Machinist

Exam Preparation
Helping you Succeed

1. The point-to-point is the simplest form of NC. The operation done in this system is :

   (A) Drilling
   (B) Boring
   (C) Straight line milling
   (D) Any one of the above

2. A point-to-point control system is suitable only for one of the following applications. Select it.

   (A) Milling profile
   (B) Turning
   (C) Drilling
   (D) Grinding

3. On which unit is the tool setting done on NC machine ?

   (A) On special device away from the machine
   (B) On machine itself while the other operations are being performed
   (C) On the NC machine during idle time
   (D) On the pre-setting setting

4. Which one of the following is NOT the advantage of CNC machine ?

   (A) Reduces inspection time
   (B) Reduces tooling time
   (C) Higher initial cost
   (D) Higher rate of production

5. CNC lathe machines have built-in coordinate measuring system. The zero position on the co-ordinate system is called :

   (A) Reference point
   (B) Machine zero point
   (C) Work zero point
   (D) Programme zero point

6. A CNC lathe is programmed and set to turn a diameter of 50 mm, but on trial run the diameter is found to be 50.1 mm. Which one of the following will ensure correct diameter during production ?

   (A) Increase the tool offset by 0.1 mm
   (B) Decrease the tool offset by 0.1 mm
   (C) Increase the tool offset by 0.05 mm
   (D) Decrease the tool offset by 0.05 mm

7. CNC machines are not normally operated. They are controlled by means of a :

   (A) Programme
   (B) Operator
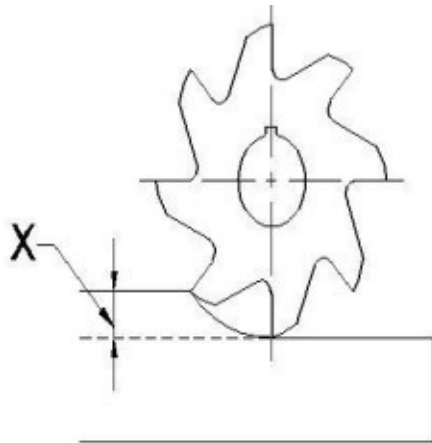   (C) Cam
   (D) Plug board system

8. Which one of the following statements about programming of NC machine is correct ?

(A) Programming is only possible on central server

(B) Programming can be entered on control panel and can be read from a tape

(C) Modification of the programme on the control panel is not possible

(D) Entering the programme is only possible directly on the control panel

9. Which one of the following statements refers to contouring control ?

(A) This system can provide feed control in three axes

(B) This system provides only one feed axis

(C) This system provides motion in two axes (But not simultaneously)

(D) This does not provide any feed axis

10. Which one of the following statements refers to block structure ?

(A) It refers to the rule for writing programme block

(B) It can be done by keyboard system on the CNC machine

(C) It is electrical control fixed on CNC machine

(D) The measuring system is calibrated for this point

11. Command G00 means

(A) Feed motion in circular path

(B) Rapid motion

(C) Feed motion in straight line

(D) Slow motion

12. Which one of the following is the command for feed motion in straight line ?

(A) G00

(B) G01

(C) G02

(D) None of the above

13. Toughness of a cutting tool means ability to withstand _____ without failure.

(A) high temperature

(B) wear

(C) impact forces

(D) chemical reaction

1041. Which knob is used to control rapid motion in VMC?

(A) Mode selection

(B) Rapid traverse

(C) Feed rate over ride

(D) Spindle speed over ride

1042. Which function is used for movement command in part program?

(A) Preparatory function

(B) Miscellaneous function

(C) Speed function

(D) Feed function

1043. What is the distance from X0, Y0, Z0 of machine zero to work zero in x,y,z direction?

(A) Tool length offset

(B) Zero offset

(C) Tool wear offset

(D) Wear offset

1044. Which is the offset from nose of the holder to overhanging length of cutter?

(A) Work offset

(B) Wear offset

(C) Tool offset

(D) Geometry offset

1045. Which is pressed to stop temporarily during auto mode operation?

(A) Cycle stop

(B) Feed hold

(C) Dry run

(D) Machine lock

1046. Which mode is to check the movement of tool without work piece?

(A) Jog

(B) Machine lock

(C) Dry run

(D) Auto mode

1047. Which mode is to check the program block one by one?

    (A) Jog mode   (B) Edit mode
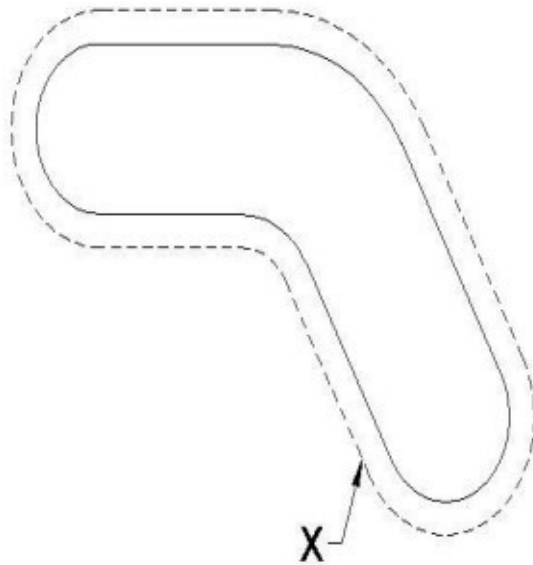
    (C) Auto mode   (D) Single block

1048.

What is marked as 'x'?

    (A) Depth of cut   (B) Feed

    (C) Cutting speed   (D) rpm

1049.

What is marked as 'x'?

(A) Programmed path

(B) Compensated path

(C) Roughing path

(D) Finishing path

1050. What is perpendicular distance between original and final surface of the work piece?

(A) Feed

(B) Cutting speed

(C) Depth of cut

(D) rpm