Chapter 1
Introduction to Word Address Programming

1.1 Objectives

After completion of this chapter the reader will be able to:

1. Understand the meaning of common terminology associated with writing a CNC program.
2. Explain the structure of a CNC program.
3. Understand the proper arrangement and use of addresses in a block.
4. Distinguish between G and M codes in a program.

1.2 Introduction

This chapter will cover the fundamental concepts and terminologies used in CNC programming. The reader will be introduced to *programming characters, addresses, words,* and *blocks*. Implementaion of G-code are coverned by varios standards and can be different between countries, for example, the standarized version of G-code used in the United States is the RS-247-D which was setteld by Electronic Industries Alliance [1]. In many European countries other standards maybe used, for example, in Austria the Deutsches Institut fr Normung e.V. (DIN 66025) is used.

1.3 Program Terminology

Addresses

The address consists of an alphanumeric character, or punctuation mark, known as programming characters. For example, in CNC programming letters and Semicolon

(;) are considered a programming characters. However, in EMCO WinNC the following list of programming characters are used:

Address list

C.....chamfer
F.....feed rate or thread pitch
G.....path function
H.....number of the correction value address in the offset register (offset)
I,J,K.....circle parameter, scale factor, K also number of repetitions of a cycle, mirror axes
M.....miscellaneous function
N.....block number 1 to 9999
O.....program number 1 to 9499
P.....dwell, subprogram call
Q.....cutting depth or shift value in cycle
R.....raduis, retraction height with cycle
S.....spindel speed
T.....tool call
X,Y,Z.....position data (X also dwell)
;.....block end (1.1)

Words

The combination of an address and a number, as shown in 1.2, is defined as a word. The letter describes the required operation of the numerical value that follows the address. In example 1.2 the G letter followed by the numerical value of 1 executes a linear interpolation command.

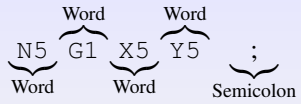
Example of a word

$$\underbrace{\mathtt{G}}_{\text{Address}}\ \overbrace{\mathtt{1}}^{\text{Number}} \Leftarrow \text{Word} \qquad (1.2)$$

Blocks

A block is defined as a line of information that consists of at least a single word or a collection of words, as depicted in 1.3, that describes a specific CNC operation.

Example of a single Block



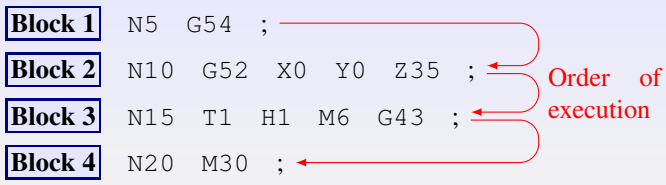
(1.3)

Note: The Semicolon is automatically generated in EMCO WinNC to indicate the end-of-block.

Program

The CNC program consists of a sequence of blocks which are stored in the Machine Control Unit (MCU). The MCU execute the program block by block according to the listed order of each block.

Example of a program



1.4 Program and Sequence Numbers

Program Number (O)

CNC programs are stored in the MCU by unique numeric codes known as program numbers. The controller recognize programs by program numbers, and can store several different programs at once as shown in Fig. 1.1. WinNC program numbers range from O0001 to O9499. Program numbers alone is not enough to better identify the purpose of the code therefrom, comment are used to describe such program. Comment describing the code can be added to the program by plac-

ing characters between parentheses following the program number (e.g. **O**0010 (WHEEL-TOP-SIDE)), as illustrated in Fig. 1.1.

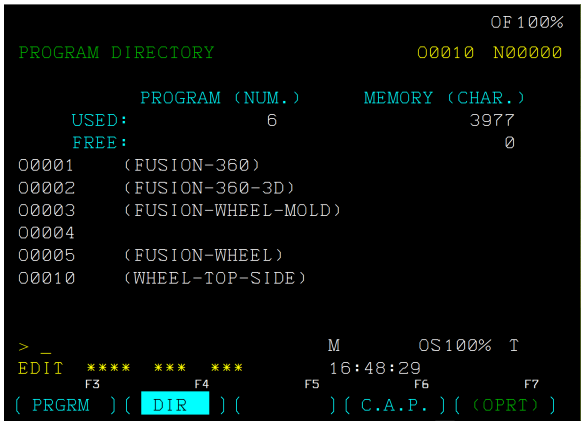


Fig. 1.1: WinNC program directory where programs are stored with a unique numeric code (Program number).

Sequence Number (**N**)

Sequence numbers (also known as block numbers) are optional words that are coded in the beginning of a block and are generated automatically by WinNC. The MCU execute the program block by block according to the order in which they appear disregarding what sequence number is used. However, sequence numbers are used to help CNC operators to locate specific lines of a program when editing data or when performing checkout operations. In WinNC sequence numbers range from **N**1 to **N**9999.

Example of a program number and sequence numbers

Program number { **O**0001

Sequence numbers {
N5 G54 ;
N10 G52 X0 Y0 Z35 ;
N15 T1 H1 M6 G43 ;
N20 M30 ;

1.5 G Codes (Preparatory Functions)

G codes are preparatory functions and are designated by the address **G** followed by one or two numerical values. The numerical values following the G address specify the mode of movement for the CNC machine along its programmed axes. The preparatory term means that the word (address **G** and numerical value) prepares the controller for the information that follow in the block. A G code is coded at the beginning of a block in order to set the control of a specific mode to act on the other words in the block.

Currently, many G codes are standardized however, other machine controllers still have unique G codes for a particular CNC control. Note that there are differences between the G codes used by the CNC milling machines and those used by the lathe machines. However, some of the G codes remain the same between the two machines and Table 1.1 illustrates a few of these G codes.

Table 1.1: Some of the standardized G codes that have the same functionality in both CNC milling and lathe machines.

G code	Mode	Function
G0	Model	Rapid positioning
G1	Model	Linear interpolation at the programmed feed and speed
G4	Non-Model	Dwell
G90	Model	Absolute programming
G91	Model	Incremental programming

The G codes fall into two major categories model and non-model:

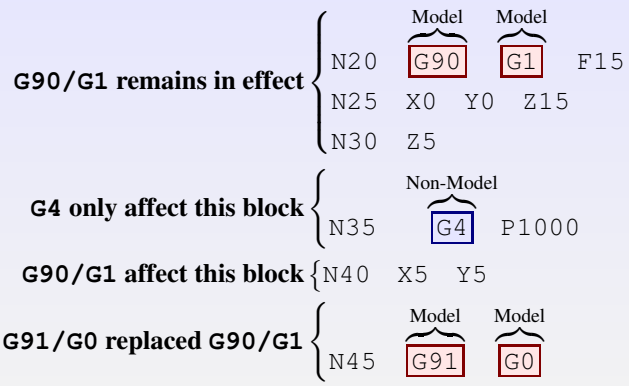
G code categories

Model:
Remain in effect for all blocks unless replaced by another modal G code.

Non-Model:
Only affect the block in which it appears.

The model G code remain in effect for the block in which it appears and the subsequent blocks until another model G code is coded. On the contrary, the non-model G code only affect the block in which it appears and not the subsequent blocks. The following example illustrates the differences between the model and non-model G codes:

Model and non-model G codes



In the above example note that the two model G codes (G90 and G1) remains in effect for the subsequent blocks until being replaced by G91 and G0 model codes. Observe that in block number N35 the non-model G code (G4) only affect this block and has no effect on the subsequent blocks.

1.6 Dimension Words

The movement of machine axes are specified by dimension words. The programming axes are based according to the Cartesian coordinates system as shown in Fig. 1.2. The direction of movement along each axis is governed by the right-hand rule.

Table 1.2: Dimension words and the type of information they store.

Address	Type of information stored
X,Y,Z	Linear axes and X also used for dwell
U,V,W	Axes which are parallel to X,Y and Z axes
A,B,C	Rotation axes
I,J,K	Axes used as supplementary of the X,Y and Z axes
R,Q	Axes used as supplementary of the Z axes, R also used for radius

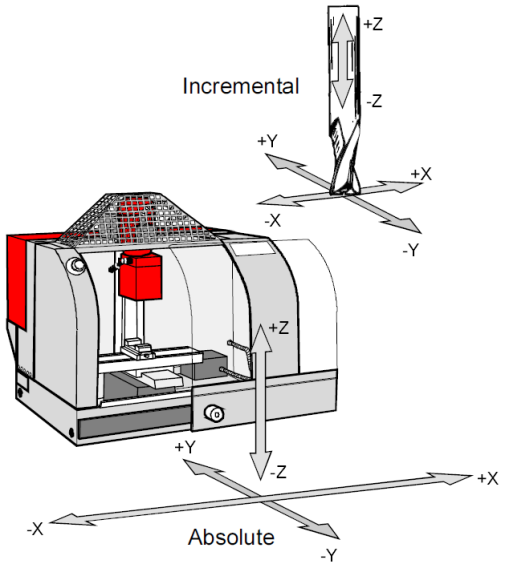


Fig. 1.2: The coordinate system of a milling machine where the axes are laid out according to the Cartesian coordinate system.

1.7 Feed Rate (The `F` Code)

The feed rate is defined as the rate at which the cutting tool is moved along the programmed axes into the part. The amount of feed rate is specified as a numerical value following the address `F` (e.g. `F150`). In the metric system, the feed rate is expressed in millimeters per minute ($^{mm}/_{min}$), and in the English system in inches per minute ($^{in}/_{min}$). Note, when programming with `G94` word the `F` address specifies the feed rate in $^{mm}/_{min}$ and when using `G95` word the feed rate is expressed in $^{rev}/_{min}$ (rpm), this is summarized in Table 1.4.

The `F` code is model and remains in effect in the program for all the tool movements in the subsequent blocks until being replaced by another `F` code.

Table 1.3: Dimension words and the type of information they store.

Block	Meaning
`G94 F150`	`G94` sets the type of feed rate to $^{mm}/_{min}$ `F150` specifies a feed rate of 150
`G95 F100`	`G95` sets the type of feed rate to $^{rev}/_{min}$ `F100` specifies a feed rate of 100

1.8 Spindle Speed (The `S` Code)

The `S` address is used to specify the spindle speed in revolutions per minute (rpm) when programming in milling and lathe machines. However, in lathe machine when programming with `G96` word the `S` address specifies the spindle speed in meters per minute ($^{m}/_{min}$) and the control continually computes the spindle speed corresponding to the current diameter to achieve a constant cutting speed.

Table 1.4: Dimension words and the type of information they store.

Block	Meaning
`G97 S3000`	`G97` sets the type of spindle speed to $^{rev}/_{min}$ (rpm) `S3000` specifies a spindle speed of 3000
`G96 S180`	`G96` sets the spindle speed to constant cutting speed in $^{m}/_{min}$ `S180` specifies a constant cutting speed of 180

1.9 M Codes (Miscellaneous Functions)

1.10 Tool Length Offset and Cutter Radius Compensation (H Codes)

1.11 Summary

1.12 Chapter Project