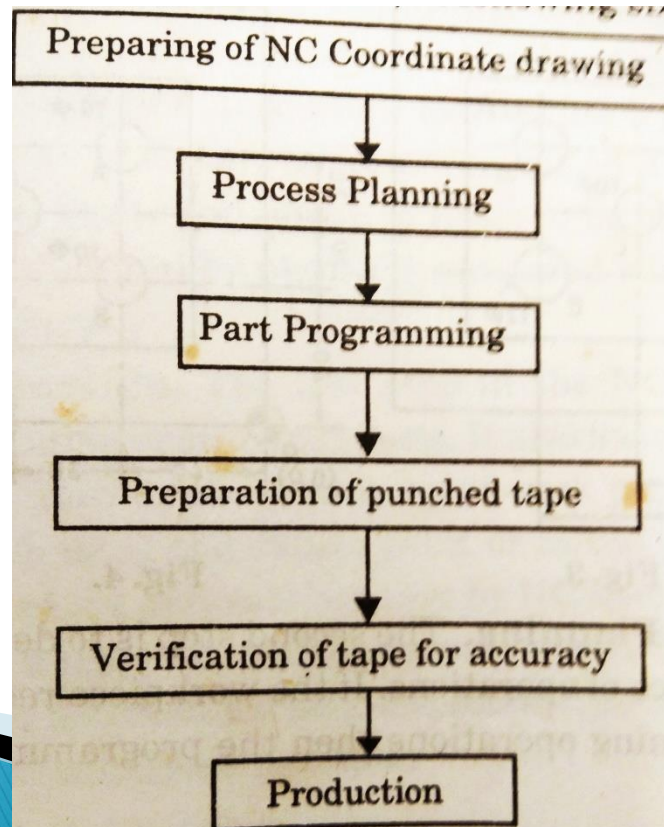# FUNDAMENTALS OF PART PROGRAMMING –

Part programme is an important component of the NC/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 NC/CNC machine is planned and documented. The part programme is then transferred to one of the input media, which is used to instruct the NC/CNC machine.

To utilize numerical control in manufacturing, the following steps must be accomplished.

```
┌─────────────────────────────────────┐
│  Preparing of NC Coordinate drawing  │
└─────────────────────────────────────┘
                  │
                  ▼
        ┌───────────────────┐
        │  Process Planning  │
        └───────────────────┘
                  │
                  ▼
        ┌───────────────────┐
        │  Part Programming  │
        └───────────────────┘
                  │
                  ▼
    ┌───────────────────────────────┐
    │  Preparation of punched tape   │
    └───────────────────────────────┘
                  │
                  ▼
    ┌───────────────────────────────┐
    │  Verification of tape for accuracy  │
    └───────────────────────────────┘
                  │
                  ▼
        ┌───────────────────┐
        │    Production      │
        └───────────────────┘
```

1. Preparing of NC Coordinate drawing-

The first step in making a NC program is to convert the existing blue print or shop drawing into an NC coordinate drawing.
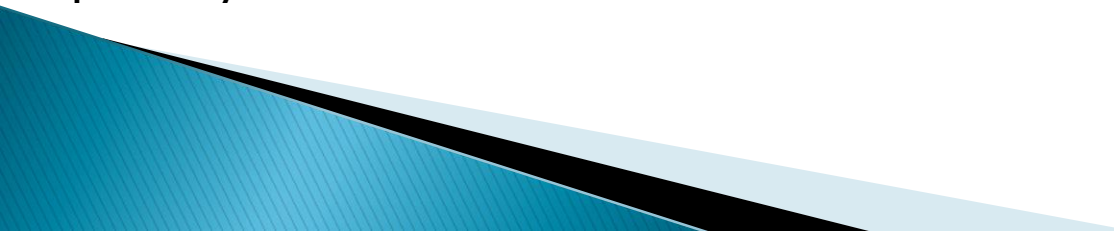
2. Process Planning-

The engineering drawing of the workpart must be interpreted in terms of the manufacturing processes to be used. this step is referred to as process planning and it is concerned with the preparation of a route sheet. The route sheet is a listing of the sequence of operations which must be performed on the workpart. It is called a route sheet because it also lists the machines through which the part must be routed in order to accomplish the sequence of operations.

3. Part programming-

A part programmer plans the process for the portions of the job to be accomplished by NC. Part programmers are knowledgeable about the machining process and they have been trained to program for numerical control. They are responsible for planning the sequence of machining steps to be performed by NC and to document these in a special format.

4. Tape preparation-

A punched tape is prepared from the part programmers NC process plan. In manual part programming, the punched tape is prepared directly from the part program manuscript on a typewriter like device equipped with tape punching capability.

5. Tape verification-

After the punched tape has been prepared, a method is usually provided for checking the accuracy of the tape. Some times the tape is checked by running it through a computer program which plots the various tool movements (or table movements) on paper. In this way, major errors in the tape can be discovered. The "acid test" of the tape involves trying it out on the machine tool to make the part. A foam or plastic material is sometimes used for this tryout. Programming errors are not uncommon, and it may require about three attempts before the tape is correct and ready to use.

6. Production-

The final step in the NC procedure to use the NC tape in production. This involves ordering the raw workparts specifying and preparing the tooling and any special fixturing that may be required, and setting up The NC machine tool for the job. The machine tool operator's function during production is to load the raw workpart in the machine and establish the starting position of the cutting tool relative to the workpiece. The NC system then takes over and machines the part according to the instructions on tape. When the part is completed, the operator removes it from the machine and loads the next part.

Part programming-

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 an NC computer program. There are two ways to program for NC:
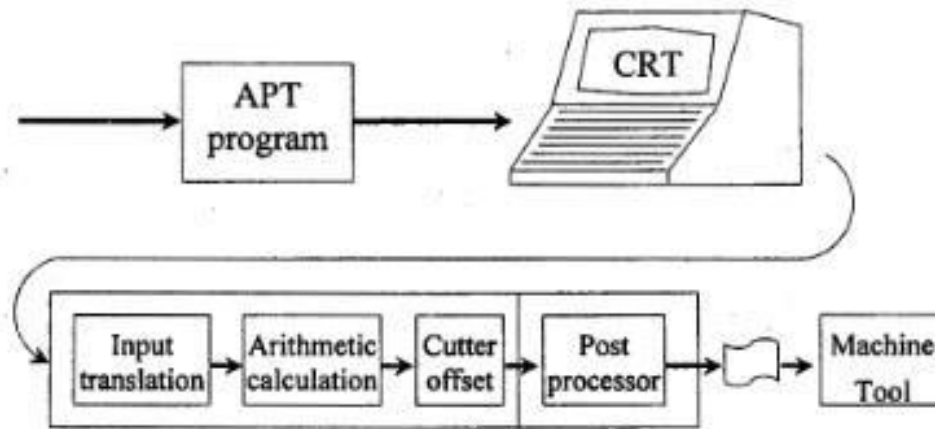
Manual part programming

Computer-assisted part programming

Manual part programming-

To prepare a part program using the manual method, the programmer writes the machining instructions on a special form called a **part programming manuscript**. The instruction must be prepared in a very precise manner because the typist prepares the NC tape directly from the manuscript. The manuscript is a listing of the relative tool and workpiece locations. It also includes other data, such as preparatory commands, miscellaneous instructions, and speed /feed specifications, all of which are needed to operate the machine under tape control.

Computer-assisted part programming-

In computer-assisted part programming, much of the tedious computational work required in manual part programming is transferred to the computer. This is especially appropriate for complex work piece geometries and jobs with many machining steps. Use of the computer in these situations results in significant savings in part programming time. Also one of the high level language such as APT, ADAPT, SPLIT, 2CL, ROMANCE and AUTOSTOP 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.

Computer's job in computer–assisted 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-We determine what type of dimensioning system the machine uses, whether an absolute or incremental dimensional system

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

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.
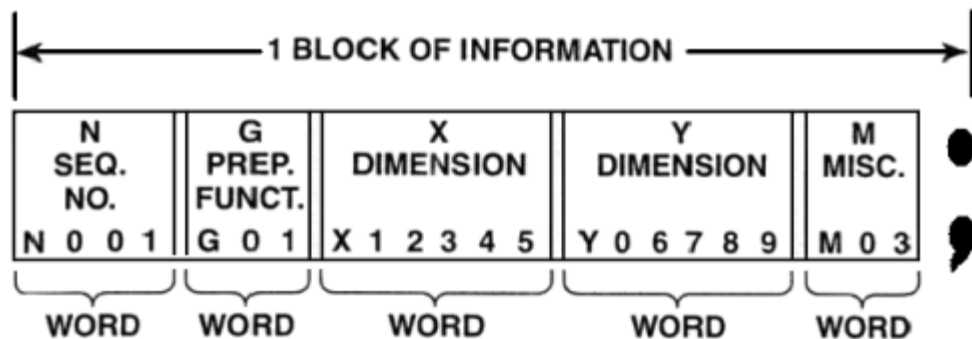

Figure 4.2 : A Block of Information

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.

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)
(ii) Preparatory Function (**G**-Words)
(iii) Coordinates (**X-, Y- and Z**-Words)
(iv) Feed Function (**F**- Word)
(v) Spindle Speed Function (**S**- Word)
(vi) Tool Selection Function (**T**-Word)
(vii) Miscellaneous Function (**M**-Word)
(viii) End of Block (**EOB**)

N – Sequence number –This is used to identify the block
G - Preparatory function –This word is used to prepare the controller for instructions that are to follow.
X – X-coordinate –These give the coordinate positions of the tool.
Y – Y-coordinate –These give the coordinate positions of the tool.
F – Feed rate –This specifies the feed in a machining operation
S - Spindle speed –This specifies the cutting speed of the process, the rate at which the spindle rotates.
T - Tool number –This word would be needed only for machines with a tool turret or automatic tool changer.
M - Miscellaneous function –The m-word is used to specify certain miscellaneous or auxiliary functions which may be available on the machine tool.
EOB – End of block –To identify the end of the instruction

Programming Formats –

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

(a) Fixed block format

(b) Tab sequential format

(c) Word address format

```
0050 00 +0025400 +0012500 +0000000 0000 00
0060 01 +0025400 +0012500 –0010000 0500 08
0070 00 +0025400 +0012500 +0000000 0000 09
```

(a) 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.

```
0050 TAB 00 TAB +0025400 TAB +0012500
```

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

```
0060 TAB 01 TAB TAB TAB -0010000 TAB 0
0070 TAB 00 TAB TAB TAB -0000000 TAB 0
```

b) 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.

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

(c) 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:

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

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.

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

Common Format of a Block
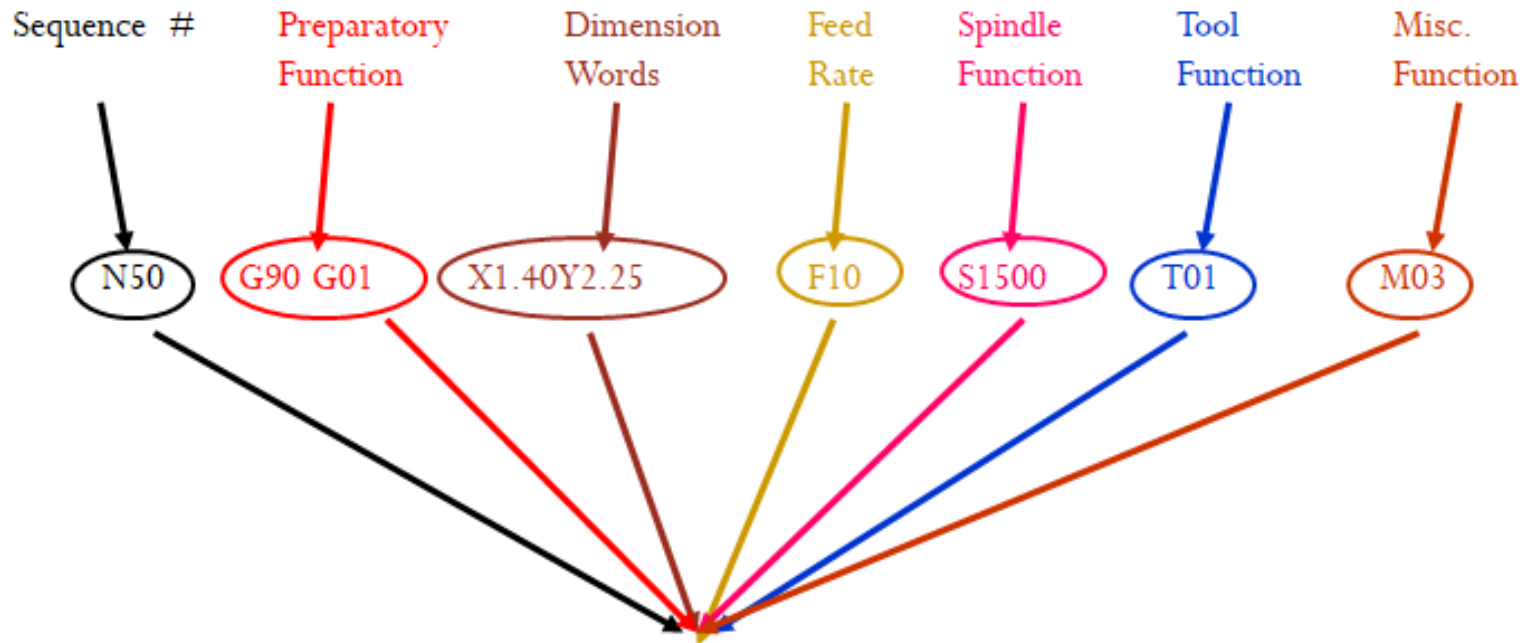
Programming Key Letters

O – Program number (Used for program identification)
N – Sequence number (Used for line identification)
G – Preparatory function
X – X axis designation
Y – Y axis designation
Z – Z axis designation

R – Radius designation
F – Feed rate designation
S – Spindle speed designation
H – Tool length offset designation
D – Tool radius offset designation
T – Tool Designation
M – Miscellaneous function

Modal commands: Commands issued in the NC program that will stay in effect until it is changed by some other command, like, feed rate selection, coolant selection, etc.

Nonmodal commands: Commands that are effective only when issued and whose effects are lost for subsequent commands, like, a dwell command which instructs the tool to remain in a given configuration for a given amount of time.

Table of Important G codes

G00 Rapid Transverse
G01 Linear Interpolation
G02 Circular Interpolation, CW
G03 Circular Interpolation, CCW
G17 XY Plane,G18 XZ Plane,G19 YZ Plane
G20/G70 Inch units
G21/G71 Metric Units
G40 Cutter compensation cancel
G41 Cutter compensation left
G42 Cutter compensation right
G43 Tool length compensation (plus)
G43 Tool length compensation (plus)
G44 Tool length compensation (minus)
G49 Tool length compensation cancel
G80 Cancel canned cycles
G81 Drilling cycle
G82 Counter boring cycle
G83 Deep hole drilling cycle
G90 Absolute positioning
G91 Incremental positioning

Table of Important M codes

M00 Program stop
M01 Optional program stop
M02 Program end
M03 Spindle on clockwise
M04 Spindle on counterclockwise
M05 Spindle stop
M06 Tool change
M08 Coolant on
M09 Coolant off
M10 Clamps on
M11 Clamps off
M30 Program stop, reset to start