Hamilton-Juniors lab-off-site learning packet day 7

Instructor Mark Hamilton **Date**_____

Program/Class AEM JR

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 is worth ____ points

Junior lesson 4

Name_____

Introduction to CNC code

G Words

G words, commonly called G codes, are major address codes for preparatory functions which involves tool movement and material removal. These include rapid moves, lineal & circular feed moves, dwell, and canned cycles. Through the years many of the G-codes have been standardized, but others are unique to a particular controller. The commonly used G-codes are briefly summarized below:

There are many other special feature and advanced machining G-codes that aren't commonly used, and therefore not discussed in this basic module. See the "G" glossary listing for explanations of all of the G-codes

Note: always consult the operator's manual for the G-code definitions for your specific controller.

G-codes for Milling (includes drilling)

Rapid positioning (G00)
Interpolation (G01, G02, G03)
Dwell (G04)
Plane selection (G17, G18, G19)
Automatic Reference Returns (G28, G29)
Cutter compensation (G40, G41, G42)
Tool offset (G43-G49)
Work Coordinate Offset (G52-G59)
Unit input (G20, G21; or G70, G71)
Fixed canned cycles (G80-G89)
Positioning Input (G90, G91)
Set Work Coordinates (G92)
Return Points (G98, G99)

G-codes for Lathes

Rapid positioning (00)
Interpolation (G01, G02, G03)
Dwell (G04)
ZX plane selection (G18)
Unit input (G20, G21)
Automatic Reference Return (G28, G29)
Tool Nose Radius Compensation (G40, G41, G42)
Work coordinate offset (G52-G59)
Fixed cycles:
Finishing Cycle (G70)
Turning Cycle (G71)
Facing Cycle (G72)
Peck-drilling (G74)
Grooving Cycle (G75)
Threading Cycles (G76, G92)
O.D./I.D. turning Cycle (G90)
Constant Surface Speed (G96, G97)
Free Rate unit (G98, G99)

Hamilton-Juniors lab-off-site learning packet day 7

Questions: (each question is worth 5 points)

1. What are G words commonly called?

2. What are G codes used for?

3. Do all the G codes work on every machine?

4. What should you always consult before putting G codes into a machine?

5. What does a G00 command a machine to do?

6. What does a G01 command a machine to do?

7. What does G40, G41 & G42 control on a machine?

8. What does a G43 control on a machine?

9. What does a G54 control on a machine?

10. What does a G71 control on a CNC lathe?

11. What does a G70 control on a CNC lathe?

12. What does a G76 control on a CNC lathe?