

MONTHLY TEST FOR THE MONTH

Q. ID: ITISKILL4894CN

February 2026

Govt ITI MULABAGAL

Answer Key

Duration: 60 Mins

Total Marks: 50

Q.ID: ITISKILL4894CN

1. What is the purpose of MDI mode?

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

Answer: B) Move the axes by manually

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

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

Answer: D) 3

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

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

Answer: D) M08

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

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

Answer: A) Jog mode

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

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

Answer: B) G03

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

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

Answer: B) Can be used again and again

7. Which mode is used to edit the program?

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

Answer: D) Edit mode

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

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

Answer: C) Circular interpolation clockwise direction

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

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

Answer: D) The spindle rotation has to stop

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

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

Answer: C) Tool has return to reference point

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

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

Answer: D) Incremental Jog mode

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

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

Answer: A) Drilling

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

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

Answer: C) M09

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

- A) The radius of the component B) The centre axis of the component

- C) The diameter of the component D) The length of the component

Answer: D) The length of the component

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

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

Answer: C) Rapid traverse

16. 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) G02 x 42.0 z 0.0 F 0.1;
C) G00 x 42.0 z 0.0 F 0.1; D) G01 x 42.0 z 0.0 F 0.1;

Answer: D) G01 x 42.0 z 0.0 F 0.1;

17. How the part programming mistakes can be avoided?

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

Answer: C) CNC (Computer numerical control) machine only

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

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

Answer: B) The cutting tool has to be changed

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

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

Answer: A) Improved strength of the components

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

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

Answer: D) M03

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

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

Answer: D) M30

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

- A) High cutting speed B) Mass production

- C) Feed back control D) High feed

Answer: B) Mass production

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

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

Answer: B) M05

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

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

Answer: B) Machine zero

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

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

Answer: C) Auto mode

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

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

Answer: A) Preparation of new programmable Data and correction / Alteration of an earlier program

27. The CNC keyboards have with following keys?

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

Answer: C) Alphanumerical control keys electronic

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

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

Answer: A) G00

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

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

Answer: D) Siemens and FANUC

30. What is the full form of MDI?

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

Answer: C) Manually Data Input

31. What do you mean by MPG?

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

Answer: B) Manually Pulse Generator

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

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

Answer: D) M11

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

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

Answer: C) Table position

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

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

Answer: A) G21

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

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

Answer: D) Jog mode

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

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

Answer: B) The diameter of the component

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

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

Answer: B) G42

38. What do you mean by simulation?

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

Answer: B) Cross checking of programmable DATA

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

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

Answer: D) MDI mode

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

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

Answer: D) M08

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

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

Answer: C) Encoder

42. 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) 2L, 2C
C) 5C
- B) 3L,2C
D) 2C, L

Answer: B) 3L,2C

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

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

Answer: B) G32

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

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

Answer: A) Auto mode

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

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

Answer: C) Incremental jog mode

46. Which command activates in automatic mode?

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

Answer: B) Program executed

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

A) G40

B) G41

C) G42

D) G32

Answer: B) G41

48. Which code is suitable for multiple turning cycle in FANUC CNC control system?

A) G94

B) G92

C) G71

D) G96

Answer: C) G71

49. Which axis is parallel to Z - axis?

A) X - axis

B) Spindle axis

C) Y-axis

D) Tool axis

Answer: B) Spindle axis

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

A) Skill of machine operator

B) Based on electrical power supply of the machine

C) Use of jigs, cams, templates and tracers

D) Combinations of co ordinates obtained from drives along the axis

Answer: D) Combinations of co ordinates obtained from drives along the axis