

**Student: SURESH C**  
**Score: 17/51 (33.33%)**

**Code: 7423**

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

4. The CNC keyboards have with following keys?

- A) Alphabetic control keys (Incorrect)  
 B) **Alphanumerical control keys electronic**  
 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 (Incorrect)  
 C) **G03**  
 D) G04

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

- A) G50 (Incorrect)  
 B) **G42**  
 C) G40  
 D) G41

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

- A) **G21**  
 B) G27  
 C) G20  
 D) G28 (Incorrect)

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

- A) M09 (Incorrect)  
 B) **M11**  
 C) M10  
 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  
 B) **M30**  
 C) M39 (Incorrect)  
 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 (Incorrect)  
 D) G01

14. Which mode is used to edit the program?

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

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

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

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

- A) **Jog mode**  
 B) Edit mode  
 C) Auto mode (Incorrect)  
 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  
C) Edit mode  
**B) Auto mode (Correct)**  
D) Manual mode

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

- A) 2 (Incorrect)  
C) 4  
**B) 3**  
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  
C) Can be used again but it has to be modified every time (Incorrect)  
**B) Can be used again and again**  
D) Cannot say

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

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

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

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

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

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

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

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

25. Which axis is parallel to Z - axis?

- A) X - axis  
C) Tool axis (Incorrect)  
**B) Spindle axis**  
D) Y-axis

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

- A) Higher flexibility (Incorrect)  
C) Reduced scrap rate  
B) Improved quality  
**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

C) Tool returns from reference point to work place

B) The cutting tool has to be changed (Incorrect)

D) Rewind the program once again

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

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

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

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

30. What is the use of G00 activates the command?

- A) Linear interpolation (Incorrect)  
C) Tool returns from reference point  
**B) Rapid traverse**  
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  
C) The cutting tool has to be changed  
B) Spindle rotation in counter clockwise direction  
D) Switch of the coolant pump (Incorrect)

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

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

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

- A) Automatic mode  
C) Jog mode  
B) MDI mode  
**D) Reference mode (Correct)**

34. What is the purpose of MDI mode?

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

35. What is the full form of MDI?

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

