

MONTHLY TEST FOR THE MONTH

Q. ID: ITISKILL4894CN

February 2026

Govt ITI MULABAGAL

Question Paper

Duration: 60 Mins

Total Marks: 50

ID: ITISKILL4894CN

Student Name: _____ Roll No: _____

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

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

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

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

3. Which code is suitable for tool nose radius compensation at right side?

- A) G40
- B) G50
- C) G42
- D) G41

4. Which one of the following M-code is suitable for coolant pump ON?

- A) M11
- B) M08
- C) M09
- D) M10

5. Which point in any CNC machine the position measuring systems has a starting point to each axis?

- A) Tool geometrical value
- B) Work zero
- C) Machine zero
- D) Reference mode

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

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

7. What is the use of Goo activates the command?

- A) Linear interpolation
- B) Rapid traverse
- C) Return to reference point
- D) Tool returns from reference point

8. Which axis is parallel to Z - axis?

- A) Y-axis
- B) X - axis
- C) Spindle axis
- D) Tool axis

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

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

10. Which one of the code is used to rewind the program?

- A) M40
- B) M30
- C) M38
- D) M39

11. What do you mean by MPG?

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

12. Which code suitable for circular interpolation counter clockwise direction?

- A) G01
- B) G02
- C) G03
- D) G04

13. What is the purpose of MDI mode?

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

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

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

15. Why in CNC machine tool, the part program entered in edit mode is stored into the computer memory?

- A) Cannot say
- B) Can be used only once
- C) Can be used again and again
- D) Can be used again but it has to be modified every time

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

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

17. Which of the following is not advantage of CNC machine?

- A) Improved quality
C) Improved strength of the components
- B) Reduced scrap rate
D) Higher flexibility

18. Which mode is used while loading / unloading the work piece in CNC machine?

- A) Edit mode
C) Automatic mode
- B) Jog mode
D) Simulation mode

19. Which one of the following M-code is suitable for chuck close?

- A) M10
C) M13
- B) M11
D) M09

20. Which command activates in automatic mode?

- A) Tool changes automatically
C) Work offset settings are taken
- B) Program executed
D) Program simulation

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

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

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

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

23. What is the activity of code M06 based on FANUC CNC control system?

- A) The cutting tool has to be changed
C) Spindle rotation in counter clockwise direction
- B) End of the program with rewind
D) Switch of the coolant pump

24. Which G-code is suitable for data input in mm?

- A) G21
C) G28
- B) G20
D) G27

25. What do you mean by simulation?

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

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

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

27. Which mode is used to edit the program?

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

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

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

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

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

30. Which is the advantage of CNC machine over conventional machines?

- A) High cutting speed
C) Feed back control
- B) High feed
D) Mass production

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

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

32. What is the indication of 'z' axis in CNC lathe machine?

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

33. Which mode the program will be executed continuously one block after another block?

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

34. Which one of the following CNC control systems are used on CNC machines?

- A) Excel and MS power point
C) Siemens and FANUC
- B) Adobe reader and Microsoft
D) MS office only

35. Which code is suitable for multiple turning cycle in

FANUC CNC control system?

- A) G94
- B) G96
- C) G71
- D) G92

36. Which mode is used to move the turret in micron level?

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

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

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

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

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

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

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

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

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

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

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

42. What is the full form of MDI?

- A) Mounting of Diagonal Impact
- B) Manually Data Input
- C) Mechanical Data Implantation
- D) Moderated Data input

43. The CNC keyboards have with following keys?

A) Numerical Data based keys only

B) Alphabetic control keys

C) CNC codes

D) Alphanumerical control keys electronic

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

45. How the part programming mistakes can be avoided?

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

46. Which mode execute that program continuously one block after another block?

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

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

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

48. A CNC machine tool has a continuous part control both linear and circular along x,y,z and control of table rotation about x axis and z axis, so the machine should be called?

- A) 2C, L
- B) 2L, 2C
- C) 3L,2C
- D) 5C

49. Which one of the following code is suitable for rapid traverse?

- A) G00
- B) G03
- C) G04
- D) G02

50. Which one of the following M-code is suitable for spindle rotation stop?

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