CNC Machining Mill Exam – Up Academy

Part 1: Applied Machining Word Problems (Questions 1–9 | 9 points)

Select the best answer.

1. If G99 is active, what Z value does the tool return to after a drilling cycle?
   A. Z0
   B. R plane
   C. Clearance plane
   D. Last Z depth

2. A part has a tolerance of ±.005. What are the acceptable limits of a 1.000 inch feature?
   A. 0.990 to 1.010
   B. 0.995 to 1.005
   C. 1.000 to 1.010
   D. 0.998 to 1.002

3. A surface is specified to be flat within .002. What type of GD&T control is this?
   A. Parallelism
   B. Flatness
   C. Profile
   D. Straightness

4. During inspection, the diameter measured is 1.005, and the tolerance is 1.000 ±.010. Is this acceptable?
   A. Yes
   B. No
   C. Only with bonus tolerance
   D. Only if using LMC

5. You're using a 3/8-16 UNC thread. What should the minor diameter be?
   A. 0.375
   B. ~0.317
   C. ~0.332
   D. ~0.344

6. What G-code would you use to drill three holes in a line with peck depth?
   A. G83
   B. G81

C. G84
D. G73

7. A work offset is set incorrectly. What defect might result?
   A. Tool breakage
   B. Mislocated features
   C. Burrs
   D. Surface finish issues

8. The part is sticking out 5.0 inches. What safety or accuracy concerns arise?
   A. Tool wear
   B. Vibration and deflection
   C. Wrong RPM
   D. Material overheating

9. A G43 H01 Z.1 command is issued. What happens next?
   A. Rapid retract
   B. Tool change
   C. Tool length offset applied
   D. Spindle stops

Part 2: Geometric Dimensioning & Tolerancing (Questions 10–18 | 9 points)

Choose the best answer.

10. What does MMC stand for and how is it used in inspection?
    A. Minimum Material Condition – smallest hole
    B. Maximum Material Condition – smallest hole/largest pin
    C. Maximum Machining Control – used for tool paths
    D. Minimum Measurement Calibration – for CMMs

11. Explain the difference between form and orientation tolerances.
    A. Form is absolute; orientation is relative to a datum
    B. Orientation is stricter than form
    C. Form applies only to holes
    D. They are the same

12. What symbol is used to represent perpendicularity?
    A. ⌖
    B. ⏥

C. ⊥
D. ∠

13. Define true position in your own words.
    A. The surface finish quality
    B. The exact location of a feature within a tolerance zone
    C. The number of tools used
    D. The maximum cutting depth

14. What tool would be best for measuring parallelism on a block?
    A. Micrometer
    B. Caliper
    C. Surface plate with height gauge
    D. Thread gauge

15. What is the difference between profile of a surface and profile of a line?
    A. One is 2D, the other is 3D
    B. Line is for external only
    C. Surface controls all points; line controls a cross-section
    D. They're identical

16. What does a composite tolerance frame mean?
    A. It stacks two tolerances
    B. It combines two datums
    C. It applies location and orientation together
    D. It's for surface finish

17. How does a CMM differ from a height gauge?
    A. CMM is manual, height gauge is automated
    B. CMM is 3-axis capable and programmable
    C. CMM only measures height
    D. They're used the same

18. Explain what RFS means in tolerance analysis.
    A. Required Flat Surface
    B. Regardless of Feature Size
    C. Rounded Face Specification
    D. Radial Form Standard

Part 3: Decimal Lingo & Shop Math (Questions 19–27 | 9 points)

Choose the correct answer.

19. How would you say 0.005 in the shop?
    A. Five inches
    B. Five mils
    C. Five thousandths
    D. Fifty tenths

20. Convert 0.0003 to shop lingo.
    A. Thirty ten-thousandths
    B. Three thousandths
    C. Three tenths
    D. Three millionths

21. What is one tenth in decimal?
    A. 0.01
    B. 0.1
    C. 0.001
    D. 0.0001

22. If a dimension reads .010, what is the tolerance zone between ±.005?
    A. 0.005 to 0.015
    B. 0.010 to 0.020
    C. 0.000 to 0.020
    D. 0.002 to 0.018

23. Answer each: 0.1 = ____, 0.01 = ____, 0.001 = ____
    A. Ten thou, one thou, one tenth
    B. One tenth, ten thou, hundred thou
    C. One tenth, one hundredth, one thousandth
    D. Tenth, hundredth, thousandth

24. A caliper reads 0.253, how many thousandths is that?
    A. 253
    B. 25.3
    C. 2.53
    D. 0.253

25. A tolerance is .005, and the feature measures 0.998. Is it in spec for a 1.000 nominal?
    A. No
    B. Yes
    C. Only if MMC is used
    D. Depends on datums

26. 0.0009 equals how many tenths of a thousandth?
    A. 9
    B. 90
    C. 0.9
    D. 1

27. 0.0011 is how many tenths of a thousandth?
    A. 1.1
    B. 11
    C. 110
    D. 0.011

Part 4: Decimal Conversion (Question 28 | 1 point)

28. Convert 'fifty thousandths' to decimal form.
    A. 0.005
    B. 0.0005
    C. 0.05
    D. 0.5

Part 5: Matching – Milling Machine Components (Questions 29–33 | 5 points)

Match the component to its function.

29. Rotating shaft that holds and drives the tool
    A. Table
    B. Saddle
    C. Spindle
    D. Knee

30. Surface where workpiece is secured
    A. Table
    B. Quill
    C. Saddle
    D. Spindle

31. Supports the table and allows Y-axis motion
    A. Knee
    B. Quill
    C. Saddle
    D. Spindle

32. Supports the saddle and moves vertically
    A. Table
    B. Spindle
    C. Quill
    D. Knee

33. Houses the spindle and moves it up/down
    A. Saddle
    B. Table
    C. Quill
    D. Spindle

Part 6: Matching – Vocabulary Review (Questions 34–38 | 5 points)

Match the term to its definition.

34. Allowable deviation of an angle from a datum
    A. Datum
    B. Chamfer
    C. Angularity
    D. Blueprint

35. Reference feature from which measurements are taken
    A. Angularity
    B. Datum
    C. Tolerance
    D. Chamfer

36. Document showing all manufacturing instructions
    A. Tolerance
    B. Blueprint
    C. Datum
    D. Chamfer

37. Acceptable variation in a part's size or geometry
    A. Chamfer
    B. Blueprint
    C. Angularity
    D. Tolerance

38. Angled cut on the edge of a part
    A. Datum
    B. Chamfer
    C. Blueprint
    D. Angularity