Hamilton-Senior lab-off-site learning packet day 4

Instructor   Mark Hamilton   **Date**_____

Program/Class_____ AEM SR _____

Instructional Objective(s):

Materials:
White board

Method of Instruction:
Research

Activities:
Read through handouts, watch videos and do activities
Answer the questions provided

Closure:
Answer questions on the last page

Assessment:
Answer sheets will be collected and graded.

Objectives:
1. Students will list 6 rules of program format with 80% accuracy
2. Students will list the first 7 lines of a CNC mill program with 80% accuracy

Programmers will have their own style of writing a program, even though the codes are the same often times people put the codes in different orders in the blocks or even the blocks in different orders or sometimes their operations in different orders. So what should you do as a beginner? Keep an open mind and ask other operators why they program the way they do you may be surprised at the answers you get.

Objective 1: the programming format
- There is no positional requirement for the address codes. They may be placed in any order within the block.
- Each individual can format their programs many different ways, programming format and programming style are very important parts of programming, but there are some programming formats that can be moved around and some need to stay a certain way and there are some program standards that are just good to follow.
- The point is that a programmer needs to have an organized program format that is consistent and efficient so that any CNC machinist in your shop can understand it.
- Some of those standard program rules to consider are as follows
    1. Program X, Y, and Z in alphabetical order, if more than one axis is programmed on a line they should be listed together and in order.
    2. G codes are usually put at the beginning of the line of information and M codes at the end of the line.
        - This is a good idea because the G code preps the machine for an action and then the M codes turns something on before the action is started. For example coolant on M08 so that the coolant starts as the tool moves into position.
    3. Some CNC machines allow you to write more than one M code per line and some wont, on the Haas machine only one M code may be programmed per block and all M codes are activated or will cause an action to occur after everything else on the line has been executed.
    4. Program format is the sequence and series of commands in a program that a machine may accept and execute.
    5. Program format is the order of the machine code listed in the program lines that consist of command words.
    6. Command words beginning with a letter and then numbers for each word along with the presents of or absence of a decimal point and presence of or absence of a plus or minus sign.
    o Program format defines the language of the machine tool

Objective 2: the explanation of a sample programs start up lines
- First the program
    o G54 G90
    o M06 T01 (put tool information in parentheses)
    o G00 X_____ Y_____
    o M03 S_____
    o G43 H01 Z.1 M08

- The first line of code should contain an absolute (G90) command along with a work offset (G54 is the default) part zero command.
- The second line or block of a program should be a tool number (T1) and a tool change (M06) command
- The third line may contain rapid G00 command to position to an X Y coordinate location.
- The forth line of code should specify a spindle speed command (S____) along with a spindle on clockwise command (M03).
- The fifth line or block contains a read tool length compensation command (G43) and a tool length offset register number (H01). Then a Z axis positioning move (Z1.0), and an optional coolant on command (M08).

- **Another format that was commonly used with the Fanuc control was as follows**
    1. **G90 G80 G40**
    2. **G00 G49 G17**
    3. **G54 G90**
    4. **M06 T01** (1/2 inch flat end mill)
    5. **X.5 Y-1.5**
    6. **M03 S2000**
    7. **G43 H01 Z1.0 M08**

The program format above is a format that is used on older CNC milling machines.
We will break down each line one at a time.
- Lines 1 & 2 were used on older machines because these older machines would keep modal codes activated until they were cancelled. Therefor if you had been using a drill cycle for example the first move the machine would make would be a drilling cycle.
- Before running a new program it was important to cancel old commands from the previous program.
    o Explanation of each code
        - G90 puts the machine in the absolute positioning mode
        - G80 cancels all canned cycles such as drilling, boring or tapping
        - G40 cancels cutter comp that may have been activated
        - G00 puts the machine in the rapid positioning mode
        - G49 cancels previously activated tool lengths
        - G17 puts the machine in the X & Y plane selection
    o Newer CNC machines cancel all old codes with the use of the M30 (program end and reset)
- **Line 3:** G54 and G90
    o G54 calls up the work coordinates offset in which you have set your part zero location.
        - G54 – G59 are available on most machines
    o G90 puts the machine in the absolute positioning mode.
        - This means where you have set zero on your part will always be the zero point.
- **Line 4:** M06 T01
    o M06 will activate the tool changer and T01 will select tool number one from the carriage.
    o It is a good idea to put the tool size in parentheses so that when running the program in the future you know exactly which tool to use.
    o Putting parentheses around a message allows the message to be seen in the program but the machine will not try to execute the information inside the parentheses.

- **Line 5:** X.5 and Y-1.5 will rapid the tool to that position form the zero point that you have set in the G54 offset.
- **Line 6:** M03 S2000 will start the spindle in the clockwise direction (M03) at an RPM of 2000

- **Line 7:** This calls up the length of the tool that you have established and stored in the number one spot in the tool library. (G43)
  - **H01** retrieves the tool length information stored for tool number one
  - **Z1.0** will move the cutter 1.0 inches above the Z zero point you set in the G54 offset
  - **M08** turns on the coolant flood

It is vital to have lines 3 through 7 in every program you write. Missing one of the lines will cause the machine to crash.
Can you imagine starting to push a cutter through material without turning the spindle on, or sending a cutter to the top of your part without telling the machine the length of the cutter?

Crash, Boom, Smash with possible blood shed from you!!!

Don't be this Guy

Hamilton-Senior lab-off-site learning packet day 4

Senior Questions lesson 4

Name_____

1. What is meant by a program format?

2. Why is it a good practice to try to keep the program format the same with others with in a shop?

3. Why are G-codes put at the beginning and the M-codes put at the end of the line?

4. Why is there a need for the first two safe startup lines in the program?

5. G90 will put the machine in what positioning system?

6. What is the G54 telling the machine to find in the controller?

7. What does M06 activate on the machine?

8. Why should you put the size of the tool in parentheses?

9. Why does the message need to be inside parentheses?

10. What does M03 do in the program?

11. Why is there an H01 in the G43 line?

12. What does M08 turn on in the program?

13. Why is it so important to have all the information in lines 3 thru 7 in a program?