

Duration: 50 Mins

Total Marks: 50

Q.ID: ITISKILL1797M0

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

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

Answer: D) Table position

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

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

Answer: B) 3

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

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

Answer: A) Jog mode

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

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

Answer: C) G03

5. The CNC keyboards have with following keys?

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

Answer: C) Alphanumerical control keys electronic

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

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

Answer: A) G32

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

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

Answer: D) Drilling

8. Which command activates in automatic mode?

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

Answer: D) Program executed

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

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

Answer: A) Siemens and FANUC

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

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

Answer: C) M08

11. Which axis is parallel to Z - axis?

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

Answer: A) Spindle axis

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

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

Answer: B) MDI mode

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

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

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

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

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

Answer: D) M30

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

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

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

Answer: B) The length of the component

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

Answer: C) Auto mode

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

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

Answer: C) M11

24. Which mode is used to edit the program?

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

Answer: C) Edit mode

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

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

Answer: D) Auto mode

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

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

Answer: A) The diameter of the component

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

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

Answer: D) G41

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

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

Answer: A) Encoder

19. What do you mean by MPG?

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

Answer: D) Manually Pulse Generator

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

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

Answer: D) G00

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

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

Answer: B) M08

28. What is the purpose of MDI mode?

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

Answer: A) Move the axes by manually

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

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

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

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

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

Answer: B) M03

22. Which mode, the cutting tool returns back to its home position?

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

Answer: B) Reference mode

30. What do you mean by simulation?

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

Answer: A) Cross checking of programmable DATA

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

31. 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) G03 x 42.0 z 0.0 F 0.1;

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

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

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

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

Answer: B) Rapid traverse

33. 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

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

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

Answer: B) Jog mode

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

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

Answer: D) G71

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

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

Answer: B) Tool has return to reference point

37. 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) 5C
C) 2L, 2C D) 3L,2C

Answer: D) 3L,2C

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

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

Answer: A) Incremental jog mode

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

- A) Circular interpolation clockwise direction B) Circular interpolation counter clockwise direction

C) Linear interpolation D) Cross sectional interpolation

Answer: A) Circular interpolation clockwise direction

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

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

Answer: C) G42

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

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

Answer: A) M09

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

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

Answer: A) The spindle rotation has to stop

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

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

Answer: C) The cutting tool has to be changed

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

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

Answer: D) Improved strength of the components

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

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

Answer: D) G21

46. 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 but it has to be modified every time
C) Cannot say D) Can be used again and again

Answer: D) Can be used again and again

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

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

Answer: A) Mass production

48. What is the full form of MDI?

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

Answer: C) Manually Data Input

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

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

Answer: A) Incremental Jog mode

50. 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
