either inch units (G70) or metric units (G71). Because hardwired NC circular interpolators work only in one quadrant and many CNC systems allow full-circle interpolation, a (G74) code emulates NC circular interpolation for CNC controllers. G75 returns the CNC back to the full-circle circular interpolation mode.

X,Y,Z,A,B and C-Codes. These words provide the coordinate positions of the tool. *X,* Y and Z define the three translational (Cartesian) axes of a machine. A. B. C are used for the three rotational axes about the *X,Y,* and Z axes. For a three axis there can be only three translational axes. Most applications only require *X.* Y.and Z codes in part programs. However, for four-, five-,or six-axis machine tools. A, B, and C are also used. The coordinates may be specified in decimal number (decimal programming) or integer number (BLU programming). For a controller with a data format of "3.4", to move the cutter to (1.12,2.275. 1.0). the codes are :

<div align="center">X1.1200 Y2.2750 Z1.OOO</div>

In BLU programming, the programmer also may need to specify leading zero(s), or trailing-zero formats. A leading-zero format means that zeros must be entered in the space proceeding the numeric value. In this format, the controller locates the decimal point by counting the digits from the beginning of a number. In trailing-zero format, it is reversed. The number specified is in the BLU unit. The data format "3.4" implies that a BLU equals 0.0001 in. (fourth decimal place). By using the data from preceding example, the leading-zero program would be

<div align="center">X.0112 Y002275 Z00l</div>

In the trailing-zero format, the program looks like   X.11200 Y22750 Z10000

For circular motion, more information is needed. A circular are is defined by the start and end points, the center, and the direction. Because the start point is always the current tool position, only the end point, the circle center. And the direction needs to be specified. I, J. and K words are used to specify the center. Usually, circular interpolation works only on either *X- Y, Y-Z.* or *X-Z* planes. When interpolating a circular are on the X- Y plane, the I word provides the X. coordinate value of the circle center and the J word provides the Y coordinate value. X and Y words specify the end point. Clockwise or counterclockwise motions me specified by the G-code (G02 versus G03). There are many variations in circular interpolation programming. Each NC controller vendor has its own form and formal. Also

they can depend on the combination of absolute or incremental, full-circle on or off modes. The following example is based on absolute programming with full circle on for a hypothetical controller.

F-Code. The F-code specifics the feed speed of the tool motion. It is the relative speed between the cutting tool and the work piece. It is typically specified in in./min (ipm). From a machinability data handbook, feed is given in in./rev (ipr). A conversion has to be done either by hand or on-hoard the controller. Some Controllers offer a G-code that specifies the ipr programming mode. When the ipr programming mode is used, the tool diameter and the number of teeth must he specified by the operator. The F-code must be given before G01, G02 or G03 can be used. Feed speed can be changed frequently in a program, as needed. When an F-code is present in a block, it takes effect immediately. To specify a 6.00-ipm feed speed for the cutting motion in Figure 10.7, one would program   .

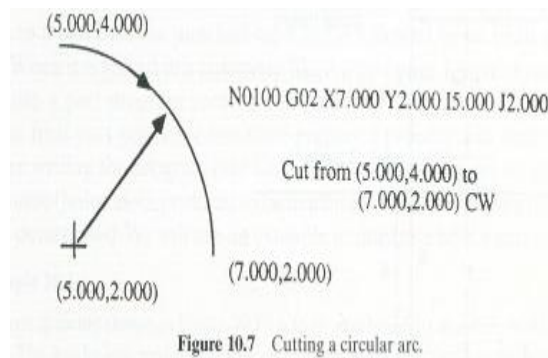N0100 G02 X7.000 Y2.000 15.000 .: .O(I ,c,: OIJ

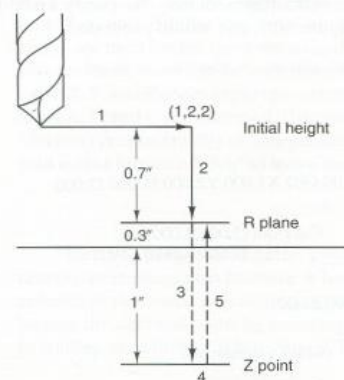Figure 10.7 Cutting a circular arc.   Figure 10.8   Drill cycle.

S-Code. The S-code is the cutting-speed code. Cutting speed is the specification of the relative surface speed of the cutting edge with respect to the work piece. It is the result of the tool (or workpiece in turning) rotation. Therefore, it is programmed in rpm. The Machinability Data Handbook (Machinability Data Center, 1980) gives these values in surface feet per minute (sfpm), and conversion is required before programming is done. When a controller is equipped with a sfpm programming option, the operator must specify the tool diameter. The S-code is specified before the spindle is turned on The S-code does not turn on the spindle. The spindle is turned on by an M-code. To specify a 1000-rpm spindle speed, the . program block is

N0010 S1000

I-code. The I-code is used to specify the tool number. It is used only when an automatic tool changer is present. It specifies the stot number on the tool magazine in which the next tool is located. Actual tool change does not occur until a tool-change M-code is specified.
R –Code. The R -code is used for cycle parameter. When a drill cycle is specified, one must give a clearance height (R plane) (see-Figure 10.8). The R-code is used to specify this clearance height. In Figure 10.8, the drill cycle consists of five operations:
1. Rapid to location (1.2.2).
2. Rapid down to the R plane.
3. Feed to the x point. the bottom of the hole.
4. Operation at the bottom of the hole, for example, dwelling.
5. Rapid or feed to either the R plane or the initial height.
The cycle may be programmed in one block, such as (cycle programming is vendor-specific)

N0010. G81 X1.000 Y2.000 Z0.000 R 1.300

M-Code. The M-code is called the miscellaneous word and is used 10 control miscellaneous functions of the machine. Such functions include turn the spindle on/off, start/stop the machine, turn on/off the coolant, change the tool, and rewind the program (tape) (Table 10.2). M00 and M01 both stops the machine in the middle of a program. M01 is effective only when the optional stop button on the control panel is depressed. The program can be resumed through the control panel. M02 marks the end of the program. M03 turns on the spindle (clockwise). The spindle rpm must be specified in the same line or in a previous line. M04 is Similar to M03, except it turns the spindle on counterclockwise. M05 turns off the spindle. M06 signals the tool-change operation. On a machine equipped with an automatic tool changer, it stops the spindle, retracts the spindle to the tool-change position, and then changes the tool to the one specified in the T -code. M07 and M08 turn on different modes of coolant. MO9 turns off the coolant. M30 marks the end of the tape. It stops the spindle and rewinds the program (tape). On some controllers, more than one M-code is allowed in the same block.

TABLE 10.2 M-CODES

M00	Program stop		M06	Tool change
M01	Optional stop		M07	Flood coolant on
M02	End of program	M08	Mist coolant on	
M03	Spindle CW	M09	Coolant off	
M04	Spindle CCW		M30	End of tape

10.2 MANUAL PART PROGRAMMING

10.2.1 Part Programs

In manual part programming, the machining instructions are recorded on a document, called a part-program manuscript (see Figure 10.9) by the part programmer. The manuscript is essentially on ordered list of program blocks. The manuscript is other entered as a computer file or punched on a paper tape. Each symbol on the numeric or special characters, corresponds to a perforation(s)on the tape (or magnetic bit pattern on a disk) and is referred to as a character. Each line: of the manuscript is equivalent to a block on the: punched tap and is followed by an EOB (end-of-block) character. When it is stored in a computer file, a tape-image format is used.

Because a part program records a sequence of tool motion and operations to produce the final part geometry one must prepare a process plan with setups and fixtures before writing the program (see Chapter 13).The work piece location and orientation, features (holes. slots, pockets) to be machined, tools and cutting parameters used need to be determined. We will use an example to illustrate how a part is programmed.

Example 10.1

The part drawing shown in Figure 10.10 is to he machined from a 4-in. x 4-in.x 2-in, work piece. The work piece material is low-carbon steel. We will use a hypothetical 3-axis CNC machining center for the process. The process plan for the part is as follows:

1. Set the lower left bottom corner of the part as the machine zero point (floating-zero programming).
2. Clamp the work piece in a vice.
3. Mill the slot with a ¾-in four flute flat ,end mill made of carbide. Form the machin ability data hand book, the recommended feed in 0.005 in./tooth/rev and the recommended cutting speed is 620 fpm.
4. Drill two holes with a 0.75-in –diameter twist drill. Use 0.18-ipr feed and 100-rpm speed.

Figure 10.11 shows the setup, fixturing and cutter path. Write a part program for the part.
Solution   The cutting parameters need be converted into rpm and ipm.
Milling :

$$\text{rpm} = \frac{12V}{\pi D} = \frac{12 \times 620 \text{ fpm}}{\pi \times 0.75 \text{ in}} = 3157 \text{ rpm}$$

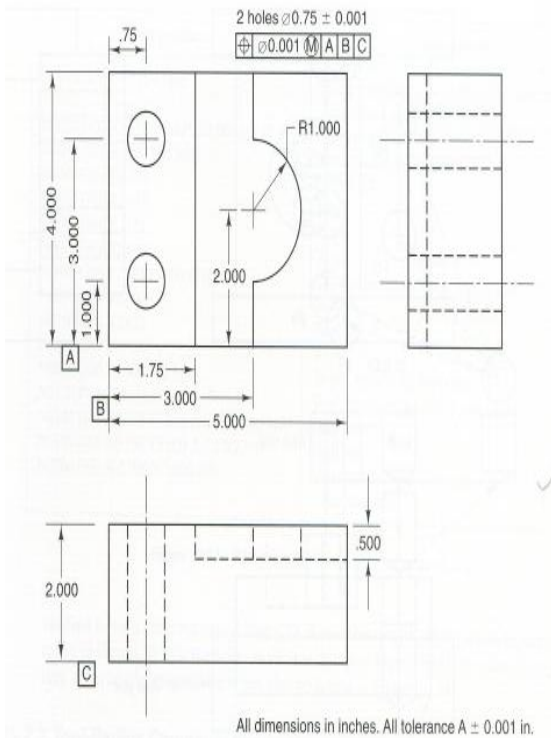$$Vf = nf \text{ rpm} = 4 \text{ tpr} \times 0.005 \text{ iprpt} \times 3157 \text{ rpm} = 63 \text{ ipm}$$

Figure 10.10 An example part.

All dimensions in inches. All tolerance A ± 0.001 in.
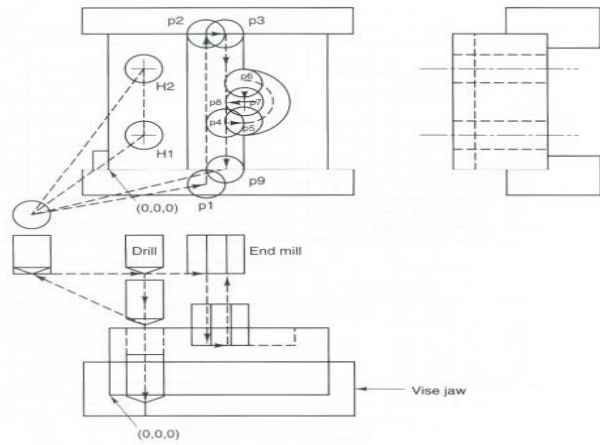

Figure 10.11 Setup and cutter path for the example part.

The coordinates of each point (cutter location) are

p1': (1.75 + 0.375, −0.1 − 0.375,400) = (2.125,−0.475,4.000)
p1: (2.125, −0.475,2.000 − 0.5000) = (2.125,−0.475,1.500)
p2: (2.125,4.000 + 0.100,1.500) = (2.125,4.100,1.500)
p3: (3.000 − 0.375,4.100,1.500) = (2.625,4.100,1.500)
p4: (2.625,2.000 − 1.000 + 0.375,1.500) = (2.625,1.375,1.500)
p5: (3.000,2.000 − 1.000 + 0.375,1.500) = (3.000,1.375,1.500)
p6: (3.000,2.625,1.500)
p7: (3.000,2.000,1.500)
p8: (2.625,2.000,1.500)
p9: (2.625,−0.100,1.500)
p9': (2.625,−0.100,4.000)

Drilling:

$$\text{rpm} = \frac{12V}{\pi D} = \frac{12 \times 100 \text{ fpm}}{\pi \times 0.75 \text{ in}} = 509 \text{ rpm}$$

$$Vf = f \text{ rpm} = 0.018 \text{ ipr} \times 509 \text{ rpm} = 9.16 \text{ ipm}$$

For the milling operation, the cutter is smaller than the slot, and two passes are required. The cutter first moves to p1 (the prime denotes the upper point).There must be clearance between the cutter and the workpiece, so that the cutter will not touch the workpiece during rapid positioning. We will use 0.1in as the clearance. The cutter then plunges down to p1, which is on the slot bottom level. Both p2 and p3 are outside the workpiece to ensure the slot edges are completely cut. The center of the cutter overhangs the edge by 0.1 in. The cutter moves to p4 from p3
to clear the slot. p5 is the beginning of a circular interpolation and p6 is the end. From p6, the cutter moves to p7 to clear the center of the circular area and then moves to p8. After the milling operation, a drill is installed in the spindle through an automatic tool change. The two holes are drilled using a drilling cycle.

Combining the information from the process plan and the cutter-location data, a part program can be written. The program for the example part is shown in Figure 10.12. A step-by-step explanation is presented on the right-hand-side of the figure. The part program is Verified using a program called Mac CNCS, and the results are in Figure 10.13 the result in figure 10.13 is the same as what is shown in Figure 10.11: Thus the program is correct. A three-dimensional view can also be found in Figure 10.14.
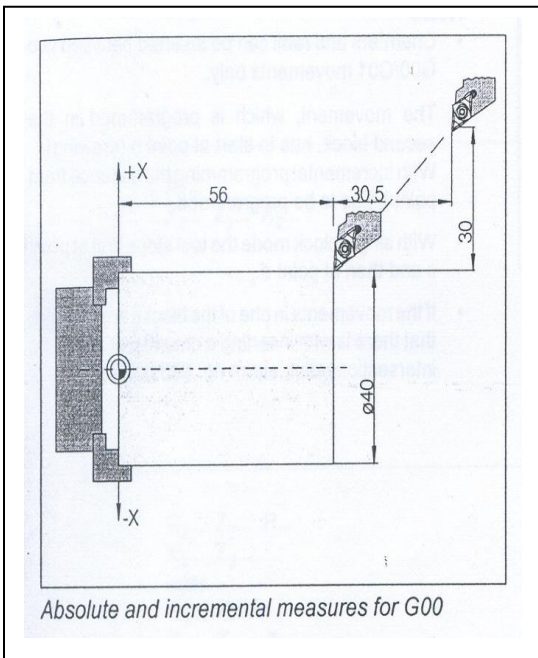
Part program	Explanation

N0010 G70 G90 T08 M06	Set the machine to the inch format and absolute dimension programming.
N0020 G00 X2.I25 V-0.475 Z4.000 53157	Rapid to p1 Down feed to pl. spindle CW
N0030 G01 Z1500 F63 M03	Feed to p2
N0040 G01 Y4.100	To p3
N0050 G01 X2.625	To p4.
N0060 G01 Y.1375	To p5.
N0070 G01 X3.000	Circular interpolation to p6.
N0080 G03 Y2.625 I 3.000 J2.000	To p7. To p8
N0090 G01 Y2.000	To p9.
N0100 G01 X2.625	To p9 spindle off, tool sign.2.
N0110 G01 Y-0.100	Tool change, set new feed and speed.
N0120 G00 Z4.000 T02 M05	Drill hole 1.
N0130 F9.16 S509 M06	Drill hole 2.
N0140 G81 X0.750 Y1.000 Z-0.1 R2.100 M03	Move to home position, stop the machine.
N0150 G81 X0.750 Y3.000 Z-0.1 R2.100	
N0160 G00 X.1.000 Y-1.000 M30	

Figure 10.12 Part program for the part in Figure 10.10
LATHE
Description of G Commands

Absolute and incremental measures for G00

G00 Positioning (Rapid Traverse)
Format
N…. G00 X(U)….Z(W)….
The slides are traversed at maximum speed to the Programmed target point
Incremental and absolute commands can be used at the same time.

Note
 • A programmed feed F will be suppressed while G00
 • The maximum feed is defined by the producer of the machine.
 • The feed override switch is limited to 100%

Example
Absolute G90
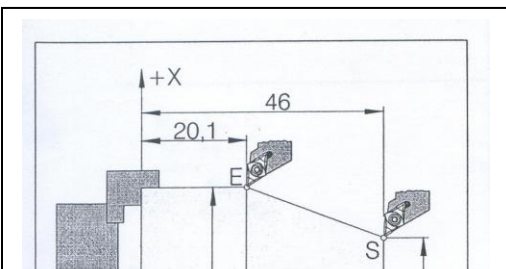N50 G00 X40 Z56
Incremental G91
N50 G00 U-30 W-30.5

G01   Linear Interpolation (Feed)

Format
N…. G01 X(U)….Z(W)….F…

Linear slide movements (face, longitudinal, taper turning)

at the programmed federate.

Example

Absolute G90
N50
…
N20 G01 X40 Z20.1 F0.1

Incremental G91
N.. G95 F0.1
…   N20 G01 X20 W-25.9

FOR LATHE	FOR MILLING
Survey of **G commands** for command Definition C	**G commands**
G00. ....Positioning (rapid traverse)	G00* Positioning (Rapid Traverse)
G01 .....Linear interpolation (feed)	G01 Linear Interpolation
G02 .....Circular interpolation clockwise	G02 Circular Interpolation Clockwise
G03 .....Circular interpolation counterclockwise	G03 Circular Interpolation Counterclockwise
G04+ ...Dwell	G04# Dwell
G7.1 ....Cylindrical Interpolation	G09# Exact Stop
G10 .....Data setting	G10 Data Setting
G11 .....Data setting Off	G11 Data Setting Off
G12.1 ..Polar Coordinate Interpolation ON	G15* End Polar Coordinate Interpolation
G13.1 ..Polar Coordinate Interpolation OFF	G16 Begin Polar Coordinate Interpolation
G17 .....Plane selection)0(	G17* Plane Selection XY
G18 .....Plane selection ZX	G18 Plane Selection ZX
G19 .....Plane selection YZ	G19 Plane Selection YZ
G20 .....Longitudinal turning cycle	G20 Measuring in Inches
G21 .....Thread cutting cycle	G21 Measuring in Millimeter
G24 .....Face turning cycle	G28# Approach Reference Point
G28+ ...Return to reference point	G40* Cancel Cutter Radius Compensation
G33 .....Thread cutting	G41 Cutter Radius Compensation left
G40 .....Cancel cutter radius compensation	G42 Cutter Radius Compensation right
G41 .....Cutter radius compensation left	G43 Tool Length Compensation positive
G42 .....Cutter radius compensation right	G44 Tool Length Compensation negative
G70 .....Inch data Input	G49* Cancel Tool Length Compensation
G71 .....Metric data Input	G50* Cancel Scale Factor
G72+ ...Finishing cycle	G51 Scale Factor
G73+ Stock removal in turning	G52# Local Coordinate System
G74+ Stock removal in facing	G53# Machine Coordinate System
G75+ Pattern repeating	G54* Zero Offset 1
G76+ Deep hole drilling, cut-in cycle in Z	G55 Zero Offset 2
G77+ Cut-in cycle in X	G56 Zero Offset 3
G78+ Multiple threading cycle	
G80 .....Cancel cycles (G83 up to G85)	
G83 .....Drilling cycle'	
G84 .....Tapping cycle	
G85 .....Reaming cycle	
G90. ....Absolute programming	
G91 Incremental programming	
G92+ ...Coordinate system setting, spindle speed	

I it
G94 .....Feed per minute
G95. ....Feed per revolution
G96   Constant cutting speed
G97. ....Direct spindle speed programming
G98. ....Return to initial plane
G99 .....Return to withdrawal plane

1. ....Initial status
+ ....Blockwise effective

Survey of **M commands**
M00 Programmed stop
M01 Programmed stop, conditional
M02 Program End
M03 Main spindle on CW   M04   Main spindle on CCW
M05* Main spindle off   M06 Tool change
M08 Coolant ON   M09* Coolant OFF
M10 Lock dividing head   M11   Unlock dividing head
M19 Oriented spindle Stop
M25 Release Clamping Device   M26   Close Clamping Device
M30 Program End
M71 Puff blowing ON   M72*   Puff blowing OFF
M98 Sub program Call   M99   Sub program End

* Initial status

G57   Zero Offset 4
G58   Zero Offset 5
G59   Zero Offset 6
G61   Exact Stop Mode
G62   Automatic Corner Override
G63   Thread Cutting Mode On
G64*   Cutting mode
G68   Coordinate System Rotation ON
G69   Coordinate System Rotation OFF
G73   Chip Break Drilling Cycle
G74   Left Tapping Cycle
G76   Fine Drilling Cycle
G80*   Cancel Drilling Cycles (G83 bis G85)
G81   Drilling Cycle
G82   Drilling Cycle with Dwell
G83   Withdrawal Drilling Cycle
G84   Tapping Cycle
G85   Reaming Cycle
G86   Drilling Cycle with Spindle Stop
G87   Back Pocket Drilling Cycle
G88   Drilling Cycle with Program Stop
G89   Reaming Cycle with Dwell
G90*   Absolute Programming
G91   Incremental Programming
G92#   Coordinate System Setting
G94*   Feed per Minute
G95   Feed per Revolution
G97*   Revolutions per Minute
G98*   Retraction to Starting Plane (Drilling Cycles)
G99   Retraction to Withdrawal Plane

1   Einschaltzustand
2   Nur satzweise wirksam

UNCONVENTIONAL MACHINING PROCESSES

New machining methods have been developed mainly because of the difficulties in machining hard, high-strength, and temperature-sensitive materials. These processes have been used for removing excess materials from the work piece to obtain the desired shape and size and have come to be known as unconventional machining processes. Table 1 lists these processes and indicates the types of energies and methods of material removal.

Table 1 Unconventional machining processes

Energy type	Mechanics of material removal	Energy source	Process
Mechanical	Plastic shear Erosion	Mechanical motion of tool/job Mechanical/ fluid motion	Conventional Machining Abrasive jet machining (AJM) Ultrasonic machining (USM)
Electrochemical	Ion displacement	Electric current	Electrochemical machining (ECM)
Mechanical and electro-chemical	Plastic shear and ion dis-placement	Electric current and mechanical motion	Electrochemical grinding (ECG)
Chemical	Corrosive Reaction Fusion and vaporization	Corrosive agent electric spark High speed electrons	Chemical machining (CHM) Electric discharge machining (EDM) Electron beam machining (EBM)
Thermal		Powerful radiation Ionized substance	Laser beam machining (LBM) Ion beam machining (IBM) Plasma arc machining (PAM).

ELECTROCHEMICAL MACHINING (ECM)

Electrochemical machining (ECM) refers to the process of metal removal in which electrolytic action is utilized to dissolve the work piece metal. It is, in effect, the reverse of electroplating with some modifications.
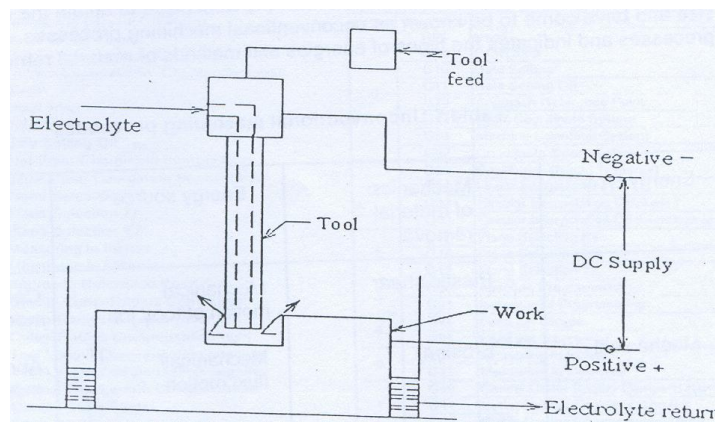
Fig3 Electrochemical machining

The electrochemical machining process is illustrated in Fig.3. The workpiece (Which must be a conductor of electricity) is placed in a tank on the machine table and connected to the positive terminal of a dc supply. The tool electrode, shaped to form the required cavity in the workpiece, is mounted in the tool holder and connected to the negative terminal of the supply. An electrolyte flows through the gap between the tool and the work piece and is then pumped back to the working zone; either through the tool or externally, depending on the application.

The action of the current flowing through the electrolyte is to dissolve the metal at the anode, i.e., the workpiece. The electrical resistance is lowest (and hence the current is highest) in the region where the tool and workpiece are closest. Since the metal is dissolved from the workpiece most rapidly in this region, the form of the tool will be reproduced on the workpiece.

There is no mechanical contact between the workpiece and the tool, and any tendency of the workpiece metal to be plated on the tool is counteracted by the flow of the electrolyte, which removes the dissolved metal from the working zone. Hence there is neither   tool wear nor plating of the workpiece material on the tool so that one tool can produce a large number of components during its life. The voltage and current are constant in the electrochemical machining process.

SUMMARY OF ECM CHARACTERISTICS

Mechanics of material removal	Electrolysis
Medium	Conducting lectrolyte
Tool Materials	Cu, brass, steel
Material removal *ratet* Tool wear rate	∞
Gap	50-300 μm
Maximum material removal rate	15×10^3 mm^3/min
Specific power consumption (typical)	7 W/mm^3 /min
Critical parameters	Voltage, current, feed rate, electrolyte, and electrolyte

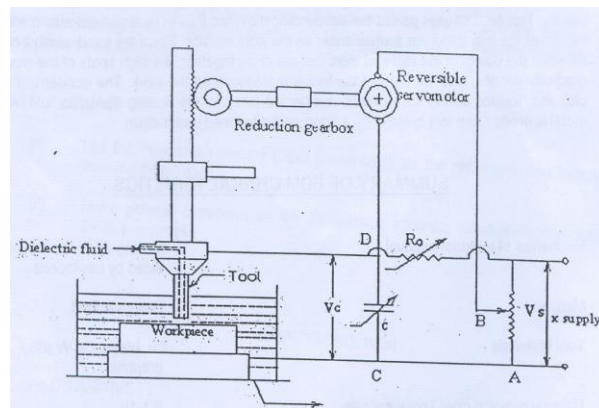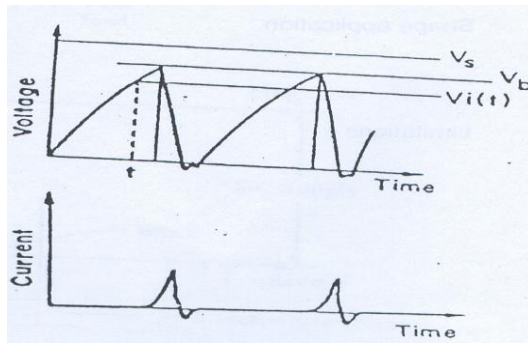|	
	conductivity
Material application	All conducting metals and alloys
Shape application	Blind complex cavities, curved surfaces, through cutting, large through cavities.
Limitations	High specific energy consumption (about 150 times that required for conventional processes), not applicable with electrically non-conducting materials and for jobs with very small dimensions, expensive machine

ELECTRIC DISCHARGE MACHINING (EDM)

Dielectric fluid return
Fig. 4 Electric discharge machining

Electric-discharge machining (EDM), or spark machining, as it is also called, is based on the eroding effect of an electric spark on both the electrodes used to produce it. It has been found *from* observation that if both electrodes are made of the same material, greater erosion occurs on the positive electrode. Hence, to obtain maximum metal removal with minimum wear on the tool' electrode; the work is made the positive and the tool the negative electrode. The principle of the electric-discharge machining process is illustrated by the simplified diagram in Fig. 4. The tool is mounted on the chuck attached to the machine spindle whose vertical feed is controlled by the servomotor through a reduction gearbox; The workpiece is placed in a tank filled with a dielectric fluid; a depth of at least 50mm over the work surface is maintained to eliminate the risk of fire: The tool and work piece are connected to a dc relaxation circuit fed either from a dc generator, or more commonly, a mercury-arc or selenium-type rectifier. Dielectric fluid is circulated under pressure by a pump, usually through a hole (or holes) in the tool electrode. A spark gap of about 10-125 µm is maintained by the servomotor.

When the power supply is switched on, the condenser voltage V_c begins to increase exponentially toward the supply voltage V_s and no current flows. As the voltage V_c builds, it reaches the gap breakdown voltage V_g (determined by the gap width and the dielectric fluid); a spark is produced across the gap, the dielectric fluid ionizes and the condenser is discharged. The surrounding dielectric then deionizes so that it again becomes an effective insulator, and the cycle is repeated. In this way a rapid succession of sparks is obtained, the interval between successive sparks being of the order of 100µs.

Each spark generates a localized high temperature of the order of 12000oC in its immediate vicinity. This heat causes part of the surrounding dielectric fluid to evaporate; it "also melts and vaporizes the metal to form a small crater on the work surface. Since the spark always occurs between the points of the tool and work that are close together, the high spots of the work are gradually eroded, and the form of the tool is reproduced on the work. The condensed metal globules, formed during the process, are carried away by the flowing dielectric fluid. As the metal is eroded, the tool is fed in by a servo controlled feed mechanism.

SUMMARY OF EDM CHARACTERISTICS

Mechanics of material removal	Melting and evaporation aided by cavitations
Medium	Dielectric fluid
Tool Materials	Cu, brass, Cu-W alloy, graphite
Material removal rate! Tool wear rate	0.1-10
Gap	10-125 Jlm
Maximum material removal rate	5x103 mm3/min
Specific power consumption (typical)	1.8 $W/mm3/min$
Critical parameters	Voltage, capacitance, spark gap, di-electric circulation, melting temperature.
Material application	All conducting metals and alloys
Shape application	Blind complex cavities, micro holes for nozzles, through cutting of noncircular holes, narrow slots.
Limitations	High specific energy consumption (about 150 times that in conventional machining); when forced circulation of dielectric is not possible, removal rate is quite low; surfa::;e tends to be rough for larger removal rates; not applicable to non-conducting materials.

INTRODUCTION TO UNCONVENTIONAL MACHINING PROCESSES

OBJECTIVE : To study the characteristic features of ECM and EDM processes.

OUTLINE OF PROCEDURE

Learn the name of the major parts of each machine. Understand the process of material removing in each case and identify the variable that affect the material removal rate and dimensional accuracy.

OBSERVATIONS

(i)   List the important variables (input parameters) for the processes and indicate their available range.

(ii)   Make general comments on the dimensional accuracy achievable from each these processes.

EXERCISE NO: IV(b)

OBJECTIVE

Demonstration of NC machine tools.

Demonstration of Injection Molding.

Demonstration of RP.

Question Bank for laboratory quizzes (ME Lab.)
All answers must be brief and to the point. Sketches must be neat and self-explanatory.

<u>LATHE</u>

1   What are the units of cutting speed and feed on machine tools?
2   What is the use of back gear arrangement in a lathe headstock?(in belt drive machine)
3   How is the rotation imparted to a part, which is to be turned between centers?
4   What are the different ways of mounting the work on a lathe?
5   What is the use of Center drilling?
6   What is the function of chasing dial?
7   What is the difference between the lead and the pitch of a MUL TISTART thread?
8   Calculate the gear ratio between the spindle and the lead screw for cutting a screw with X threads per mm. the lead screw pitch is mm.
9   Why is the main spindle of a lathe hollow?
10   List the type of surfaces produced by turning.
11   Sketch the plan/top view of different types of cutting tools you have used during the lathe exercise and indicate their respective names
12   What are the instruments for measuring the diameters of turned shafts?
13   What is the instrument normally used for-measuring lengths of various parts?
14   Explain how you can determine the taper angle of a taper pin.
15   Calculate the time required for single pass straight turning of a cylindrical bar (diameter D1 length L) at a spindle speed of N rpm and feed f in appropriate units.
16.   Main scale of a Vernier has 10 divisions/cm and the least count of the instrument is 0.01mm. What should be the length of each division on the Vernier scale?
17.   How are the spindle speeds changed?
18.   What is the relation between spindle speed and cutting speed?
19.   What is the relation between feed rate (mm/rev.) and spindle speed?
20.   Are the back gears used to get lower or higher spindle speeds?
21.   Draw figures of left hand and right hand turning tools.
22.   How is the size of lathe specified?
23.   Why are follow-rest and steady-test used?
24.   What is Live Center and Dead center of the lathe?

MILLING

1. How is a milling cutter mounted?
2. What is the main difference between a horizontal and a vertical milling machine?
3. Explain what is meant by a universal milling machine.
4. Why are helical tooth milling cutters usually preferred over straight tooth cutters for slab milling?
5. Why is down milling generally avoided?
6. What are the advantages of up milling?
7. What" special - attachment is needed in the milling machine to perform down milling?
8. In what respect does a slitting saw differ from a narrow milling cutter? .
9. How are milling cutters generally classified?
10. What is the difference between a plain milling cutter and a side-milling cutter?
11. How does a universal dividing head differ from a plain dividing head?
12. When does a universal indexing become essential?

SHAPING

1. What is the driving mechanism on the shaping machine?
2. Why is quick return effect important?
3. What happens to the quick return ratio when the stroke length is reduced?
4. How is feeding done on a shaping machine?
5. Why is clapper box provided on a shaper?

DRILLING AND FITTING

1. To which elements (tool and work) the speeds and feeds are provided on
2. Lathe (ii) Milling machine (m) Shaper and (iv) Drilling machine.
3. What type of speed variation mechanism is provided in the drilling machines you have studies?
4. What material is generally selected for the machine tool structure?
5. What types of guides are used for the main sideways of the basic machine tools?
6. How are the sizes of various basic machine tools specified?
7. Why are square threads used on driving screws of machine tools?
9. Which of the following process are intermittent? (a) Milling (b) Drilling (c) Shaping (d) Turning
10. What makes the simultaneous rotation of the spindle and the feed motion possible on drilling machines?
11. What are the functions of flutes on a twist drill?
12. Explain how power is transmitted from the drill spindle the drill shank?
13. Why are drilled holes generally slightly larger than drill diameter?
14. What is the primary purpose of reaming?
15. Sketch a twist drill and name its principle pats.
16. Name the principal kinds of reamer.
17. How is the diameter of a drilled hole measured?
18. What is the approximate order of magnitude by which hole diameter increases after reaming?
19. What will happen when the drilling is done with dull drill?
20. What are the types tapes in a hand operated tap set?
21. What is a course file used for soft work materials?
22. Describe very briefly the important features of files?
23. Why the number of flutes on a reamer always even?
24. How is the drill held in spindle?

25. Why is the cutting fluid generally not used during drilling cast iron?
26. How are the files classified?
27. Distinguish between second cut and double cut files.
28. What is a scraper?
29. Does a saw blade cut on a return stroke?
30. What operation other than hole drilling can be performed on drilling machine?

CNC

1. What is NC machine?
2. What are the advantages of using NC machine over traditional machine tools?
3. What is "part program" used in CNC machine?
4. What is punched tape? What are the disadvantages of using "punched tapes" used in earlier generation of NC machine tools? .
5. What is CNC (computer numerical control), dNC (direct numerical control), DNC (distributed numerical control)? Specify the advantages of each over other.
6. What are the different components of NC machine tool?
7. What is MCU in NC and write down its function?
8. Clearly explain with figure point-to-point motion control and continuous path mo ion control used in NC machines? Which is control system is generally used to perform machining on the workpieces ?
 9. What is closed loop control system and open loop control system used in NC machine? What are the differences between them? What are the motors used for each type?
10. What is adaptive control system?
11. Write down the differences between incremental positioning and absolute positioning system used in NC machine?
12. Which system of co-ordinate system is used in NC machine? Write down the directions of positive X,Y,Z axis in case of lathe.
 13. Briefly explain about the different types of tape format
   a) Fixed sequential format   '
   b) Block-address format
   c) Tab-sequential format,
   d) Word address format
14. What is ATC in CNC machine?
15. Write down the functions of few G codes like G01, 002 003,004 etc.
16. N0100 G02 X9.000 Y8.000 I5.000 J2.000 F6.000

   Explain each .term for this block of part program.

17. What is BLU programming used in NC part programming?
18. Explain the terms "tool length compensation" and "cutter diameter compensation".
19. What is liner interpolation and what is circular interpolation?
20. What is canned cycle used in NC part program?
21. What is fixed zero system and floating zero system?
22. What is rapid traverse, fine feed and creep feed?
23. What are the different types of tool used in NC machine?

UNCONVENTIONAL MACHINING

1. What lead to the development of unconventional machining processes?
2. How material removal takes place in AJM?
3. Show graphically how material removal rate varies with' nozzle tip distance in AJM?
4. How material removal takes place in USM?
5. What are the typical grain sizes used in USM?
6. Why is USM tool made of ductile metal?
7. What will happen if USM is carried out at low frequency?
8. How material removal takes place in ECM?
9. What are the common tool materials for ECM?
10. What kind of material cannot by ECM?
11. How material removal takes places in EDM?
12. Why is work piece connected to the positive terminal in EDM?
13 What the most commonly used dielectric fluid in EDM?
14. Can EDM be used for machining non-conducting materials?
15. Which unconventional machining process uses fusion and vaporization on material removal?

by- amans@