A tracer or stylus scans the model and the motion of the tracer i transmitted to the cutting tool by means of servomechanism. Th main disadvantage of the copying method is the time spent o producing the master, as the master is made without automatio and it has to be produced to a high degree of accuracy. Also t recover the cost of manufacture of master, the batch size has to b kept large. A more flexible system, widely used, is the automati lathe machine. The set up of the required dimensions of the part are established by pairs of microswitches and stoppers. This sequence controlled machine is quite flexible but the main disadvantage is that the setting of the limit switches and stoppers for adopting the machine to produce a new product requires many working hours and highly skilled and experienced operator. Also, since the number of limit switches and stoppers is limited, the maximum number of programmed operations on a part is also limited. These problems of automation of medium and small volume production have been overcome by numerically controlled machine tools where the machining operations/processes are controlled with the help of instructions given to the machine tool in a coded form. The simplest definition of numerical control (NC) given by Electronic Industries Association (EIA) is: "*A system in which the actions are controlled by direct insertion of numerical data at some point. The system must automatically interpret at least some portion of this data*".

This definition, as applied to the control of machine tools, may be elaborated as, controlling a machine tool by means of a prepared programme, which consists of blocks or series of numbers, alphabets or alphanumerics. These numbers define the required position of each machine slide, feed, cutting speed and depth of cut. In addition the codes are used to control other functions like coolant ON/OFF, tool change, etc. The data for preparing the coded instructions, called *part programme*, is taken from the finished component drawing. In case of the manufacture of complicated parts, the system calculates the additional data points automatically.

NC MACHINES

The control systems and machine tools in numerically controlled machine tools have varying complexities and capabilities. In the initial stages, the NC machine tools had NC systems added to the machine but only to control the position of the workpiece relative to the cutting tool. The operator had to select the cutting tools, speed and feeds etc. In the next stage, the capabilities of the machine tools improved and in addition to maintaining cutter/workpiece relationship, the material removal was also controlled by the numerical control system. The mechanical design of the machine tool was also improved with the development of recirculating ball screw and better slideways. These machines are referred to as *NC machines*. The instructions to the NC machines are fed through an external medium i.e. paper tape or magnetic tape. The information read from the tape is stored into the memory of the control system called 'buffer storage' and is processed by the machine step by step. So when the machine is working on one instruction block, the next block is read from the tape and stored in the memory of the machine control system. Since the part cannot be produced without a tape being run through the machine, these machines are also called *tape controlled machines*. The tape has to be run repeatedly depending on the number of components to be produced. Also if there is even a minor change in the design of the component, the tape has to be discarded and new tape with changed programme has to be produced.

CNC MACHINES

In case of computer numerical control machine tools, a dedicated computer is used to perform all the basic NC functions. The complete part programme to produce a component is input and stored in the computer memory and the information for each operation is fed to the machine tools i.e. motors, etc. The part programmes can be stored in the memory of the computer and used in future. The conventional NC machine tools are not much in use these days. CNC machine tools are widely used due to many new control features available on these machines.

Some of the additional features available in CNC machine tools are:

(i) The part programme can be input to the controller unit through key-board or the paper tape can be read by the tape reader in the control unit.

(ii) The part programme once entered into the computer memory can be used again and again.

(iii) The part programme can be edited and optimised at the machine tool itself. If there is any change in the design of the component, the part programme can be changed according to the requirements.

(iv) The input information can be reduced to a great extent with the use of special sub-programmes developed for repetitive machining sequences) For common operations such as drilling holes on a pitch circle, special cycle programmes can be built and stored in the computer memory. These sub-programmes or subroutines can be retrieved and used any number of times within a part programme; only certain parameters have to be specified and the computer control carries out the necessary calculations and subsequent actions.

(v) The CNC machines have the facility for proving the part programme without actually running it on the machine tool. The control system processes the part programme and the movement of the cutting tool in each operation is shown on the monitor screen (video display unit). The shape of the component which will be produced after machining is also shown on the screen without actual machining taking place)

(vi) CNC control unit allows compensation for any changes in the dimensions of the cutting tool. When a part programme is written, part programmer has a particular type and size of cutting tool in mind. But while actually using the part programme on the the machine tool that particular cutter may not be available. CNC control system allows the compensations to be made for difference between the programmed cutter and the actual cutter used)

(vii) With the CNC control systems, it is possible to obtain information on machine utilisation which is useful to the management. The control system can provide the information such as number of components produced, time per component, time for setting up a job, time for which a particular tool has been in use, time for which machine has not been working and fault diagnosis, etc.

DIRECT NUMERICAL CONTROL

(Direct Numerical Control (DNC) is a manufacturing system in which a large number of machines are controlled by a computer through direct connections. All the machines are linked to a main frame computer which sends information to individual machines as and when required. The part programmes for all the components, which are to be manufactured on the machines in DNC system, are stored in the memory of the computer. When a machine needs control commands, they are communicated by the computer immediately. There are two alternative system configurations for linking the computer with the machine tools. In the first configuration the main computer is directly linked to the machine and only minimum facilities are provided at each machine tool so that each machining operation can be completed. In this system there could be delays at the main computer in providing details to the machine tool, while it is processing some other information. In the second configuration, the main computer is connected to machine tool through a mini computer called *satellite computer*. The main computer stores the part programmes for all the components to be machined on a particular machine. The satellite computer receives and stores the part programmes. The satellite computer then controls the machine tool operation. The advantage of this system is that the machines can be used independent of the main computer as the main computer is not actively involved in the operation of the machine tool.)

ADVANTAGES OF CNC MACHINES

Some of the advantages of CNC machine tools are briefly discussed below.

(a) Reduced Lead Time

The time between the receipt of a design drawing by the production engineer and manufacturer getting ready to start production on the shop floor, including the time needed for planning, design and manufacture of jigs, etc. is called *lead time*. Since special jigs and fixtures are often entirely eliminated in CNC machines, the whole of the time needed for their design and manufacture is saved. Consider the position of two manufacturers, one equipped with conventional machines and the other with CNC machine tools. For a job requiring special jigs and fixtures and cutting tools, the manufacturing unit with conventional machines will have to spend considerable time in design and manufacture of special tooling before taking up the production of components whereas the unit with CNC machines can start production within a short period of the work being planned and material being available.

(b) Elimination of Operator Errors

(The machine is controlled by programme of instructions stored in the memory of the computer. The programme is checked before it goes to the machine so no errors will occur in the job. Fatigue, boredom or inattention by an operator will not affect the quality or duration of the machining)

(c) Operator Activity

The operator is relieved of tasks readily performed by the machine and is free to attend to matters for which his skills and abilities are essential. Pre-setting of tools, setting of components and preparation and planning of future jobs fall into this category. It is possible for two work-stations to be prepared on a single machine table, even with small batches. Two setting positions are used, and the operator can set up one station while machining takes place at the other. The two jobs need not be identical. Also it is possible for one operator to attend to more than one CNC machine at a time.

(d) Lower Labour Cost

Actual cutting time in the CNC machines is more than conventional machines since the setting time, etc. are lower. Also one operator can run two or more machines or multiple pallet machines are used resulting in reduced labour cost.

(e) Smaller Batches

By the use of pre-set tooling and pre-setting techniques, down-time between batches is kept at a minimum. Periodic machining of small batches is found to be economical and brings about a rapid stock turnover. Larger storage facilities for work-in-progress are not required. Machining centres eliminate some of the set-ups needed for successive operations on one job and time spent in waiting until each machine for successive operations is free. The components circulate round the machine shop in a shorter period, inter-departmental transport costs are saved and 'Progress Chasing' is reduced.

(f) Longer Tool Life

Tools can be used at optimum speeds and feeds because these functions are controlled by the part programme. Programmed speeds and feeds can be overridden by the operator if difficulty in machining is encountered, for example, operator can change the speed and feed if the material has different properties.

(g) Elimination of Special Jigs and Fixtures

Standard locating fixtures are often not used on CNC machines, and cost of special jigs and fixtures is frequently eliminated. The capital cost of jig storage facilities is also reduced. The storage of a part programme is very simple matter, it may be kept for many years and the manufacture of spare parts, repeat orders or replacements is made much more convenient.

(h) Flexibility in Changes of Component Design

The modification or changes in component design can be readily accommodated by re-programming and altering the concerned instructions. There are no jigs and fixture to be altered, hence, savings are effected in time and cost.

(i) Reduced Inspection

The time spent on inspection and in waiting for inspection to begin is greatly reduced. Normally, it is necessary to inspect the first component only. If there is any difference in the dimensions of the machined component, the programme is checked and corrected, if needed.

(j) Less Scrap

Since the operator errors are eliminated; a proven part programme results in an accurate component. However, tool settings, as in a boring bar, do not come under the control of the part programme and periodic checks must be made to ensure that the settings remain correct. Drill or tap breakages are not unknown, but since the tools are operating under controlled optimum conditions, the incidence of breakages should be very small.

(k) Accurate Costing and Scheduling

In CNC machines, the time taken in machining is predictable, consistent and results in a greater accuracy in estimating and more consistency in costing. Consistent operation enables the accurate compilation of shop loading schedules and thus results in a balanced loading and a more predictable output.

DISADVANTAGES OF CNC MACHINES

As with every system, the CNC systems too have certain disadvantages which are given below:

(a) Higher Investment Cost

CNC machine tools represent a more sophisticated and complex technology. This technology costs more to buy than its non - CNC counterpart. The higher cost requires manufacturing management to use these machines more aggressively than ordinary equipment. High machine utilization is essential in order to get reasonable returns on investment. Machine shops must operate their CNC machines two or three shifts per day to achieve this high utilization.

(b) Higher Maintenance Cost

Because CNC is a more complex technology and machines are used harder, the maintenance problem becomes more acute.

(c) Costlier CNC Personnel

Certain aspects of CNC machine operations require a higher skill level than conventional operations. Part programming and CNC maintenance are two areas where personnel with the required skill are in short supply. The problems of finding, hiring and training these people must be considered a disadvantage to the CNC shop.

(d) Planned Support Facility

Since most of the preparatory work for CNC operation is done away from the machine, planned support facilities will be essential e.g. part programming, tape preparation and tool pre-setting are fundamental services that must be considered.

However, the advantages of CNC systems outweigh the disadvantages considerably and the CNC machines have been widely accepted by the industry. CNC is being used in drilling, turning, boring, milling and grinding machines. In addition to metal removing operations, CNC is used on machines like pipe bending, coil winding, flame cutting, welding, wire-cut EDM and many other areas.

PARTS SUITABLE FOR CNC MACHINES

The following factors should be considered while selecting components for machining on CNC machine tools:

- The number of operations per component are large.
- Operations are very complex.
- Size of batches is medium.
- Batches are often repeated.
- Labour cost for the component is high.
- The component requires substantial tooling.
- Components require 100% inspection.
- Set-up and inspection times are high.
- Ratio of cutting time to non-cutting time is high.
- A large variety of components is produced.
- The component requires highly skilled operator.

EVALUATIVE QUESTIONS

1. Explain the meaning of numerical control. What are the areas where numerical control can be used?
2. What are the differences between NC and CNC machines? Discuss the factors due to which CNC machines have completely taken over from NC machine tools.
3. Discuss the advantages of employing CNC machines over manual machines.
4. Discuss the requirements under which CNC machines are best suited.
5. "CNC is a management control system ?" Discuss.

3

CLASSIFICATION OF NUMERICAL CONTROL MACHINES

OBJECTIVES

At the end of this unit you should understand:

1. The different types of NC/CNC systems and their uses.
2. The different types of feedback devices used on CNC machines.
3. The method of giving coordinates on a drawing for CNC machines.
4. How to identify the axis in CNC machines.

INTRODUCTION

A large variety of components is manufactured in industries and every component has its own geometric parameters and hence different machining requirements. For meeting the machining requirements of different components a single CNC system cannot give optimum results. So the CNC machines are designed to meet specific requirements to make them cost effective. CNC/NC machines are classified as follows:

(a) Based on feedback control
(b) Based on control system features

CLASSIFICATION BASED ON FEEDBACK CONTROL

Based on feedback control, the NC/CNC system are classified as open-loop and closed-loop control systems.

(i) Open-loop Control System

Machine tool controls in which there is no provision to compare the actual position of the cutting tool or work piece with the input command value, are called open-loop systems. In the open-loop control system, the electric motor continues to run until the absence of power, from input command signal, indicates that the programmed location has been attained. There is no monitoring of

the actual displacement of the machine slide. In the open-loop control system, the control may indicate a movement of 50mm whereas actually the slide may have moved only 49.8 mm. Fig. 3.1. shows the block-diagram of an open-loop control system.

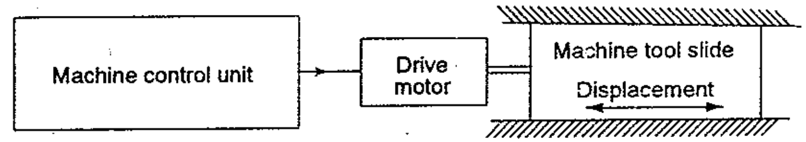


Fig. 3.1. *Block diagram of an open-loop system*

In open-loop control system the actual displacement of the slide may vary with change in external conditions and due to the wear of components of the drive mechanism i.e. backlash errors in leadscrew, etc. Since there is no provision of feedback in the control system periodical adjustments are required to compensates for the changes due to various factors. Although open-loop systems are less accurate compared to a closed loop system, these are adequate for many applications. Open-loop systems are less expensive than closed-loop systems due to the absence of monitoring devices and their maintenance is not complicated.

(ii) Closed-loop Control System

In a closed-loop control system the actual output from the system i.e. actual displacement of the machine slide, is compared with the input signal. The closed loop control systems are characterized by

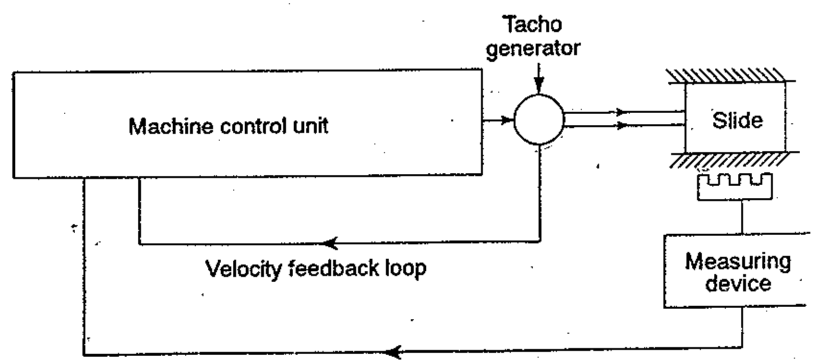


Fig. 3.2. *Block diagram of a closed-loop system*

the presence of feed-back devices in the system. In the closed-loop control system the displacement can be achieved to a very high degree of accuracy because a measuring or monitoring device is used to determine the displacement of the slide. The feedback from the monitoring device is then compared with the input signal and the slide position is regulated by the servo system until it agrees with the desired position. Fig. 3.2 shows a closed loop control system with a provision for feedback for the displacement of position of machining slide. In order to measure the speed of the motor and compare the actual speed with the programmed speed, a velocity feedback system is added to the system.

Feedback Devices

As discussed in the closed loop control system, two types of feedback required are:

(i) Velocity feedback to measure and monitor the speed of the drive motor.

(ii) Positional feedback to measure and monitor the position or displacement of the machine slides.

Velocity Feedback

Velocity feedback is normally provided by a device called *tachogenerator*. A tachogenerator is simply a voltage generator that gives voltage output which is proportional to its speed. The tachogenerator is normally built in the servomotor case and is directly fitted on the servomotor shaft. The output voltage from the tachogenerator is used as feedback to monitor the motor speed. Rotary encoders are also used to provide feedback for velocity control.

Position Feedback

The ideal methods of measuring the displacement or position of the cutting tool will be to continuously measure the position of the cutting tool edge relative to the datum point. This will result in accurate displacements and it will take into account the tool wear, etc. But measurements from cutting edge are not possible due to the presence of chips, coolants, holding devices and in some cases, due to the component geometry itself. The positional feedback is provided by measuring the slide movements with measuring device. The position measuring devices used are either rotary or linear measuring transducers.

Rotary or Angular Position Measuring Transducers

Angular position measuring transducers operate by measuring the angular speed of a rotating element, normally of a leadscrew. From the known value of lead of the leadscrew, movement of worktable or machine slide is calculated by the control system. Most commonly used angular position measuring transducers operate on the *photo electric principle*. Such a transducer is shown in Fig. 3.3. The transducer consists of a disc fitted on the axis of

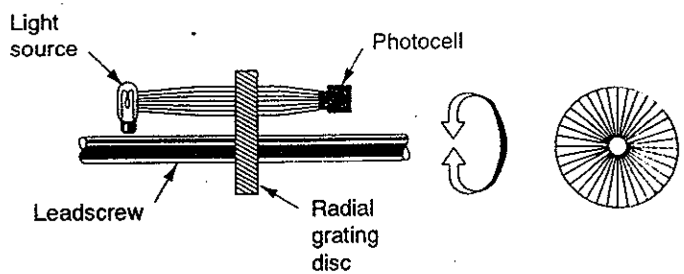


Fig. 3.3. *Angular position measuring transducer*

leadscrew. The disc is made up of uniform alternate transparent and opaque areas. A light source is fitted on one side of the disc and photocell on the other side. When the disc rotates with the rotation of the leadscrew, the photocell will sense light and dark areas alternatively. As the dark area of disc is gradually uncovered, the light intensity falling on the photocell goes on increasing until it reaches a maximum when the transparent part of the disc comes in from of the light source. As the disc continues to rotate the dark area starts to reduce the light intensity falling on the photocell which will gradually reduces to zero when the dark area comes between the photocell and the light source. The photocell gives output voltage based on the intensity of light falling on it and the output from photocell resembles a sine-wave, which is converted into square shaped pulses to make it useful for control purposes. The number of output pulses are then counted. As the output from the photocell is related to the rate at which the transparent areas of the disc come in front of the light source, the rotary speed of leadscrew is calculated from the known number of lines engraved on the rotating disc. The displacement of the slide is then calculated from the lead of the leadscrew. The direction of rotation of the leadscrew is sensed by putting a second photocell in the circuit. The second photocell is positioned in such a way that the output from this photocell is identical to that of first photocell but the output from the two photocells will be out of

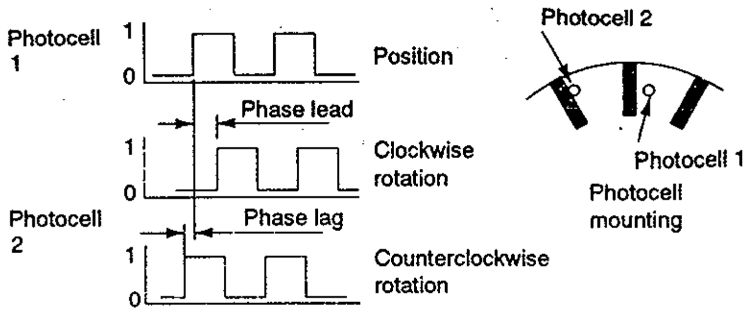


Fig. 3.4. *Two photocells sense direction of movement by phase difference*

phase. This phase difference is used to determine the direction of rotation of leadscrew as shown in Fig. 3.4. Position measurement by angular position measuring transducer is indirect as the output of the transducer has to be converted into table displacement.

Linear Position Measuring Transducers

Linear position measuring transducer also operates on the photo electric principle. The linear measuring system measures the displacement of the machine slide from a fixed datum. A linear measuring system consists of a precision linear scale engraved with close spaced alternate transparent and opaque parallel lines as one unit and a photocell and light source as the second unit. One of the units is fixed on the stationary element of the machine tool and the other unit is fixed to the moving worktable. A pulse is generated by the photocell as it is exposed to light source through the transparent areas of the linear scale. From the known number of engraved lines per unit length on the linear scale and by counting the number of pulses, the displacement of the worktable can be established.

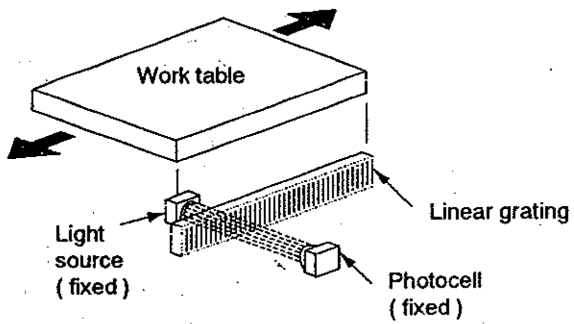


Fig. 3.5. *Linear position measuring transducer*

As in the case of rotary transducers, a second photocell is used

to detect the direction of movement. The linear system may have either a glass scale in which case light passes through the transparent area or a stainless steel scale in which case the light is reflected from the transparent areas. Principle of linear measuring system is shown is Fig. 3.5.

CLASSIFICATION BASED ON CONTROL SYSTEM FEATURES

Based on control system features, the NC/CNC control systems are classified as

(i) Point-to-point control system
(ii) Straight line control system
(iii) Continuous path or contouring control system.

(i) Point-to-point Control System

Point-to-point control is one where accurate positional control is required only to place the machine slides in fixed position and the machine tool slide is required to reach a particular fixed coordinate point in the shortest possible time. The machining operations are performed at specific points and there is no machining while the machine table/slides move from one point to the next. No machining takes place until the machine slides have reached the programmed coordinate point and slide movement ceases. Since there is no machining when the machine slides move form one point to other point, all the slide movements are made in rapid traverse to save time. Also the path of movement is not important but care must be taken to ensure that the cutting tool should not hit the workpiece while moving from one position to the next. The

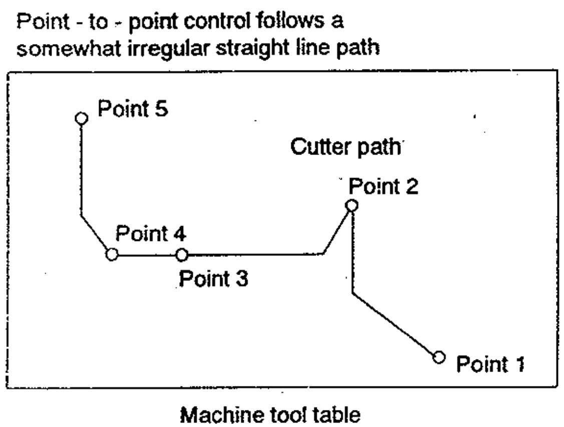


Fig. 3.6. *Point-to-point system*

movement along different axis may be sequential or simultaneous and each axis is controlled independently. The simultaneous movement along the axis results in reduced cycle time. Point-to-point system is suitable for drilling, boring, tapping, punch presses and jig boring machines. An example of machining by point-to-point control system is shown in Fig. 3.6.

(ii) Straight Line Control System

Straight line or straight cut CNC system is an extension of point-to-point control system with the provision of machining along a straight line as in case of milling and turning operations. This is obtained by providing movement at controlled feed rate along the

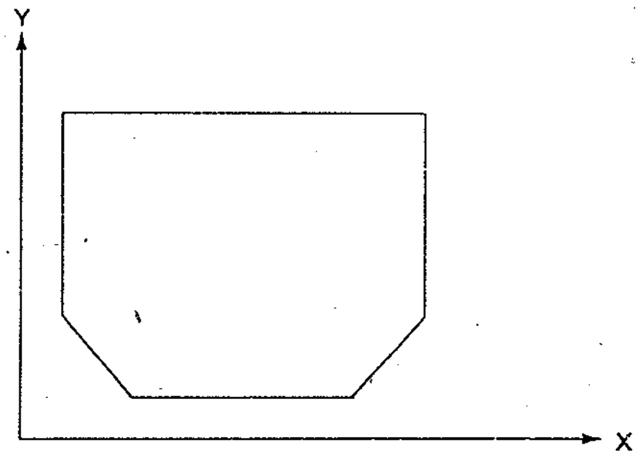


Fig. 3.7. *Straight line system*

axis in the line of motion. It is possible to machine along diagonal lines with movement in two axis at a controlled feed rate. However, in such cases the control system must be capable of calculating and displacing the slides simultaneously at suitable feed rates to reach the desired points because in this case the feed rates along different axis will have to be different. Fig. 3.7 shows the application of straight line control system.

(iii) Continuous Path or Contouring System

The contouring system is a high technology and most versatile control systems. The contouring system generates a continuously controlled motion of the tool and workpiece along different coordinate axis. This control system enables the machining of profiles, contours and curved surface. A system designed for continuous

path machining can, of course, be used for point-to-point and straight line machining but that will result in under utilization of the system. In contouring system, the movement of several machine slides has to be controlled simultaneously so that their relative positions and velocities are established at every point and continuously throughout the operation.

The method by which continuous path system moves from one point to another point is called *interpolation*. There are three types of interpolation i.e. linear, circular and parabolic. Most of the NC/CNC system are capable of providing linear and circular interpolation only. But few control system use parabolic interpolation also. Linear interpolation means moving from one programmed point to another programmed point in straight line. Linear interpolation enables machining along a straight line including taper cuts. While programming with linear interpolation, the coordinates of the end point of line act as the beginning of next line. The control system calculates the intermediate points and decides the speeds of the axis motors if simultaneous movement in two axes is required as in case of taper cuts.

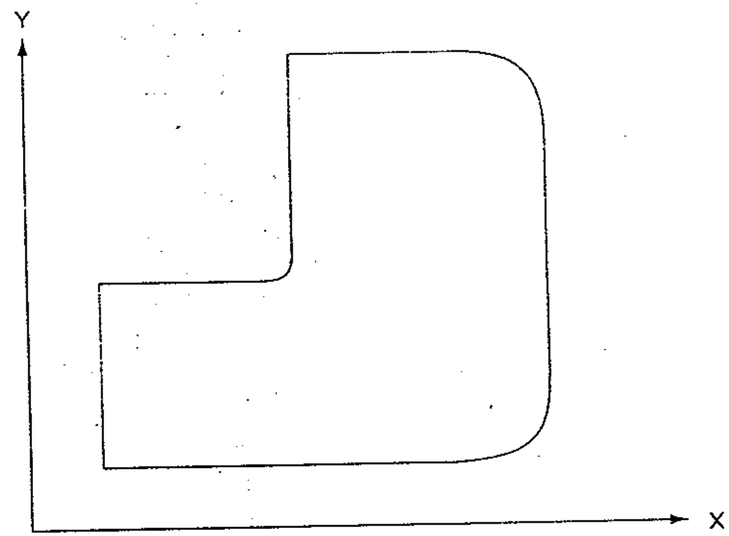


Fig. 3.8. *Contouring system*

Circular interpolation is used to machine circles and arcs. In circular interpolation also the current point acts as the starting point for the circular interpolation and the programmer has to specify end point of the arc and the radius of the arc. Fig. 3.8 shows the use of contouring system with linear and circular interpolation.

METHOD OF LISTING THE COORDINATES OF POINTS IN NC/CNC SYSTEM

Two types of co-ordinate systems are used to define and control the position of the tool in relation to the workpiece. Each system has its own applications and the two co-ordinate systems may be used independently or may be mixed within a CNC part programme according to the machining requirements of the component. The co-ordinate systems used are:

(1) Absolute co-ordinate system
(2) Incremental co-ordinate system

In addition, the dimensions can be given in metric system and in inch (FPS) system.

Absolute Co-ordinate System

In the absolute system the co-ordinates of a point are always referred with reference to the same datum. The datum positions in the *X*-axis, *Y*-axis and *Z*-axis are defined by the user/programmer before starting the operation on the machine. A major advantage of using absolute system is that it is very easy to check and correct a programme written using this method. If a mistake is made in the value of any dimension in a particular block, it will affect that dimension only and once the error is corrected there will be no further problems.

Incremental Co-ordinate System

In the incremental system the co-ordinates of any point are calculated with reference to the previous point i.e. the point at which the cutting tool is positioned is taken as datum point for calculating the coordinates of the next point to which movement is to be made. It is difficult to check a part programme written in incremental dimension mode.

The difference between absolute system and incremental system can be better appreciated with the help of component shown in Fig. 3.9. The co-ordinates of points P1, P2, P3 and P4 in absolute and incremental system are given in the table below:

Point	*Absolute system*	*Incremental System*
P1	1, 3	1, 3
P2	3, 2	2, –1
P3	4, 2	1, 0
P4	4, 3	0, 1

The coordinates of a point can be given either in inch system or in metric system.

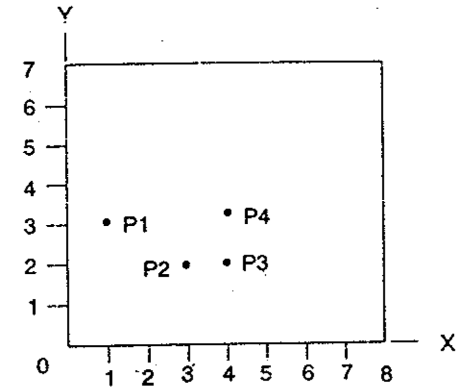


Fig. 3.9.

In addition, the position of a point may be defined using the polar co-ordinates where the distance of the point from a specified datum point and the angle from a specified datum axis give the coordinates of the points.

Most of the CNC/NC machines are capable to work in the absolute as well as incremental coordinate systems. While adopting a single system of co-ordinates will serve to simplify the programming, the two systems may many times be mixed to save time and avoid duplication of effort in making the programme. It is always convenient to write the main programme in absolute system and subroutines are written in incremental mode. However, the programmer has to tell the control system whether absolute or incremental mode and whether inch system (FPS) or metric system of units will be used. This is achieved by giving relevant codes in the programme. These codes will be discussed in the chapter on part programming.

AXIS IDENTIFICATION IN NC/CNC MACHINES

Most of the machines have two or more slideways, disposed at right angles to each other, along which the slides are displaced. Each slide can be fitted with a control system and for the purpose of giving commands to the control system the axis have to be identified. The basis of axis identification is the 3-dimensional cartesian co-ordinate system and the three axis of movement are identified as X, Y and Z axis. The possible linear and rotary movements of machine slides/workpiece are shown in Fig. 3.10. Rotary movements about X, Y and Z axis are designated as A, B and C respectively.

The main axis of movement and the direction of movement along these axis is identified as follows:

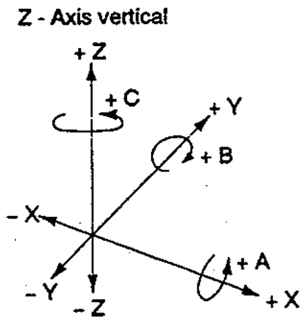
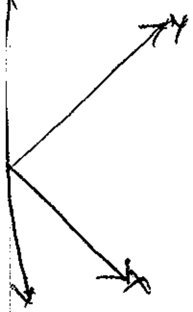


Fig. 3.10. *Possible linear and rotary movements of machine slides*



Z

1. Co-ordinate system

Z-axis: The Z-axis of motion is always the axis of the main spindle of the machine. It does not matter whether the spindle carries the workpiece or the cutting tool. If there are several spindles on a machine, one spindle is selected as the principal spindle and its axis is then considered to be Z-axis. On vertical machining centres, the Z-axis is vertical and on horizontal machining centres and turning centres, the Z-axis is horizontal. Positive Z movement (+ Z) is in the direction that increases the distance between the workpiece and the tool. Convention of designating the Z-axis on milling, drilling and turning machines is shown in Fig. 3.11.

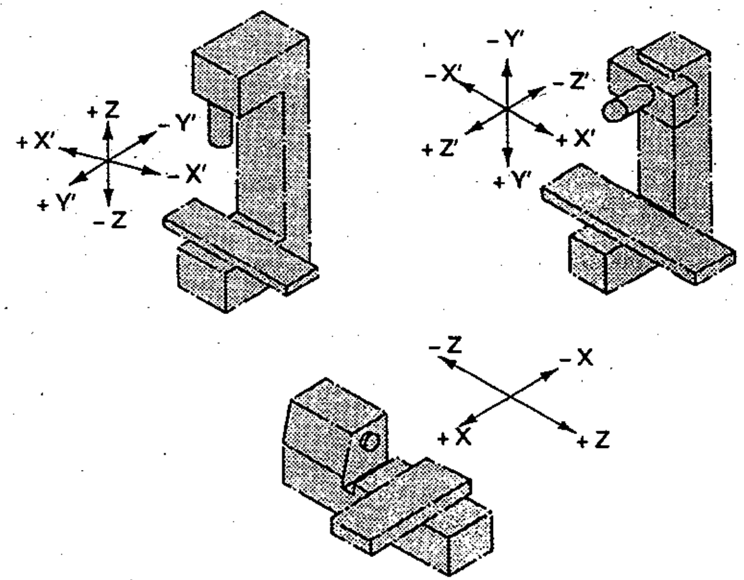


Fig. 3.11. *Designation Z-axis*

X-axis: The X-axis is always horizontal and is always parallel to the work holding surface. If the Z-axis is vertical, as in vertical milling machine, positive X-axis (+X) movement is identified as being to the right, when looking from the spindle towards its supporting column.

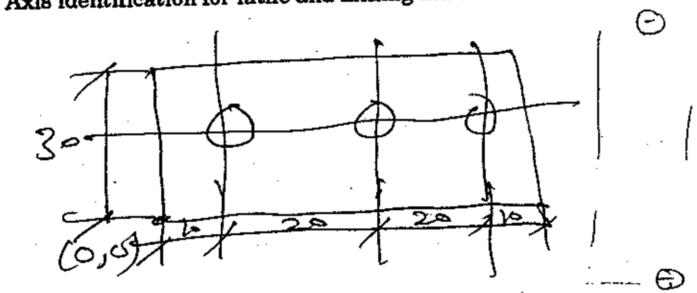
If Z-axis is also horizontal as in turning centres, positive X-axis motion is to the right, when looking from the spindle towards the workpiece.

Y-axis: The Y-axis is always at right angles to both the X-axis and Z-axis. Positive Y-axis movement (+ Y) is always such as to complete the standard 3-dimensional co-ordinate system.

Rotary axis: The rotary motion about the X, Y and Z-axis are identified by A, B, C respectively. Clockwise rotation is designated positive movement and counter-clockwise rotation as negative movement. Positive rotation is identified looking in + X, + Y and + Z directions respectively.

EVALUATIVE QUESTIONS

1. What are the different types of CNC systems? Discuss the relative merits and demerits of open loop and closed loop control systems.
2. Discuss the salient features of point to point, straight line and contouring CNC systems.
3. Why is feedback necessary in machine tools? Discuss the methods used for position and velocity feedback in CNC machines.
4. With the help of suitable figure, explain the following with reference to CNC machines:
   (a) Absolute and incremental dimensioning.
   (b) Axis identification for lathe and milling machines.



+ x

4

CONSTRUCTIONAL DETAILS OF CNC MACHINES

OBJECTIVES

At the end of this unit you should understand:

1. The need for design changes in CNC machines compared to conventional counterparts.
2. The areas where design changes are required in CNC machines.
3. How sliding friction is converted into rolling friction in CNC machines.
4. Advantages of using ball screw and nut assembly.
5. Methods of swarf removal.
6. Safety precautions for the safety of machines and the operator.

INTRODUCTION

The basic design of a conventional machine tool is not suitable for CNC machines. Many design changes are required for CNC machines as compared to their conventional counterparts, due to a number of additional requirements which CNC machines are expected to meet. The manual hand wheel controls in the conventional machines are replaced by axis drive motors in CNC machines. If the axis drive motors have to operate against heavy loads due to friction at the sliding surfaces or due to inertia of moving components or due to some other factors, the motors will have to develop high power output which in turn will ask for motors of large size. In order to limit the size of drive motors and avoid other related problems, the design of CNC machine should be such as to minimise the friction between the sliding surfaces.

Higher cutting speeds and feeds and improved tooling used in CNC machines subject the machine tool to high multidirectional forces. Also the set-up time and the change-over time between the jobs are considerably reduced in CNC machines and most of the time of the machines is spent in actually cutting the material.

Higher percentage of cutting time will results in faster wear of slideways, guideways, lead screw and gears, etc. The higher percentage of cutting time means higher rates of metal removal requiring an efficient system for removal of swarf from the machining area. In addition, safety of the operator working on the machine is very important in CNC machines.

In order to take care of above and many other factors, there is a need for special consideration to be given to the design of CNC machine tools in the following areas:

(1) Machine structure
(2) Slideways
(3) Spindle mounting
(4) Drive units
(5) Elements of transmission and positioning slides
(6) Location of transducers
(7) Tool and work holding devices
(8) Swarf removal
(9) Safety

MACHINE STRUCTURE

The design and construction of CNC machine should be such that it meets the following main objectives:

(i) High precision and repeatability
(ii) Reliability
(iii) Efficiency

To meet the requirements of high precision, repeatability and high efficiency, the numerically controlled machine tools should have a structure that is correctly designed to withstand normal weight distribution. The higher cutting speeds and feeds in CNC machines result in rapid acceleration and deceleration of the slides and the machines are subjected to fluctuating and variable forces during the machining operations. The machine structure should not bend due to the heavy cutting forces. All the parts of the machine structure should remain in relative relationship regardless of the magnitude and direction of the stresses developed due to these forces. Another source of inaccuracy in the CNC machines is the thermal distortion of the machine structure. The design of machine tool structures should be such that the thermal distortion is minimum. The machine tool should be protected from external heat sources and the internal heat sources e.g., headstock motor should be placed centrally so that thermal effects are equally distributed. The machine tool should be provided with an efficient and foolproof lubrication and cooling system. Also the machine structure design should be such that removal of swarf is easy and the chips, etc. do not fall on the slideways.

SLIDEWAYS

In the conventional machine tools, there is a direct metal to metal contact between the slideway and the moving slides. Since the slide movements are very slow and machine utilisation is also low, this arrangement is adequate for conventional machine tools. However, the demand on slideways is much more in CNC machines because of rapid movements and higher machine utilisation. The conventional type of arrangement with metal to metal contact does not meet the requirements of numerically controlled machine tools. The design of slideway in a CNC machine tools should:

(a) Reduce friction
(b) Reduce Wear
(c) Satisfy the requirements of movement of the slides
(d) Improve smoothness of the drive

To meet these requirements in CNC machine tool slideways, the techniques used include hydrostatic slideways, linear bearings with balls, rollers or needles and surface coatings.

Hydrostatic Slideways

In the hydrostatic slideways, air or oil is pumped into small pockets or cavities machined into the carriage or slides which are in contact with the slideway. The pressure of the fluid gradually reduces to atmospheric pressure as it seeps out from the pockets, through the gap between the slide and the slideways. The hydro-

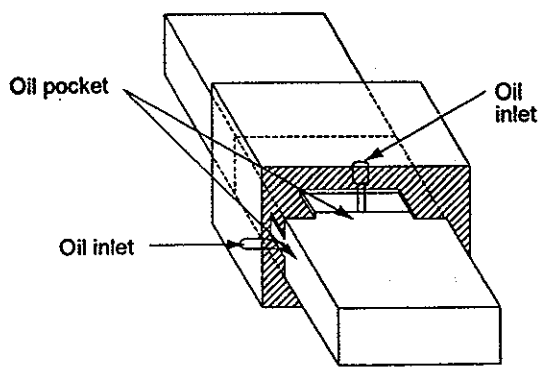


Fig. 4.1. *Hydrostatic slideway*

static slideway provides almost a frictionless condition for the movement of the slide. For efficient operation, it is very important that the fluid and slideways are kept clean. Also, the hydrostatic slideways need a very large surface area to provide adequate support. A hydrostatic slideways is shown in Fig. 4.1.

Linear Bearing with Balls, and Rollers

The sliding friction, due to direct metal to metal contact, between the slide and slideways is replaced with rolling friction by the use of antifriction ball or roller bearings. A linear ball bush shown in Fig. 4.2 uses recirculating balls within a bush type of bearing. These are designed to run along precision ground shafts and offer frictionless movement over varying strokes of length with high linear precision.

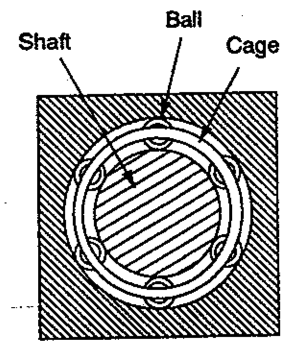


Fig. 4.2. *Linear ball bushing*

For movement along a flat plane, recirculating linear roller bearings are used. The main characteristic of the linear roller bearings is that there is a continuous roller circulation which allows unlimited linear movement. A linear roller bearing, also called a *tychoway*, consists of hardened and precision ground supporting elements and a number of cylindrical rollers. As in case of roller bearings, the rollers are guided between shoulders of the supporting elements with very close tolerances. The guiding element prevents the rollers from falling out and sliding against each other. Also the guiding element assists in smooth return of the rollers to the loading zone. The rollers are in contact with guideways machined on the bed of the machine. This arrangement provides smooth and easy movement but the machine bed has to be machined to an accurate form. Also the machine bed surfaces coming in contact with rollers have to be hardened. To reduce the problem of accurate machining of machine bed, hardened steel guides with special guide forms may be attached to the bed of machine and the rollers can move on the rails. The linear roller bearings can be mounted horizontally for load carrying applications such as machine tool table or they can be mounted vertically to provide support, guidance and motion for the vertical elements of the machine tool. A linear roller bearing is shown in Fig. 4.3. Vee and flat roller arrangement shown in Fig. 4.4 can also be used to provide frictionless linear movement.

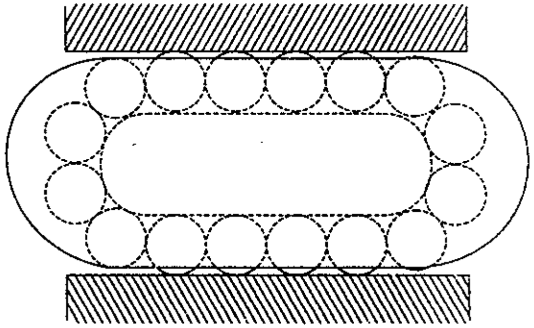


Fig. 4.3. *Linear roller bearing*

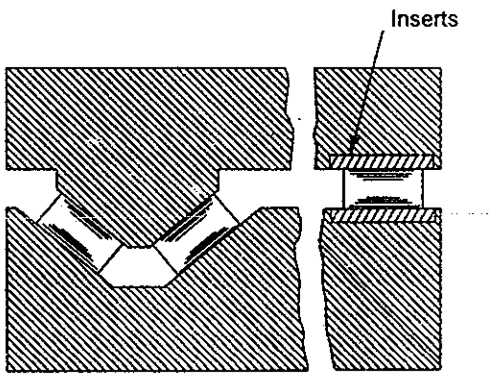


Fig. 4.4. *Vee and flat roller*

Surface Coatings

The guiding surfaces of the machines are sometimes coated with low friction material such as polytetrafluoroethylene (PTFE) or replaceable strips of low friction material are used. When the strips wear to such an extent that the alignment is in error these can be replaced.

SPINDLE

At the high cutting speeds and high material removal rates, the

spindle carrying the workpiece or the tool are subject to deflection and thrust forces. To ensure increased stability and minimise torsional strain, the machine spindle is designed to be short and stiff and the final drive to the spindle is located as near to the front bearing as possible. The rotational accuracy of the spindle is dependent on the quality and design of bearings used. The ball or roller bearings are suitable for high speeds and high loads because of low friction, lower wear rate and lesser liability to incorrect adjustment and ease of replacement when necessary. For efficient service and accuracy the bearings should be of high quality. The vibrations and noise in the spindle can be reduced by using toothed belts and accurate and balanced gears. Adequate supply of lubricants should be ensured to the spindle bearings.

DRIVE UNITS

Drive motors are required to perform the following functions:

(i) To drive the main spindle (Spindle drive)

(ii) To drive the saddles or carriage (Axis drive)

In addition there may be some more motors in the CNC machine for services such as coolant pumps, swarf removal, etc.

Spindle Drive

In CNC machines, large variation in cutting speed is required. The cutting speed may vary from 10 meters per minute to 1000 meters per minute or more. The cutting speeds are provided by rotation of the main spindle with the help of an electrical motor through suitable gear mechanism. The multichange gear boxes with fixed speed ratios used in conventional machine tools are not suitable for CNC machine tools. To obtain optimum cutting speeds and feeds, the drive mechanism should be such as to provide infinitely variable speeds between the upper and the lower limits. The infinitely variable speed systems used in CNC machines employ either electrical motors (A. C. or D. C.) or fluid motors.

Electrical Motors

The spindle is normally driven by an electrical motor either an Alternating Current (A.C.) motor or a Direct Current(D.C.) motor. The drive may be direct from the motor to the machine spindle or indirect, through belt or gear transmission. If belt drive is employed, toothed belts are used and if gear drives are employed, it is constant meshing type of gear box where gears giving various ratios are usually in constant mesh and are operated by remote controlled electromagnetic or hydraulic clutches. Stepped drives have limited use in CNC machines.

A.C. induction motors are used to drive main spindle directly. The A.C. induction motors are more reliable, easily maintainable and less costly compared to other electrical motors. Speed variation in A.C. motors can be obtained by pole change method. The stepless speed variation can also be obtained by varying the frequency of the A.C. current with the help of frequency convertors.

D.C. motors are being extensively used for stepless speed variation of spindle. The stepless variation of speed is obtained by varying the D.C. voltage applied to the motor.

Fluid Motors

Fluid motors are also being used for driving the spindle. Pressurised oil or air supplied by a pump, running at constant speed, is directed on to the blades of the motor which are capable of giving very high rotational speeds.

Axis Drive

All the axis in a CNC machine are controlled by servomotors. The movement along the different axis is required either to move the cutting tool or the work material to the desired positions. In order to accomplish accurate control of position and velocity, stepper motors are used for axis drive. The principal of working of a stepper motor is that on receiving a signal i.e. pulse, from the control unit, the motor spindle will rotate through a specified angle called step. The step size depends on the design of the motor and lies between 1.8 degree and 7.5 degree, which means that one rotation of the spindle can be divided into 200 parts. If a single pulse is received from the control system the motor spindle will rotate by one step. The control unit generates pulses corresponding to the programmed value of movement required of the tool or work. The rate of movement of tool or work is controlled by the speed at which the pulses are received by the stepper motor. The distance travelled by the carriage is calculated by the known value of lead of the axis lead screw and by counting the number of pulses. The rate at which pulses are sent to the stepper motor is accurately governed by the control system. Hence there is no need of providing positional or velocity feedback system. The use of stepper motor considerably simplifies the system as feedback devices are not used. The cost of the machine tool is also less.

However stepper motors are suitable only for light duty machines due to low power-output.

Elements of Motion Transmission

The conventional machines use lead screw for motion transmission purposes. The lead screw with acme-threads is not suitable for CNC machines due to high friction between the lead screw and the nut and poor power transmission efficiency and inaccuracy due to back lash. These problems have been overcome with the use of recirculating ball screw and nut arrangement. Here again, the approach is to replace sliding friction by rolling friction. The connection between the screw and the nut is through an endless stream of recirculating steel balls. The screw thread is, actually, a hardened and ground ball race in which the steel balls, in the nut, circulate. The balls rotate between the screw and the nut and at some point the balls are returned to start of the thread in the nut. The rigidity of the drive system and positioning accuracy can be further improved by pre-loading the nut assembly. A recirculating ball screw is shown in Fig. 4.5.

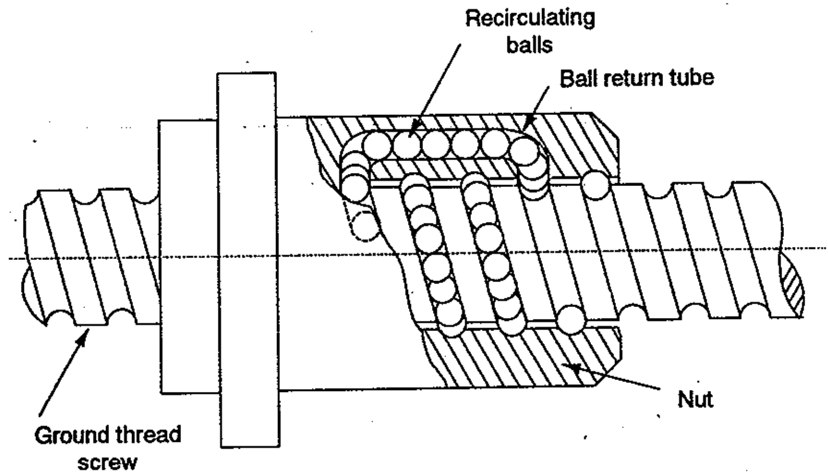


Fig. 4.5. *Recirculating ball screw and nut*

The advantages of using ball screw and nut assembly are:

(i) *High Efficiency*: As compared to conventional lead screw the efficiency of ball screw and nut assembly is very high (over 90%). The power requirement for the ball screw arrangement is also less due to reduced friction.

(ii) *Reversibility*: The ball screw and nut assembly is reversible which makes it possible to backdrive the unit i.e. by applying axial force to either nut or screw, the unconstrained member can be made to rotate.

(iii) *Wear and Life*: The recirculating rollers reduce wear to a minimum and the ball screw, therefore, has longer life without loss of accuracy.

(iv) *No Stick Slip*: Stick-slip is the phenomenon which occurs when small movements between two lubricated elements are required. The lubricating medium tries to cause the mating elements to stick to each other to resist motion and results in a jerky motion as the mating elements try to stick and then slip during their relative movement. Since the sliding metal to metal contact is substituted by rolling contact, the stick-slip phenomenon is eliminated in the ball screw and nut assembly.

Location of Transducers/Control Elements

In CNC machines the control of all machine functions is totally transferred to a computerised control system. The control unit should be situated so that it is convenient for the operator to operate the machine from the central place. The facilities which a control unit should offer are:

(i) Indicate the current status and position of various machine tool features and give feedback.

(ii) Allow manual or semi-manual control of machine tool elements.

(iii) Enable machine tool to be programmed.

The control unit part for allowing manual control and programming of the machine may be housed on the machine structure itself or a separate control panel may be installed near the machine or it may be mounted on a swing arm to allow it to be adjusted according to the position of the operator.

The facilities for indication of present status of the machine features and to give feedback have to be provided at suitable place on the machine tool itself so that actual movement of slides, etc. can be monitored and fedback to the control system. To monitor the position of the slides, two types of transducers are used i.e. linear transducers and rotary transducers.

The linear transducers should be positioned:

(i) near to the sliding surface and lead screw

(ii) in an accessible position for maintenance purposes.

Rotary transducers should be located:

(i) at the driving end of leadscrew

(ii) at the free end of the leadscrew

(iii) on the nut if a fixed screw and rotating nut system is used.

Tool and Work Holding Devices

As already discussed the cutting time in CNC machine ranges from 70 to 80%, the tooling required for these machine tools needs to be specially designed. The requirements of tool and work holding devices and cutting tools for CNC machines are discussed in the Chapter on "Tooling for CNC Machines".

Swarf Removal

CNC machines are designed to work at optimum cutting conditions with the improved cutting tools on a continuous operation basis. Since the cutting time is much more in CNC machines, the volume of swarf generated is also more. Unless the swarf is quickly and efficiently removed from the cutting zone, it can affect the cutting process and the quality of the finished product. Also the swarf cannot be allowed to accumulate at the machine tool because it may hamper the access to the machine tool. In addition some auxiliary functions like automatic component loading or automatic tool change may also be affected by accumulation of swarf. To avoid these problem an efficient swarf control system should be provided with the CNC machine tools with some mechanism to remove the swarf from the cutter and cutting zone and for the disposal of swarf from the machine tool area itself.

Swarf Removal from Cutting Zone

The swarf removal from the cutting zone is generally taken care of by the design configuration of the machine. Slant bed and vertical bed turning centres have the advantages over flat bed or horizontal bed configuration in that the swarf does not accumulate on the guideways. Similarly horizontal machining centres are advantageous to vertical machining centres. But swarf removal by gravity is not adequate in CNC machines. To supplement the gravity system, multiple coolant jets are arranged around the cutting tool and the coolant, under pressure, takes away the accumulated swarf from the cutting area. Also on some machines it is possible to programme 'coolant wash' stage in the part programme, where the cutting area is flooded with pressurised coolant and the swarf from the cutting tool and the workpiece is washed away. Compressed air jets are also used for swarf clearance from the cutting zone.

Swarf Disposal From Machine Tool

Continuously operating linear or rotary conveyors are used for removing the swarf from the machine tool. The system is such that the swarf from the cutting zone falls directly on the conveyor and is immediately taken away. The swarf from the conveyor is taken to disposal bins, which can then be collected and removed from the machine area. A linear transport conveyor system is shown in Fig. 4.6.

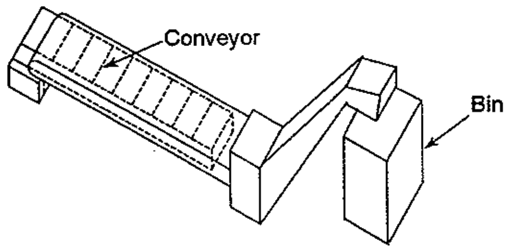


Fig. 4.6. *Linear conveyor for swarf removal*

Guarding and Safety

Since the CNC machines are under continuous automatic operation, there is a need to protect the machine guideways and to ensure safety of the operator since the machines run at high speeds with automatic auxiliary operations.

Protection of Machine Guideways

Protection of machine guideways, drive screws and transducers, etc. is very important for efficient working and long life of the machine. Various types of collapsible guards and covers are used to protect those elements. All the sliding elements are fitted with wipers and the drive screws are normally protected by using telescopic covers. Jets of cutting fluid are used to wash away swarf and clear the tool work area.

Safety of Operator

Safety of operator is very important aspect which cannot be overlooked. To ensure safe working conditions the CNC machine tools are provided with metallic or plastic guards. Where it is not possible to provide effective guards, proximity protection is provided by pressure mats or light barriers.

Perimeter Guards: The overall guards or perimeter guards serve as an enclosure for the machine tool. The perimeter guards protect the operator against flying swarf and from any accident by hitting against the moving components when the machine is working. The access to the machine is provided through large sliding doors for setting up the machine and for loading/unloading of the workpiece. The doors have various types of inter-lock switches fitted on them. If the door is opened when the machine is working, the control unit will flash a warning signal, or activate a auditary signal like a buzzer. On some machines the power to the machine may be cut off if the doors are kept open beyond a certain period of time. During set-up period, the warning signal can be cancelled by the operator. The guards are fitted with transparent windows so that the machining area is visible from the operator side.

Pressure Mats: The pressure mats are used on milling, drilling or grinding machines where the machine table can move to the either side of the machine. Since the tables move at a rapid rate, it may cause some accident if the operator is standing too close to the machine. The pressure mats are placed around the machine and if someone crosses the mat, a warning signal is generated.

Light Barrier: Light barriers are also provided on milling, drilling and grinding machines. The light barrier consists of a light source, usually infra-red, sending a beam to light sensitive cell. If anything obstructs the light beam, a warning signal is generated. The light barriers are placed around the machine. They can be made inactive by the operator, if required.

EVALUATIVE QUESTIONS

1. Why do CNC machines require design changes over their manual counterparts? Discuss.
2. What are the different areas where design changes are required in CNC machine? Give reason for each area.
3. What are the factors considered while designing the structure of a CNC machine?
4. Discuss how sliding friction is converted into rolling friction in CNC machines?
5. Discuss the advantages and disadvantages of using a Ball Screw and Nut Assembly in CNC machines.
6. Explain why CNC machines have to be fitted with comprehensive guards and interlocking devices?
7. Explain the following:
   (a) Optimum cutting conditions can be obtained on CNC machines but not on conventional machines. Why?
   (b) Factors considered while designing the spindle of a CNC machine.
   (c) Axis drive motors used in CNC machines.
   (d) Methods of swarf removal in CNC machines.

5

FUNDAMENTALS OF PART PROGRAMMING

OBJECTIVES

At the end of this unit you should understand:

1. The basic principles of manual part programming.
2. The use of various codes in part programming.
3. How to develop part programme for various jobs.
4. Meaning and usage of cutter radius compensation.

INTRODUCTION

Part programme is an important component of the CNC system. The shape of the manufactured components will depend on how correctly the programme has been prepared. Part programme is a set of instructions which instructs the machine tool about the processing steps to be performed for the manufacture of a component. Part programming is the procedure by which the sequence of processing steps and other related data, to be performed on the CNC machine is planned and documented. The part programme is then transferred to one of the input mediums, which is used to instruct the CNC machine.

NC WORDS

The combination of binary digits (bits) in a row on the tape denotes a character. A NC word is a collection of characters used to form an instruction. Typical NC-words are X-position, Y-position, feed rate, etc. A collection of NC words is called a block and a block of words is a complete NC instruction. Following are the NC-words used in the formation of blocks. All the NC words may not be used on every CNC machine.

(i) Sequence Number (N-Word)

The first word in every block is the sequence number. The sequence number is used to identify the block. The sequence number is preceded by word N and is written as N 0001, N 0002,

N 9999, etc. The programme is executed from lowest block number to highest unless instructed otherwise. It is customary to start with block No. 0001 or 0010 and proceed in steps of 5 or 10, so that accidently omitted block may be inserted easily.

(ii) Preparatory Function (G-Words)

The preparatory word prepares the control unit to execute the instructions that are to follow. The preparatory function is represented by two digits preceded by G i.e. G00 .... G99. The preparatory function enables the controller to interpret the data which follows and it precedes the coordinate words. For example G01 is used to prepare the controller for linear interpolation. Some of the preparatory functions are given in Table 5.1.

TABLE 5.1 PREPARATORY FUNCTIONS (G CODES)

Code	*Function*
G00	Rapid traverse
G01	Linear interpolation
G02	Circular interpolation (clockwise)
G03	Circular interpolation (counter clockwise)
G04	Dwell
G05	Hold/delay
G17	XY plane designation
G18	ZX plane designation
G19	YZ plane designation
G33	Thread cutting
G40	Cutter compensation-cancel
G41	Cutter compensation-left
G42	Cutter compensation-right
G63	Thread cutting cycle
G70	Dimensioning in inch units
G71	Dimensioning in metric units
G80	Canned cycle-cancel
G81-G89	Canned Cycles
G90	Absolute dimensioning
G91	Incremental dimensioning
G92	Zero preset
G94	Feed rate mm/min
G95	Feed rate mm/rev

(iii) Coordinates (X-, Y- and Z-Words)

These words give final coordinate positions for X, Y, Z motions. In two-axis CNC system only two-coordinate words would be used. To specify angular positions around the three-coordinate axis additional, i.e. a-word and b-word, are used. In addition, the words I, J, K, are used to specify the position of arc centre in case of circular interpolation. Different CNC systems use different formats for expressing coordinates of a point. In some systems, the decimal point is not coded but the control system automatically provides a decimal point at a pre-set position. But in such cases leading zeros may have to be given in the programme, e.g., x = 5.60 will be written as x 000560 in a particular system where x dimension is to be followed by 6 digits and the control system places decimal after leaving two least significant digits. However, some CNC control systems accept the decimal form. Here, we will adopt this convention of expressing the data in decimal form. While giving the data positive sign is optional but negative sign has to be given in case of negative dimensional positions.

(iv) Feed Function (F-Word)

The feed function is used to specify the feed rate in the machining operation. The feed rate is expressed in millimeters per minute (mm/min) or mm/rev. If the feed is 200 mm/min, it will be represented as F 200. The appropriate G code should be specified to instruct the machine whether the feed value is in mm/min or in mm/rev. (G94 or G95).

(v) Spindle Speed Function (S-Word)

The spindle speed is specified either in revolutions per minute (r.p.m.) or as meters per minute. If the speed is given in meters per minute, the control unit calculates the rev/minute using the appropriate formulae. If the machine is required to run at 800 rpm the speed will be specified as S 800.

(vi) Tool Selection Function (T-Word)

The T-word is needed only for machines with programmable tool turret or automatic tool changer (ATC). Each tool pocket on the tool turret or ATC has a distinct tool number. The T-word in the part programme specifies which tool is to be used in the operation. The tool number for a particular operation is specified as T00 to T99. Also with each tool code, the corresponding tool length offset is also specified with the help of two additional digits i.e. T01.01 where second 01 denotes the tool length offset for tool No. 01.

(vii) Miscellaneous Function (M-Word)

The miscellaneous function word is used to specify certain miscellaneous or auxiliary functions which do not relate to the dimensional movements of the machine. The miscellaneous functions may be spindle start, spindle stop, coolant ON/OFF, etc. An

example of M-word is M02 which indicates end of programme. The miscellaneous functions are given in Table 5.2.

TABLE 5.2 MISCELLANEOUS FUNCTIONS (M CODES)

Code	*Function*
M02	Programme stop
M03	Spindle start (clockwise)
M04	Spindle start (counter clockwise)
M05	Spindle stop
M06	Tool change
M08	Coolant on
M09	Coolant off
M30	Programme stop and tape rewind

(viii) End of Block (EOB)

The EOB symbol identifies the end of instruction block.

The use of NC words will be made more clear in the later part of this chapter with the help of suitable examples.

Important Note: Part programme commands for axis motion are given with the assumption that it is the cutting tool that moves. So when an axis motion command X 200.5 or Y-35.6 is given to the system, it is expected that the cutting tool will move to the programmed position. However, if a machine tool moves the workpiece instead of the tool, it must execute the motion command in a direction opposite to that programmed. This fact is taken into account by machine tool and control system designer. So in all cases the part programme axis motion should be written assuming that the cutting tool moves. Also it is the cutter path which is defined in the programme, so in case of programming for milling cutters, the axis motion and cutter positioning should be programmed with respect to centre the cutter.

The extensively used G codes are discussed here:

Rapid Traverse Function (G00)

In order to reduce the cycle time, the non-machining movements are executed in the rapid traverse mode in NC/CNC machines. The G code for rapid traverse is G00. All the movements in rapid traverse are performed at the maximum feed rate available on the CNC machine tool, which depends on the design of the machine. The instruction block with rapid traverse function is written as follows:

N - G00 X— Y— Z— EOB

In addition some more functions like M, S & T can also be given in the same block. The instruction block with G00 will move the cutting tool and/or workpiece to the position with the coordinates given in that block at the rapid feed rate. Since all the movements are executed at rapid feed rate, it should be ensured that before the start of rapid movement the cutting tool position is such that it will not hit the workpiece during these movements. G00 is a modal function i.e. if a number of consecutive blocks with rapid traverse are included in the programme, then G00 is written in first block only and it remains active in the subsequent blocks.

Linear Interpolation Function (G01)

Any machining along a straightline, including taper lines, is done using the linear interpolation function G01. The general format for writing an instruction block using G01 is

N — G01 X— Y— Z— F— EOB

The above instruction block will move the cutting tool to a position specified by the coordinates in this block. The feed rate at which the cutting tool is required to move is also specified while using G01. However, if the feed rate has been defined in one of the previous instruction blocks, the same feed rate will remain active in the current instruction block also.

The use of G00 and G01 in writing the instructional blocks is discussed with reference to Fig. 5.1.

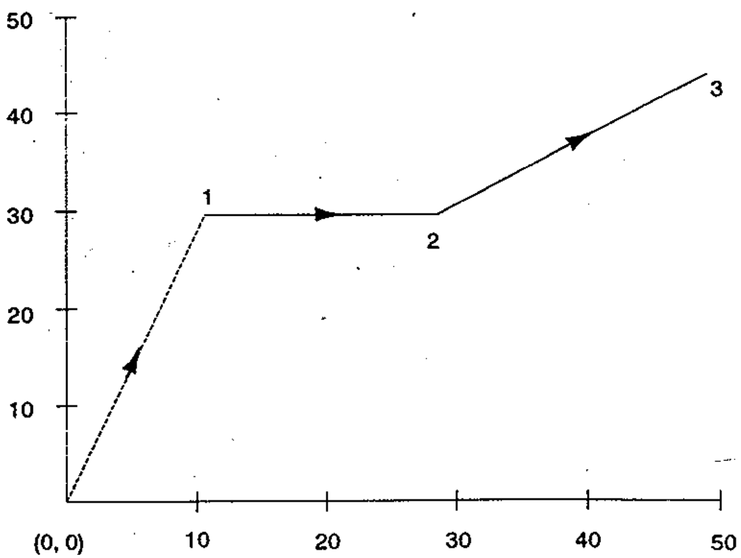


Fig. 5.1.

(1) Starting Point is (0, 0) and tool is 20 mm above the job surface.
(2) Machining is to be done along 1-2-3
(3) Z = 0 is at the surface of workpiece
(4) Depth of groove is 3 mm

N1 G90 G71 G94 M03 S800 EOB	Absolute mode, metric mode, feed in mm/min and spindle start at 800 rpm CW.
N2 G00 X 10.00 Y 30.00 EOB	From starting point (0, 0), the cutting tool moves at rapid feed rate to point 1 with no change in Z coordinate.
N3 G00 Z 2.00 EOB	In rapid feed rate, the cutting tool moves to a point 2mm above the job surface.
N4 G01 Z-5.00 F 200 EOB	In linear interpolation, the cutting tool moves to a depth 3 mm inside the workpiece at feed rate of 200 mm/min.
N5 G01 X 30.00 EOB	In linear interpolation, the cutting tool moves to point 2 (writing G01 is optional).
N6 G01 X 50.00 Y 45.00 EOB	In linear interpolation, the cutting tool moves to point 3.
N7 G00 Z 20.00 EOB	Tool moves to a point 20 mm above the job surface at rapid feed rate.
N8 G00 X –10.00 Y 0.00 EOB	Move to point X-10.00 to clear the job for loading/unloading.
N9 MO2 EOB	Programme end

Circular Interpolation Function (G02/G03)

If the cutting tool is required to move along an arc, circular interpolation functions (G02 or G03) are used to execute the instructions. G02 is used if the direction of interpolation is clockwise (CW) and G03 is used for counter clockwise (CCW) interpolation. The following information is required for writing an instruction block for circular interpolation.

(i) G-code depending on the direction of interpolation

G02 for clockwise interpolation
G03 for counter clockwise interpolation.

(ii) Co-ordinates of the target point i.e. X, Y and Z coordinates of the end point of the arc. If the interpolation is in X-Y plane, then only X & Y coordinates are needed.

(iii) Coordinates of the centre point of the arc. The coordinates of the centre point of the arc are written in incremental mode w.r.t. the starting point of the arc. The coordinates of centre point of the arc are listed with address letters, *I*, *J*.*K*, where the X coordinate of arc centre is written under address *I*, Y coordinate of the arc centre is written under address *J* and Z coordinate is written under address *K*. If there are only two axis involved in the circular interpolation then the coordinates of arc centre corresponding to these two axis only are given in the instruction. For example, in a CNC lathe since only two axis (X or Z) are used, the coordinates of arc centre are specified under address letters *I* and *K*. The radius of the arc is also accepted by some control system. In such systems, there is no need to give *I*, *J*, *K* values, simply the radius of the arc is programmed under *R* address.

(iv) If the feed rate of the cutting tool has been defined in the previous block, then it may not be given in the current block because the same feed rate will be active in circular interpolation also.

The instruction blocks for the circular motion along the tool path shown in Fig. 5.2(a) and Fig. 5.2(b) are as under:-

For Fig. 5.2(a)

G02 X18.00 Y24.00 I8.00 J 0 F 150 EOB
or
G02 X18.00 Y 24.00 R8.00 F 150 EOB

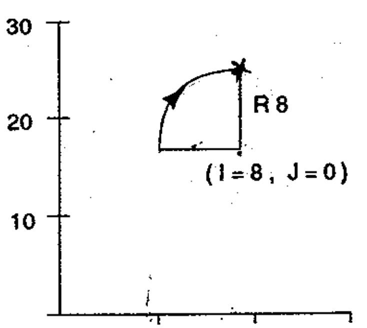


Fig. 5.2(a)

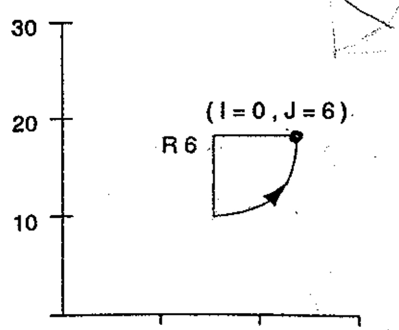


Fig. 5.2(b)

For Fig. 5.2(b)

G03 X22.00 Y18.00 I0 J6.00 F150 EOB
or
G03 X22.00 Y18.00 R6.00 F 150 EOB

Dwell Function (G04)

Dwell function is used in case it is desired that the cutting tool should not immediately return after touching the programmed position but should wait at the programmed position for some time, before executing the next instruction block in the part programme. For example, in case of machining a groove on a CNC lathe machine, to remove the material equal to desired depth of cut, the tool should remain at the required depth for some time to ensure the complete removal of the material. This is achieved by giving a dwell (delay) in the instruction block using G04. Dwell time is indicated as under:

G04 U 4000
or
G04 U4

where G04 is the G-code for dwell and dwell time is 4 seconds. U is the address letter, for dwell time and is different for different control systems. In the above instruction block dwell time is 4 seconds. In some cases the dwell time is indicated as under:

G04 4000 or G04 4

Here again, the meaning is the same as above but the format of writing the instruction is different. If the next statement in the part programme has G00 or G01 function, the tool will stop at the previous position for the time indicated with G04.

Some of the G-codes are modal i.e. these codes remain active until cancelled or superceded by a G-code of same class. For example, G90 is a modal code, it will remain active until it is superceded by G91 i.e. the control will change from absolute dimension to incremental dimension system. It is a good practice to cancel any model functions like G41, G42, G81, etc. in the programme after their function have been performed and are no longer required. Similarly G00, G01, G02, G03, G70, G71, G90, G91, G94, G95 are also modal. But other codes like G04 are non-modal i.e. these codes are active only in the block in which these are programmed.

Programming Formats

Format is the method of writing the words in a block of instruction. The following are the three programme formats being used for part programming:

(i) Fixed block format
(ii) Tab sequential format
(iii) Word address format

The numerical control systems are designed to understand and work with one type of programme format but control systems which can understand and work with more than one type of format are also being used in CNC machines.

(i) Fixed Block Format

In the fixed block format, instructions are always given in the same sequence. All instructions must be given in every block, including those instructions which remain unchanged from the preceding blocks. For example, if some coordinate values (i.e. x, y or z coordinates) remain constant from one block to next block these values have to be specified in the next block also. In this system, only data is provided in the programme and the identifying address letters are not given, but the data must be input in a specified sequence and characters within each word must be of the same length.

(ii) Tab Sequential Format

In this programme format, instructions in a block are always given in the same sequence as in case of fixed block format and each word is separated by the TAB character. If the word remains same in the succeeding block, the word need not be repeated but TAB is required to maintain the sequence of words. Since the words are written in a set order, the address letters are not required.

(iii) Word Address Format

In the word address format, each data is preceded and identified by its address letter. For example, X identifies the x-coordinate, F identifies the feed rate and so on. If a word remains unchanged, it need not be repeated in the next block. A typical instruction block in word address format will be as follows:

N010 X0000 Y0000 F 200 S 0800 T 010.01 M 30 EOB

N - Sequence number
G - Preparatory function
X - X-coordinate
Y - Y-coordinate

F - Feed rate
S - Spindle speed
T - Tool number
M - Miscellaneous function
EOB - End of block

To illustrate the programme formats discussed above, let us consider the job shown in Fig. 5.3 where two holes are to be drilled

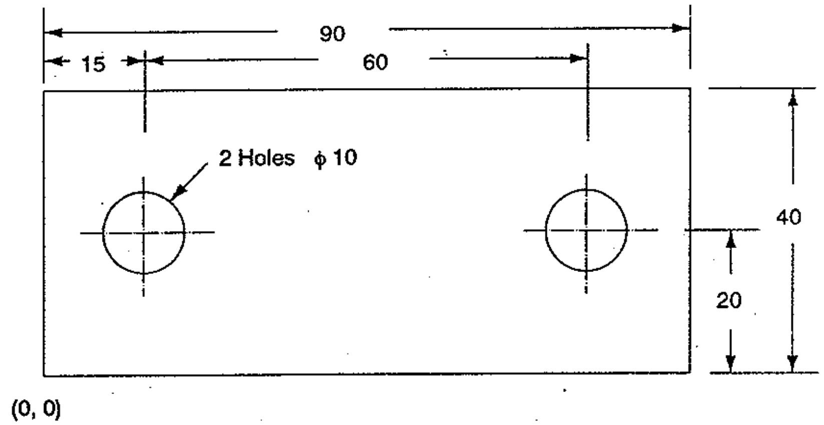


Fig. 5.3.

using a CNC drilling machine. The holes are to be drilled using a 10mm diameter drill at 500 rpm and feed rate of 200 mm/min. No G, M or T codes are required. The information to be coded is as follows:

	Hole 1	*Hole 2*
N-word	001	002
X-word	15.00	75.00
Y-word	20.00	20.00
F-word	200	200
S-word	500	500

In the fixed block format, the two instruction blocks will read as:

N	*X*	*Y*	*F*	*S*	
001	15.00	20.00	200	500	EOB
002	75.00	20.00	200	500	EOB

In TAB sequential format, the instruction blocks are

N		*X*		*Y*		*F*		*S*	
001	TAB	15.00	TAB	20.00	TAB	200	TAB	500	EOB
002	TAB	75.00	EOB						

In word address format, the two instructions will read as

N001	X 15.00	Y 20.00	F 200	S 500	EOB
N002	X 75.00	EOB			

WRITING A PART PROGRAMME

The first instruction in any part programme is to inform the control system about the various set-up conditions for the machining task to be taken up. Many part programmes will start in a similar way. The first block of instructions should specify the following:

- block number (N-number)
- co-ordinate value-absolute or incremental (G 90 G 91)
- dimensional units - metric or inch (G 70 or G 71)
- tool number if applicable (T-word)
- spindle speed (S-word)
- feed function (G 94 or G 95)

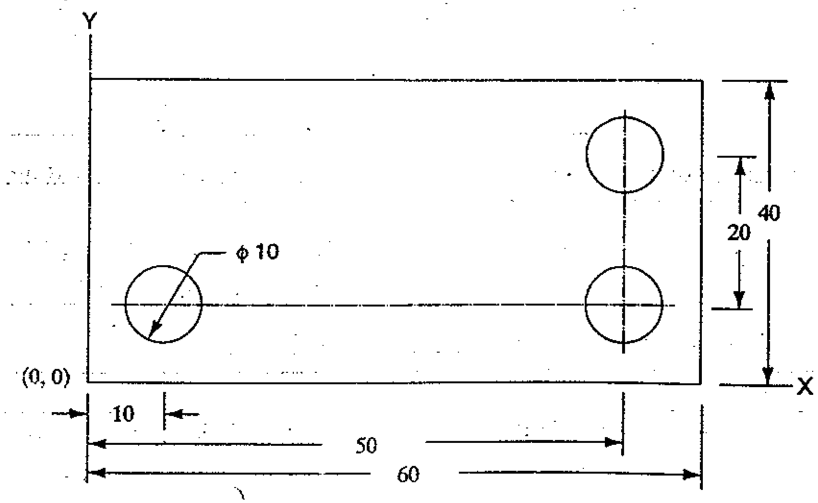
The first block of programme will look as follows:

N 0001 P121 G 90 G 70 G 94 M03 S 800 EOB

Each block is terminated by typing end of block (EOB) character.

Machining in Point-to-Point

In a point-to-point CNC system the workpiece or the cutting tool moves from one point to other point and machining is done at specific points and no machining is done when the spindle/workpiece is moving between two points. To illustrate the point-to-point machining, let us consider the workpiece shown in Fig. 5.4 where three holes are to be drilled at different places.



Note: (i) The depth of hole is 10 mm
(ii) Z = 00 at the surface of the workpiece.
(iii) The cutting tool is positioned above the workpiece surface.

N010 G71 G90 G94 EOB	*Metric mode, Absolute system and Feed in mm/min.*
N020 M03 F200 S1000 EOB	*Spindle start CW at 1000 rpm, feed rate is 200mm/min.*
N030 G00 X10.00 Y10.00 EOB	*Move in rapid to point P(10, 10)*
N040 G00 Z 2.00 EOB	*Move in rapid to a point 2mm above the workpiece.*
N050 G01 Z-10.00 EOB	*Drill hole (feed = 200 mm/min.)*
N060 G00 Z 2.00 EOB	*Move in rapid to point 2mm above workpiece surface*
N070 G00 X 50.00 EOB	*Move in rapid to X = 50*
N080 G01 Z-10.00 EOB	*Drill hole*
N090 G00 Z 2.00 EOB	*Same as N060*
N100 G00 Y 30.00 EOB	*Move to Y = 30*
N110 G01 Z-10.00 EOB	*Same as N080*
N120 G00 Z 20.00 EOB	*Move in rapid to a point 20 mm above workpiece surface*
N130 G00 X00 Y00 EOB	*Move in rapid to (X0, Y0)*
N140 M02	*Programme end*

Machining along Straight Line

Machining along straight line is done using linear interpolation. The lines may be horizontal, vertical or inclined at an angle in any direction. Machining in linear interpolation is done using G01 code in the programme. G01 commands the cutting tool/workpiece from the present position to the position with assigned coordinates. The general format for writing a command is

N — G01 X— Y— Z— F — EOB

In case of straight line machining with a milling machine, machining can be started in either of the two ways. Firstly, the tool is taken to required depth of cut outside the workpiece and then the tool is programmed to machine the component along the straight line. Secondly, the tool may be plunged to required depth of cut into the workpiece and then machining along straight line is started.

Let us consider the example of part programme for straight line milling on the component shown is Fig. 5.5. Machining is to be done at AB and BC. CNC part programme for this job is given below: 2 check again

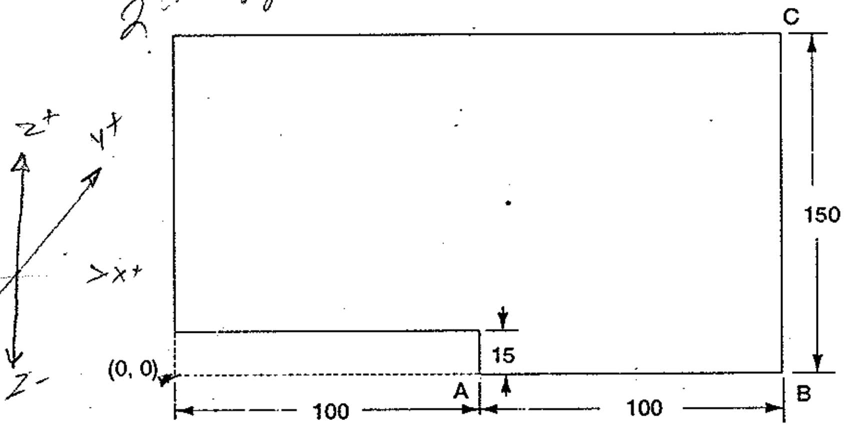


Fig. 5.5. *Z = 0 is 50 mm above the surface of the workpiece and depth of cut is 5 mm*

N0010 G71 G90 G94 EOB	
N0020 F200 S2000 EOB	*Set feed 200 mm/min. and speed 2000 rpm.*
N0030 M03 M08 EOB	*Spindle on clockwise, coolant on*
N0040 G00 Z2.00 EOB	*Spindle moves to Z = 2 in rapid mode.*
G01 N0050 X 100.00 Y0.00 EOB	*Move to X = 100, Y = 0.*
N0060 Z-55.00 EOB	*Spindle down to required depth of cut.*
N0070 G01 X 200.00 EOB	*Linear interpolation to X = 200.00*
N0080 G01 Y 150.00 EOB	*Move to Y = 150.00*
N0090 G00 Z10 M09 EOB	*Rapid spindle retract to Z = 10 and coolant off.*
N00100 G00 X-10.00 Y 0.00 EOB	*Rapid to X = –10 & Y = 0*
N00110 M02 EOB	

Lathe Operations

In case of CNC lathe operations, only two axes (X-axis and Z-axis) are involved. The Z-axis is the axis of the spindle and X-axis is the direction of transverse motion of the tool post. To develop CNC part programme for lathe operations the following procedure is adopted.

(i) Move the cutting tool to a point near the job in the rapid mode (G00).

(ii) Set linear interpolation (G01) and move to the required depth of cut in X-direction.

(iii) Move along Z-axis to the required length of the job as per drawing.

(iv) Set rapid mode (G00) and retract the tool along X-axis.

(v) Move to start point in G00 mode.

To illustrate the procedure consider the CNC part programme for the component shown in Fig. 5.6. The operations to be done are:

(i) Facing operation

(ii) To reduce the diameter of the bar from 30 mm to 26 mm.

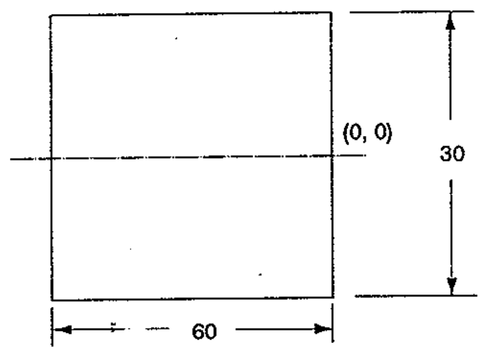


Fig. 5.6.

The bar stock available is 50 mm diameter rod

N0010 P02 G71 G90 G94 EOB	
N0020 T01 F 200 M03 S 800 EOB	*(Tool No. 1, feed rate 200 mm/min and speed 800 rpm)*
N0030 G00 X22.00 Z 1.00 EOB	*(In rapid mode, move to point X = 22, Z = 1)*
N0040 G00 X0 EOB	*(Move to X = 0, Z remaining constant)*
N0050 G01 Z0 EOB	*(Go to Z = 0 in G01 mode)*
N0060 X30.00 EOB	*(Move to X = 30 in G01 mode. This is facing operation)*
N0070 Z–60.00 EOB	*(Move to Z = – 60 in G01 mode. This is a dummy cut parallel to axis of the job.)*

N0080 G00 X 32.00 EOB	*(Withdraw the tool by 1mm i.e. to X = 32 so that when the tool is taken back to Z = 0, it does not leave a scratch mark on the job.)*
N0090 G00 Z0 EOB	*(Move in rapid to Z = 0)*
N0100 G01 X 26.00 EOB	*(Move to X = 26 in G01 mode. It gives a depth of cut of 2 mm)*
N0110 G01 Z – 60.00 EOB	*(Move to Z = – 60 to turn the job)*
N0120 G00 X 32.00 EOB	*(Withdraw the tool to X = 32 in rapid movement)*
N0130 Z 20.00 EOB	*(Go to Z = 20)*
N0140 M 02 EOB	*(End of programme)*

Taper Turning in Linear Interpolation

As already discussed, the G01 code is used for taper turning also because taper turning is also machining along a straight line at an angle. In case of taper turning simultaneous motion is required along both X-axis and Z-axis. The part programme for taper turning operation is similar to the simple turning operation except that in this case both the coordinates i.e. X and Z values of the final point are to be given in the programme. Consider the taper turning job shown in Fig. 5.7.

The raw material available is 20 mm dia bar.

The operations involved are:

(i) Facing

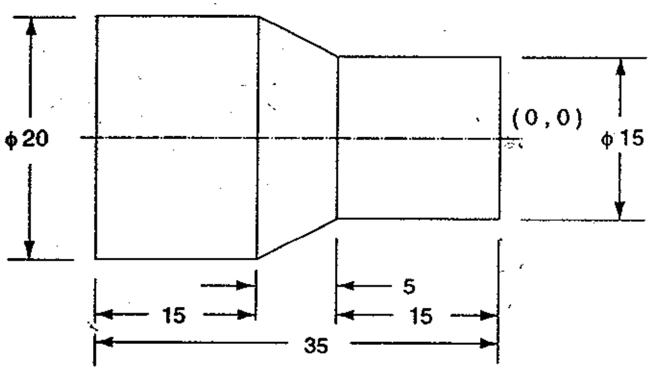


Fig. 5.7.

(ii) Turn to 15 mm diameter over 15 mm length.

(iii) Taper turning

The part programme for this job is given below:

	Absolute mode	*Incremental mode**
N0010	P03 G71 G90 G94 EOB	N0010 P03 G71 G91 G94 EOB
N0020	T01 S1000 M03 EOB	N0020 T01 S1000 M03 EOB
N0030	G00 X22 Z0.5 EOB	N0030 G00 X 11.00 Z – 0.5 EOB
N0040	G01 X0.00 F200 EOB	N0040 G01 X-11.00 F 200 EOB
N0050	Z0.00 EOB	N0050 Z-0.5 EOB
N0060	X 20.00 EOB	N0060 Z11.00 EOB
N0070	X 15.00 EOB	N0070 X –3.5 EOB
N0080	Z–15.00 EOB	N0080 Z-15.00 EOB
N0090	X 20.00 Z-20.00 EOB	N0090 X 2.5 Z-5.00 EOB
N0100	Z–35.00 EOB	N0100 Z-15.00 EOB
N0110	G00 X 25.00 Z 20.00 EOB	N0110 G00 X 2.5 Z 55.00 EOB
N0120	M02 EOB	N0120 M02 EOB

*For incremental mode the starting position of the cutting tool has been assumed at (X = 0, Z = 1)

Multipass Turning Operation

As in case of conventional machining, the depth of cut is limited in CNC machine also. So the desired material removal is accomplished in a number of cuts. Each time the tool is fed against the workpiece by 2 to 3 mm for rough cut and 0.75 to 1 mm for finished cut and turning is done upto desired length. The tool is then taken back to the starting position and the cycle is repeated. To illustrate the multipass turning operation consider the job shown in Fig. 5.8 where diameter is to be reduced to 30mm from the available raw material bar of 40 mm diameter, maximum depth of

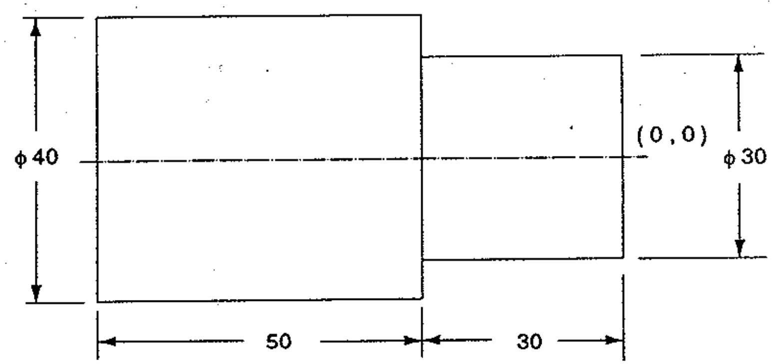


Fig. 5.8.

cut being limited to 3 mm. The part programme for this component is given below:

N01	G90 G71 G94 F500 S1000 T01 EOB
N02	G00 X41.00 Z1.00 M03 EOB
N03	G01 X37.00 EOB
N04	G01 Z-30.00 EOB
N05	G00 X 41.00 Z 1.00 EOB
N06	G01 X 34.00 EOB
N07	G01 Z-30.00 EOB
N08	G00 X 41.00 Z1.00 EOB
N09	G01 X 31.00 EOB
N10	G01 Z-30.00 EOB
N11	G00 X 41.00 Z1.00 EOB
N12	G01 X 30.00 EOB
N13	G01 Z -30.00 EOB
N14	G01 X 41.00 EOB
N15	G00 Z 25.00 EOB
N16	M02 EOB

Thread Cutting Operation on CNC Lathe

Thread cutting can be done on CNC lathe using G33 code. But additional parameters based on type of thread, depth of thread and pitch, etc. are to be given in the programme. Thread cutting operation on a CNC lathe is performed by moving the slide in synchronisation with the job on which threads are to be cut. The required thread pattern and the shape of the thread is generated by the thread cutting tool. In thread cutting operation, the axis feed rate is calculated by the control system from the programmed values of pitch of the thread and spindle speed. To attain full depth of the thread, a number of cuts have to be taken, and each time the depth of cut is incremented by a small value. The part program to cut threads of 50 mm length with pitch = 0.75 mm and depth of thread = 0.46 mm is discussed below. Here the lead (pitch) of the thread is programmed under K word. The programmed motions are shown in Fig. 5.9 and the part programme will look as under:

Note: depth of cut is 0.2 mm per pass, except in last pass where depth of cut is 0.06 mm.
(The starting tool position is X = 0 and Z = 10)

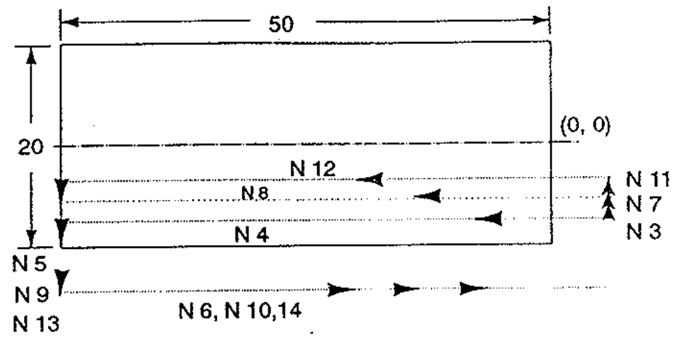


Fig. 5.9.

N001	G91	G94 G71 EOB
N002	G00	X 25.00 Z – 5.00 EOB
N003	G01	X – 5.2. F 100 EOB
N004	G33	Z-55.00 K0.75 EOB
N005	G00	X 5.2 EOB
N006	G00	Z 55.00 EOB
N007	G01	X-5.4 F 100 EOB
N008	G33	Z-55.00 K0.75 EOB
N009	G00	X 5.4 EOB
N010	G00	Z 55.00 EOB
N011	G01	X-5.46 F 100 EOB
N012	G33	Z-55.00 K0.75 EOB
N013	G00	X 6.00 EOB
N014	G00	Z 25.00 EOB
N015	M02	EOB

Since thread cutting involves repeated cutting of the profile, each time the thread cutting should start from the same point. This aspect is taken care of by the control system.

Machining Using Circular Interpolation

Circular profiles can be produced on a CNC lathe using G02 or G03 codes. In addition to X and Z values, the parameters for the centre of the arc are also given in the programme with I & K words. I, K denote the centre of the arc for circular profile.

To illustrate the use of circular interpolation function consider the profile shown in Fig. 5.10. The CNC programme for circular profile will be as follows:

N01 G90 G71 G94 EOB
N02 G00 X0 Z1.00 EOB
N03 G01 Z0 EOB
N04 G02 X10.00 Z-5.00 I0 K-5.00 F 150 EOB Clockwise circular interpolation upto (X = 10 Z = –5)

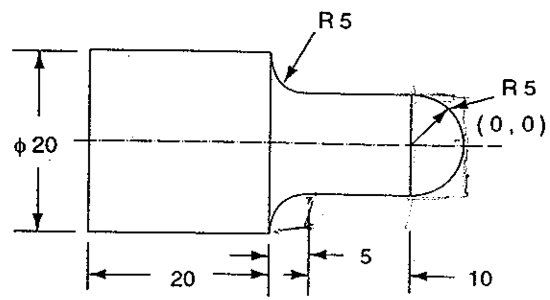


Fig. 5.10.

N05 G01 Z-15.00 EOB

N06 G03 X 20.00 Z-20.00 I5.00 K0 EOB — Counter clock-wise circular interpolation to X = 20 Z = –20

N07 M02 EOB

Internal Features

Internal features like boring, internal taper and internal threading can be generated on a lathe in the same manner as the external features. But care should be taken in giving dimensions for the movement of the cutting tool.

Programming for CNC Milling Machine Operations

As already discussed in case of CNC milling machine, motion is possible along all three axes. The Z-axis is the axis of the spindle and any movement of the tool/workpiece which takes the cutting tool away from the workpiece is the positive Z motion and any movement of the cutting tool towards or into the workpiece is negative Z motion. Having gone through the part programming for CNC lathe operations, it will not be difficult to understand the part programming for CNC milling machines or for machining centres. The only difference being the addition of Y-axis.

Cutter Radius Compensation

As stated earlier the part programme is developed for the cutter path with reference to the center of the tool rather than the point on the periphery where the actual cutting takes place. At the time of writing a part programme a cutter of suitable diameter is selected and programme is developed for centre line of the cutter. But when actual machining is done, if a cutter of smaller diameter

is used, it will result in a larger workpiece and if a cutter with larger diameter is used it will result in a smaller workpiece.

The difference in the programmed diameter of the cutter and the diameter of the actual cutter is accounted for by cutter radius compensation. The difference in the diameter of the cutter is entered into the control system. The control system will then generate a new cutter-path. The new path will be separated from the programmed cutter path by difference in the radius of programmed cutter and the actual cutter. It is necessary to indicate whether compensation is to be made to the right or to the left of the tool when machining. The following three G-codes are used for cutter radius compensation.

G–41 - Compensation applied to shift the programmed cutter path to left.

G–42 - Compensation applied to shift the programmed cutter path to right.

G–40 - Cancel cutter radius compensation.

The direction in which the cutter path has to be shifted is decided by looking in the direction of cut. In Fig. 5.11, if the direction of cut is programmed in clockwise direction of the over size cutter, compensation would be provided to shift the cutter path towards left of the programmed path (G 41) and if the direction of cut is programmed counter clockwise the compensation would be applied to shift the cutter path towards right from the programmed path (G42). The facility of cutter radius compen-

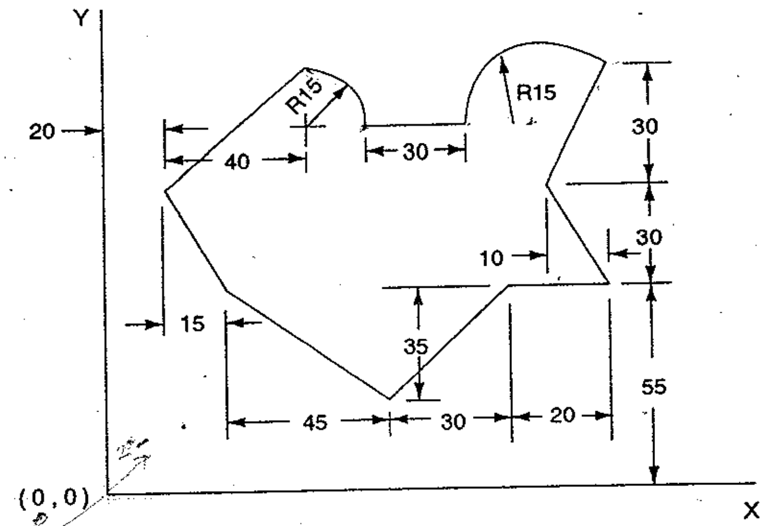


Fig. 5.11.

sation is very useful as cutters of different diameters can be used without changing the part programme. By using cutter radius compensation, the job of the programmer is also simplified. The part programme can be developed assuming the cutter radius as zero i.e. on the actual drawing dimensions of the component. The cutter path can then be shifted by using the relevant G-code.

Part programming for the component shown in Fig. 5.11 using cutter radius compensation is given below:

Programmed cutter diameter = 0 mm i.e. programming is done as per the part drawing.

Diameter of cutter available = 28 mm.

The difference in radius (0 – 28)/2 = – 14 mm is stored in the memory for cutter radius compensation under address D 01.

```
N001 G71  G90 G94 EOB
N002 G00  X–20.00 Y–20.00 Z 20.00 EOB
N003 G00  Z–5.00 EOB
N004 G01  G41 D01 X20.00 Y85.00 F400 S1000 M03 EOB
N005 G01  X60.00 Y115.00 EOB
N006 G02  X75.00 Y 100.00 I0.00 J–15.00 EOB
N007 G01  X 105.00 EOB
N008 G02  X 130.00 Y 115.00 I15.00 J 0.00 EOB
N009 G01  X 120.00 Y 85.00 EOB
N010 G01  X 130.00 Y 55.00 EOB
N011 G01  X 110.00 EOB
N012 G01  X 80.00 Y 20.00 EOB
N013 G01  X 35.00 Y 55.00 EOB
N014 G01  X 20.00 Y 85.00 EOB
N015 G00  X G40 X 0.00 Y0.00 EOB
N016 M02  EOB
```

Part programme for the component shown in Fig. 5.12 with cutter radius compensation and direction of cut programmed in

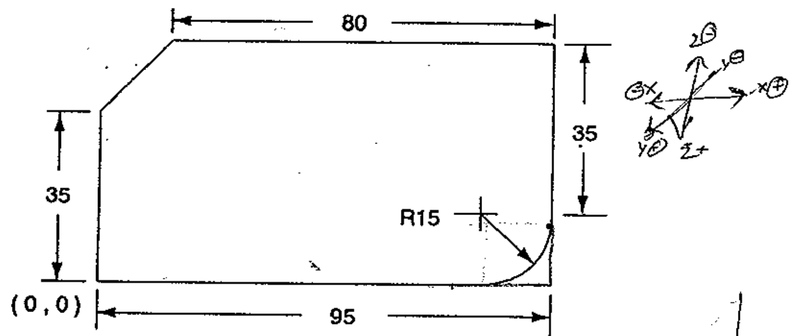


Fig. 5.12.

anticlockwise direction is given below. The following information is available. The cutter radius compensation is stored in D02.
Z = 0 is at the top surface of the workpiece.

Feed = 65 mm/minute
Speed = 1000 rpm
Depth of cut = 10 mm

```
N005  G71  G90 G94 EOB
N010  G00  X–20.00 EOB
N015  G00  Z–10.00 EOB
N020  G01  G42 D02 X0 Y0 F200 S1000 M03 EOB
N0025 G01  X 80.00 EOB
N0030 G02  X95.00 Y 15.00 I0 J 15.00 EOB
N0035 G01  Y50.00 EOB
N0040 G01  X15.00 EOB
N0045 G01  X0 Y35.00 EOB
N0055 G01  X0 Y0 EOB
N0060 G40  EOB
N0065 G00  X–20.00 Z20.00 EOB
N0070 M02  EOB
```

Zero Points and Reference Points

On each CNC machine, zero points and reference points are defined. The part programme for any component is developed relative to these points.

Machine Zero: The machine zero point is at the origin of the coordinate measuring system of the machine. The machine zero point is fixed and cannot be shifted. The machine zero point is also called 'Home position'.

Work Zero: Workpiece zero or datum may be defined as a point, line or surface on the component drawing to which all the dimensions referenced. For writing the part programme, the programmer should know the relationship between the workpiece zero coordinates and machine zero coordinates. In other words, all the coordinate values for slide movements have to be defined with reference to the machine zero. However this complicates the part programmers job. To simplify the part programme writing, the CNC machines have the facility of floating zero or zero shifting.

Zero Shift: The zero shifting facility is available on CNC machines. This facility allows the machine tool zero point to be shifted to any position within the programmable area of the machine. The zero shift or datum shift facility allows the user to shift the machine zero to coincide with the workpiece zero. Part programming is then simplified.

EVALUATIVE QUESTIONS

1. What is a part programme? Discuss the step in writing a part programme.
2. Explain the terms preparatory function and miscellaneous function stating where these are used in a part programme.
3. Explain the use of following codes:
   (i) G00, G01, G02, G04.
   (ii) G71, G90, G91, G94, G95.
   (iii) G17.
   (iv) G80, G81.
   (v) M02, M03, M05, M30.
4. Write a part programme for the component shown in Fig. 5.13. The machining parameters are given below:
   Cutting speed = 800 rpm
   Feed = 200 mm/min
   Depth of cut should not exceed 2mm.

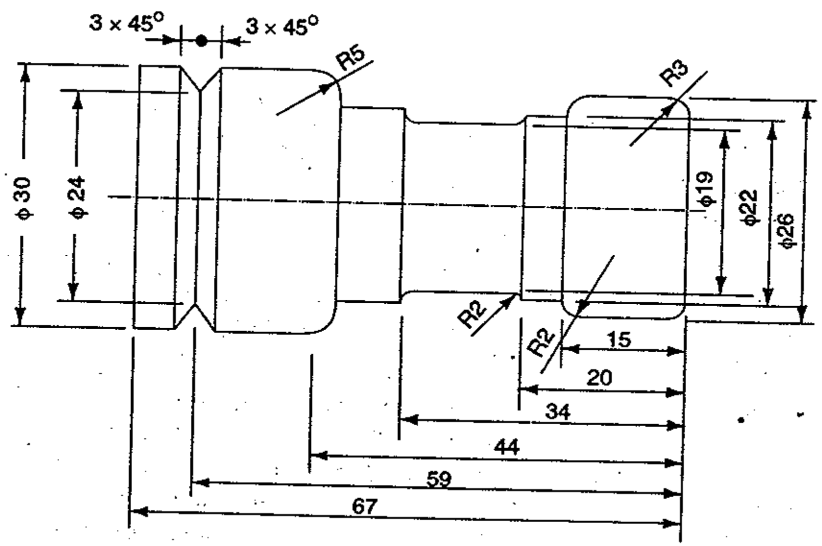


Fig. 5.13.

5. Write a part programme for the component shown in Fig. 5.14. Assume suitable machining data.
6. Explain the thread cutting operation on a CNC lathe. Rewrite the part programme for the threading job shown in Fig. 5.9 using absolute dimensioning.
7. What is cutter radius compensation? Discuss when it is used and how it is included in the part programme?

8. Write a part programme for the workpiece shown in Fig. 5.11 using incremental dimensioning.
9. What do you understand by machine zero and work zero? Explain.

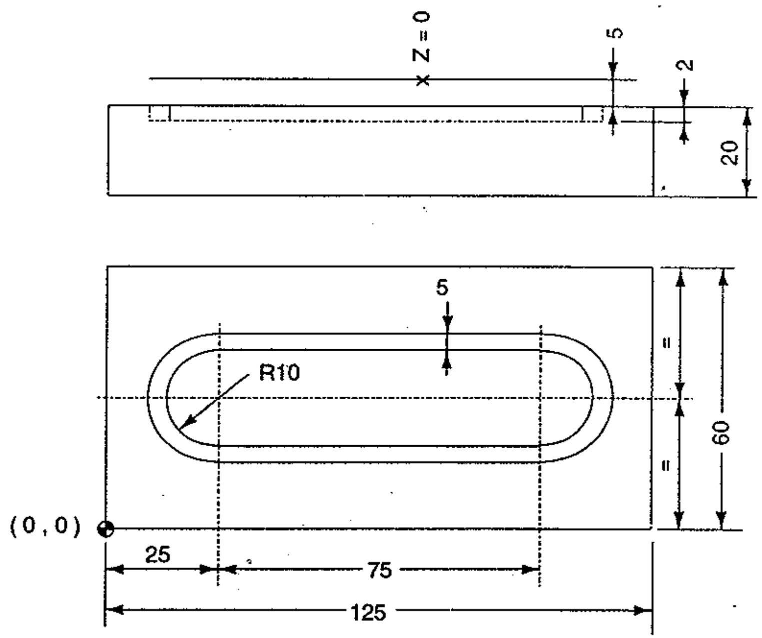


Fig. 5.14.

6

PART PROGRAMMING USING SUBROUTINES, DO LOOPS AND CANNED CYCLES

OBJECTIVES

At the end of this unit you should understand :

1. The significance of using subroutines, Do loops and canned cycles.
2. Know-how to develop subroutines and Do loops for different applications.

INTRODUCTION

In the previous chapter, we have discussed the fundamentals of part programming procedure and practice for simple components. However, in actual practice the components to be programmed are complex in nature and the resulting part programmes are proportionately longer if written in the way we have discussed. Also if some repetitive features, such as grooves or holes, are present in a component, identical sections of the programme will have to be repeated to produce the same feature at different points. Simpler ways of achieving such repetitions are provided by the facility of repetitive programming techniques available to the part programmer. Use of repetitive programming techniques significantly reduces the length of resulting part programme, shortens the time required to develop the programme and reduces the computer memory space required for the programme. The common techniques used for repetitive programming are:

(i) Subroutines
(ii) Do loops
(iii) Fixed cycles or canned cycles

SUBROUTINES

Subroutines, also called, subprogrammes, are a powerful time saving technique. The subroutines provide the capability of pro-

gramming certain fixed sequence or frequently repeated patterns. Subroutines are, in fact, independent programmes with all the features of a usual part programme. Subroutines are stored in the memory under separate programme numbers. Whenever a particular feature is required within the programme, the associated subroutine is called for execution. The subroutines may be called any time and repeated any number of times. After execution of subroutine the control returns to main programme. To describe and use a subroutine, the following information is required, in the form of codes and symbols.

- Identification (start) of subroutine
- End of subroutine
- A means of calling a subroutine

Here we will use letter L followed by a number i.e. L221, to identify the start of a subroutine. L221 means start of subroutine No. 221. Miscellaneous code M17 will indicate the end of subroutine. The subroutine can be called anywhere in the main programme by just giving the subroutine number preceded by letter L. For example, if subroutine No. 221 is to be called and used in the main programme, then L221 is entered at appropriate place.

To understand the use of subroutine consider that it is required to mill a square pocket 40 × 40mm at various positions on a flat plate. The following information is available.

Depth of profile = 3 mm
Z = 0 is at the surface of the flat plate

The tool should retract back to a position 5 mm above the flat plate surface while moving from one position to the other position.

L 101	EOB	- subroutine No. 101
N 100	G 91 EOB	- set incremental mode
N 105	G01 Z-8.00 F300 EOB	- cutter move to required depth at given feed rate
N110	G01 X 40.00 EOB	- move 40mm in X-direction
N115	G01 Y 40.00 EOB	- move 40mm in Y-direction
N120	G01 X-40.00 EOB	- move – 40mm in X-direction
N125	G01 Y-40.00 EOB	- move – 40mm in Y-direction
N130	G00 Z 8.00 EOB	- Cutter move 8 mm in Z-direction (i.e. 5mm above the flat surface)
N135	G90 EOB	- absolute mode
N140	M17 EOB	- end of subroutine

This subroutine can now be called any number of times to mill square profiles of 40 × 40 mm. In order to mill three such profiles, the above subroutine can be called and used in the main programme, at different positions as follows:

N 005	G90 G71 G94 M03 S500 EOB	
N 010	G00 X 50.00 Y 50.00 EOB	- move to starting point at first position
N 015	G00 Z 5.00 M08 EOB	- spindle move to a point 5mm above surface and coolant ON
N 020	L101 EOB	- call subroutine 101
N 025	G00 X 120.000 Y 120.00 EOB	- move to starting point at 2nd position
N 030	L 101 EOB	- call subroutine 101
N 035	G00 X 200.00 Y 200.00 EOB	- move to starting point at 3rd position
N 040	L101 EOB	- call subroutine 101
N 045	G00 X0.00 Y0.00 Z20.00 EOB	- move to X = 0, Y = 0, spindle retract to Z = 20 and coolant OFF.
N050	M02 EOB	- end of programme

From the above programme we can see that the length of the part programme has been considerably reduced using a subroutine. The main programme is written in absolute mode (G90) and the subroutine is written in incremental mode (G91) since this has to be used at different locations.

The use of subroutines depends on the experience and imagination of the programmer and on the capabilities of machine control system. Some of the situations where subroutines can be used are:

(a) A number of grooves in a shaft. The groove is programmed in its entirety in subroutine. The main programme will only move the cutting tool to the starting point of the next groove and the groove will be cut by calling the subroutine.

(b) A complex contour requiring number of roughing and finishing passes. The entire contour may be programmed in a subroutine, resulting in a much shorter programme.

(c) A complex hole pattern requiring extensive machining at each hole location i.e. centering, predrilling, drilling, reaming and chamfering. Here the subroutine will contain the location of all the holes to be made. The main programme will include the tool changes, spindle and coolant ON/OFF and the corresponding Z motions. The main programme will call the subroutine and all the X-Y motions will be controlled by the subroutine.

DO LOOPS

The ability to write the programmes with Do loops enables the programmer to instruct the control unit to jump back to an earlier part of the programme and execute the intervening programme blocks a specified number of times. The Do loops statement is given in the main programme itself and it is necessary to give the following information in the form of symbols or codes.

- start of the loop
- number of repeats of the loop
- end of the loop

Do loop is used for repetitive programming in cases such as turning and milling operations where it is not possible to remove the entire material in a single pass and more than one cut have to be taken to machine the components to required size or where uniform repetition is required like cutting uniformly spaced grooves in a shaft or drilling of a pattern of holes in plate, etc. The use of do loop is explained with the help of a part programme for the component shown in Fig. 6.1. There are two taper turning steps and one straight turning. Since the material cannot be removed in a single pass, the part programme without Do loop

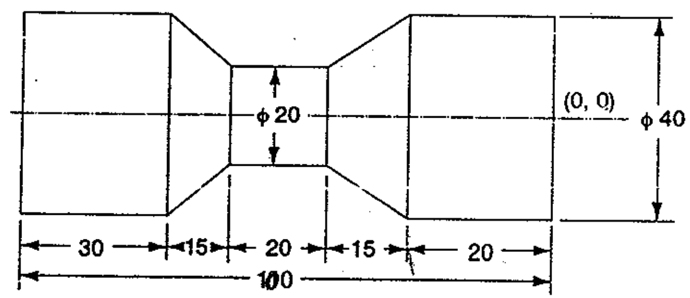


Fig. 6.1.

will involve calculation of intermediate points. Keeping the maximum depth of cut as 2mm, the material will be removed in five passes. The programme using Do loop is shown below:

N001	G90	G71 G94 M03 S400 EOB
N005	G00	X 40.00 Z 1.00 EOB
N010	G01	Z–100.00 F 400 EOB
N015	G00	X 60.00 Z–20.00 EOB
N020	G73	(Start Do Loop - 5 times) EOB
N025	G91	EOB
N026	G01	X–2.00 EOB
N030	G01	X–10.00 Z–15.00 EOB
N035	G01	Z–20.00 EOB

N040 G01 X 10.00 Z–15.00 EOB
N045 G00 X 2.00 EOB
N050 G00 Z 50.00 EOB
N055 G00 X–2.00 EOB
N060 G06 (End of Do Loop) EOB
N065 G90 EOB
N070 G00 X 50.00 Z 15.00 EOB
N075 M02 EOB

The tool motions in the Do loop are repeated five times. The cutting tool moves each time 2 mm towards the component and the steps in the body of the loop are repeated.

CANNED CYCLES

Canned cycle or fixed cycle may be defined as a set of instructions, inbuilt or stored in the system memory, to perform a fixed sequence of operations. The canned cycles can by brought into action with a single command and as such reduce the programming time and effort. Canned cycles are used for repetitive and commonly used machining operations. The canned cycles are stored under G code address. G81 to G89 are reserved for fixed canned cycles and G80 is used to cancel the canned cycle.

Fixed Cycles for Lathe Operations

Commonly available fixed cycles for lathe operations are:

(i) Canned cycle for turning
(ii) Canned cycle for threading
(iii) Canned cycle for rough turning
(iv) Canned cycle for finish turning

Canned cycles for turning and threading are discussed here:

Fixed Cycle for Turning

As discussed in the previous chapter, the depth of cut is limited in CNC machines also. In order to machine the component to required dimensions, a number of cuts may have to taken. One way of writing a part programme for achieving the required diameter on a CNC lathe machine has been discussed in Chapter 5 i.e. by repeating the same steps. However, since the same steps are being repeated everytime, the part programme becomes unnecessarily lengthy, occupies large computer memory and the part programmer has to spend more time in writing the part programme. In order to save part programming time and computer memory, fixed cycle for turning are available in the control

system. The programmer has to first write an instruction block to position the cutting tool at the starting point and then call the fixed cycle for turning as follows:

N5 G81 X–2.0 Z–30.00 F 200 EOB

where,

G81 is the code for the fixed turning cycle
X–2.0 denotes that the depth of cut is 2mm
Z–30.00 denotes that the length to be machined is 30 mm.

The cycle is executed as follows:

Step 1 : The cutting tool moves by 2 mm in the X direction at a given feed rate of 200 mm/min. i.e. it takes required depth of cut.
Step 2 : The cutting tool moves 30.00 mm in negative Z direction at feed rate of 200 mm/minute.
Step 3 : The cutting tool moves back by 2 mm in X direction at rapid traverse.
Step 4 : The cutting tool moves back in Z direction by 30 mm at rapid traverse.

So after the cycle has been executed, the cutting tool is repositioned at the same point from where it started. Also it may be noted that the four tool motions have been accomplished by a single instruction block, which may otherwise have been written in four instruction blocks.

To understand the use of a fixed turning cycle, consider the component shown in Fig. 6.2. It is a case of simple step turning where it is required to reduce the diameter from 20 mm to 16 mm and 12 mm.

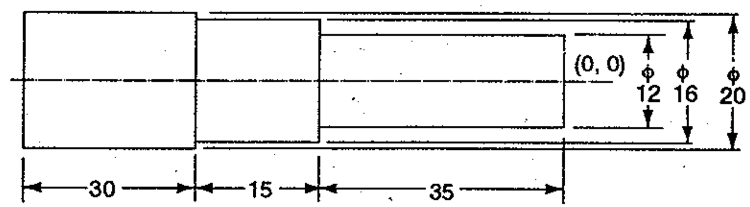


Fig. 6.2.

The steps required to make this component are:

(i) Turn to 16 mm diameter over a length of 50 mm
(ii) Turn to 12 mm diameter over a length of 35 mm.

The depth of cut should not exceed 1.5 mm and the speed of the workpiece is 300 rpm and feed rate is to be kept at 200 min/minute.

N1 G91 G94 G71 M03 S800 EOB
N2 G00 X20.00 Z0 EOB
N3 G81 X–2.00 Z–50.00 F200 EOB
N4 G81 X–4.00 Z–50.00 F 200 EOB
N5 G81 X–6.00 Z–35.00 F 200 EOB
N6 G81 X.–8.00 Z–35.00 F 200 EOB
N7 G80 EOB
N8 G00 X 25.00 Z 10.00 EOB
N9 M02 EOB

Threading Cycle

In order to call and use a fixed cycle for thread cutting on a CNC lathe machine, the information is given as follows:

N2 G84 X–0.3 Z–30.00 K 1.5 EOB

where,

G84 is the fixed cycle code for thread cutting.
X0.3 is the depth of cut in one pass
Z–30.00 is the length of thread
K 1.5 specifies the pitch of the thread

The following four tool motions are executed with the use of above instruction:

1. The cutting tool moves 0.3 mm in negative X direction at a feed rate depending on the pitch of the thread to be cut.
2. Threads are cut over a length of 30 mm. The feed of the cutting tool is automatically set according to the pitch of the thread to be cut, programmed under address K.
3. The cutting tool retracts by 0.3 mm at a rapid feed rate.
4. The cutting tool moves back to the starting position at a rapid feed rate.

To illustrate the use of fixed cycle for thread cutting, consider that threads with pitch of 0.75 mm are to be cut over a length of 50 mm. The workpiece is shown in Fig.5.9. Here again let us assume that the starting point is at X = 25 and Z = 5. The part programme for this component using fixed threading cycles is given below : (Depth of cut in first two passes is 0.2 mm and in the third pass it is only 0.05 mm).

N001 G91 G94 G71 M03 S500 EOB
N002 G00 X 25.00 Z 5.00 EOB
N003 GS4 X–5.2 Z–50.00 K 0.75 EOB
N004 G84 X–5.4 Z–50.00 K0.75 EOB
N005 G84 X–5.45 Z–50.00 K0.75 EOB
N006 G80 EOB

N007 G00 X 6.00 Z 25.00 EOB
N008 M02 EOB

We can see that while using G33, the number of instructional blocks required to cut the same threads was 15, with G84 the number of blocks required are only 7.

Fixed Cycles for CNC Milling Machine and Machining Centre Operations

On the CNC milling machines and machining centres also, a number of canned cycles or fixed cycles are available to reduce the programming effort and computer memory space required for a part programme. The common canned cycles available are:

1. Drilling cycle
2. Boring cycle
3. Threading (Tapping) cycle.

In addition, there are fixed cycles for pocket milling, PCD drilling, etc. Some of these cycles are discussed here :

Drilling Cycle

Fixed cycle for drilling a hole is available, where the complete drilling cycle is completed by giving the information in a single block.

To understand the use of drilling cycle, consider the workpiece shown in Fig. 6.3. The part programme for this component is given below:

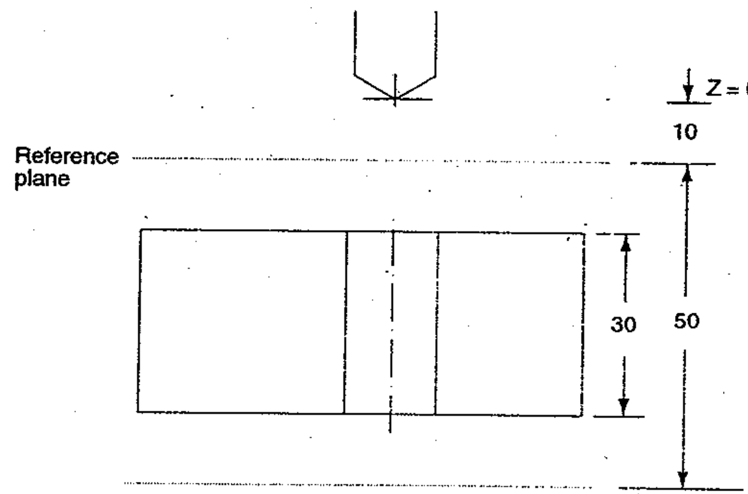


Fig. 6.3.

N01 G71 G94 G90 EOB

N02 G00 X10.00 Y 15.00 EOB
N03 G00 Z–10.00 EOB
N04 G81 Z–50.00 M03 S800 F 150 EOB
N05 G80 EOB
N06 M02 EOB

The drilling cycle is explained below:

N01 - *Metric mode feed rate in mm/min and absolute coordnate system*

N02 - *Positioning block i.e. position the drilling tool at X = 10.00 and Y = 15.00*

N03 - *Drilling tool moves to reference plane in rapid traverse.*

The reference plane is selected above the workpiece surface to avoid the drill striking the workpiece while moving in rapid traverse.

N04 - *Call drilling Cycle. The spindle starts rotating at 800 rpm in clockwise direction and the hole is drilled at the required position at the given feed rate of 150 mm/minute. The drilling tool is positioned at reference plane after the drilling operation is completed.*

N05 - *The drilling cycle is cancelled*

N06 - *End of programme*

Deep Hole Drilling Cycle (Peck Drilling Cycle)

When the depth of hole is more ($l/d > 10$) it is desirable to withdraw the drill from the hole at regular intervals to avoid clogging due to chips. This is called *wood peck drilling*. In the CNC machining centres, peck drilling cycle is available. By using the peck drilling cycle, the drill is retracted upto reference plane at rapid feed rate every time after drilling the hole to a specified incremental depth. Consider the workpiece shown in Fig. 6.4. The final depth of the 10 mm diameter hole is 70 mm, the reference plane is 10 mm above the surface and the overtravel required is also 10 mm. The total movement of the drill is 90 mm. However, the hole is not drilled in a single pass. Each time the drill is fed to a specified depth and withdrawn to reference plane before again feeding the drill further into the workpiece. Here the total tool travel from reference plane to final position of the drill is programmed as Z value and the incremental depth after which the tool has to be withdrawn is programmed as K valve. The typical format for using deep hole drilling cycle is given below:

N01 G71 G94 G91 M03 S1000 EOB
N02 G00 X 10.00 Y 10.00 EOB
N03 G00 Z–10.00 EOB

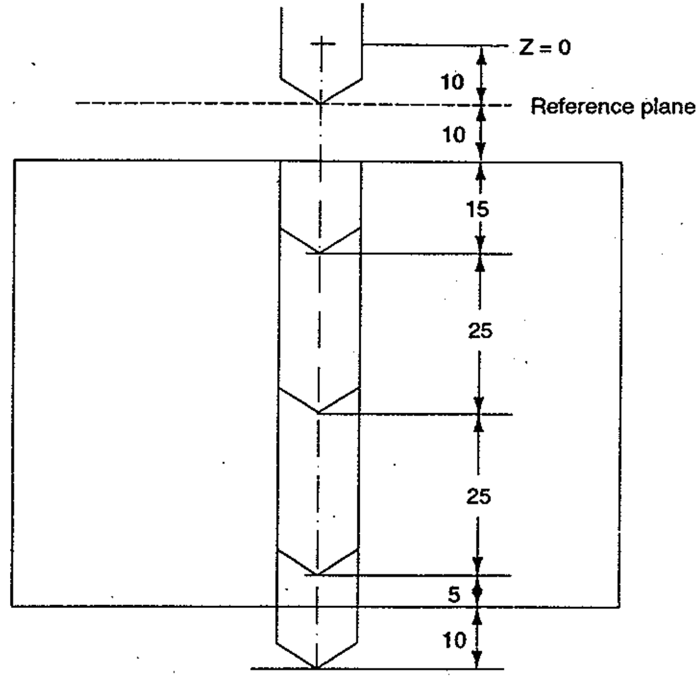


Fig. 6.4.

N04 G82 Z–90.00 K 25.00 F100 EOB
N05 G80 EOB
N06 M02 EOB

Here the deep hole drilling cycle is called using G82

Boring Cycle

In the boring cycle the boring tool is fed to the required depth at the given feed rate. When the tool has reached the required depth, the rotation of the tool is stopped and the tool is withdrawn at a rapid feed rate upto the reference plane. The programming format for using boring cycle (G83) is as under:

N001 G91 G71 M03 S600 EOB
N002 G00 X 10.00 Y 10.00 EOB
N003 G00 Z–10.00 EOB
N004 G83 Z–60.00 F100 EOB
N005 G80 EOB
N006 M02 EOB

Threading (Tapping) Cycle

The tapping operation, involves positioning of tap at required X and Y position, moving it rapidly to reference plane and feeding into the predrilled hole in the workpiece at given feed rate. The spindle rotation is then reversed and the tap is brought back to reference plane at the programmed feed rate. The spindle rotation is again reversed to prepare for next tapping operation.

Fixed cycle for tapping is available on CNC machining centers. The use of tapping cycle is illustrated with the help of Fig. 6.5. The part programme using tapping cycle (G84) is given below:

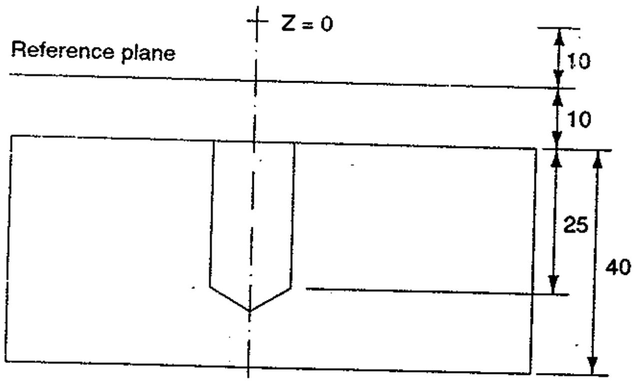


Fig. 6.5.

```
N001  G71  G91 M03 S500 EOB
N002  G00  X10.00 Y10.00 EOB
N003  G00  Z–10.00 EOB
N004  G84  Z–25.00 F60 EOB
N005  G80  EOB
N006  M02  EOB
```

EVALUATIVE QUESTIONS

1. What is a subroutine? Write a subroutine for drilling a series of holes for the workpiece shown in Fig. 6.6.
2. What are the parameters required to define and use a 'Do loop' in a part programme. Write a part programme using Do loop for the workpiece shown in Fig. 6.6.
3. What are the fixed cycles? What is the difference between a fixed cycle and a subroutine? Discuss how a fixed cycle can be useful in writing a part programme.

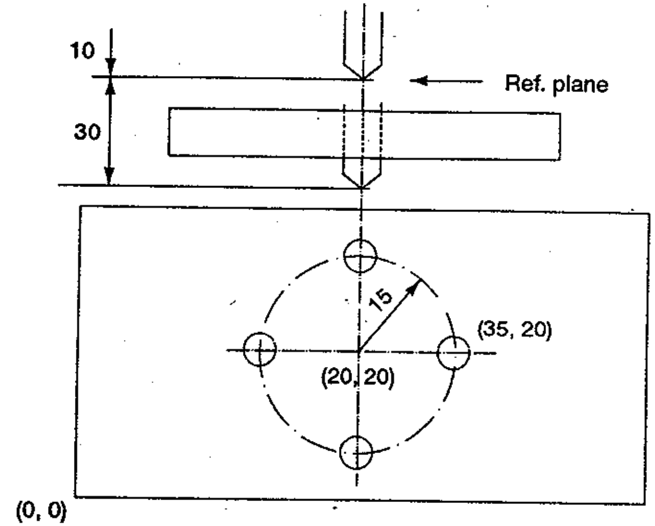


Fig. 6.6.

7

COMPUTER-AIDED PART PROGRAMMING

OBJECTIVES

At the end of this unit you should understand:

1. What is meant by computer aided part programming.
2. What are the benefits of computer aided part programming.
3. How to write simple part programmes using APT and COMPACT-II programming languages.

INTRODUCTION

The method discussed in previous chapters to develop part programme is called manual part programming. The manual part programming is a time consuming process and needs an expert part programmer who should have through knowledge of the various machining processes, materials, speeds and feeds, part programming codes and capabilities of various machine tools, etc. Manual part programming is a labour oriented task and needs skilled programmers. Also, if a person is expert in programming one machine, he will not be able to develop part programme for another machine, since the format or the type of information required by the two machines may be different. With the modern NC/CNC machines where more than three axes are to be controlled it may not be possible to develop part programmes by manual programming methods.

All these problems have been overcome and part programming has been considerably simplified with the use of computer aided part programming, where the computer generates the part programme required to machine the component. The process of generating part programmes in computer aided part programming is partly done by part programmer and partly by the computer.

The part programmer's job in the computer aided part programming is to first define the geometry of the component from the

component drawing. The geometry or shape of the component is split into simple elements like points, lines, arcs, full circles, distances and directions and these elements are assigned specific numbers to identify their position. The geometry of elements of the component is defined using simple abbreviated english like terms having specific meaning which is understood by the computer and control system. The instructions to define a point and straight line may be written as:

P1/0, 0	(co-ordinates of point P1 are (0, 0))
L1/P2, P4	(line L1 passes through points P2 and P4)

The programmer may be able to see the geometric construction on the video display unit depending upon the system capabilities. The second part of the programmer's job is to give additional information regarding the machining sequences, type of operation, tool sizes, etc. From the geometry of the component, the system generates the data required to machine the component. This data is called cutter location (CL) data. The data generated upto this point is independent of the machine and can be used on any machine capable of doing the required operations. The data does not contain G or M codes.

The cutter location data is then post-processed in the computer to translate it into a form which a particular machine control system can understand. The post-processing involves addition of G Codes, M codes and other machine dependent information in the required format.

The part programme at this stage is machine dependent and can be used for a specific machine only.

The advantages of using computer-aided part programming are:

(i) Part programming is considerably simplified.

(ii) The part programmes generated are accurate and efficient.

(iii) All arithmetic calculations are done by the computer, resulting in saving in time and elimination of errors.

(iv) The part programming for different machines can be done by a single person, which can then be post processed for specific machines.

(v) Such system can deal with many axes for simultaneous movement.

(vi) If new machines are added, only a post processor may be needed to integrate the new machines with the existing system.

COMPUTER AIDED PART PROGRAMMING LANGUAGES

Large number of part programming languages have been developed from time to time, to meet particular needs. But the most widely used part programming languages are APT and COMPACT II. These two languages are discussed here in brief :

APT

APT stands for automatically programmed tools. This is the most widely used and most comprehensive part programming language available. APT is a three-dimensional system which can be used to control up to five axes. In programming using APT, it is assumed that the workpiece remains stationary and cutting tool does all the moving. The APT part programme consists of four types of statements.

(i) Geometry statements: These are also called definition statements and are used to define geometric elements like point, circle, arc, plane, etc.

(ii) Motion statements: The motion statements are used to define the cutter path.

(iii) Post processor statements: These statements are applicable to specific machine tools and are used to define machining parameters like feed, speed, coolant on/off, etc.

(iv) Auxiliary statements: These are miscellaneous statements used to identify the part, tools, tolerances, etc.

1. *Geometry Statements*

The geometry of the components is defined with the help of points, lines, circles, arcs and planes, etc. The general form of APT geometry statement is:

Symbol = Geometry type/descriptive data

For example, line L1 is defined as passing through two points P1 and P2 as follows:

P1 = POINT/ 3.0, 4.0, 0.0
P2 = POINT/ 3.0, 2.0, 0.0

L1 = LINE/P1, P2
L1 = LINE/3.0, 4.0, 0, 3.0, 2.0,0

There are various methods of defining a point, line, circle or plane. Some of the methods of defining the basic geometric elements are listed below:

Point

P1 = POINT/X, Y, Z
(Point with coordinates X, Y, Z)
P2 = POINT/CENTER C1
(Point at the centre of the circle 1)

P3 = POINT/INTOF, L1, L2
(Point at intersection of lines L1 & L2)
P4 = POINT/XLARGE, INTOF, L1, C1
(Point at intersection of line L1 & circle C1, where X coordinate has higher value)
P5 = POINT/YSMALL, INTOF, C1, C2
(Point at intersection of circles C1 and C2, but with small value of Y coordinate).

Here it has been assumed that lines and circles have already been defined in the geometry statements.
There are many other ways to define point which have not been discusses here.

Line

In APT, a line can be defined in many ways. Some of the definitions of a straight line are:

L1 = LINE/P1, P2
(line passing through points P1 and P2)
L2 = Line/P2, PARLEL, L1
(line passing through point P2 and parallel to line L1)
L3 = LINE/P1, ATANGL, 60, L2
(line passing through point P1 and at an angle 60 degree to line L2)
L4 = LINE/P1, ATANGL, -120, L2
(line passing through point P1 and at an angle -120 degree to line L2)
L5 = LINE/PARLEL, L31, YLARGE, 10
(line parallel to line L31 and offset by 10 mm towards Y large coordinate)

Circle

Circle is also a common geometric element. Some statement used to define a circle are:

C1 = CIRCLE/CENTER, P1, RADIUS, 10
(circle with centre point P1 and radius 10 mm)
C2 = CIRCLE/CENTER, P1, TANTO, L1
(circle with center at P1 and tangent to line L1).
C3 = CIRCLE/ P1, P2, P3
(Circle passing through points P1, P2, P3)
C4 = CIRCLE/XSMALL, L1, YSMALL, L2, RADIUS, 5
(Circle passing through the intersection of Lines L1 and L2 and radius of 5 mm (with X small on L1 and Y small on L2))

Plane

A plane can be defined as passing through three points as:

PL1 = PLANE/P1, P2, P4

2. *Motion Statements*

Motion statements are also called machine control instructions. The general format for APT motion statement is

motion command/descriptive data

For example, a motion statement is

GOTO/P2

or

FROM/P1

The motion statements define the tool path as per the part geometry specified in the part programme. If the movements have to be made in incremental mode, GODLTA command is used. For example, the statement

GODLTA/5.0, 4.0, 0.0

instructs the cutting tool to move from its present position 5 mm in X direction, 4 mm in Y direction and the Z coordinate remains same.

The other common motion commands in APT are:

FROM/P1
GO/PAST
GO LFT
GO FWD
GO BACK
GO UP
GO DOWN

3. *Post Processor Statements*

Post processor statements are machine specific and control the operation of the machine spindle, feed rate and other features of machine tool like tool change, etc. Some of the post processor statements are:

COOLNT/
END
FEDRAT/
MACHIN/
RAPID
SPINDL/
TURRET

All these APT words except the words without slash (/) require descriptive data. For example, when the coolant pump is required to be switched on; the postprocessor statement is given as:

COOLNT/ON

Similarly descriptive data is required with other APT words. Some examples of post processor statements are:

SPINDL/500	*(Spindle speed = 500 rpm)*
FEDRAT/200	*(Feed rate = 200 mm per minute)*
MACHIN/TURNING CENTRE 1	*(Machine tool to be used for the operation)*

4. *Auxiliary Statements*

Auxiliary statements are used for part identification, cutter size definition, defining the dimensional tolerances and other functions to prepare the control system to accept and execute the part programme. Some of the auxiliary APT words are:

PART NO/	*(Part number for identification of the component)*
CUTTER/	*(Cutter size definition)*
FINI/	*(Programme finish)*
INTOL	*(Inside tolerances)*
OUTTOL	*(Outside tolerances)*
CLPRNT	*(Print cutter location data)*

For example, the statement

CUTTER/20

means that the diameter of the cutter to be used is 20 mm.

Example of Part Programme Using APT

Consider the component shown in Fig. 7.1 (a). It is assumed that the component has already been cut to rough shape and it is required to write a part programme only for finish cut. The APT part programme for this component is given below:

The cutter path is shown in Fig. 7.1 (b).

```
PARTNO/SAMPLEJOB 1
MACHIN/MILLING 1
CLPRNT
P0 = POINT/ 0, -20.0, 0
P1 = POINT/ 0, 0, 0
P2 = POINT/ 150.0, 0, 0
P3 = POINT/ 150.0, 50.0, 0
P4 = POINT/ 100.0, 50.0, 0
```

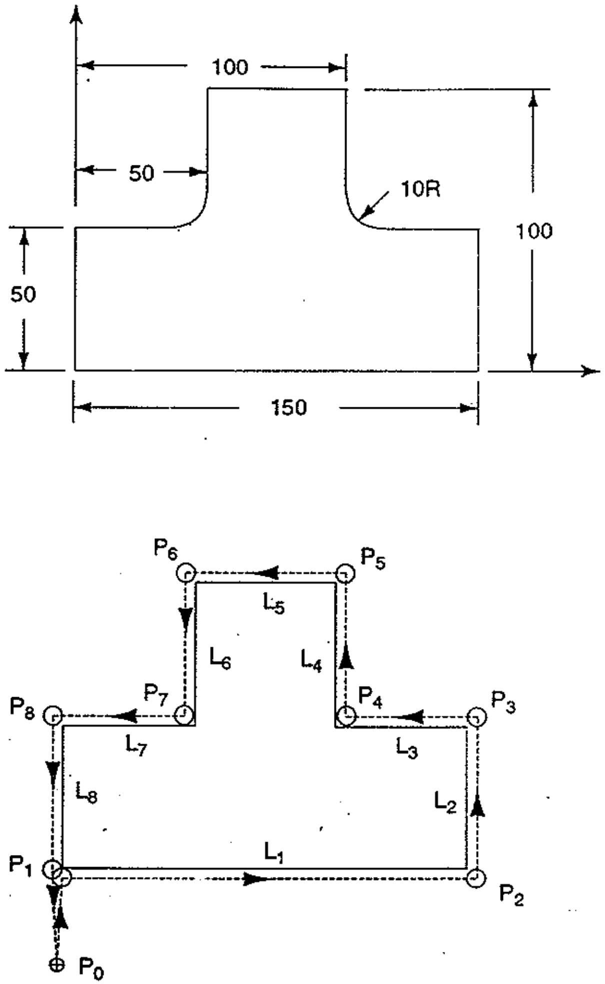


Fig. 7.1(a, b).

```
P5 = POINT/ 100.0, 100.0, 0
P6 = POINT/ 50.0, 100.0, 0
P7 = POINT/ 50.0, 50.0, 0
P8 = POINT/ 0, 50.0, 0
L1 = LINE/ P1, P2
L2 = LINE/ P2, P3
L3 = LINE/ P3, P4
L4 = LINE/ P4, P5
L5 = LINE/ P5, P6
L6 = LINE/ P6, P7
L7 = LINE/ P7, P8
L8 = LINE/ P8, P1
```



```
PL1 = PLANE/ 0, 0, – 20.0, 150.0, 0, – 20.0, 0, 100.0, 20.0
CUTTER/ 10.0
SPINDL/ 580
FEDRAT/ 80
COOLNT/ ON
FROM/ P0
GO/TO, L1, TO, PL1, TO, L8
GORGT/ L1, PAST, L2
GOUP/ L2, PAST, L3
GOLFT/ L3, TO, L4
GOUP/ L4, PAST, L5
GOLFT/ L5, PAST, L6
GODOWN/ L6, TO, L7
GOLFT/ L7, PAST, L8
GODOWN/ L8, PAST, L1
RAPID
GOTO/ P0
COOLNT/ OFF
FINI
```

Compact II

A computer assisted part programme written using COMPACT II consists of a series of statements which provide information or instructions to the machine control system. Every statement begins with a major word, which is then followed by a set of associated minor words. The major words indicate the type of operation to be performed by the control system and the minor words give the details about the location and manner in which the operation is to be accomplished. Minor words are also used to define cutting parameters like speeds, feeds, etc. or the cutting tool description. The COMPACT II programme involves five different groups of instructions. These five groups are:

(1) Initialization statements
(2) Geometry statements
(3) Tool change statements
(4) Motion statements
(5) Programme termination statement

All these statements are discussed here briefly.

1. *Initialization Statement*

Initialization statement must come first in any programme. The initialization statement usually includes four sub-statements i.e. machine, identification, set-up and base statements.

(i) The machine statement is the first statement of the programme. The machine statement gives information about the machine tool. It consists of a major word MACHIN, followed by the name of the machine tool link. An example of machine statement is

MACHIN, MILL

(ii) The identification statement is the second statement in the programme and is used to identify the part programme. The statement consists of the major word IDENT, followed by the part name or number or any other alphanumeric combination.

IDENT, SAMPLE PROGRAM
IDENT, PART NO 100357

(iii) The set-up statement is the third statement in the programme. It is used to specify home position and absolute zero of the machine tool. Home position or gauge length reference point (GLRP) is the point from which the tooling gauge length are measured. For example, in a machining centre, GLRP is the centre point at the surface of the quill. Set-up, in fact, is used to specify programme zero when a floating zero machine is used. For a lathe machine, the set-up statement is of the form

SET UP, 10X, 7.5Z

It means that the set-up point is 10 mm from spindle centre line (where X = 0) and 7.5 mm away from the point where Z = 0
For a milling machine the set-up statement is of the form

SET UP, 5LX, 0LY, 15LZ

which indicates that the set-up point has coordinates (5, 0, 15) with reference to machine absolute zero.
The set-up statement is also used to include the travel limits of the machine tool by using LIMIT parameter.

SET UP, 10X, 15 Z, LIMIT (X 0/20, Z 0/55)

(iv) The base statement is the fourth statement in the programme. It is used to establish the base or coordinate system of the part for easier programming. It is advantageous to reference the base to absolute zero although it can be referenced to any other point. The base statement

Base, 10XA, 3YA, 5ZA

means that location of base point with reference to the absolute zero is (10, 3, 5).

2. Geometry Statements

The shape of a workpiece is defined using the geometric elements

i.e. points, lines, circles and planes. In COMPACT II, the various geometric elements are defined as follows:

Point

A point is defined by using the major word DPT, followed by minor words which describe how that point is specified. A point can be defined in several ways. Some of the methods to define a point are given below:

DPT1, 10XB, 5YB, 3ZB
(Point 1 is defined as having coordinates (10, 5, 3) with reference to the base point)

DPT2, LN1, LN2
(Point 2 is defined as the intersection of lines LN1 & LN2)

DPT3, LN1, CIR1, XL
(Point 3 is at the intersection of lines LN1 and circle CIR1 and where the X values is large)

DPT4, CIR1, CIR2, YL
(Similar to DPT3)

DPT5, CIR1, 40CCW
(Point 5 is on circle 1 and 40 degree counter clockwise with respect to reference axis).

Line

A line is defined by major word DLN, followed by minor words which describe how that line is specified. A line can be defined as passing through two points, as passing through a point and making certain angle with reference axis or as perpendicular to one of the coordinate system axis. Some of the geometry statements used in defining a line are:

DLN1, PT1, PT2
DLN2, PT1, PERLN2
DLN3, PT2, PARLN1
DLN4, PT3, 30CW

Circle

The major word used to define a circle is DCIR, followed by minor words to specify that circle. A circle may be defined by specifying its centre and radius, providing three points through which it passes, being tangential to two existing lines or having some relationship with an existing circle. Some of the methods of defining a circle are:

DCIR1, PT1, 20R
DCIR2, PT1, PT2, PT3

DCIR3, CIR1/10R
DCIR4, LIN1, LIN2, 10R
DCIR5, LNI/5YS, LN2/7XL, 10R

Plane

A plane is defined by using the major word DPLN followed by appropriate minor words. The statements used to define a plane are:

DPLN1, PT1, PT2, PT3
DPLN2, 10 ZA

The plane 2 is defined as a plane perpendicular to Z axis and its intercept (10mm) on Z axis is given in the definition.

3. *Tool Change Statements*

The tool change statement contains all information relevant to machining i.e. tool configuration, feed rate, spindle speed, etc. Machines equipped with automatic tool changers are programmed using the major word ATCHG and MTCH is used for the machines requiring manual tool change. The tool change statement initiates a number of functions like, the axis are moved to tool change point, the spindle motion is stopped, the tool magazine is indexed and the spindle is started at required speed and feed rate after the tool change. The tool change statement for milling machines is:

ATCHG, TOOL3, GL16, 20TD, 800RPM, 200MMPM

The above tool change statement indicates that:

- Tool No. 3 is commanded to mount in the spindle
- GL16 means that tool No.3 has gauge length of 16 mm
- 20 TD means the cutter diameter is 20 mm
- The spindle speed is 800 rpm and feed rate is 200 mm per minute after the tool number 3 is mounted.

In case of a lathe machine, the tool change statement includes tool gauge length in both X and Z axis. In addition the radius of the tool nose may be given in the statement.

ATCHG, 8GLX, 10 GLZ, TOOL2, 1TLR, 600 RPM, 50 MMPR

4. *Motion Statements*

Motion statements define the geometrical shape of the component. Major words are used to identify whether the motion is linear or circular and the minor words specify and terminate the path of the tool.

1. For *linear motion*, there are two major words i.e. MOVE and CUT. MOVE generates rapid motion and is used to position the tool in the non-cutting movements. CUT is used to move the cutting tool at the given feed rate for metal removal. Some examples of motion statements used to generate linear motion are:

MOVE, PARLN1, PASTLN2
CUT, PARLN1, PASTLN3
CUT, PARLN4, OFFLN2/XS
CUT, PARLN2, ONLN3
CUT, PARLN3, TOLN6

2. For motion along a *circular path*, three major words are used i.e. CONT, ICON, and OCON. These words indicate the location of the tool center with respect to the circular arc to be obtained after machining. CONT is used when the tool center falls on the contour whereas OCON (outside contour) and ICON (inside contour) are used to produce convex or concave surfaces, respectively. The minor words are used to define the direction of motion and start and finish points. Some of the statements for circular motion are:

ICON, CIR1, CW, S(PASTLN2), F(180)
ICON, CIR2, CCW, S(TANLN4), F(TANLN5)
OCON, CIR3, CW, S(TANLN4), F(TANLN3)

The words S and F in the above statements indicate the starting and finish points of contour.

5. *Programme Termination*

The programme is terminated by the major word END. END statement is always the last statement in the programme

Example of Part Programme Using COMPACT II

The complete COMPACT-II part programme for the component shown in Fig. 7.2 is given below:

MACHIN, MILL
IDENT, SAMPLE JOB2
SETUP, 30LX, 30LY, 30LZ
BASE, 20XA, 20YA, 20ZA
DLN1, 0YB
DLN3, 150XB
DLN2, LN3/6XS, LN1, 60 CCW
DLN4, 75YB
DLN5, 0XB
DCIR1, LN1/12YL, LN5/12XL, 12R
DCIR2, LN3/12XS, LN4/12YS, 12R
ATCHG, TOOL1, 10GL, 800RPM, 200MMPM, 20TD

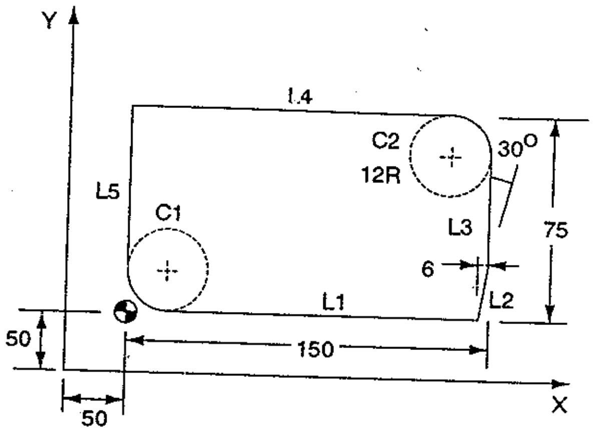


Fig. 7.2.

```
MOVE, OFFLN1/1YS, OFFLN5/XS, 5ZB
CUT-25ZB
CUT PARLN1, PASTLN2
CUT PARLN2, PASTLN3/YS
OCON, CIR2, CCW, S(TANLN3), F(TANLN4)
OCON, CIR1, CCW, S(TANLN5), F(TANLN1)
END
```

All the programme statements are self explanatory. It may be noted that cut motion along LN3 and LN5 have not been programmed separately because the tool motion from the current tool location to the starting point of circular motion is automatically programmed in circular motion statements.

EVALUATIVE QUESTIONS

1. What is computer-aided part programming? How is it different from manual part programming?
2. What are the different statement used to write a part programme using APT? Discuss each statement with the help of suitable example.
3. Write a part programme, using APT, for the component shown in Fig. 7.3. Assume suitable datum, speed, feed and depth of cut.
4. What are the different statements used to write a part programme using COMPACT II? Discuss each statement with the help of suitable examples.
5. Write a part programme, using COMPACT II, for the component shown in Fig. 7.3. Assume suitable machining data.

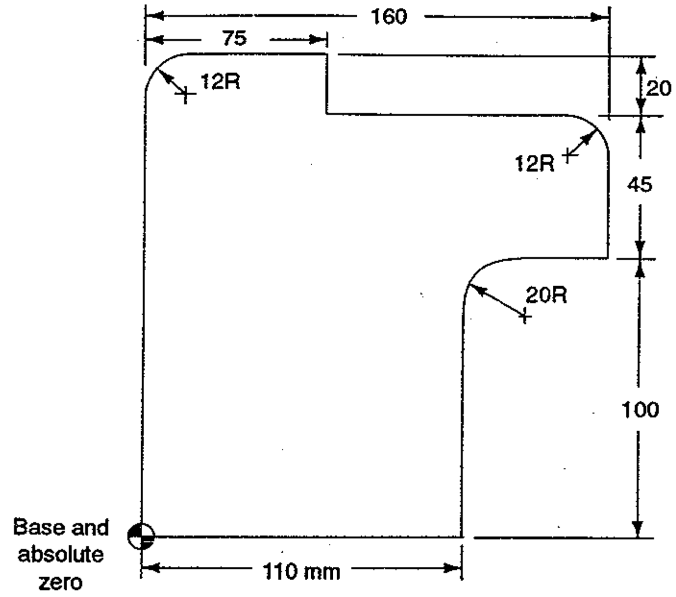


Fig. 7.3.

8

TOOLING FOR CNC MACHINES

OBJECTIVES

At the end of this unit you should understand:

1. The special tooling requirements of CNC machines.
2. Meaning and usage of pre-set and qualified tools.
3. Advantages of using automatic tool changers and multi-pallets on CNC machines.

INTRODUCTION

① To use fully limit

The special design features of CNC machines have resulted in use of higher cutting speeds and feeds, leading to considerable saving in the cycle time. To fully exploit the higher metal removal rates of the CNC machines, the tooling used should be able to withstand the higher cutting forces in the process and help to reduce the down time to a minimum possible. The tooling used on CNC machines should be:

(a) Rigid to withstand high metal removal rates

(b) Capable of being pre-set and re-set in the shortest possible time to keep the down time to minimum.

(c) Accurate enough to produce repetitive accuracy on the job.

In conventional machines, the cutting tool cuts metal for about 25% of the total machining time whereas the CNC machine tools are expected to cut metal for 70 to 80% of the time. Since CNC machines are very costly, the down time on these machines has to be reduced to a minimum. The tooling for CNC machine tools includes the cutting tools, and tool and work holding device.

Cutting Tools

To minimize the tool change and tool setting time, the CNC machines use pre-set and qualified tools.

Tool Pre-setting

The tools are set to known dimensions away from the machine tool. The pre-setting of tools can be planned and carried out in

advance, so that tools are available to ensure continuity of production and minimise down time due to tool set-up on job changes. Special pre-setting devices are available for the pre-setting of tools. Pre-setting is done with the tool held in the tool holder so that the assembly i.e. tool holder and cutting tool can be straightway fitted on the machine. Tool presetting fixture is shown in Fig.8.1.

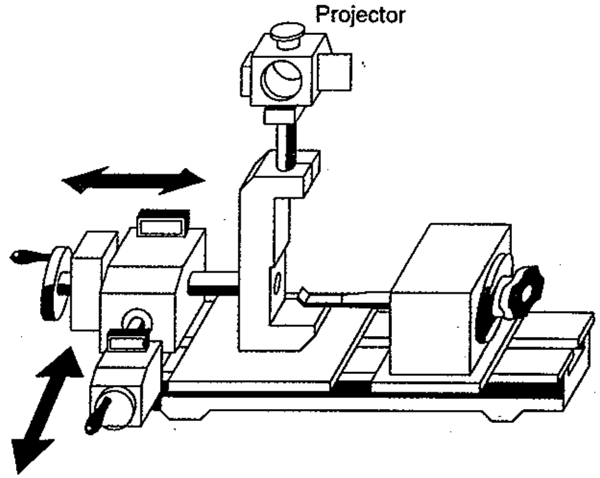


Fig. 8.1.

Qualified Tools

Qualified tools are the tools on which the position of the cutting edge is guaranteed within close limits of accuracy from a specified datum on the tool holder. The qualified dimensions are applied to the tool tip from the three datums. Usually the datums are formed

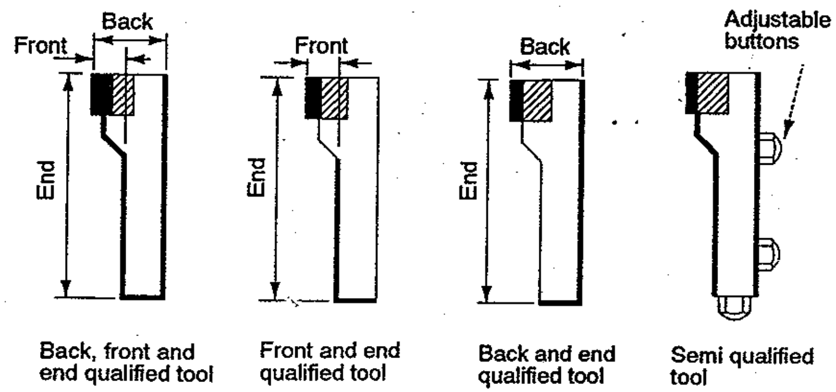


Fig. 8.2.

by the tool holder. Since the dimensions of the tool holder are fixed and known, precise position of the tool tip is known and accurate positioning on the machine tools is possible. Hard metal inserts are ideally suited for qualified tooling since their dimensions and geometry are known. Fig.8.2 shows the qualified tools.

Semi-qualified tools with three adjustable buttons on the tool shank are also available. These are called semi-qualified because the buttons can be adjusted to a qualified dimension and can be used on different machines. The semi-qualified tools have to be regularly checked and maintained to ensure that the dimensions remain correct.

Indexable Inserts

At the high cutting speeds and feeds used on CNC machines, the brazed tools are normally not used. Indexable carbide inserts are used to take up higher cutting speed and to minimise the tool change time. Since the actual cutting time is more in CNC machines, the consumption of cutting tools is also more. The tool change time with the brazed tool is many times more compared to tool change time with indexable inserts, where the inserts is taken out and indexed. Harder and special grade carbides should be used to achieve faster rate of metal removal. The indexable inserts of tungsten carbide are further coated with a wear resistant layer of titanium carbide or titanium nitride for better results. Indexable inserts are available in various shapes and with varying geometry. The chip breakers can be formed on the inserts either by shaped grooves in the insert itself or by way of clamping arrangements. In case the solid HSS tooling is employed in some machining applications, proper cutting fluid should be used to get improved surface finish, increased tool life and accurate dimensions. The overhang of the tool should be kept minimum. As the cutting speeds and feed rate are high in case of CNC machines, the temperature of the cutting edge of the cutting tool is also very high. To ensure proper cooling, coolant-fed tools are used where the coolant is fed to the cutting edge through a centrally drilled hole in the shank of the tool. Also there is an interlock in the system so that if the coolant supply is not there, the machine will either give a signal or may stop.

Tool Holders

Tool holders used on CNC machine tools should be of quick changing type and should be capable of being pre-set. Since in the modern CNC machines, there is a provision for automatic tool changing, the quick changing type tool holders can reduce the tool change time to about 3 to 5 seconds.

Work Holding Devices in CNC

The numerically controlled machines are capable of performing a number of operations using different tools, on different faces of a component in a single setting. This requires that the component should be accessible from different sides without changing of clamps or re-positioning of component. The work holding device has to bear multidirectional cutting forces. So additional demands are made on work holding devices in numerically controlled machine tools. To reduce the clamping/unclamping time, hydraulic and pneumatic actuation is widely used in work holding devices. In general, a workholding device for CNC applications should have the following features:

(i) It should restrict the linear and rotary motion of the component.

(ii) The component should not distort or deflect due to cutting forces in the process.

(iii) It should facilitate quick loading and unloading of the component.

(iv) It should be fool proof i.e. it should ensure that component cannot be loaded wrongly.

(v) It should not interfere with the cutting tools.

(vi) It should permit number of operations on different planes in a single setting.

(vii) It should provide for easy removal of chips.

(viii) It should be adaptable to automated loading/unloading of components.

(ix) It should be safe.

Automatic Tool Changers

The CNC machines are designed to perform a number of operations in a single setting of the workpiece. To reduce the down-time in change-over from one operation to the next, the CNC machines are equipped with automatic tool change facility. The tool is automatically selected and changed based on the tool control function (T-word) in the part programme.

Turning centres are available with the tool turret containing 8-12 tools. As the tool change command is received by the control system, the tool turret moves to a fixed tool change position and the required tool comes to the cutting position.

On the machining centres, automatic tool changers (ATC) are provided to reduce the idle time between change over from one operation to another. The ATC consists of a tool magazine for storing the tools and a tool change unit for transfering the tool from tool magazine to spindle. The tool already fitted on the spindle is removed and replaced in the tool magazine. The initial position of various tools is fed to the control system, which then keeps updating the data regarding tool number fitted in the particular pocket in the tool magazine. Tool magazines with upto 60 tools are quite common in India. The tool change cycle consists of two parts:

(i) Tool Selection Cycle

The tool for successive operation is selected during the previous machining operation. The selected tool comes to the tool change position, whenever the tool selection command is received by the system.

(ii) Tool Transfer Cycle

In this part of the tool change cycle, the tool which is lying selected in the magazine, is transferred to the spindle and the tool which is in the spindle is transferred to the magazine. Before the tool transfer takes place, the spindle is turned off and moves to the tool change position, so that the tools will not hit the workpiece.

Multipallet Machines

To further reduce the non-productive time, the CNC machines are provided with automatic pallet change systems. Twin-pallet CNC machines are very common but machines with upto 5 pallets are also being used. The multipallet system enables the operator to load the workpiece on one pallet while machining of workpiece on second pallet is going on. This helps in rapid change of workpiece, thereby reducing the idle time of the machine.

EVALUATIVE QUESTIONS

1. What are the distinctive requirements of cutting tools used on CNC machine? Discuss.
2. What is tool pre-setting? How do the preset tools help to increase productivity?
3. What are the special features of work holding devices used in CNC machines? Discuss.
4. Discuss the methods used to reduce the idle time on CNC machine.

GLOSSARY

Absolute Co-ordinate System: Dimensioning system in which all the coordinates are measured from a fixed datum.

Address: A coded identification to indicate the destination of data in the memory.

Alphanumeric: A character which is either numeric digit or letter of alphabet.

Automation: The use of machines for the performance of repetitive tasks. Automated operations are often controlled by a program which automatically checks the sequence validity, monitors and corrects deviations.

Axis: A general direction along which the relative movements of the tool and workpiece are made. The axis may be linear or rotary.

Automatic Tool Changer (ATC): A device for automatically changing the cutting tools under the instructions given in the part program. The required tool is automatically selected from the tool magazine and loaded on the machine spindle.

Binary: A numbering system with base 2. The system employs two digits 0 and 1 and forms the basis of computer operation and design.

Binary Coded Decimal: A form of binary system in which each decimal digit is expressed as a 4-bit binary combination. Large numbers are then coded by coding each digit forming the number.

Block: A group of NC words, characters or digits, used to describe one instruction in a part programme. Each block has a distinctive meaning to convey to the control system. Each block is terminated by 'end of block' character.

Buffer: A temporary storage for holding the data between two stages. In CNC machines, the buffer stores the data temporarily when it is transmitted from the control unit to the machine tool.

Canned Cycle: Fixed sequence of operations, which are permanently stored in the control system. The fixed cycles can be called and used by a single command in the part programme.

Cartesian Coordinates: Also known as rectangular coordinates, is a system of coordinates where a point in space is defined by means of three orthogonal axis i.e. X,Y,Z.

Character: A letter, digit or other symbol which is used as single unit of information in a part programme.

Circular Interpolation: A control system feature used for generating circular profiles in CNC systems.

Closed Loop Control: A control system where a feedback signal is used to continuously monitor the output i.e. velocity or tool position and compare it with input data.

Command: An instruction given to the control system.

Compensation: A correction applied to a controlled axis. In CNC machines compensation is used to compensate for the difference in the radius of cutter used for machining from the cutter specified in the original part programme.

Computer Aided Part Programming: Preparation of a part programme with the aid of a computer.

Computer Numerical Control: A numerical control system where the machine tool is controlled by one or more integrated microcomputers.

Continuous Path Control: A numerical control system in which the relative movements of cutting tool and workpiece are continuously controlled. This type of system is used to machine the contours on a component.

Coordinate: A set of data to specify a particular location.

Cutter Location Data: A data set which defines the cutter centre line for machining a particular component. The CL data is post-processed to generate a part programme for a specific CNC machines.

Cycle: A set of operations which is repeated regularly in the same sequence.

Datum: A reference point from which individual coordinate dimensions are measured.

Direct Numerical Control: The use of a shared computer for distribution of part programs to a number of NC machines and other equipment.

Dwell: A programmed delay in the machining cycle.

EIA Standard Code: A standard code for numerical control punched tape input, proposed by Electronic Industries Association, USA.

End of Block: A character entered in a part programme to denote end of an instruction block.

Feedback: A feature of closed loop control system where the output of the control system is monitored and compared with the input command signal.

Feed Rate: The rate at which the slide moves (mm/min or mm/rev.)

Fixed Block Format: A numerical control format for writing the part programmes. In this system each word in the part programme is identified by its position.

Fixed Zero: A fixed point on a numerically controlled system, about which all machine movements are referenced.

Fixture: A mechanical device for holding the workpiece in a precisely defined position for the machining operations.

Floating Zero: A characteristic of numerical control system which allows the zero reference point to the shifted to anywhere over the full travels of the machine slides.

Format: A specified arrangement of data in a part programme.

G-Code (G Function): A preparatory code which prepares the machine tool for a particular mode of operation.

Incremental Co-ordinate System: A system of giving the dimensions on a drawing in which each point is referenced to the last position rather than a fixed datum.

ISO Standard Code: A standard code for tape punching for numerical control system.

Linear Interpolation: A control system feature used to machine along straight lines.

Loop (Do Loop): repetition of a sequence of operations in a part programme.

Machine Control Unit: The control system including CNC control system, input and output unit and video display.

Magnetic Tape: An input medium for NC systems.

Manual Part Programming: The preparation of a part programme in the required programme format for a particular CNC machine.

M-Function: Also called miscellaneous function or auxiliary function is a coded command for non machining functions like coolant ON/OFF, Spindle ON/OFF etc.

Modal Function: A NC function which remains active until cancelled or superseded by a function of the same group.

Numerical Control: A system to control the machine tool operations with a part programme.

Open Loop System: A control system in which there is no feedback from the output of the system.

Parity Check: A method of checking the binary data for any syntax errors.

Part Programme: A complete set of instructions for the purpose of machining a component on CNC machine.

Peck Drilling cycle: An automatic programmable drilling cycle for deep hole drilling. The drill tool after drilling to a specified depth comes back to the reference point at rapid feed rate and again starts drilling.

Photocell: An light sensitive electronic component which gives an output voltage when light falls on it. The output being proportional to intensity of light falling on it.

Point-to-Point Control: A system where controlled motion is required only to reach a given point, with no machining during the slide traverse from one point to other point.

Pre-Set Tooling: A system in which the tools are set away from the machine tool. This allows quick tool changes and reduces machine downtime.

Programme Proving: Method of verifying the correctness of a part programme.

Programme Stop: An instruction in a CNC part programme which will stop the machine.

Punched Card: An input medium for NC control systems.

Punched Tape: An input medium for NC systems in the form of paper or plastic tape.

Qualified Tooling: The tooling on which the position of cutting edge is guaranteed from specified datum points on tool holder.

Recirculating Ball Screw & Nut Assembly: A leadscrew system in which the contact between the lead screw (ball screw) and the nut is through balls. Here the sliding friction is replaced by rolling friction.

Reference Point: A point within the limits of travel of the machine slides to which the workpiece movements are referenced.

Semi-Qualified Tooling: Cutting tools on which the position of the cutting edge can be adjusted to known dimensions relative to various points on the tool holder.

Sequence Number: A number used to identify the instructional blocks in a part programme.

... Motor: An electric motor in which the rotary move- ... ach rotation of the motor is further ... CNC machines for

Sub-Routine: A sequence of operations, which is separately defined and stored. The subroutine can be called at any point in the main part programmes.

Tab Sequential Format: A NC format for writing the part programme in which each word is separated by a TAB character.

Techogenerator: A feedback device used in closed loop control systems, which generates output voltage which is proportional to the rotational speed of the motor.

Tape Reader: Equipment to read and decode the information from a punched tape. The tape readers may be working on mechanical, optical or pneumatic principle.

Transducer: A device that measures the output and converts into a signal which is understood by the control systems.

Word Address Format: A part programming format in which each word is preceded by an address letter to identify its function.

Zero Shift: A facility on CNC machines to shift the machine Zero to any point within the programmable limits.