

Duration: 90 Mins

Total Marks: 51

Q.ID: ITISKILL6543VC

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

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

Answer: A) The diameter of the component

2. What is the full form of MDI?

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

Answer: C) Manually Data Input

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

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

Answer: A) Encoder

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

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

Answer: A) M30

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

Answer: C) 3L,2C

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

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

Answer: D) M11

7. What do you mean by MPG?

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

Answer: B) Manually Pulse Generator

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

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

Answer: B) Incremental Jog mode

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

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

Answer: A) Drilling

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

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

Answer: C) Jog mode

11. How the part programming mistakes can be avoided?

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

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

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

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

Answer: A) Tool has return to reference point

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

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

Answer: A) Mass production

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

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

Answer: B) M03

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

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

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

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

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

Answer: B) 3

17. 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

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

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

Answer: A) Rapid traverse

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

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

Answer: D) The length of the component

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

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

Answer: B) G03

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

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

Answer: B) Reference mode

22. Which mode is used to edit the program?

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

Answer: D) Edit mode

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

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

Answer: C) Improved strength of the components

24. 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

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

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

Answer: D) Auto mode

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

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

Answer: D) M09

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

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

Answer: A) Machine zero

28. What is the purpose of MDI mode?

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

Answer: C) Move the axes by manually

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

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

Answer: C) Siemens and FANUC

30. Which axis is parallel to Z - axis?

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

Answer: B) Spindle axis

31. What do you mean by simulation?

- A) Cross checking of programmable DATA
B) Editing of related part program

C) Finding out of tool geometrical values

D) Finding out of work offset values

Answer: A) Cross checking of programmable DATA

C) Circular interpolation counter clockwise direction

D) Circular interpolation clockwise direction

Answer: D) Circular interpolation clockwise direction

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

A) Spindle position

B) Work position

C) Table position

D) Spindle speed

Answer: C) Table position

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

A) M10

B) M09

C) M08

D) M11

Answer: C) M08

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

A) G42

B) G50

C) G41

D) G40

Answer: A) G42

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

A) Edit mode

B) MDI mode

C) Auto mode

D) Jog mode

Answer: D) Jog mode

34. 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) G01 x 42.0 z 0.0 F 0.1;

C) G00 x 42.0 z 0.0 F 0.1;

D) G02 x 42.0 z 0.0 F 0.1;

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

41. Which command activates in automatic mode?

A) Work offset settings are taken

B) Program simulation

C) Program executed

D) Tool changes automatically

Answer: C) Program executed

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

A) Rewind the program once again

B) The cutting tool has to be changed

C) Tool returns from reference point to work place

D) The spindle rotation has to stop

Answer: D) The spindle rotation has to stop

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

A) G04

B) G02

C) G03

D) G00

Answer: D) G00

36. 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

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

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

A) M09

B) M01

C) M06

D) M08

Answer: D) M08

37. 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) Cannot say

C) Can be used again and again

D) Can be used again but it has to be modified every time

Answer: C) Can be used again and again

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

A) Incremental jog mode

B) MDI mode

C) Auto mode

D) Jog mode

Answer: A) Incremental jog mode

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

A) Linear interpolation

B) Cross sectional interpolation

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

A) G32

B) G01

C) G34

D) G30

Answer: A) G32

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

A) G32

B) G40

C) G42

D) G41

Answer: D) G41

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

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

Answer: D) G21

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

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

Answer: D) Auto mode

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

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

Answer: A) G71

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

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

Answer: C) The cutting tool has to be changed

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

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

Answer: A) M05