

C) Auto mode D) Edit mode

Answer: D) Edit mode

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

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

Answer: C) The cutting tool has to be changed

17. What do you mean by simulation?

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

Answer: C) Cross checking of programmable DATA

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

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

Answer: B) G21

19. Which axis is parallel to Z - axis?

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

Answer: A) Spindle axis

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

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

Answer: C) G42

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

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

Answer: B) M30

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

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

Answer: C) Reference mode

23. How the part programming mistakes can be avoided?

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

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

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

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

Answer: D) Auto mode

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

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

Answer: B) Table position

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

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

Answer: D) G41

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

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

Answer: D) Auto mode

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

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

Answer: D) G03

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

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

Answer: A) Jog mode

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

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

Answer: D) Encoder

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

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

Answer: B) M11

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

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

Answer: C) Siemens and FANUC

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

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

Answer: D) The length of the component

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

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

Answer: C) Improved strength of the components

35. Why in CNC machine tool, the part program entered in edit mode is stored into the computer memory?

- A) Can be used again but it has to be modified every time B) Can be used only once
C) Cannot say D) Can be used again and again

Answer: D) Can be used again and again

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

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

Answer: A) The diameter of the component

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

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

Answer: B) Mass production

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

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

Answer: C) Circular interpolation clockwise direction

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

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

Answer: D) Tool has return to reference point

40. What is the purpose of MDI mode?

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

Answer: B) Move the axes by manually

41. What do you mean by MPG?

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

Answer: D) Manually Pulse Generator

42. 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

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

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

Answer: B) M09

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

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

Answer: B) Incremental jog mode

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

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

Answer: D) The spindle rotation has to stop

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

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

Answer: A) M08

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

- A) G04 B) G00
C) G03 D) G02

Answer: B) G00

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

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

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

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

Answer: C) Machine zero

50. 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