



AMART

Flexible Instruction Day 3

Assignment #3

Learning Standard	Framework (pa.gov) Pick learning standards from the crosswalk
Task # & Description	1402 Demonstrate the basic use of G and M codes.

CONTACT INFORMATION/ASSISTANCE

Instructor Name: David Hockenberry

Instructor E-Mail: hockenberryd@butlertec.us

I will be available by email from 7:40-12, and 12:30-3:10 to answer your questions. Please reach out if you are having difficulty completing your assignments.

INSTRUCTIONS

Complete all activities independently using your classroom knowledge. No internet is required.

Name: _____

Session: _____

____/____ points

G-Code and M-Code Match Assignment

AMART – Advanced Manufacturing and Robotics Technology

Directions

Match each G-code or M-code with its correct function. Complete all sections independently using your classroom knowledge.

Common G-Codes Reference Chart

G-Code	Function
G00	Rapid Positioning
G01	Linear Feed Move
G02	Clockwise Arc
G03	Counterclockwise Arc
G04	Dwell
G17	XY Plane Selection
G18	ZX Plane Selection
G19	YZ Plane Selection
G20	Inch Programming
G21	Metric Programming
G28	Return to Home
G40	Cutter Compensation Cancel
G41	Cutter Compensation Left
G42	Cutter Compensation Right
G43	Tool Length Compensation
G54	Work Offset 1
G55	Work Offset 2
G80	Canned Cycle Cancel

G-Code	Function
G81	Drill Cycle
G83	Peck Drill Cycle
G90	Absolute Programming
G91	Incremental Programming

Common M-Codes Reference Chart

M-Code	Function
M00	Program Stop
M01	Optional Stop
M03	Spindle Clockwise
M04	Spindle Counterclockwise
M05	Spindle Stop
M06	Tool Change
M08	Coolant On
M09	Coolant Off
M19	Spindle Orient
M30	Program End and Rewind
M97	Local Subprogram Call
M98	Subprogram Call
M99	Return from Subprogram

Part 1 – G-Code Identification

Match the G-code with the correct description.

G-Code Letter

G00 _____

G-Code Letter

G01 _____

G02 _____

G03 _____

G20 _____

G21 _____

G28 _____

G40 _____

G43 _____

G54 _____

Descriptions

- A. Cutter compensation cancel**
- B. Rapid positioning move**
- C. Clockwise circular interpolation**
- D. Linear feed move**
- E. Tool length compensation positive**
- F. Counterclockwise circular interpolation**
- G. Return to machine home position**
- H. Work offset coordinate system**
- I. Programming in inches**
- J. Programming in millimeters**

Part 2 – M-Code Identification

Match the M-code with the correct description.

M-Code Letter

M00 _____

M01 _____

M03 _____

M-Code Letter

M04 _____

M05 _____

M06 _____

M08 _____

M09 _____

M30 _____

M98 _____

Descriptions

- A. Spindle stop**
 - B. Program end and rewind**
 - C. Coolant on**
 - D. Optional stop**
 - E. Tool change**
 - F. Subprogram call**
 - G. Program stop**
 - H. Spindle clockwise rotation**
 - I. Coolant off**
 - J. Spindle counterclockwise rotation**
-

Part 3 – Multiple Choice

Circle the best answer.

1. Which G-code is commonly used for rapid movement?

- A. G01**
 - B. G00**
 - C. G03**
 - D. G54**
-

2. Which code turns coolant on?

- A. M05**
- B. M30**

C. M08

D. G20

3. Which G-code is used for a straight cutting move?

A. G01

B. G02

C. G03

D. G28

4. Which M-code stops the spindle?

A. M03

B. M04

C. M05

D. M08

5. Which G-code commands a clockwise arc?

A. G00

B. G01

C. G02

D. G03

6. Which code is used to call a tool change?

A. M06

B. M09

C. M98

D. G43

7. G20 programs the machine in:

A. Millimeters

B. Inches

C. Degrees

D. RPM

8. Which M-code ends and rewinds the program?

- A. M01
 - B. M03
 - C. M30
 - D. M98
-

9. Which G-code is commonly used to return to machine home?

- A. G54
 - B. G28
 - C. G43
 - D. G40
-

10. Which code turns coolant off?

- A. M08
 - B. M09
 - C. M05
 - D. M00
-

Part 4 – True or False

Write TRUE or FALSE.

- _____ G-codes control machine movements.
 - _____ M-codes control machine functions.
 - _____ G00 is normally used for rapid positioning.
 - _____ M03 starts the spindle clockwise.
 - _____ G01 is used for controlled feed moves.
 - _____ M08 turns coolant on.
 - _____ G54 is a work offset.
 - _____ M30 ends the CNC program.
-