MTTC 107 CERTIFICATION REVIEW TEST

(MILL AND LATHE OPERATOR)

1. Given an 18 ¾” length of stock, how many 2 ½” pieces can be cut from it?
   a. 7.5
   b. 7
   c. 8
   d. 1

2. How often should you check the lubrication levels on a machine?
   a. Daily
   b. yearly
   c. never
   d. only on Fridays

3. On a blueprint, what do you call a dimension in a box (boxed dimension)?
   a. Base
   b. Bain
   c. Basic
   d. basal

4. What type of tool is pictured below?
   a. Cutoff tool
   b. Threading tool
   c. Grooving tool
   d. Boring tool

5. What type of tool is pictured below?
a. Turning
b. Grooving
c. Boring
d. Back boring

6. When should you wear your safety glasses in the shop?
   a. Only when your shop supervisor is around
   b. When you have time
   c. Always
   d. Only when you are running a machine

7. What is the RPM formula for drilling?
   a. 3.82 X tool diameter/SFM
   b. 3.1417 X RPM/IPM
   c. IPR X CFM
   d. 3.82 X SFM/Tool diameter

8. What is the code for rapid traverse?
   a. G01
   b. G00
   c. G95
   d. G11

9. What is the code for linear feed (must have an F word with it)?
   a. G01
   b. G02
   c. G10
   d. G03

10. What is the code for program stop?
    a. G00
    b. M01
    c. M00
    d. M06

11. What is the code for optional stop?
    a. M00
    b. M02
    c. M03
    d. M01

12. What is the code for program end rewind?
    a. G30
    b. H30
    c. D30
    d. M30

13. What is the code for zero return?
    a. G38
    b. G05
14. What is the code for constant surface speed (SFM)?
   a. G96
   b. G97
   c. G94
   d. G95

15. What is the code for fixture offset (work offset #1)?
   a. G55
   b. G54
   c. G56
   d. G50

16. What is the code for incremental positioning?
   a. G20
   b. G21
   c. G22
   d. G91

17. What is the code for absolute positioning?
   a. G91
   b. G20
   c. G21
   d. G90

18. What is the code for inch mode?
   a. G54
   b. G21
   c. G19
   d. G20

19. What is the code for metric mode?
   a. G20
   b. G22
   c. G98
   d. G21

20. What is the code for tool change ON THE MILL?
   a. G06
   b. M08
   c. M06
   d. M19

21. What is the code for spindle activation clockwise OR FORWARD?
   a. M01
   b. M02
   c. M4
   d. M3
22. What mode would you select for changing a tool on a mill or lathe?
   a. MOI
   b. MEM
   c. HAND JOG
   d. MANUAL
   e. MDI

23. What is the difference between the 1.384 dimension and the .945 dimension?
   a. One is diameter
   b. One is Basic
   c. One is metric
   d. One is inch

24. What is the measurement reading on the 0-1” micrometer below?
   a. 1.049”
   b. 0.109”
   c. 0.149”
   d. 0.151”
25. What is the reading on the 1-2 inch micrometer below?

![Micrometer Image]

a. 0.413”
b. 1.488”
c. 1.413”
d. 1.422”

26. What is the reading on the 0-1” depth micrometer below?

![Depth Micrometer Image]

a. 1.660”
b. 0.535”
c. 0.520”
d. 0.510”

27. What screen gives you the information for height offset value?

a. Offense
b. Offsell
c. Offer
d. offset

28. What mode on a CNC milling center would you select to send the machine home?

a. ZRN
b. Zero Return
c. MDI
d. Manual
e. All of the above
29. If you were boring a hole specified at 2.125” deep with a tolerance of +/- .001 and measured the part at 2.122” deep, what would you add or subtract from the Tool length offset to bore the hole to proper depth?
   a. +.003
   b. -.003
   c. -.002
   d. +.002

30. A print calls out a finished part length of 3.25 inches. How long would you cut the stock for each part if you leave 1/8” for sizing?
   a. 2.375”
   b. 3.75”
   c. 3.875”
   d. 3.375”

31. Which of the buttons below would you depress if you were using M01 in a program?
   a. Orient Spindle
   b. Coolant
   c. Block delete
   d. Option Stop
32. What button below would you push to manually input and physically execute programming commands?

- a. MDI/DNC
- b. Hand Jog
- c. Zero Set
- d. Edit

33. What button would you press to view the distance to go while running a program?

- a. Alarm Mesgs
- b. Param DGNOS
- c. Setng GRAPH
- d. Posit
34. The tolerance range is set on a dial indicator. If this dial indicator is checking the diameter of a part turned on a lathe, what would you add or subtract from the geometry offset (X) to machine the part at the mean dimension?

a. +.015  
b. -.015  
c. +.085  
d. -.085

35. A part dimension states that the depth of a bored hole must be within .001” (+/- .0005). When machining the first part how should the TLO (tool length offset) be set?

a. Increase the length by ten or twenty thousandths  
b. Decrease the length by ten or twenty thousandths  
c. Set it right on the money  
d. Let the programmer set it up
36. You are milling a part using cutter diameter compensation. The part measures 2.004” after the machining is done (using the .250” to 2.250 dimensions below). What must you do to the cutter diameter value to cut the part to size?

- Add .004”
- Subtract .004”
- Add .001”
- Subtract .001”

37. What does this symbol mean?

- Keyway
- Slot
- Profile of a line
- Profile of a surface

38. What does this symbol mean?

- Pitchfork
- Runout
- Total runout
- Total burnout

39. What does this symbol mean?

- Squareness
- Flatness
- Paradigm
- Parallel
- None of the above
40. What does this symbol mean?
   a. Corner
   b. Squareness
   c. Flatness
   d. Perpendicularity
   e. None of the above

41. For the tool shown (in the turret positioned as shown for a conventional slant bed turret lathe), which would require an M04 code for spindle activation?
   a. The one on the right
   b. The one on the left
   c. neither
42. Standing in front of the mill, which is the axis direction to choose to move the tool to the right in the diagram below? (Mark the appropriate box in the diagram with an X)

![Diagram of a milling machine](image)

43. What type of dimension is this?
   a. Inch
   b. Metric
   c. Incremental
   d. Basic
   e. None of the above

44. What considerations should you take into account when adding soluble oil to a CNC machine?
   a. Cost
   b. Smell
   c. Concentration
   d. Temperature
   e. None of the above

45. You are turning a shaft and it measures 1.752” on the diameter, and the print calls out a 1.750” +/- .0005 diameter. What must you add or subtract in the geometry offset (X value) to make the part cut to the dimension?
   a. Subtract .0002”
   b. Subtract .02”
   c. Subtract .002”
46. On a standard CNC lathe, what is the axis that controls the diameter?
   a. Z  
   b. C  
   c. Y  
   d. X

47. On a standard CNC lathe, what is the axis that controls length?
   a. Z  
   b. C  
   c. Y  
   d. X

48. Why are machines homed (or sent to “zero return” position)?
   a. To give you time to finish your coffee when you get to work in the morning  
   b. So the machine has a frame (or point) of reference  
   c. So the machine can warm up  
   d. None of the above

49. Standing in front of a horizontal machining center, what axis moves toward and away from you?
   a. Z  
   b. C  
   c. Y  
   d. X

50. While standing in front of a horizontal machining center, what is the axis that goes up and down (toward the ceiling and floor)?
   a. Z  
   b. C  
   c. Y  
   d. X
51. What is the name of the measuring instrument pictured below?
   a. Telescope
   b. Snap ring
   c. Fiddler handle
   d. Telescoping gage
   e. None of the above