Student: Ashok
Score: 23/50 (46.00%)   **Code: 6269**

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

A) G02   **B) M03 (Correct)**
C) G03   D) M04

---

2. What do you mean by simulation?

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

---

3. What do you mean by MPG?

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

---

4. The CNC keyboards have with following keys?

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

---

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

A) Linear interpolation   **B) Circular interpolation clockwise direction (Correct)**
C) Circular interpolation counter clockwise direction   D) Cross sectional interpolation

---

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

A) G01   B) G02
C) G03   D) G04 (Incorrect)

---

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

A) G50   **B) G42 (Correct)**
C) G40   D) G41

---

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

A) G21 (Correct)   B) G27
C) G20   D) G28

---

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

A) M09   **B) M11**
C) M10 (Incorrect)   D) M13

---

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

A) M10   B) M11
C) M09   **D) M08 (Correct)**

---

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

A) M38 (Incorrect)   **B) M30**
C) M39   D) M40

---

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

A) G71 (Correct)   B) G92
C) G96   D) G94

---

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

A) G30   **B) G32**
C) G34   D) G01 (Incorrect)

---

14. Which mode is used to edit the program?

A) Jog mode (Incorrect)   **B) Edit mode**
C) MDI mode   D) Auto mode

---

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

A) Auto mode   B) Jog mode (Incorrect)
C) MDI mode   **D) Incremental jog mode**

---

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

A) Jog mode (Correct)   B) Edit mode
C) Auto mode   D) MDI mode

---

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

A) Auto mode   B) Jog mode
C) MDI mode   D) Edit mode (Incorrect)

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

A) Jog mode   **B) Auto mode (Correct)**

C) Edit mode   D) Manual mode

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

A) 2 (Incorrect)   **B) 3**

C) 4   D) 5

20. 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 and again**

C) Can be used again but it has to be modified every time   D) Cannot say (Incorrect)

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

A) High cutting speed (Incorrect)   B) Feed back control

C) High feed   **D) Mass production**

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

A) Drilling   B) Parting

C) Grooving   D) Facing (Incorrect)

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

A) Resolver   B) Tachometer (Incorrect)

C) Encoder   D) Motor

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

A) Spindle speed   B) Spindle position

C) Table position (Correct)   D) Work position

25. Which axis is parallel to Z - axis?

A) X - axis   **B) Spindle axis**

C) Tool axis   D) Y-axis (Incorrect)

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

A) Higher flexibility   B) Improved quality (Incorrect)

C) Reduced scrap rate   **D) Improved strength of the components**

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

A) The spindle rotation has to stop (Correct)   B) The cutting tool has to be changed

C) Tool returns from reference point to work place   D) Rewind the program once again

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

A) M01   B) M06

C) M08 (Correct)   D) M09

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

A) M07   B) M08

C) M09 (Correct)   D) M30

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

A) Linear interpolation (Incorrect)   **B) Rapid traverse**

C) Tool returns from reference point   D) Return to reference point

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

A) End of the program with rewind   B) Spindle rotation in counter clockwise direction

C) The cutting tool has to be changed (Correct)   D) Switch of the coolant pump

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

A) Tool has return to reference point   B) Tool has to return from reference point (Incorrect)

C) Circular interpolation clockwise   D) Circular interpolation counter clockwise

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

A) Automatic mode   B) MDI mode

C) Jog mode (Incorrect)   **D) Reference mode**

34. What is the purpose of MDI mode?

A) For program execution   **B) Move the axes by manually (Correct)**

C) Cross checking of the program graphically   D) Editing of programmable Data

35. What is the full form of MDI?

A) Mechanical Data Implantation   **B) Manually Data Input (Correct)**

C) Moderated Data input   D) Mounting of Diagonal Impact

36. Which command activates in automatic mode?

A) Work offset settings are taken

B) Tool changes automatically (Incorrect)

C) Program executed

D) Program simulation

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

A) Can read only the program

B) Simulation of the program

C) Execute the program

D) Preparation of new programmable Data and correction / Alteration of an earlier program (Correct)

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

A) Simulation mode

B) Automatic mode (Incorrect)

C) Jog mode

D) Edit mode

39. Which mode execute thet program continuosly one block after another block?

A) Jog mode

B) Edit mode

C) MPG mode

D) Auto mode (Correct)

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

A) Auto mode

B) Incremental Jog mode (Correct)

C) MDI mode

D) Edit mode

41. 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 (Incorrect)

B) Skill of machine operator

C) Combinations of co ordinates obtained from drives along the axis

D) Based on electrical power supply of the machine

42. How the part programming mistakes can be avoided?

A) NC (Numerical control) machine only

B) CNC (Computer numerical control) machine only (Correct)

C) Both NC and CNC machine

D) Cannot be avoided in CNC machine

43. 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 (Incorrect)

B) 3L,2C

C) 5C

D) 2L, 2C

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

A) G00

B) G02

C) G03

D) G04 (Incorrect)

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

A) G40 (Incorrect)

B) G41

C) G42

D) G32

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

A) M05 (Correct)

B) M03

C) M06

D) M04

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

A) Work zero

B) Tool geometrical value

C) Machine zero (Correct)

D) Reference mode

48. 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 (Incorrect)

D) MS office only

49. 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) G02 x 42.0 z 0.0 F 0.1;

C) G01 x 42.0 z 0.0 F 0.1; (Correct)

D) G03 x 42.0 z 0.0 F 0.1;

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

A) The radius of the component

B) The length of the component (Incorrect)

C) The diameter of the component

D) Centre axis of the component