

## UNIT IV

### NC PART PROGRAMMING

#### Introduction:

The part program is a sequence of instructions, which describe the work, which has to be done on a part, in the form required by a computer under the control of a numerical control computer program. It is the task of preparing a program sheet from a drawing sheet. All data is fed into the numerical control system using a standardized format.

Programming is where all the machining data are compiled and where the data are translated into a language which can be understood by the control system of the machine tool. The machining data is as follows :

- (a) Machining sequence classification of process, tool start up point, cutting depth, tool path, etc.
- (b) Cutting conditions, spindle speed, feed rate, coolant, etc.
- (c) Selection of cutting tools.

While preparing a part program, need to perform the following steps :

- (a) Determine the startup procedure, which includes the extraction of dimensional data from part drawings and data regarding surface quality requirements on the machined component.
- (b) Select the tool and determine the tool offset.
- (c) Set up the zero position for the workpiece.
- (d) Select the speed and rotation of the spindle.
- (e) Set up the tool motions according to the profile required.
- (f) Return the cutting tool to the reference point after completion of work.
- (g) End the program by stopping the spindle and coolant.

The part programming contains the list of coordinate values along the X, Y and Z directions of the entire tool path to finish the component. The program should also contain information, such as feed and speed. Each of the necessary instructions for a particular operation given in the part program is known as an NC word. A group of such NC words constitutes a complete NC instruction, known as block. The commonly used words are N, G, F, S, T, and M. The same is explained later on through examples.

Hence the methods of part programming can be of two types depending upon the two techniques as below :

- (a) Manual part programming, and
- (b) Computer aided part programming.

#### Manual Part Programming

The programmer first prepares the program manuscript in a standard format. Manuscripts are typed with a device known as flexo writer, which is also used to type the program instructions. After the program is typed, the punched tape is prepared on the flexo writer. Complex shaped components require tedious calculations. This type of programming is carried out for simple machining parts produced on point-to-point machine tool.

To be able to create a part program manually, need the following information :

- (a) Knowledge about various manufacturing processes and machines.
- (b) Sequence of operations to be performed for a given component.
- (c) Knowledge of the selection of cutting parameters.
- (d) Editing the part program according to the design changes.
- (e) Knowledge about the codes and functions used in part programs

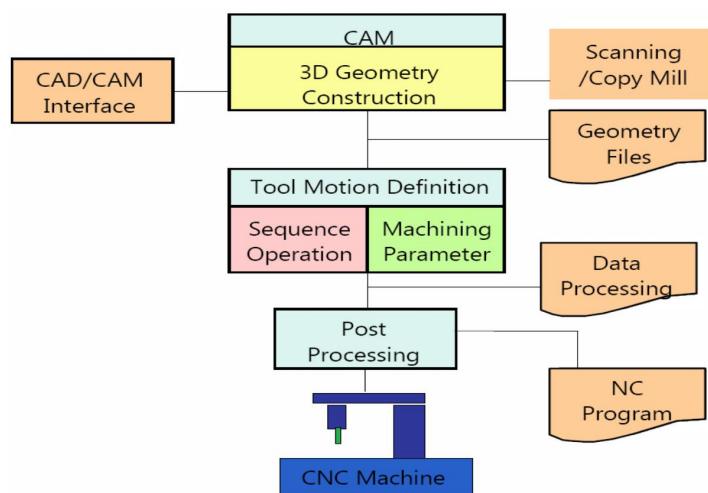
If the complex-shaped component requires calculations to produce the component are done by the programming software contained in the computer. The programmer communicates with this system through the system language, which is based on words. There are various programming languages developed in the recent past, such as APT (Automatically Programmed Tools), ADAPT, AUTOSPOT, COMPAT-II, 2CL, ROMANCE, SPLIT is used for writing a computer programme, which has English like statements. A translator known as compiler program is used to translate it in a form acceptable to MCU.

The programmer has to do only following things :

- (a) Define the work part geometry.
- (b) Defining the repetition work.
- (c) Specifying the operation sequence.

Over the past years, lot of effort is devoted to automate the part programme generation. With the development of the CAD (Computer Aided Design)/CAM (Computer Aided Manufacturing) system, interactive graphic system is integrated with the NC part programming. Graphic based software using menu driven technique improves the user friendliness. The part programmer can create the geometrical model in the CAM package or directly extract the geometrical model from the CAD/CAM database. Built in tool motion commands can assist the part programmer to calculate the tool paths automatically. The programmer can verify the tool paths through the graphic display using the animation function of the CAM system. It greatly enhances the speed and accuracy in tool path generation.

Interactive Graphic System in Computer Aided Part Programming:



## FUNDAMENTAL ELEMENTS FOR DEVELOPING MANUAL PART PROGRAMME:

The programmer to consider some fundamental elements before the actual programming steps of a part takes place. The elements to be considered are as follows :

Type of Dimensioning System:

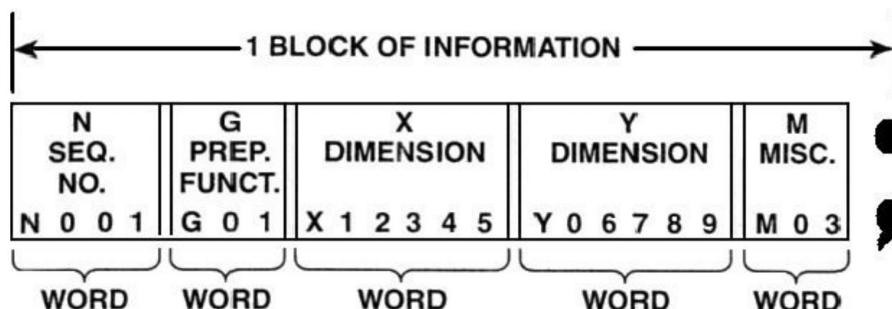
a) Axis Designation:

The programmer also determines how many axes are availed on machine tool. Whether machine tool has a continuous path and point-to-point control system that has been explained.

NC Words:

The NC word is a unit of information, such as a dimension or feed rate and so on. A block is a collection of complete group of NC words representing a single NC instruction. An end of block symbol is used to separate the blocks. NC word is where all the machining data are compiled and where the data are translated in to a language, which can be understood, by the control system of the machine tool.

Block of Information:



NC information is generally programmed in blocks of words. Each word conforms to the EIA standards and they are written on a horizontal line. If five complete words are not included in each block, the machine control unit (MCU) will not recognize the information; therefore the control unit will not be activated. It consists of a character N followed by a three digit number raising from 0 to 999.

N001 – represents the sequence number of the operation.

G01 – represents linear interpolation.

X12345 – will move the table in a positive direction along the X-axis. Y06789 – will move the table along the Y-axis.

M03 – Spindle on CW and ; – End of block.

### Standard G and M Codes:

The most common codes used when programming NC machines tools are G-codes (preparatory functions), and M codes (miscellaneous functions). Other codes such as F, S, D, and T are used for machine functions such as feed, speed, cutter diameter offset, tool number, etc. G-codes are sometimes called cycle codes because they refer to some action occurring on the X, Y, and/or Z-axis of a machine tool. The G-codes are grouped into categories such as Group 01, containing codes G00, G01, G02, G03, which cause some movement of the machine table or head. Group 03 includes either absolute or incremental programming. A G00 code rapidly positions the cutting tool while it is above the workpiece from one point to another point on a job. During the rapid traverse movement, either the X or Y-axis can be moved individually or both axes can be moved at the same time. The rate of rapid travel varies from machine to machine.

#### G-Codes (Preparatory Functions)

Code	Function
G00	Rapid positioning
G01	Linear interpolation
G02	Circular interpolation clockwise (CW)
G03	Circular interpolation counterclockwise (CCW) G20      Inch input (in.)
G21	Metric input (mm)
G24	Radius programming
G28	Return to reference point
G29	Return from reference point
G32	Thread cutting
G40	Cutter compensation cancel
G41	Cutter compensation left
G42	Cutter compensation right
G43	Tool length compensation positive (+) direction
G44	Tool length compensation minus (-) direction
G49	Tool length compensation cancels
G 53	Zero offset or M/c reference G54      Settable zero offset
G84	canned turn cycle
G90	Absolute programming
G91	Incremental programming

#### M-Codes (Miscellaneous Functions)

M or miscellaneous codes are used to either turn ON or OFF different functions, which control certain machine tool operations. M-codes are not grouped into categories, although several codes may control the same type of operations such as M03, M04, and M05, which control the machine tool spindle. Some of important codes are given as under with their function s:

Code	Function
M00	Program stop
M02	End of program
M03	Spindle start (forward CW)
M04	Spindle start (reverse CCW)
M05	Spindle stop
M06	Tool change
M08	Coolant on
M09	Coolant off
M10	Chuck clamping
M11	Chuck - unclamping
M12	Tailstock spindle out
M13	Tailstock spindle in
M17	Tool post rotation normal
M18	Tool post rotation reverse
M30	End of tape and rewind or main program end
M98	Transfer to subprogram
M99	End of subprogram

### **Tape Programming Format:**

Both EIA and ISO use three types of formats for compiling of NC data into suitable blocks of information with slight difference.

#### **Word Address Format**

This type of tape format uses alphabets called address, identifying the function of numerical data followed. This format is used by most of the NC machines, also called variable block format. A typical instruction block will be as below :

N20 G00 X1.200 Y.100 F325 S1000 T03 M09 <EOB>

or

N20 G00 X1.200 Y.100 F325 S1000 T03 M09;

The MCU uses this alphabet for addressing a memory location in it.

#### **Tab Sequential Format**

Here the alphabets are replaced by a Tab code, which is inserted between two words. The MCU reads the first Tab and stores the data in the first location then the second word is recognized by reading the record Tab. A typical Tab sequential instruction block will be as below :

>20 >00 >1.200 >.100 >325 >1000 >03 >09

#### **Fixed Block Format**

In fixed block format no letter address of Tab code are used and none of words can be omitted. The main advantage of this format is that the whole instruction block can be read at the same

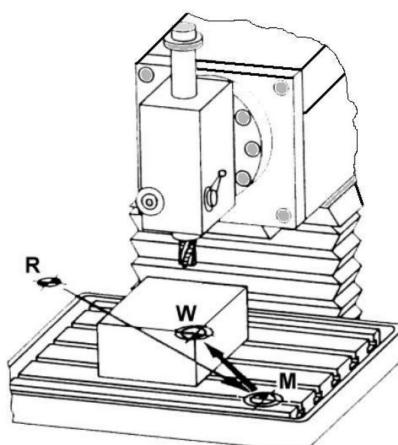
instant, instead of reading character by character. This format can only be used for positioning work only. A typical fixed block instruction block will be as below:

20 00 1.200 .100 325 1000 03 09 <EOB>

#### Machine Tool Zero Point Setting:

The machine zero point can be set by two methods by the operator, manually by a programmed absolute zero shift, or by work coordinates, to suit the holding fixture or the part to be machined.

#### Manual Setting:



The operator can use the MCU controls to locate the spindle over the desired part zero and then set the X and Y coordinate registers on the console to zero.

#### Absolute Zero shift:

The absolute zero shift can change the position of the coordinate system by a command in the CNC program. The programmer first sends the machine spindle to home zero position by a command in the program. Then another command tells the MCU how far from the home zero location, the coordinate system origin is to be positioned.

R = Reference point (maximum travel of machine)

W = Part zero point workpiece coordinate system

M = Machine zero point (X0, Y0, Z0) of machine coordinate system

The sample commands may be as follows :

N1 G28 X0 Y0 Z0 (sends spindle to home zero position or Return to reference point).

N2 G92 X3.000 Y4.000 Z5.000 (the position the machine will reference as part zero or Programmed zero shift).

#### Coordinate Word:

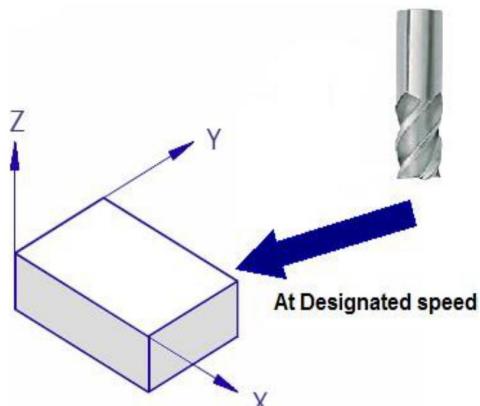
A coordinate word specifies the target point of the tool movement or the distance to be moved. The word is composed of the address of the axis to be moved and the value and direction of the movement.

### Example

X150 Y-250 represents the movement to (150, □ 250). Whether the dimensions are absolute or incremental will have to be defined previously using G-Codes.

### Linear Interpolation:

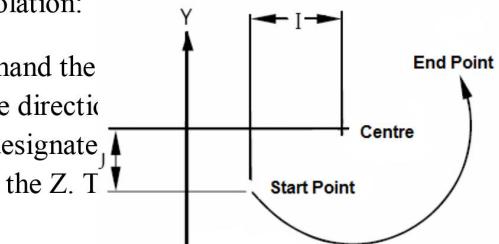
This is to command the cutter to move from the existing point to the target point along a straight line at the speed designated by the F address.



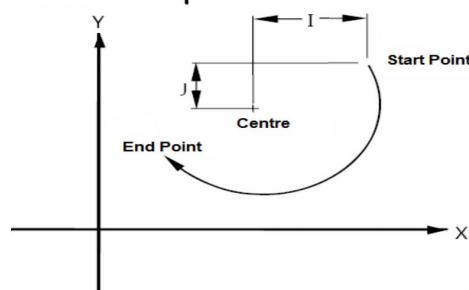
### Circular Interpolation:

This is to command the arc in clockwise direction. Circular arc is designated by I, J, K parameters of the center of the arc along the X-axis, Y, and Z along the Z. T of the arc.

Clockwise Circular Interpolation:



o the target point along a circular  
parameters of the center of the  
ce along the X-axis, J along the  
om the starting point to the center

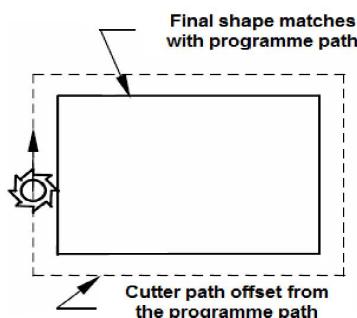


### Counter Clockwise Circular Interpolation:

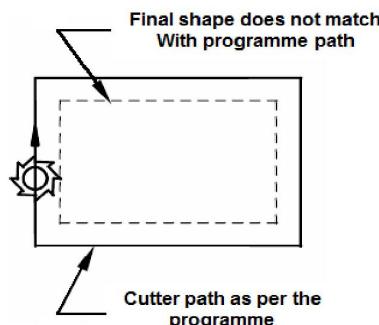
### Cutter compensation:

In NC machining, if the cutter axis is moving along the programmed path, the dimension of the workpiece obtained will be incorrect since the diameter of the cutter has not been taken into account. What the system requires are the programmed path, the cutter diameter and the position of the cutter with reference to the contour. The cutter diameter is not included in the programme. It has to be input to the NC system in the tool setting process.

### Tool path without cutter compensation:



### With cutter compensation:



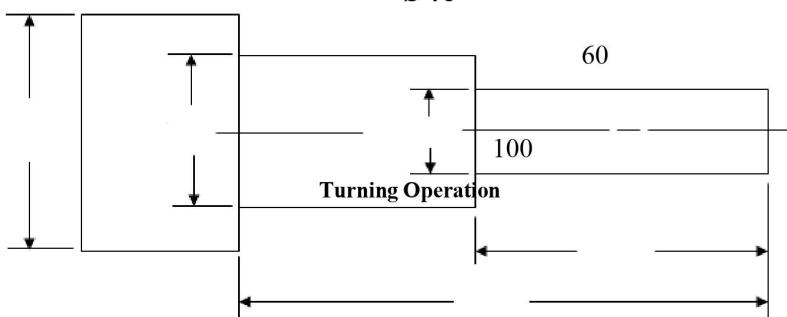
### Example :

01 (All dimensions are in mm).

$\varnothing 35$

$\varnothing 25$

$\varnothing 16$



% 1000;	(Main programme)
N01 G54 G90 G71 G94 M03 S800;	(Parameters Setting)
N05 G01 X-12.5 Z0 F2;	(Facing the job)
N10 G00 Z1;	(Retrieval of tool)
N15 G00 X00;	(Tool clearance)
N20 G01 Z-100;	(Starting cut)
N25 G00 X1 Z1;	(Clearance position)
N30 G00 X-2;	(Position of cut)
N35 G01 Z-60;	(Cutting length)
N40 G00 X-1 Z1;	(Retrieval of tool)
N45 G00 X-3;	(Position of cut)
N50 G01 Z-60;	(Cutting length)
N55 G00 X-2 Z1;	(Retrieval of tool)
N60 G00 X-4;	(Position of cut)
N65 G01 Z-60;	(Cutting length)
N75 G00 X-4.5;	(Position of cut)
N80 G01 Z-60;	(Cutting length)
N85 G00 X5 Z5;	(Final position of tool)
N90 M02;	(End of programme)

#### FIXED CYCLE/CANNED CYCLE:

A fixed cycle is a combination of machine moves resulting in a particular machining function such as drilling, milling, boring and tapping. By programming one cycle code number, as many as distinct movements may occur. These movements would take blocks of programme made without using Fixed or Canned cycles. The corresponding instructions of a fixed cycle are already stored in the system memory. The advantages of writing a part programme with these structures are :

- (a) Reduced lengths of part programme.
- (b) Less time required developing the programme.
- (c) Easy to locate the fault in the part programme.
- (d) No need to write the same instructions again and again in the programme.
- (e) Less memory required in the control unit.

#### Example:

01 (G81 Drilling Cycle) (All dimensions are in mm).

R00 – Dwell time at the starting point for chip removal.

R02 – Reference plane absolute with sign.

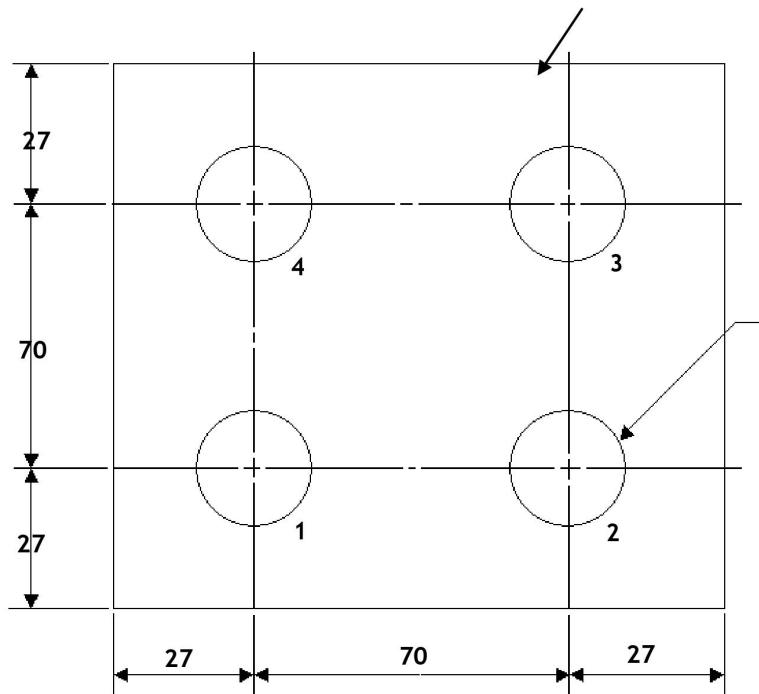
R03 – Final depth of hole absolute with sign.

R04 – Dwell time at the bottom of drilled hole for chip breaking.

R10 – Retract plane without sign.

R11 – Drilling axis number 1 to 3.

% 400;



N5 G17 G71 G90 G94 G55; N10 T1 L90;

N15 G00 D5 Z5 M3 S600 X27 Y27;

N20 G81 R02=5, R03=-33, R11=3, F50 M7; N25 X97;

N30 Y97; N35 X27;

N40 G00 G80 Z100 M9;

N45 M02;