

Duration: 30 Mins

Total Marks: 50

Q.ID: ITISKILL6528TX

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

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

Answer: C) Tool has return to reference point

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

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

Answer: A) Encoder

3. Which axis is parallel to Z - axis?

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

Answer: A) Spindle axis

4. Encodor is used in CNC machine tool to sense and control?

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

Answer: D) Table position

5. 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

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

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

Answer: B) Siemens and FANUC

7. What do you mean by MPG?

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

Answer: A) Manually Pulse Generator

8. 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

9. What do you mean by simulation?

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

Answer: D) Cross checking of programmable DATA

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

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

Answer: A) Reference mode

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

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

Answer: C) Jog mode

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

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

Answer: B) G42

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

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

Answer: C) Mass production

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

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

Answer: D) G71

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

- A) Facing
- C) Drilling

- B) Grooving
- D) Parting

Answer: C) Drilling

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

- A) M13
- C) M11

- B) M10
- D) M09

Answer: C) M11

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

- A) Improved strength of the components
- C) Reduced scrap rate

- B) Improved quality
- D) Higher flexibility

Answer: A) Improved strength of the components

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

- A) G03
- C) G02

- B) G04
- D) G00

Answer: D) G00

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

- A) Linear interpolation
- C) Circular interpolation counter clockwise direction

- B) Circular interpolation clockwise direction
- D) Cross sectional interpolation

Answer: B) Circular interpolation clockwise direction

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

- A) The length of the component
- C) The radius of the component

- B) Centre axis of the component
- D) The diameter of the component

Answer: D) The diameter of the component

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

- A) Jog mode
- C) Auto mode

- B) Edit mode
- D) MPG mode

Answer: C) Auto mode

22. 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) 3L,2C

- B) 5C
- D) 2C, L

Answer: C) 3L,2C

23. Which command activates in automatic mode?

- A) Program executed
- C) Tool changes automatically

- B) Work offset settings are taken
- D) Program simulation

Answer: A) Program executed

24. 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;
 - C) G00 x 42.0 z 0.0 F 0.1;
- B) G01 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;

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

- A) G02
- C) M03

- B) M04
- D) G03

Answer: C) M03

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

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

- B) Execute the program
- D) Simulation of the program

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

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

- A) Jog mode
- C) MDI mode

- B) Auto mode
- D) Incremental jog mode

Answer: D) Incremental jog mode

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

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

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

Answer: C) The length of the component

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

- A) M09
- C) M01

- B) M06
- D) M08

Answer: D) M08

30. Which mode is used to edit the program?

- A) Jog mode
- C) Edit mode

- B) MDI mode
- D) Auto mode

Answer: C) Edit mode

31. In Which mode we can input the program command

manually?

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

Answer: A) MDI mode

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

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

Answer: B) Jog mode

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

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

Answer: B) Rapid traverse

34. The CNC keyboards have with following keys?

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

Answer: C) Alphanumerical control keys electronic

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

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

Answer: B) M30

36. How the part programming mistakes can be avoided?

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

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

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

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

Answer: B) M08

38. 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

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

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

Answer: D) The spindle rotation has to stop

40. 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) Can be used again and again
- D) Cannot say

Answer: C) Can be used again and again

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

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

Answer: B) G03

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

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

Answer: C) 3

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

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

Answer: B) G32

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

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

Answer: B) G41

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

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

Answer: B) M09

46. 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
- B) Skill of machine operator
- 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

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

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

Answer: C) G21

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

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

Answer: C) Auto mode

49. What is the full form of MDI?

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

Answer: A) Manually Data Input

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) End of the program with rewind
- D) The cutting tool has to be changed

Answer: D) The cutting tool has to be changed
