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

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 steps of using the G41 cutter compensation left with 80% accuracy

Compensating for the size of the tool you are using is part of the math that the machinist must perform when operating a manual machine. But when using a CNC machine that has the ability to cut arcs and circles that math can become much more complicated.
I don't know about you but I am not a big fan of complicated.
I like easy!! Complicated, not so much.

Well luckily there is a code in CNC programming that does all that math for you.
Now that is some news I like to hear!!

This amazing code is called cutter compensation.

There are 3 codes used in cutter comp.
G41, G42 & G40

Objective 1: 6 steps of using the G41 cutter compensation left.
- The cutter comp commands are cutter comp left (G41) cutter comp right (G42) and cutter comp cancel (G40)
1. **G41** will select cutter compensation **left**, that is the tool is moved to the left of the programmed path to compensate for the radius of the tool
2. The programmed path will be to the left of the **direction** that the tool is traveling.
3. For example if the tool is **climb milling** in the Y direction in the positive direction the tool would be to the left of the programmed line. (**See example**)
4. The G41
5. Tool direction
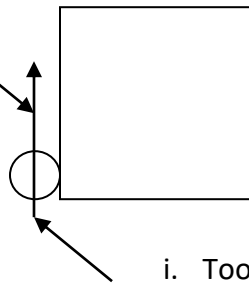
        i. Tool offset to the left of the part

NOTE: G41 will always be used when you are climb milling

6. The G41 would appear in the program as follows.
    a. G01 G41 D01 Y2.0 F5.0 ;
    b. The G01 is linear interpolation
    c. The G41 is cutter comp left
    d. The D01 picks up the tool info you put into the machine
        i. That information you put in the machine library will be the tool diameter or the tool radius depending on the machine.
    e. Y2.0 is the tool destination
    f. F5.0 is the feed rate of 5" per minute

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

Senior lesson

Name_____

1. How many codes are available when using cutter compensation?

2. What are the codes that deal with cutter comp?

3. What is the CNC machine compensating for when you use cutter comp?

4. What side of the programmed path is the G41?

5. What type of milling are you performing when using the G41?

6. What does the G01 do in the program?

7. What information about the tool do you need to put in the tool library when using cutter comp?

8. What does the D01 do in the program?

9. Why do you think you use D01 when using tool one?

10. What D number would you use if you were cutting with tool 5?