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.

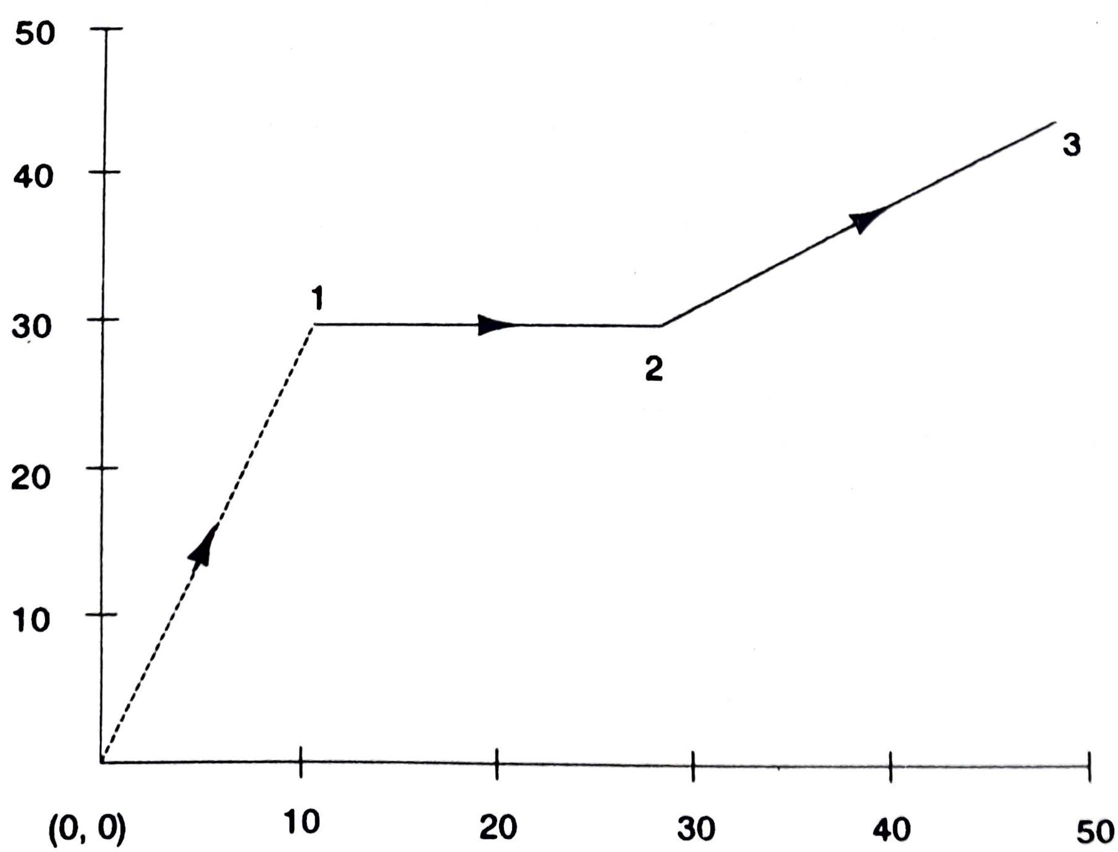

Fig. 5.1.

(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

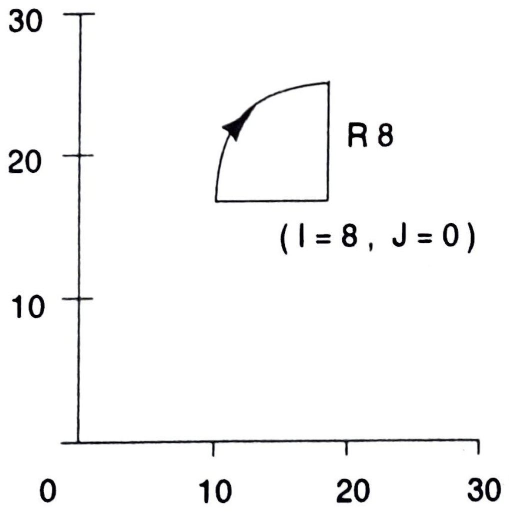

Fig. 5.2(a)

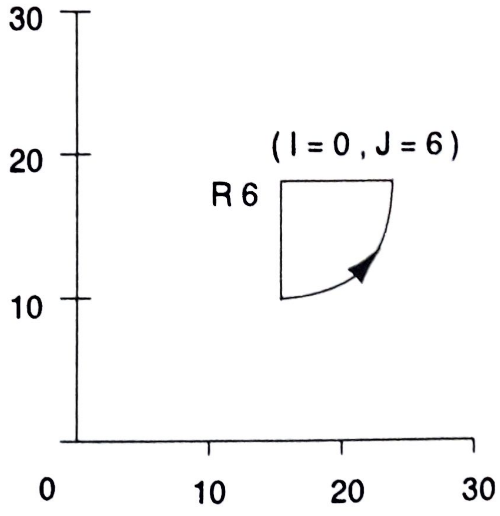

Fig. 5.2(b)

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

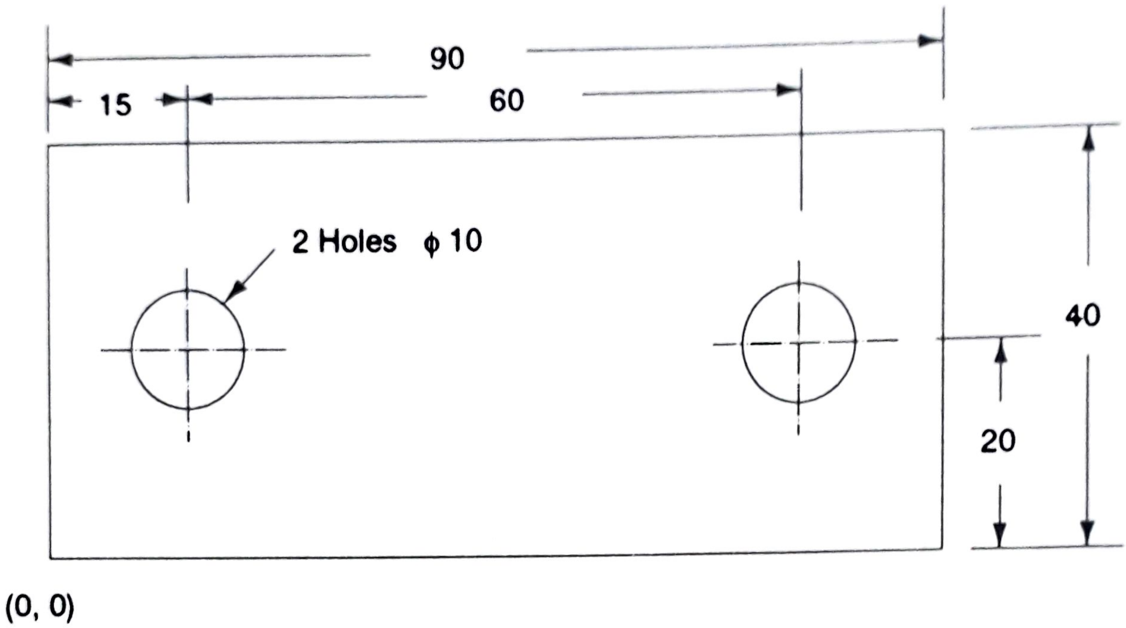

Fig. 5.3.

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.

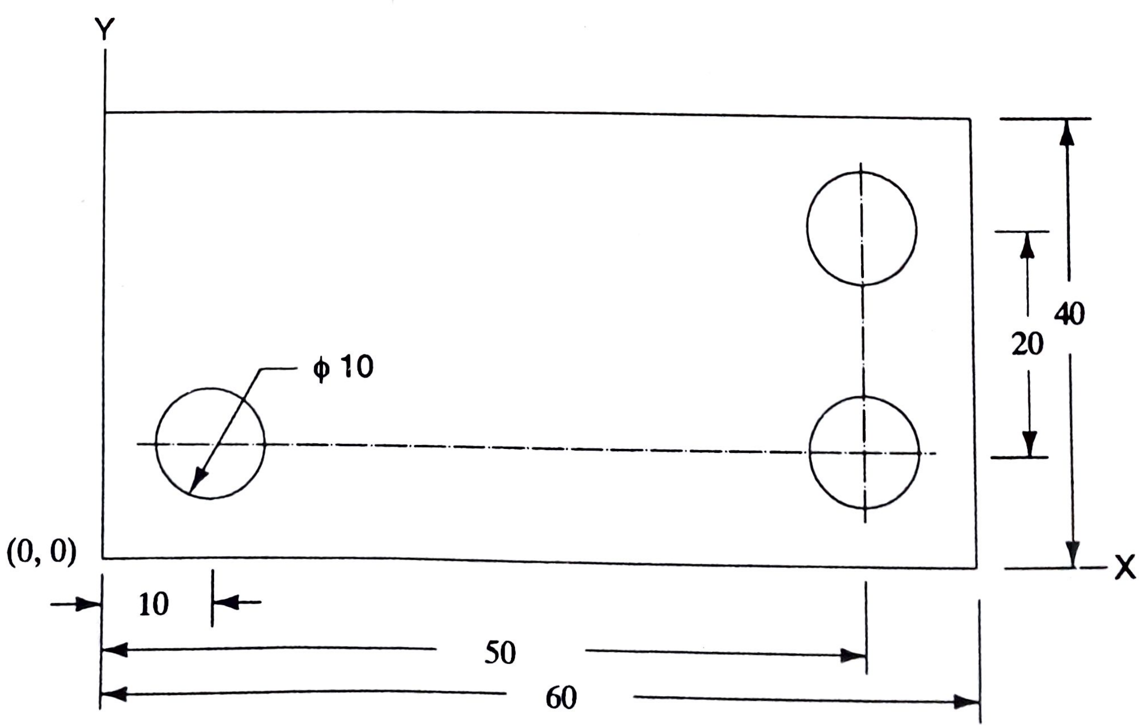

Fig. 5.4.

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:

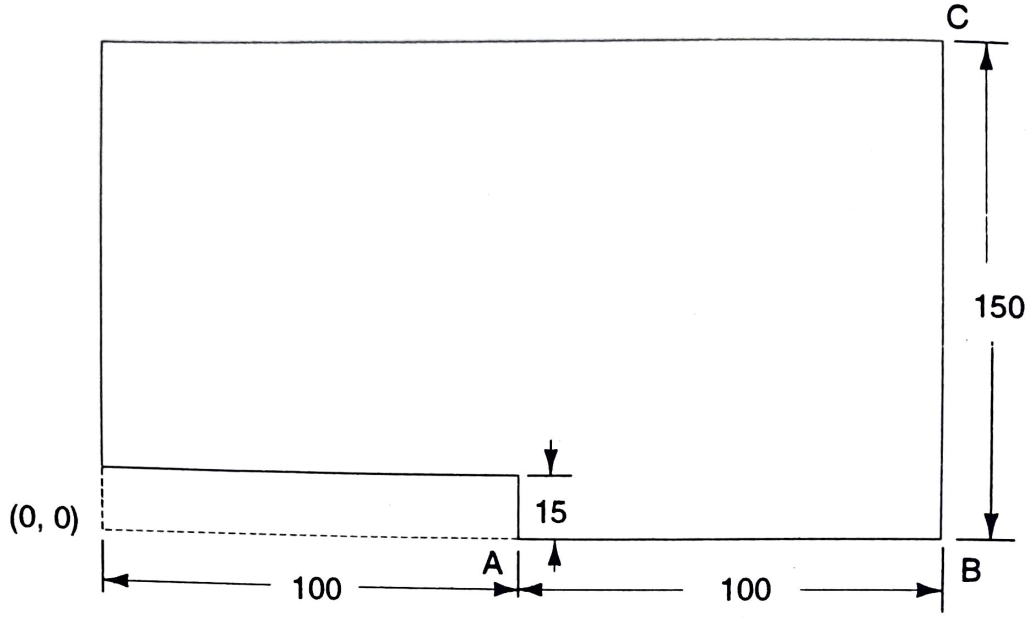

Fig. 5.5. *Z = 0 is 50 mm above the surface of the workpiece and depth of cut is 5 mm*

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.*
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.

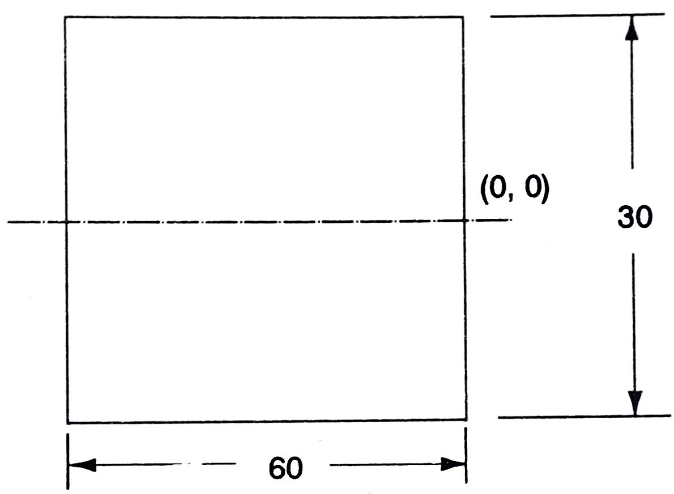

Fig. 5.6.

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

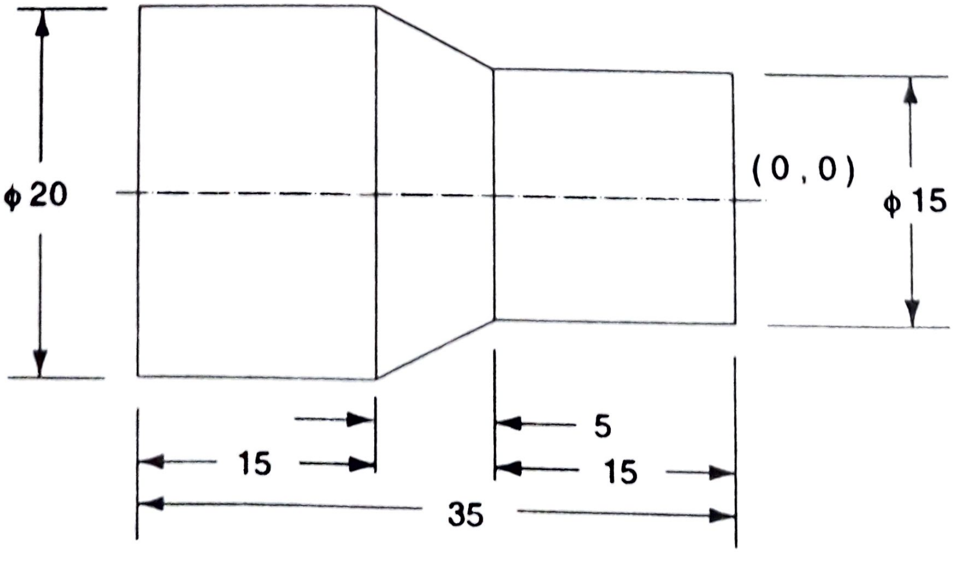

Fig. 5.7.

(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

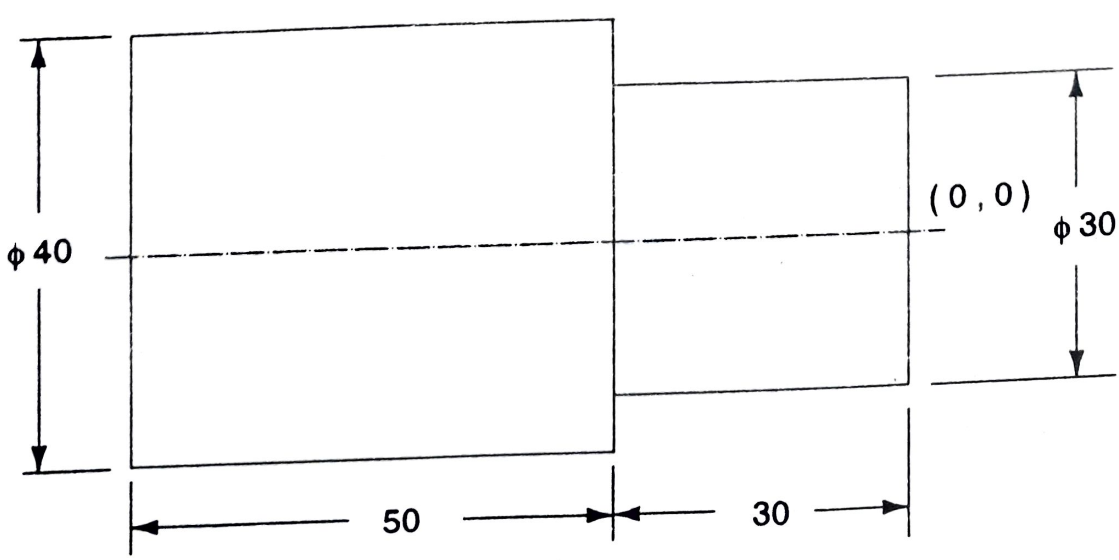

Fig. 5.8.

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)

```
N001  G91   G94 G71 EOB
N002  G00   X 25.00 Z – 5.00 EOB
N003  G01   X – 5.2. F 100 EOB
```

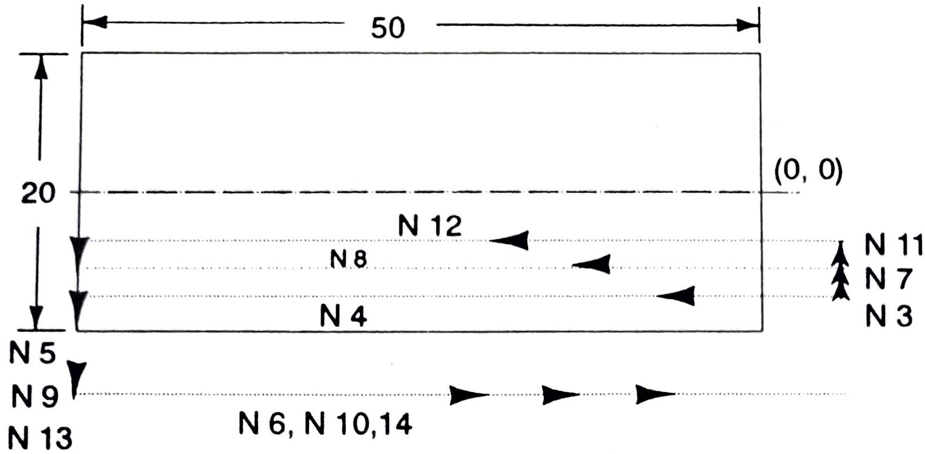

Fig. 5.9.

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)

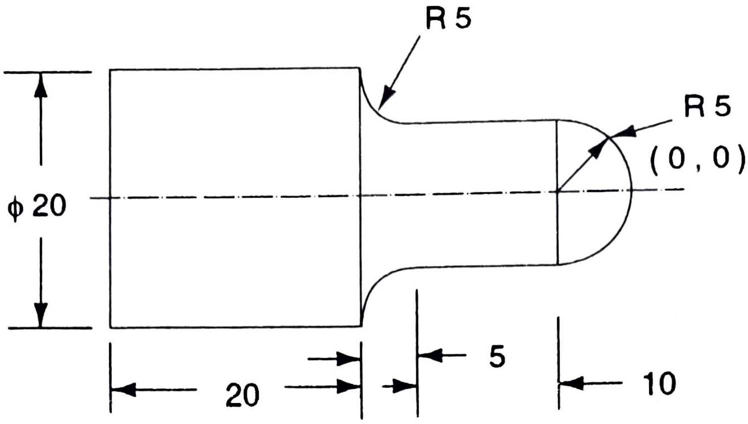

Fig. 5.10.

N05 G01 Z-15.00 EOB
N06 G03 X 20.00 Z-20.00 I5.00 K0 EOB — Counter clockwise 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-

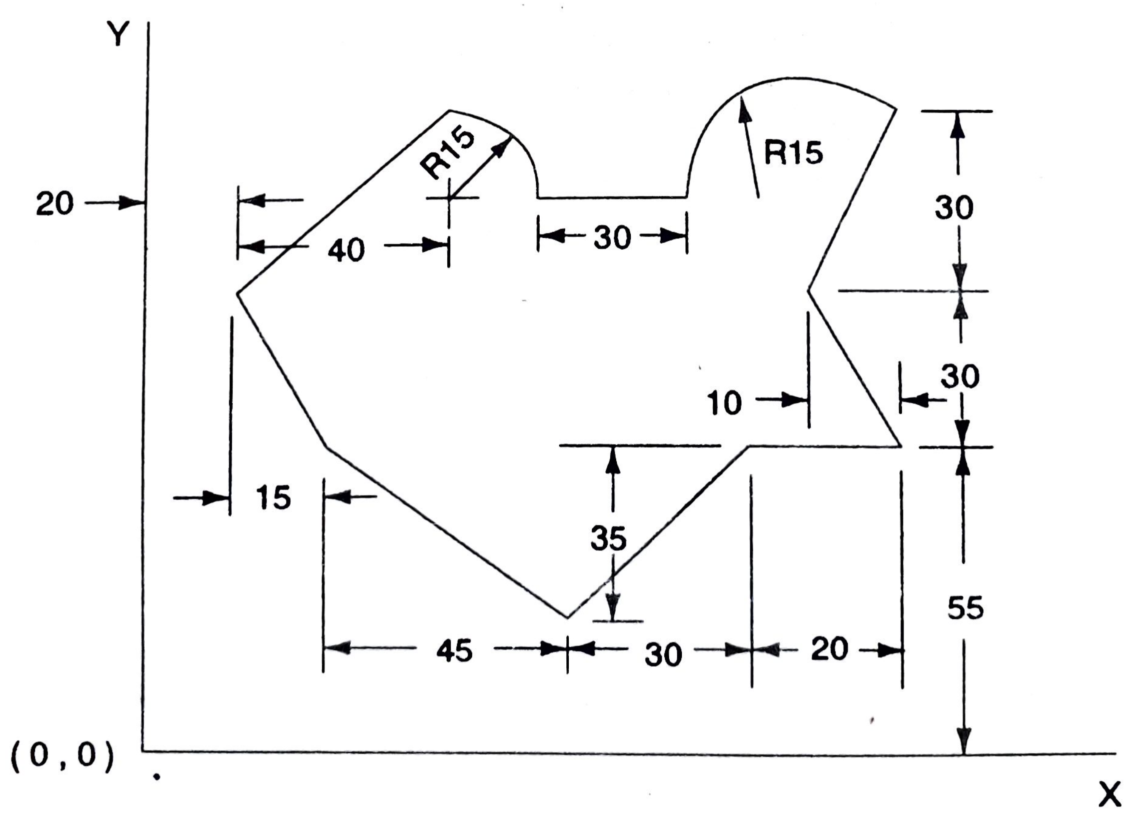

Fig. 5.11.

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 G01  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.

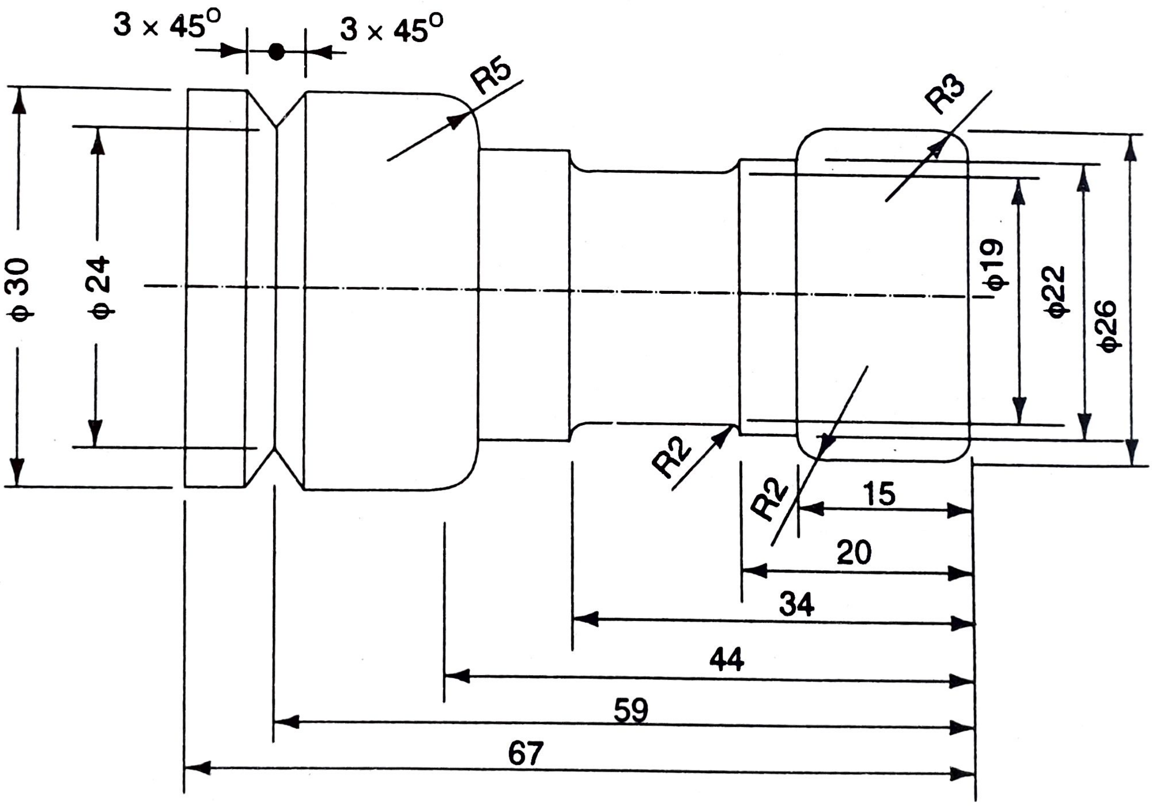

Fig. 5.13.

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.

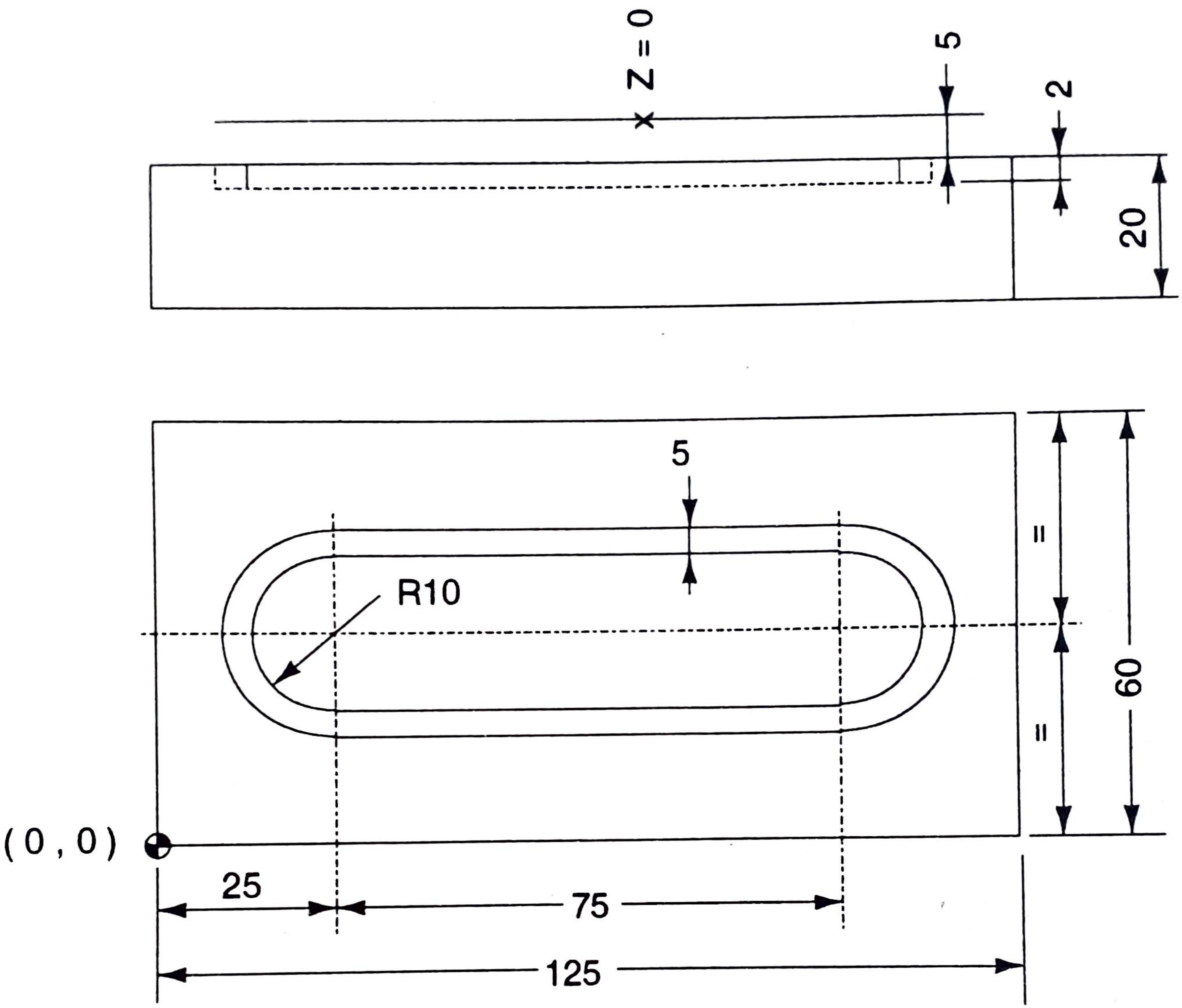

Fig. 5.14.

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 programming 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-direc tion (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

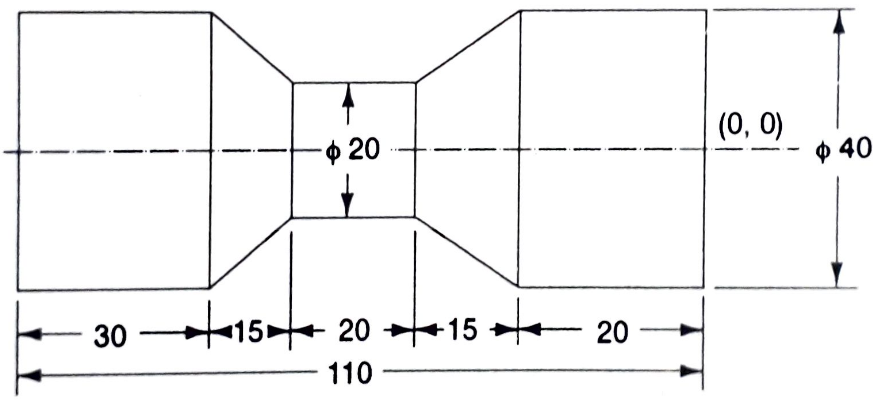

Fig. 6.1.

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

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

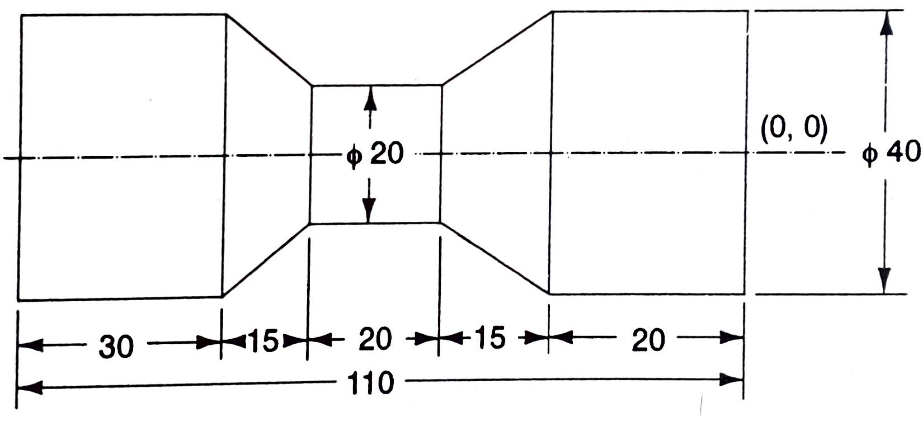

Fig. 6.1.

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.

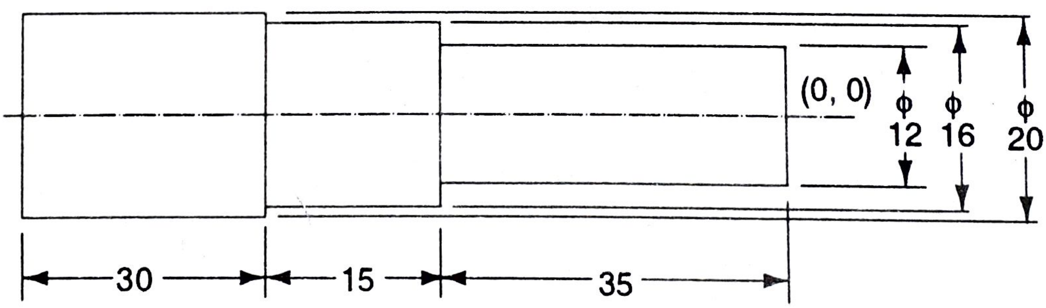

Fig. 6.2.

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  G84  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:

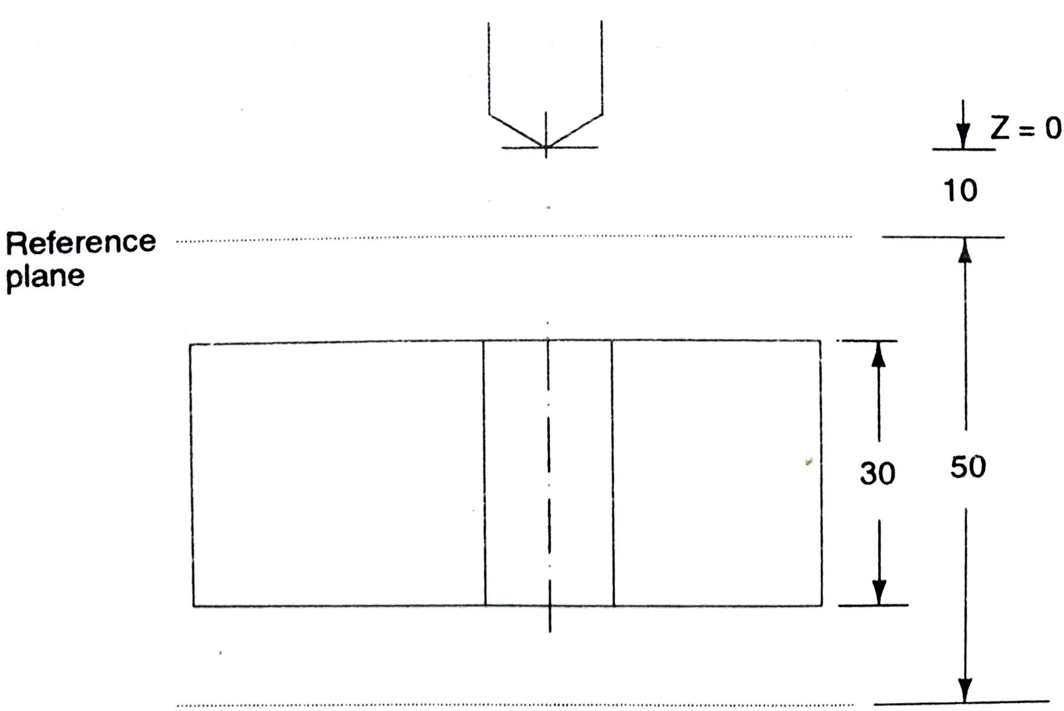

Fig. 6.3.

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

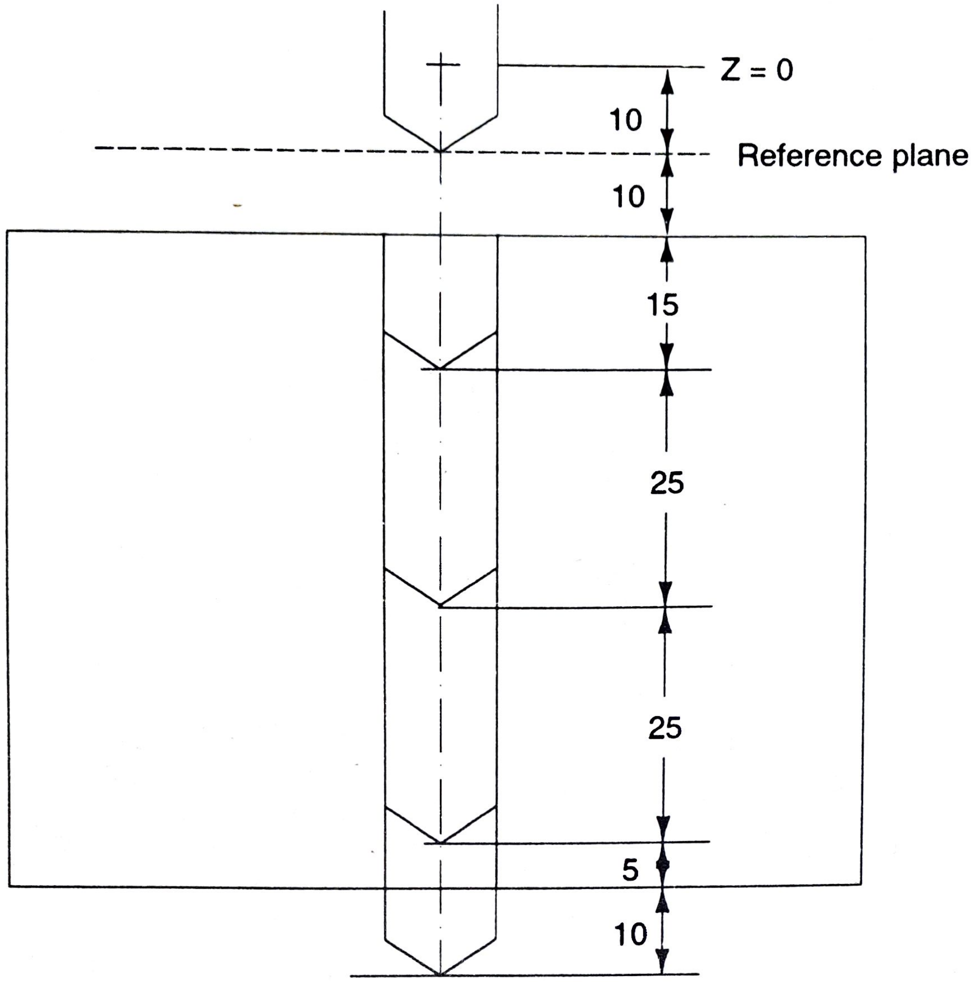

Fig. 6.4.

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:

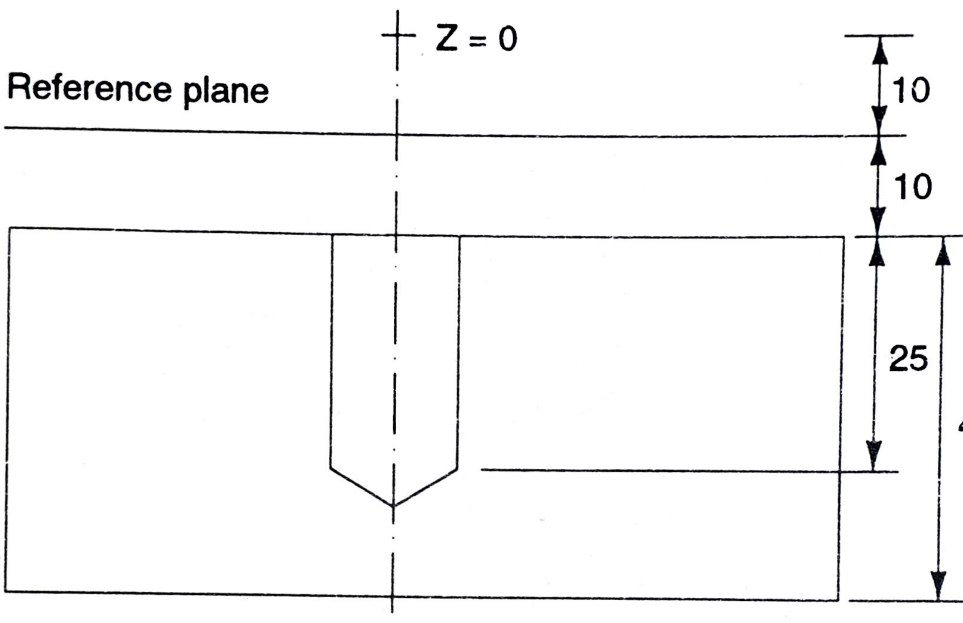

Fig. 6.5.

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.

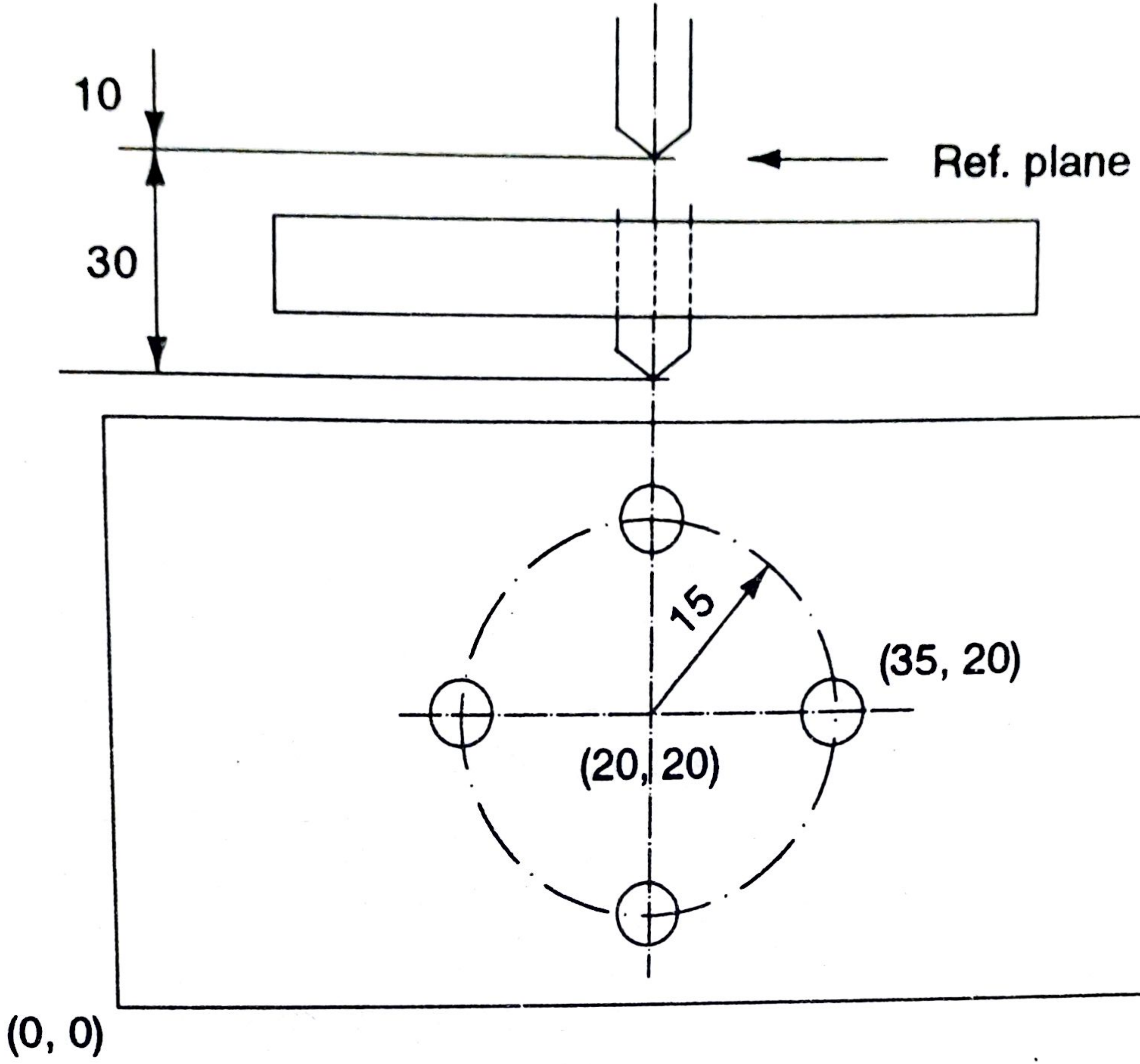

Fig. 6.6.