

Duration: 30 Mins

Total Marks: 51

ID: ITISKILL1644NN

Student Name: _____ Roll No: _____

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

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

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

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

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

10. Which command activates in automatic mode?

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

3. What do you mean by simulation?

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

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

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

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

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

12. Which mode is used to edit the program?

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

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

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

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

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

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

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

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

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

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

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

15. 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

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

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

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

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

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

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

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

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

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

19. What do you mean by MPG?

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

20. What is the full form of MDI?

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

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

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

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

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

23. 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) 2L, 2C
- D) 5C

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

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

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

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

26. 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) G01 x 42.0 z 0.0 F 0.1;
- B) G03 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;

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

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

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

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

29. What is the purpose of MDI mode?

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

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

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

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

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

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

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

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

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

34. 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) Use of jigs, cams, templates and tracers
- C) Combinations of co ordinates obtained from drives along the axis
- D) Skill of machine operator

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

- A) M08
- B) M30

C) M07

D) M09

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

A) G01

B) G03

C) G04

D) G02

37. 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

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

A) M11

B) M08

C) M10

D) M09

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

A) G42

B) G40

C) G41

D) G32

40. 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

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

A) M40

B) M39

C) M30

D) M38

42. Which axis is parallel to Z - axis?

A) Y-axis

B) X - axis

C) Tool axis

D) Spindle axis

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

A) M04

B) M06

C) M05

D) M03

44. The CNC keyboards have with following keys?

A) CNC codes

B) Alphanumerical control keys electronic

C) Alphabetic control keys

D) Numerical Data based keys only

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

A) G00

B) G04

C) G02

D) G03

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

A) Edit mode

B) Jog mode

C) Auto mode

D) MPG mode

47. 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

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

A) The length of the component

B) The diameter of the component

C) Centre axis of the component

D) The radius of the component

49. How the part programming mistakes can be avoided?

A) Both NC and CNC machine

B) NC (Numerical control) machine only

C) CNC (Computer numerical control) machine only

D) Cannot be avoided in CNC machine

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

A) Parting

B) Drilling

C) Facing

D) Grooving

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

A) Siemens and FANUC

B) Adobe reader and Microsoft

C) MS office only

D) Excel and MS power point