

MONTHLY TEST

Q. ID: ITISKILL9497CK

March 2026

Govt ITI MULABAGAL

Question Paper

Duration: 30 Mins

Total Marks: 25

ID: ITISKILL9497CK

Student Name: _____ Roll No: _____

1. Which mode is used for five positioning (Micron level) of the axis movement?

- A) Incremental Jog mode B) Edit mode
C) Auto mode D) MDI mode

2. How the complex cutting motion involving X,Y and Z moments is carried out by CNC machining?

- A) Skill of machine operator B) Combinations of co ordinates obtained from drives along the axis
C) Use of jigs, cams, templates and tracers D) Based on electrical power supply of the machine

3. Which M code is used to activate coolant pump in CNC machine?

- A) M09 B) M01
C) M06 D) M08

4. What is the purpose of MDI mode?

- A) For program execution B) Move the axes by manually
C) Cross checking of the program graphically D) Editing of programmable Data

5. Which is the operation in point to point system used?

- A) Drilling B) Facing
C) Grooving D) Parting

6. The CNC keyboards have with following keys?

- A) Alphanumerical control keys electronic B) Numerical Data based keys only
C) Alphabetic control keys D) CNC codes

7. In Which mode we can input the program command manually?

- A) Jog mode B) Edit mode
C) Auto mode D) MDI mode

8. Which mode is used for moving the turret in X and Z direction?

- A) MDI mode B) Auto mode
C) Jog mode D) Edit mode

9. What is the purpose of G28 code based on FANUC CNC control system?

- A) Tool has to return from reference point B) Circular interpolation clockwise
C) Circular interpolation counter clockwise D) Tool has return to reference point

10. How many ways CNC machine tool systems can be classified?

- A) 2 B) 5
C) 4 D) 3

11. What is the use of CNC code of M05 to develop the command?

- A) The cutting tool has to be changed B) The spindle rotation has to stop
C) Tool returns from reference point to work place D) Rewind the program once again

12. Which of the following feed back devices can sense both speed and position?

- A) Motor B) Tachometer
C) Resolver D) Encoder

13. What do you mean by simulation?

- A) Cross checking of programmable DATA B) Finding out of work offset values
C) Editing of related part program D) Finding out of tool geometrical values

14. Which one of the following is possible in edit mode?

- A) Can read only the program B) Preparation of new programmable Data and correction / Alteration of an earlier program
C) Simulation of the program D) Execute the program

15. Which one of the following code is suitable for spindle rotation in clock wise direction?

- A) M04 B) G03
C) G02 D) M03

16. Which one of the following M-code is suitable for spindle

rotation stop?

- A) M04
- B) M06
- C) M05
- D) M03

17. Encoder is used in CNC machine tool to sense and control?

- A) Work position
- B) Table position
- C) Spindle speed
- D) Spindle position

18. Which one of the following code used to switch off the coolant pump?

- A) M08
- B) M09
- C) M30
- D) M07

19. What is the indication of 'x' axis in CNC lathe machine?

- A) The diameter of the component
- B) The length of the component
- C) The radius of the component
- D) Centre axis of the component

20. Which one of the following code is used for tool nose compensation at left side?

- A) G41
- B) G40
- C) G32
- D) G42

21. What do you mean by MPG?

- A) Manually Pulse Generator
- B) Manual Procedure to Calculate Geometrical Values
- C) Manually Procure the Digital Graphics
- D) Manual Program Graphics

22. How the part programming mistakes can be avoided?

- A) Cannot be avoided in CNC machine
- B) CNC (Computer numerical control) machine only
- C) Both NC and CNC machine
- D) NC (Numerical control) machine only

23. Which G-code is suitable for thread cutting?

- A) G34
- B) G01
- C) G30
- D) G32

24. Which one of the following command the G02 code indicates?

- A) Cross sectional interpolation
- B) Circular interpolation clockwise direction
- C) Circular interpolation counter clockwise direction
- D) Linear interpolation

25. Which one of the program block may suitable to move the cutting tool 'x' axis 42.0 'z' axis 0.0 with the feed of 0.1 mm?

- A) G00 x 42.0 z 0.0 F 0.1;
- B) G01 x 42.0 z 0.0 F 0.1;
- C) G03 x 42.0 z 0.0 F 0.1;
- D) G02 x 42.0 z 0.0 F 0.1;