INFORMATION 4

UNDERSTANDING NUMERICAL CONTROL PROGRAMMING

Numerical control (NC) programming consists of a sequence of instructions written in a way that can read and run by a CNC machine. Programs generally use 2 separate types of instructions: those which tell the cutting tool where to go (such as x and z coordinates) and those which tell the machine specific things to do such as turn the power on or off. Below is an example of 1 line or block of information. Fig. 4-1.

N0G90X.5Z1.5F1

The individual words in this block are translated as follows:

N0 This is the block sequence number for the program. Block 0 is the first block in the program.

G90 Indicates absolute coordinates are used to define tool position

X.5 Specifies the X-axis position as 0.5 inch.

Z1.5 Specifies the Z-axis position as 1.5 inch. The cutting tool will move to the absolute coordinate position (0.5,1.5).

F1 Specifies a feed rate of 1 inch per minute at which the tool will advance to the specified coordinate points.

Your NC hard copy program has a lot of numbers and letters that at this time don't make sense to you. By reading the following information and doing the worksheet, you will have a simple understanding of how to read your program and what the CNC lathe is told to do.

In figures 4-2, 4-3 and 4-4 you will see the list of G and M codes used that are the standards for industry. PLEASE READ CHARTS CAREFULLY AS THEY CONTAIN INFORMATION TO BE USED ON THE FOLLOWING WORKSHEETS.

Table 4-2 spectraLIGHT NC Address Characters

N	Block number (for user reference only)
G	Preparatory codes
T	Tool select (maximum of four tools)
X	Primary X motion dimension
Z	Primary Z motion dimension
I	Arc center, X-axis dimension (circular interpolation)
K	Arc center, Z-axis dimension (circular interpolation)
F	Feed rate in inches per minute or dwell time in seconds
M	Miscellaneous functions

Table 4-3 spectraLIGHT G Codes

Interpolation Group

G00	Rapid traverse
G01	Linear interpolation
G02	Circular interpolation (clockwise)
G03	Circular interpolation (counterclockwise)

Wait Group

G04	Dwell (wait) = value of the feed rate (F code) in seconds [used primary for robotic operations]; see Notes 1 and 2 below
G05	Pause for operator intervention - press RETURN to resume machining operation; see Note 2 below

Programming Mode Group

G90	Absolute coordinate programming - all X and Z axes coordinates are relative to a (0,0) location on a lathe
G91	Incremental coordinate programming - each command is relative to the one before it in the program

Table 4-4 spectraLIGHT Miscellaneous Codes

M02 End of Program: takes effect after all motion has stopped; turns off stepper motors, spindle and accessory outlets.

M04 Spindle Motor Counterclockwise: activated concurrently with motion specified in the program block; remains in efect until superceded by another M code.

M05 Spindle Off: activated after the motion specified in the program block; remains in effect until superceded by another M code.

M08 Coolant On: turns ON AC outlet normally assigned to coolant concurrently with the motion specified in the program block; remains in affect until superceded by M09 - Coolant Off.

M09 Coolant Off: turns OFF AC outlet normally assigned to coolant after the motion specified in the program block; remains in affect until superceded by M08 - Coolant On.

M10 Clamp Air Chuck: closes air chuck accessory concurrently with the motion specified in the program block; remains in affect until superceded by M11 - Unclamp Air Chuck.

M11 Unclamp Air Chuck: opens air chuck accessory after the motion specified in the program block; remains in affect until superceded by M10 - Clamp Air Chuck.

From the above charts you will now be able to look at your hard copy and pick up information on what is happening in the program.

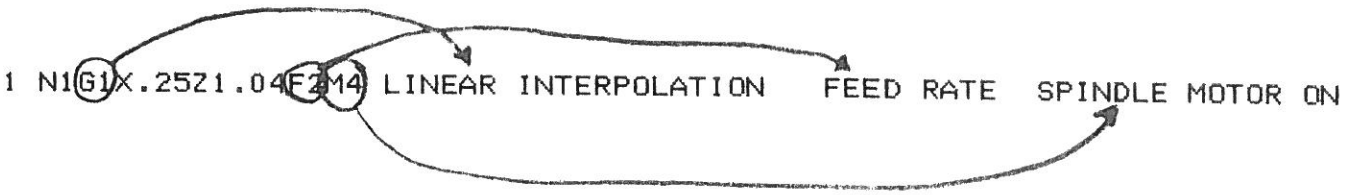
WORKSHEET 3 (10 PTS)

DIRECTIONS: TAKE HARD COPY AND

1. Neatly circle all lettered characters and numbers next to the characters other than the N, X, and Z.

2. Using previous information with charts, tell what is happening at that point. If a line has more than 1 bit of information, draw a line to the definition area.

USE EXAMPLE BELOW FOR REFERENCE.



CHECK POINT 5