

ITI Quiz - 05-Feb-2026

11:51 AM

Q. ID: ITISKILL2424TW

February 2026

Govt ITI college HD Kote

Question Paper

Duration: 30 Mins

Total Marks: 50

ID: ITISKILL2424TW

Student Name: _____ Roll No: _____

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

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

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

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

3. What do you mean by MPG?

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

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

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

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

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

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

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

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

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

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

- A) M09
- B) M30

C) M08

D) M07

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

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

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

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

11. What is the purpose of MDI mode?

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

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

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

13. What is the use of G00 activates the command?

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

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

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

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

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

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

A) Grooving
C) Facing

B) Drilling
D) Parting

17. 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) Combinations of co ordinates obtained from drives along the axis

C) Skill of machine operator

D) Based on electrical power supply of the machine

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

A) M40

B) M30

C) M39

D) M38

19. What is the full form of MDI?

A) Mechanical Data Implantation

B) Mounting of Diagonal Impact

C) Moderated Data input

D) Manually Data Input

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

A) M01

B) M09

C) M06

D) M08

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

A) Resolver

B) Encoder

C) Motor

D) Tachometer

22. Which mode is used to edit the program?

A) Auto mode

B) Edit mode

C) MDI mode

D) Jog mode

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

A) Execute the program

B) Preparation of new programmable Data and correction / Alteration of an earlier program

C) Simulation of the program

D) Can read only the program

24. How the part programming mistakes can be avoided?

A) Cannot be avoided in CNC machine

B) Both NC and CNC machine

C) NC (Numerical control) machine only

D) CNC (Computer numerical control) machine only

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

A) M03

B) M04

C) G02

D) G03

26. The CNC keyboards have with following keys?

A) CNC codes

B) Alphanumerical control keys electronic

C) Numerical Data based keys only

D) Alphabetic control keys

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

A) Rewind the program once again

B) Tool returns from reference point to work place

C) The spindle rotation has to stop

D) The cutting tool has to be changed

28. Which command activates in automatic mode?

A) Work offset settings are taken

B) Tool changes automatically

C) Program executed

D) Program simulation

29. 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) G02 x 42.0 z 0.0 F 0.1;

B) G00 x 42.0 z 0.0 F 0.1;

C) G03 x 42.0 z 0.0 F 0.1;

D) G01 x 42.0 z 0.0 F 0.1;

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

A) 2

B) 3

C) 5

D) 4

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

A) M10

B) M09

C) M08

D) M11

32. 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) 3L,2C

C) 5C

D) 2L, 2C

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

A) High cutting speed

B) Mass production

C) Feed back control

D) High feed

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

A) Work zero

B) Tool geometrical value

C) Reference mode	D) Machine zero	C) Automatic mode	D) Jog mode
35. What is the purpose of G28 code based on FANUC CNC control system? A) Tool has to return from reference point C) Circular interpolation counter clockwise B) Tool has return to reference point D) Circular interpolation clockwise		43. 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	
36. Which one of the following M-code is suitable for spindle rotation stop? A) M06 C) M03 B) M05 D) M04		44. Which mode the program will be executed continuously one block after another block? A) Auto mode C) Jog mode B) Edit mode D) Manual mode	
37. Which mode is used for five positioning (Micron level) of the axis movement? A) Incremental Jog mode C) Edit mode B) Auto mode D) MDI mode		45. Which one of the following code is suitable for rapid traverse? A) G00 C) G02 B) G03 D) G04	
38. Which one of the following M-code is suitable for chuck close? A) M11 C) M13 B) M09 D) M10		46. What do you mean by simulation? A) Finding out of work offset values C) Editing of related part program B) Cross checking of programmable DATA D) Finding out of tool geometrical values	
39. Which one of the following CNC control systems are used on CNC machines? A) MS office only C) Adobe reader and Microsoft B) Siemens and FANUC D) Excel and MS power point		47. Which code is suitable for tool nose radius compensation at right side? A) G42 C) G50 B) G40 D) G41	
40. Which mode is used while loading / unloading the work piece in CNC machine? A) Jog mode C) Automatic mode B) Simulation mode D) Edit mode		48. 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	
41. Which G-code is suitable for thread cutting? A) G30 C) G01 B) G32 D) G34		49. Which of the following is not advantage of CNC machine? A) Improved quality C) Higher flexibility B) Reduced scrap rate D) Improved strength of the components	
42. Which mode, the cutting tool returns back to its home position? A) Reference mode B) MDI mode		50. Which axis is parallel to Z - axis? A) Y-axis C) Spindle axis B) X - axis D) Tool axis	