Module 5: NC Programming

Goal

Upon completion of Module 5, the student will have an understanding of numerical control programming.

Performance Objectives

Upon completion of Module 5, the student will be able to recognize NC words and their component address codes and parameters, and detect errors in NC code blocks using the programming guidelines provided in this module.

Materials Required

None.

Activities

After reading the material contained in Module 5, become familiar with NC words and their component address codes and parameters, and the programming guidelines. Answer the questions at the end of the module. The Performance Evaluation is based on the answers to the questions.
Introduction

Before a part is machined, every step in the machining process must be described in a **part program**. The program, or list of instructions, is written in a language that can be understood by many CNC machine tools. The language is called the **EIA RS-274D standard format**, or G&M code programming.

In this module, you will learn how to write NC words using EIA RS-274D codes, and how to combine the codes to form an NC part program.

![six individual NC Words](image)

* A Typical Block of NC Code
Elements of a Part Program

Part programs use two types of instructions: *those that define the tool path* (such as X and Y axis coordinates), and *those that specify machine operations* (such as turning the spindle on or off).

An NC part program is made up of **blocks** (lines) of **code**. Each code block contains a string of **words**. An NC word is made up of an alphabetic character (the **address character**) and a number (the **parameter**). For example, the coordinate X2 has the address character “X” and the parameter “2.”

The address characters you will use for your NC part programs are N, G, X, Y, Z, I, J, K, F, S, T, M, and the semi-colon (;).

**N** words are the address characters used to number the blocks of code in your program. For instance, the first block in a program begins with the word N0. The next block begins with N1. The next blocks are N2, N3, N4 and so on. The N word is always the first word in each block.

**G** words are **preparatory codes**. Preparatory codes tell the mill what kind of cut to make and which programming mode to use. G words are always given before any motions are made.

**X** words tell the tool **where** to move to (absolute) or **how far** to move (incremental) on the X axis.

**Y** words tell the tool **where** to move to (absolute) or **how far** to move (incremental) on the Y axis.

**Z** words tell the tool **where** to move to (absolute) or **how far** to move (incremental) on the Z axis.
I, J and K words tell the tool the X, Y and Z coordinates of the center point of an arc (for circular interpolation). In absolute programming, I words represent the X coordinate, J words represent the Y coordinate and the K words represent the Z coordinate. In incremental programming, the I, J and K words specify the distance from the start point to the center point of the arc.

F words tell the tool what feed rate to use to move the tool into the workpiece. The parameter represents the rate of speed in inches per minute. For instance, the word F2 means to move the tool into the workpiece at a rate of two inches per minute.

S words specify spindle rotation. The parameter defines the speed at which the spindle rotates in revolutions per minute (RPM).

T words define the tool, if more than one tool is used. The parameter in the tool word is the number assigned to the tool. For instance, tool #1 (T1) can be a right-hand tool, while tool #2 (T2) is a left-hand tool.

M words represent miscellaneous machine operations. These operations vary from turning the spindle on and off (M03 and M05), to repeating the NC program (M47).

A semi-colon (;), or comment code, allows you to attach comments, or reminders, to each block of code about the instructions each block contains. Any characters that appear after a comment code are not acted upon by the mill.

Example: N5G90G01X2Y3F2; GO TO START POINT AT 2 IPM

Any text following the semi-colon is a comment, and has to be on the same line as the semicolon.
Some NC words are used more often in part programs than others. Here are some of the more common NC words you should know. The G codes are broken into several groups.

**G CODES**

**G00** Rapid Traverse: move rapidly to the indicated position. Rapid traverse motion does not necessarily mean the tool moves in a straight line.

**Interpolation Group**

**G01** Linear Interpolation: move in straight lines.

**G02** Circular Interpolation, Clockwise: move the tool in an arc in a clockwise direction.

**G03** Circular Interpolation, Counterclockwise: move the tool in an arc in a counterclockwise direction.

**Wait Group**

**G04** Dwell (wait): a timed pause equalling the feed rate value in seconds.

**G05** Pause: pause until the Return key is pressed. You can also use M00.

**Programming Mode Group**

**G90** Absolute Coordinates: move the tool to each place in the program defined by X, Y and Z.

**G91** Incremental Coordinates: move the tool the distance defined by X, Y and Z in the program.

**M CODES**

**M00** Pause: pause until the Return key is pressed. You can also use G05.

**M02** End of Program: this is the last instruction in the program.

**M03** Spindle On: turns on spindle motor.

**M05** Spindle Off: turns off spindle motor.

**M06** Tool Change: pauses all operations, turns off spindle, retracts spindle for tool change.

**M47** Rewind: repeats the program. If rewind is used, you must place a G05 or M00 pause at the beginning of the program.
New Terms

Address Character - An alphabetic character representing a command in an NC word

ELA RS-274D Standard Format - A standard for NC programming established in 1979 by the Electronics Industries Association, also called G&M code programming

Block - A word or group of words making up one line of an NC program

Code - In NC programming, the same as a word, typically used for G and M words, also a general term for a set of programming instructions

Parameter - Numbers that describe movement or distance, and are used with an address character command in an NC word

Part Program - A list of numerically coded instructions

Preparatory Code - NC words (G address character plus a parameter) that tell the machine tool what type of interpolation and programming mode to use

Word - The combination of an address character (the command) and a parameter (the distance or movement)
Programming Suggestions

There are nine basic guidelines to follow when writing NC programs.

Guidelines

1) The sequence of words (address characters plus parameters) in an NC block must appear in the following order when more than one address character is used per block: N, G, X, Y, Z, I, J, K, F, S, T, M, ;

2) Only one of each address character can be used per block, except some G codes. For instance, you can place one G code from the Interpolation Group and one G code from the Programming Mode Group in the same block. However, you can not place two codes from one group in the same block.

Example:
The block, N1G01G90X2X2, is incorrect because it has two X codes. The use of the G codes, however, is correct because one G code is from the Interpolation Group and the other is from the Programming Mode Group.

The block, N1G00G01X2Y2, is incorrect because it uses two G codes from the same group.

3) Some codes don't have to be repeated in every block. The mill will use the same value until you change it.

Example:
N3G01X.5Y2F1
N4Y3.5

In block N3 the tool is told to move in a straight line (G01) to X.5,Y2 at a feed rate of 1 inch per minute. In block N4 the tool is told to move to 3.5 on the Y axis, but the feed rate, linear interpolation (G01), and X value remain the same as in block N3. It's not necessary to give the mill that information again.
4) The first block in a program should move the tool away from the workpiece to a safe point (where the tool is not touching the workpiece) to turn on the spindle.

Example:
N0G00G90Z.1  
N1M3

5) Don’t forget; the second block in a program should turn on the spindle.

Example:
N1M3

6) The third instruction in a program, after the spindle is turned on, should move the tool to the start point, the point where machining begins.

Example:
N2G00G90X.5Y.5

Block N2 tells the tool to move in rapid traverse to absolute coordinates X.5 and Y.5.

7) The end of a part program should move the tool back to the start point so the tool will be ready to cut the next part.

Example:
N5G0Z.1  
N6X.5Y.5M5
N7M2

If these were the last three blocks in the program, the tool would move back to the start point X.5,Y.5,Z.1, the spindle would be turned off, and the program would end.
8) The maximum number of characters allowed in one block is 66. In order to save space, you can leave out extra zeros as long as they have no affect on the value.

For instance you can write N6G00X0.500Y1.500 as N6G0X.5Y1.5 saving six characters.

But, you can not write N2G90Z0.05M00 as N2G9Z.5M because there's no such thing as a G9, the Z value has changed from .05 to .5, and the M address character has been left without a parameter.

9) If you are just starting to program, and you're not sure about how to combine codes in a block so the correct action is performed first, put each G and M code in a separate block.

Example:

N7G0X.5Y2M5 can be written as

N7G0X.5Y2
N8M5

If you want to make sure the spindle will turn off after you move to the X and Y positions in this block, you should put the M5 in the next block.