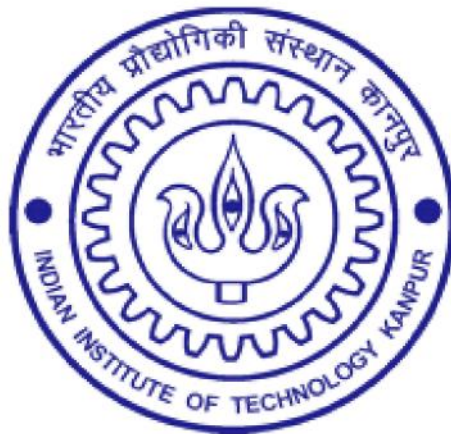


LABORATORY MANUAL

MANUFACTURING PROCESSES – 1



TA 202 LAB
Department of Mechanical Engineering
INDIAN INSTITUTE OF TECHNOLOGY KANPUR

GENERAL INSTRUCTIONS

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. It is unsafe for the students to come to the laboratory wearing garments with parts that hang about loosely. Students should preferably use half-sleeve shirts. The Students should also ensure that floor around the machine is clear and dry (not oily) to avoid slipping.
3. An apron will be issued 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.
5. The student should take the permission of the Lab Staff / Tutor before handling any machine.
6. The student should not lean on the machine when it is working.
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.
8. Students are required to clear off the chips from the machine and lubricate the guides etc. at the end of the session.
9. Laboratory reports should be submitted on A4 size sheets.
10. Reports will not be returned to the students. Students may see the graded reports 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 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) & NC Demonstration	6 hr.
II	Milling	3 hr.
III	Drilling and Fitting (D&F)	3 hr.
IV	CNC Exercise	3 hr.

PROJECT

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	VI Project groups should be formed and names to be given to tutor.
II Second turn	Project idea discussion (minimum 3 ideas) and freezing
III Third turn	Isometric view submission (as per engineering standard)
IV Fourth	Part drawing submission (as per engineering standard)
V Fifth	Lab Examination + CNC Exercise

Number of components in the project should not be more than 25 (Excluding standard parts)

LABORATORY EXERCISES SCHEDULE.

EXERCISE			
POWER TRANSMISSION Video Clip (10 MIN,)			
CLASS	LATHE	MILLING	DRILLING & FITTING
I	A+B	C	D
2	CNC DEMO. (60MIN.) A+B	D	C
3	C+D	A	B
4	CNC DEMO. (60 MIN.) C+D	B	A
5	CNC EXERCISE & Lab Exam		

NOTES: Each Section will be divided in to four groups (A, B, C & D).

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 workpiece is removed in the form of chips by interacting the workpiece with an appropriate tool. This mechanical generation of chips can be carried out by single point or multi point tools or by abrasive operations these are summarized below:

Machining Processes

Single point tool operations

1. Turning
2. Boring
3. Shaping
4. Planing

Multi-point tool operations

1. Milling
2. Drilling
3. Tapping
4. Reaming
5. Hobbing
6. Broaching
7. Sawing

Abrasive operations

1. Grinding
2. Lapping
3. Honing
4. Super-finishing

The process of chip formation in metal cutting is affected by relative motion between the tool and the workpiece 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 (Fig.1.) or boring (Fig.2.) operations. Similarly, planes may be generated by a series of straight cuts without rotating the workpiece as in shaping and planing operations (Fig.3). In shaping the tool is reciprocating and the work piece is moved crosswise at the end of each stroke. Planing is done by reciprocating the workpiece and crosswise movement is provided to the tool.

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 (Fig.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 of the workpiece may revolve while the drill is fed into it (Fig.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 of the feed motion 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 but perpendicular to each other. 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 orientation various geometries can be produced on the workpiece. Consider another case when the generatrix is a circle and the directrix 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 combinations 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 generatrices 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.

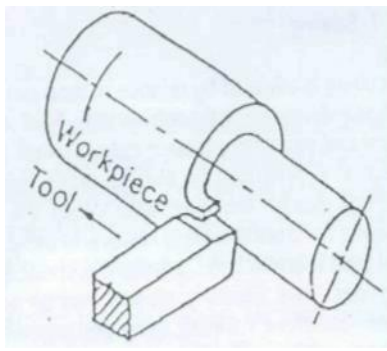


Fig. 1 Straight turning



Fig. 2 Straight boring

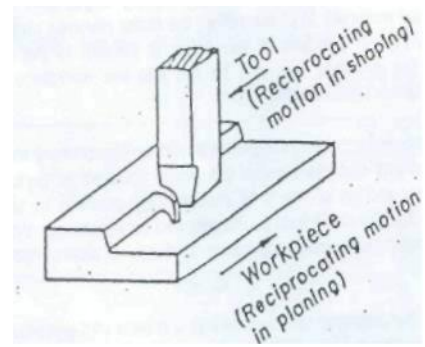


Fig. 3 Shaping and planing

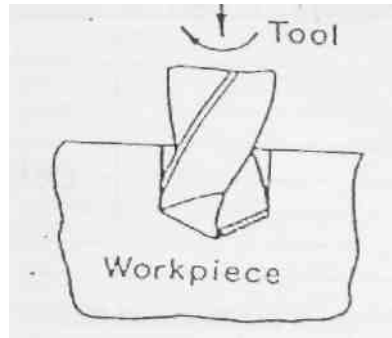


Fig. 5 Drilling

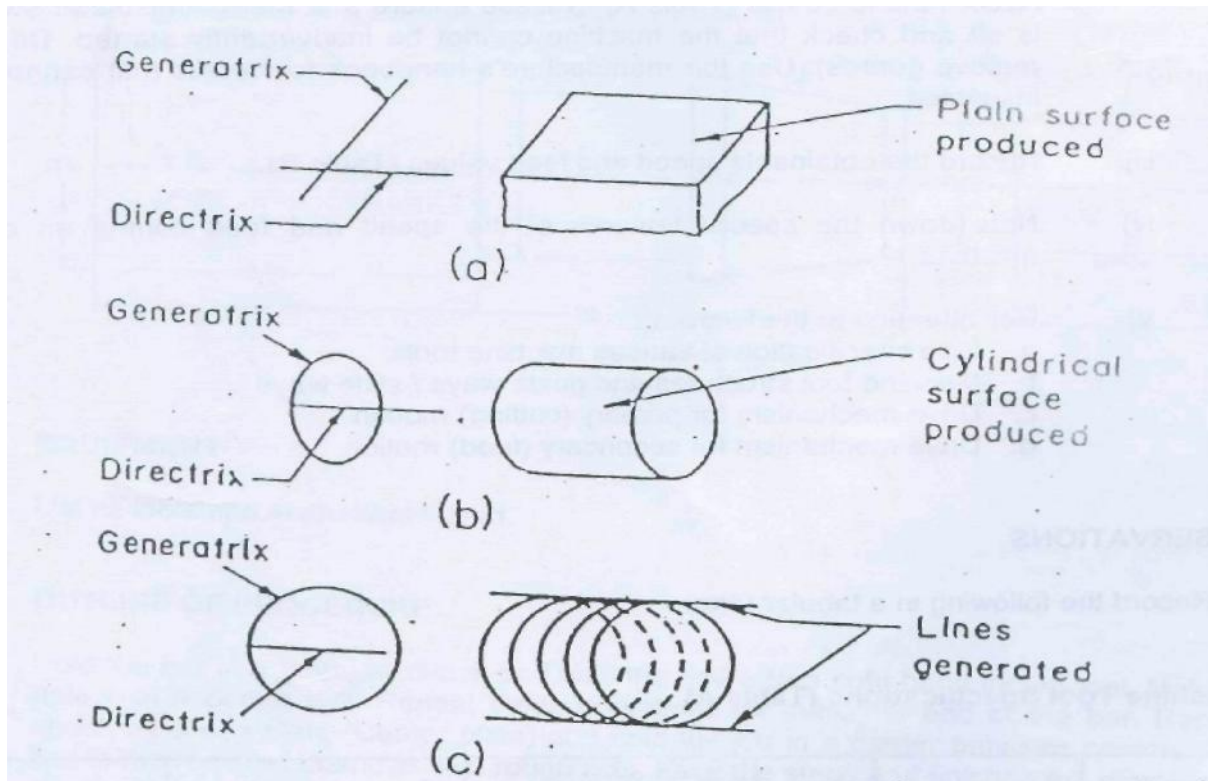


Fig.6 Concept of generatrix and directrix. (a) Rectilinear generatrix and directrix. (b) Directrix perpendicular to the plane of generatrix. (c) Directrix in the plane of generatrix.

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: (a) 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							

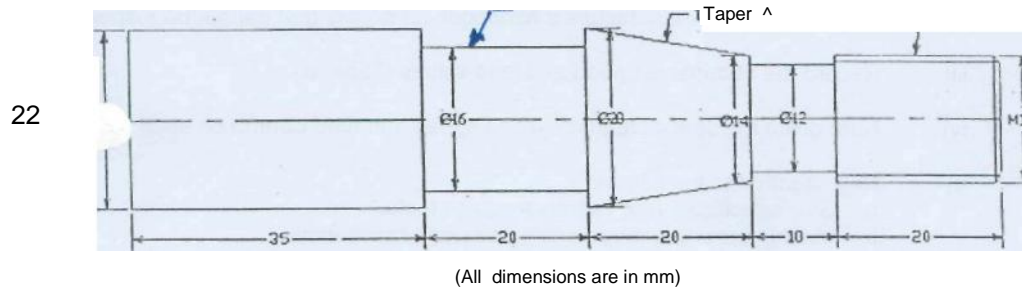
Speed and Feed Data (Table B)

No.	Lathe	
	Speed	Feed
1.		
2.		
3.		
4.		
5.		
6.		
7.		
8.		

(b) Plot the lathe speeds against No. from Table B on a semi-log graph paper and show that the speed steps are in G.P.

PART(B)

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



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 and support the work piece with revolving center. Turn the bar to the required diameter with rough cuts. Face the steps and finishes the diameters to the required sizes. Machine the groove with form tools. Machine the taper with the help of the cross-slide swiveling arrangement required surface. Cut the threads.

OBSERVATIONS

- Measure all dimensions (up to second decimal place) on the specimen turned by your group. Make a neat sketch and indicate all measured dimensions.
- Discuss briefly how tapered portion was turned.
- Show the calculation of the required gear ratio for thread cutting.
- Sketch the main drive unit of the lathe and show how the speed steps are obtained.

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

- What are the units of cutting speed and feed on machine tools?
- What is the use of back gear arrangement in a lathe headstock?
- How is the rotation imparted to a part, which is to be turned between centers?
- What are the different ways of mounting the work on a lathe?
- What is the use of Center drilling?
- What is the function of chasing dial?
- What is the difference between the lead and the pitch of a multistart thread?
- 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 Y mm.
- Why is the main spindle of a lathe hollow?
- List the type of surfaces produced by turning.
- Sketch the plan/top view of different types of cutting tools you have used during the lathe exercise and indicate their respective names.
- What are the instruments for measuring the diameters of turned shafts?
- What is the instrument normally used for measuring lengths of various parts?
- Explain how you can determine the taper angle of a taper pin.
- Calculate the time required for single pass straight turning of a cylindrical bar (diameter D length L) at a spindle speed of N rpm and feed F in appropriate units.
- 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?
- How are the spindle speeds changed?
- What is the relation between spindle speed and cutting speed?
- What is the relation between feed rate (mm/rev.) and spindle speed?
- Are the back gears used to get lower or higher spindle speeds?
- Draw figures of left hand and right hand turning tools.
- How is the size of lathe specified?
- Why are follow-rest and steady-test used?
- What is Live Center and Dead center of the lathe?

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.
- 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.

OBSERVATION : Record the following in a tabular form:

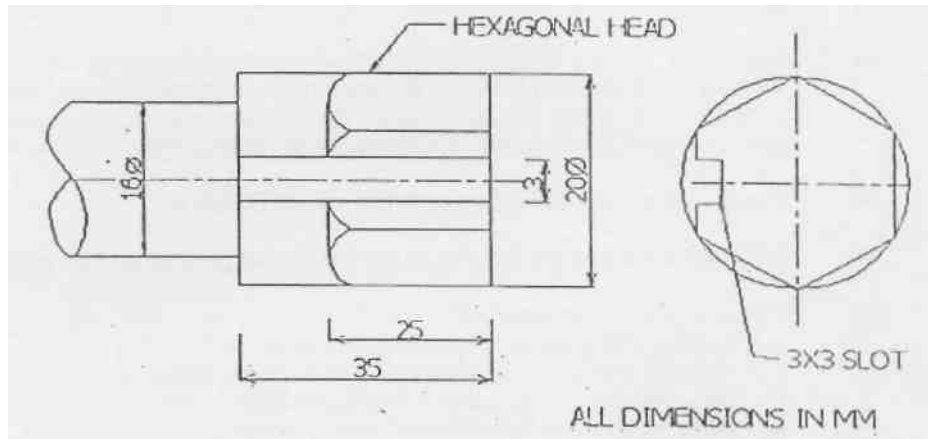
Machine Tool Specifications (Table A)

Machine	Type & Make	Size	Speed given to		Feed given to		Type of Produced
			Tool	Work	Tool	Work	
Milling m/c							

Speed and Feed Data (Table B)

No.	Milling m/c.	
	Speed	Feed
1.		
2.		
3.		
4.		
5.		

PART (B) OBJECTIVE: To machine the hexagonal head and the slot shown in the sketch on the specimen,



EQUIPMENT: List all tools / cutters and instruments used.

OUTLINE OF PROCEDURE

Fit the helical cutter on the arbor and the specimen between the centers of the dividing head and the tail center. 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

- Measure all dimensions (up to second decimal place) on the specimen milled by your group. Make a neat sketch and indicate all measured dimensions.
- Explain in brief how the required indexing was obtained with the dividing head.
- Explain up-milling and down-milling operations. Which one did you use for slot milling and why?
- Explain the advantages of using a helical milling cutter.

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

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

SHAPING: INTRODUCTION AND PRACTICE

PART (A)

OBJECTIVE: To study the characteristic features of Shaper.

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	
Shaper M/c.							

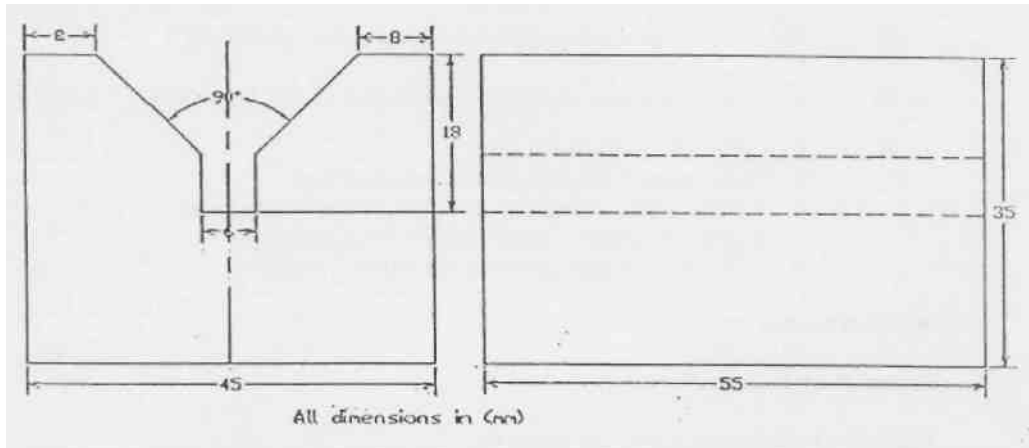
Speed and Feed Data (Table 2)

No.	Shaper M/c.	
	Speed	Feed
1.		
2.		
3.		
4.		
5.		

PART (B)

OBJECTIVE:

To machine a V-block as shown in the sketch out of the workpiece provided.



EQUIPMENT

List all tools and instruments used.

OUTLINE OF PROCEDURE

Hold the work piece in a vice and machine the bottom surface shown in the sketch. Invert the casting in the vice and machine the top surface till the desired height is obtained. Machine the inclined faces using right and left hand tools. Finally machine the groove.

OBSERVATIONS

- Measure all dimensions (up to second decimal place) on the specimen machined by your group. Make a neat sketch and indicate all measured dimensions.
- Explain the quick return mechanism.
- Explain the use of clapper box on the machine.

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

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

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.
- 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 I slide ways.
 - c. Drive mechanism for primary (cutting) motion,
 - d. Drive mechanism for secondary (feed) motion.
- Vi) Drill geometry - check for angles

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 m/c							

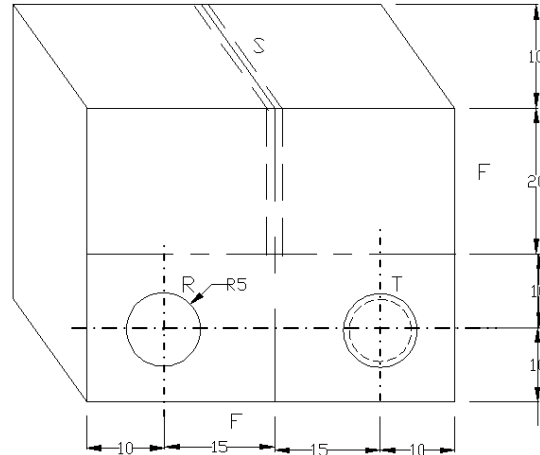
Speed and Feed Data (Table B)

No.	Drilling M/c.	
	Speed	Feed
1		
2		
3		
4		

PART (B)

OBJECTIVE:

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



R - Reamed hole 10mm
T - Tapped hole M 10
S - Saw cut
F - Filled edge

EQUIPMENT : List all tools and instruments used.

OUTLINE OF PROCEDURE

File only two surfaces which make 90° each other mild steel work piece ensuring with a trisquare that the angle is 90° . Mark two centers of the holes. Punch at the center and also punch full length of the hexa cut from top to bottom. Drill And ream the holes as required. Tap the hole a set of three taps

OBSERVATIONS

- Measure all dimensions (up to second decimal place) on the specimen made by your group. Make a neat sketch and indicate all measured dimensions.
- Explain how power is transmitted *from* drill spindle to drill shank.'
- Sketch a reamer and show its main features.
- Explain why a set of three taps was used.

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

- To which elements (tool and work) the speeds and feeds are provided on
- Lathe (ii) Milling machine (m) Shaper and (iv) Drilling machine.
- What type of speed variation mechanism is provided in the drilling machines you have studied?
- What material is generally selected for the machine tool structure?
- What types of guides are used for the main sideways of the basic machine tools?
- How are the sizes of various basic machine tools specified?
- Why are square threads used on driving screws of machine tools?
- Which of the following process are intermittent? (a) Milling (b) Drilling (c) Shaping (d) Turning
- What makes the simultaneous rotation of the spindle and the feed motion possible on drilling machines?
- What are the functions of flutes on a twist drill?
- Explain how power is transmitted from the drill spindle the drill shank?
- Why are drilled holes generally slightly larger than drill diameter?
- What is the primary purpose of reaming?
- Sketch a twist drill and name its principle parts.
- Name the principal kinds of reamer.
- How is the diameter of a drilled hole measured?
- What is the approximate order of magnitude by which hole diameter increases after reaming?
- What will happen when the drilling is done with dull drill?
- What are the types tapes in a hand operated tap set?
- Why is a coarse file used for soft work materials?
- Describe very briefly the important features of files?
- Why the number of flutes on a reamer always even?
- How is the drill held in spindle?
- Why is the cutting fluid generally not used during drilling cast iron?
- How are the files classified?
- Distinguish between second cut and double cut files.
- What is a scraper?
- Does a saw blade cut on a return stroke?
- What operation other than hole drilling can be performed on drilling machine?

Overview of Numerical Control (NC)

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 (MCU) 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.

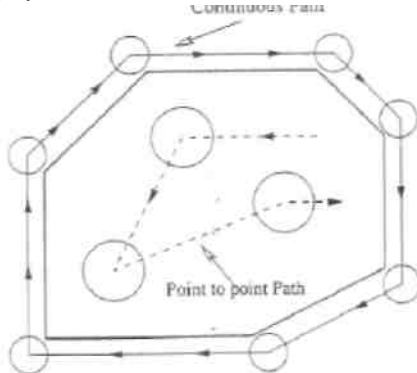


Fig. 15.15 Point-to-point and continuous path.

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 operations. In almost all the cases where PTP control is used, 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 or more axes simultaneously. Here, the paths 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 whereas open-loop systems use stepper motors.

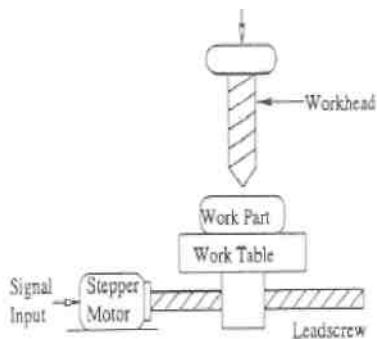


Fig. 15.16 Open-loop system.

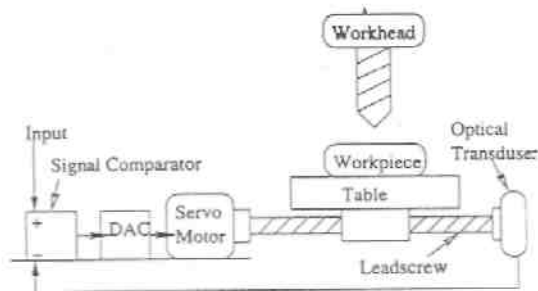


Fig. 15.17 Closed-loop system.

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 the 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:

- * Flexibility is high
- * Scheduling is easier
- * Set-up lead and processing times are less
- * Greater uniformity and accuracy in cutting
- * Lower 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 to 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 motion moves 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 programmer is facing the machine.

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

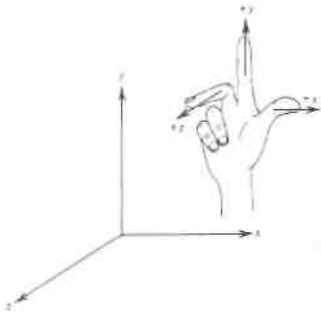


Figure 10.1 A right-hand coordinate system. (Y. Koren, *Computer Control of Manufacturing Systems*, McGraw-Hill, 1983.)

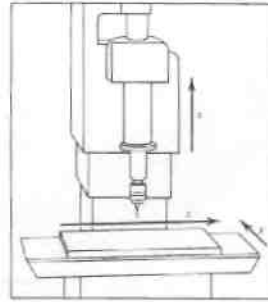


Figure 10.3 Coordinate system for a drill. (Y. Koren, *Computer Control of Manufacturing Systems*, McGraw-Hill, 1983.)

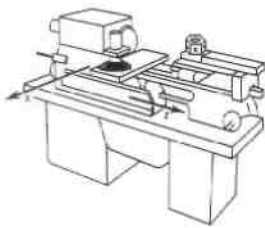


Figure 10.2 Coordinate system for a lathe. (Y. Koren, *Computer Control of Manufacturing Systems*, McGraw-Hill, 1983.)



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 are 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.

10.1.5.1 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: specifies 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 NC 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 be ".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

N0010

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 are 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

G0 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 G40-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 (G80-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 :

X1.1200 Y2.2750 Z1.000

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

X.0112 Y002275 Z001

In the trailing-zero format, the program looks like

X.11200 Y22750 Z10000

For circular motion, more information is needed. A circular arc 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 arc 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 are 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 format. 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 specifies 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-board 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 be 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 I5.000 J2.000
```

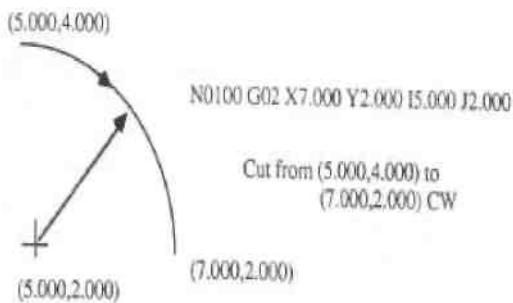


Figure 10.7 Cutting a circular arc.

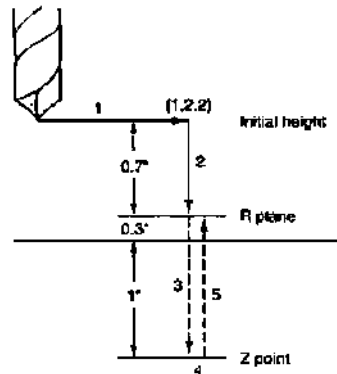


Figure 10.8 Drill cycle.

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 slot 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 to 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 stop 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. M09 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 an ordered list of program blocks. The manuscript is then entered as a computer file or punched on a paper tape. Each symbol or 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 tape 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 be 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 $\frac{3}{4}$ -in four flute flat end mill made of carbide. From the machinability data hand book, the recommended feed is 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-iplr 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:

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

$$V_f = n f \text{ rpm} = 4 \text{ tpr} \times 0.005 \text{ iprpt} \times 3157 \text{ rpm} = 63 \text{ ipm}$$

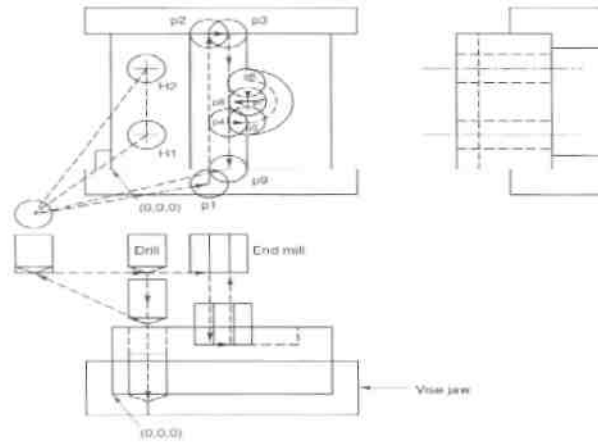
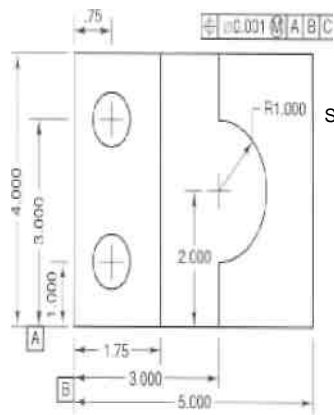


Figure 10.11 Setup and cutter path for the example part.

The coordinates of each point (cutter location) are
 $p1': (1.75 + 0.375, -0.1 - 0.375, 4.000) = (2.125, -0.475, 4.000)$
 $p1: (2.125, -0.475, 2.000 - 0.5000) = (2.125, -0.475, 1.500)$
 $p2: (2.125, 4.000 + 0.100, 1.500) = (2.125, 4.100, 1.500)$
 $p3: (3.000 - 0.375, 4.100, 1.500) = (2.625, 4.100, 1.500)$
 $p4: (2.625, 2.000 - 1.000 + 0.375, 1.500) = (2.625, 1.375, 1.500)$
 $p5: (3.000, 2.000 - 1.000 + 0.375, 1.500) = (3.000, 1.375, 1.500)$
 $p6: (3.000, 2.625, 1.500)$
 $p7: (3.000, 2.000, 1.500)$
 $p8: (2.625, 2.000, 1.500)$
 $p9: (2.625, -0.100, 1.500)$
 $p9': (2.625, -0.100, 4.000)$

Drilling: All dimensions in inches, All tolerance A * 0.001 in.

12V 12 X 100

Rpm = $\frac{TTD}{IT \times 0.75 \text{ in}}$ = 509 rpm

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

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 0.1 in 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 X2.125 V-0.475 Z4.000 S3157	Rapid to p1
N0030 G01 Z1500 F63 M03	Down feed to pl. spindle CW
N0040 G01 Y4.100	Feed to p2
N0050 G01 X2.625	To p3
N0060 G01 Y.1375	To p4.
N0070 G01 X3.000	To p5.
N0080 G03 Y2.625 I 3.000 J2.000	Circular interpolation to p6.
N0090 G01 Y2.000	To p7.
N0100 G01 X2.625	To p8
N0110 G01 Y-0.100	To p9.
N0120 G00 Z4.000 T02 M05	To p9 spindle off, tool sign.2.
N0130 F9.16 S509 M06	Tool change, set new feed and speed.
N0140 G81 X0.750 Y1.000 Z-0.1 R2.100 M03	Drill hole 1.
N0150 G81 X0.750 Y3.000 Z-0.1 R2.100	Drill hole 2.
N0160 G00 X.1000 Y-1.000 M30	Move to home position, stop the machine.

Figure 10.12 Part program for the part in Figure 10.10

FOR LATHE

Survey of **G commands** for command Definition C

G00.	Positioning (rapid traverse)
G01	Linear interpolation (feed)
G02	Circular interpolation clockwise
G03	Circular interpolation counterclockwise
G04+	Dwell
G7.1	Cylindrical Interpolation
G10	Data setting
G11	Data setting Off
G12.1	Polar Coordinate Interpolation ON
G13.1	Polar Coordinate Interpolation OFF
G17	Plane selection 0
G18	Plane selection ZX
G19	Plane selection YZ
G20	Longitudinal turning cycle
G21	Thread cutting cycle
G24	Face turning cycle
G28+	Return to reference point
G33	Thread cutting
G40	Cancel cutter radius compensation
G41	Cutter radius compensation left
G42	Cutter radius compensation right
G70	Inch data Input
G71	Metric data Input
G72+	Finishing cycle
G73+	Stock removal in turning
G74+	Stock removal in facing
G75+	Pattern repeating
G76+	Deep hole drilling, cut-in cycle in Z
G77+	Cut-in cycle in X
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
+	Block wise 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 M09*
M08	Coolant ON
M11	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

FOR MILLING

G commands

G00*	Positioning (Rapid Traverse)
G01	Linear Interpolation
G02	Circular Interpolation Clockwise
G03	Circular Interpolation Counterclockwise
G04#	Dwell
G09#	Exact Stop
G10	Data Setting
G11	Data Setting Off
G15*	End Polar Coordinate Interpolation
G16	Begin Polar Coordinate Interpolation
G17*	Plane Selection XY
G18	Plane Selection ZX
G19	Plane Selection YZ
G20	Measuring in Inches
G21	Measuring in Millimeter
G28#	Approach Reference Point
G40*	Cancel Cutter Radius Compensation
G41	Cutter Radius Compensation left
G42	Cutter Radius Compensation right
G43	Tool Length Compensation positive
G44	Tool Length Compensation negative
G49*	Cancel Tool Length Compensation
G50*	Cancel Scale Factor
G51	Scale Factor
G52#	Local Coordinate System
G53#	Machine Coordinate System
G54*	Zero Offset 1
G55	Zero Offset 2
G56	Zero Offset 3
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

* Einschaltzustand

Nur satzweise wirksam

LATHE Description of G Commands

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.

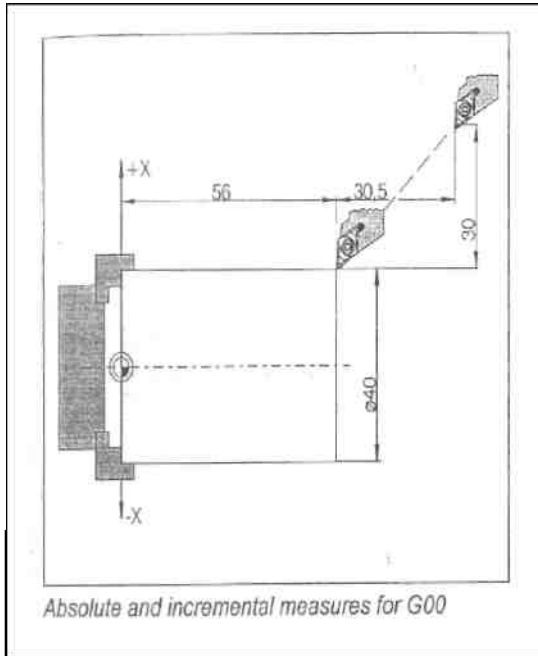
Note

- 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%

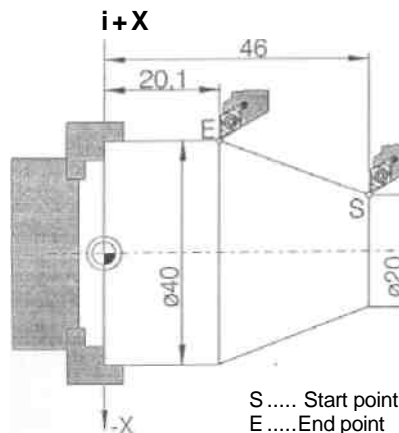
Example Absolute G90 N50

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 feedrate.

Example

Absolute G90

N50

...

N20 G01 X40 Z20.1 F0.1

S..... Start point
E.....End point

Incremental G91 N..

G95 F0.1

N20 G01 X20 W-25.9

Absolute and incremental measures for G00

CNC Questions.

1. What are the different components of NC machine tool?
2. What is MCU in NC and write down its function?
3. Clearly explain with figure point-to-point motion control and continuous path motion control used in NC machines? Which is control system is generally used to perform machining on the work pieces?
4. 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?
5. 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?