

ITI Quiz - 05-Feb-2026

11:51 AM

Q. ID: ITISKILL2424TW

February 2026

Govt ITI college HD Kote

Answer Key

Duration: 30 Mins

Total Marks: 50

Q.ID: ITISKILL2424TW

1. How the part programming mistakes can be avoided?

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

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

2. Which command activates in automatic mode?

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

Answer: B) Program executed

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

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

Answer: A) Siemens and FANUC

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

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

Answer: A) MDI mode

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

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

Answer: B) M08

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

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

Answer: A) M11

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

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

Answer: D) M08

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

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

Answer: B) G00

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

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

Answer: C) G21

10. 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

11. 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

12. 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 cutting tool has to be changed
- D) The spindle rotation has to stop

Answer: D) The spindle rotation has to stop

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

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

Answer: C) G32

14. Which mode execute the program continuously one block after another block?

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

Answer: D) Auto mode

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

- A) M30
C) M38

- B) M39
D) M40

Answer: A) M30

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

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

Answer: D) Incremental jog mode

17. What do you mean by simulation?

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

Answer: C) Cross checking of programmable DATA

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

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

Answer: A) The diameter of the component

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

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

Answer: B) G71

20. Which axis is parallel to Z - axis?

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

Answer: D) Spindle axis

21. 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) 2C, L
- B) 3L,2C
D) 5C

Answer: B) 3L,2C

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

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

Answer: D) Table position

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

- A) The cutting tool has to be changed
- B) Spindle rotation in counter clockwise direction

- C) Switch of the coolant pump

- D) End of the program with rewind

Answer: A) The cutting tool has to be changed

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

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

Answer: A) G42

25. What is the full form of MDI?

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

Answer: B) Manually Data Input

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

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

Answer: C) Jog mode

27. The CNC keyboards have with following keys?

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

Answer: B) Alphanumeric control keys electronic

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

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

Answer: D) Circular interpolation clockwise direction

29. Which mode is used to edit the program?

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

Answer: D) Edit mode

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

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

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

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

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

Answer: B) M09

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

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

Answer: D) Mass production

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

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

Answer: C) Encoder

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

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

Answer: C) G03

35. What do you mean by MPG?

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

Answer: A) Manually Pulse Generator

36. 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) G02 x 42.0 z 0.0 F 0.1;
- D) G03 x 42.0 z 0.0 F 0.1;

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

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

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

Answer: B) Auto mode

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

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

Answer: B) M05

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

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

Answer: A) Jog mode

40. 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

41. 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) Can be used again but it has to be modified every time
- C) Cannot say
- D) Can be used only once

Answer: A) Can be used again and again

42. 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

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

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

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

Answer: D) Incremental Jog mode

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

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

Answer: B) Machine zero

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

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

Answer: C) Rapid traverse

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

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

Answer: A) 3

47. Which mode, the cutting tool returns back to its home

position?

- A) Jog mode
- B) Reference mode
- C) MDI mode
- D) Automatic mode

Answer: B) Reference mode

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

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

Answer: A) Improved strength of the components

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

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

Answer: B) Tool has return to reference point

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

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

Answer: B) Drilling