CNC Machining Practice Test – Up Academy

This practice test includes 30 questions focused on applied machining concepts, trigonometry, word problems, and blueprint math.

Practice Questions (30 points)

1. 1. A chamfer is .100 x 45°. What is the actual surface length of the chamfer?
2. 2. A tool travels 12 inches along the X-axis and 5 inches along the Y-axis. What angle is formed from the X-axis?
3. 3. Use the Pythagorean theorem to calculate the length of a diagonal across a 3 in. x 4 in. part.
4. 4. A drilling operation is performed at 15° to the surface. If the vertical depth is 1.5 inches, what is the horizontal travel?
5. 5. What is the radius of a circle with a diameter of 1.250 inches?
6. 6. A G83 cycle has a peck depth of 0.100 and a final Z depth of -1.000. How many pecks will it take?
7. 7. A surface must be flat within .003 inches. What inspection tools would you use to check this?
8. 8. What does G54 do in a CNC program?
9. 9. Calculate the tolerance range for a feature dimensioned 2.500 ± .010.
10. 10. What is the decimal equivalent of 3/16 inches?
11. 11. A face mill cuts 4 slots each .750 in long. What is the total length cut?
12. 12. Convert 0.060 into shop lingo.
13. 13. A part's feature is 0.995 in a spec of 1.000 ±.010. Is it in tolerance?
14. 14. The spindle speed is set to 4000 RPM. What G-code is used to set this speed?
15. 15. What is the cosine of 60° used for in right-triangle cutting applications?
16. 16. A tool moves from (0, 0) to (6, 8). What is the total tool travel distance?
17. 17. What does the G90 code represent?
18. 18. A machinist sets up a part with a 2.0 inch stick-out from the collet. What risk does this present during turning?
19. 19. What would the feed per revolution be if G95 is used with F.008?
20. 20. A slot is 1.000 in long, and the tolerance is ±0.005. What are the min and max lengths allowed?
21. 21. What is the result of running G00 Z2.0 after finishing a drilling cycle?
22. 22. A taper cuts 0.5 inches in Z over a 6-inch part. What angle does this create?
23. 23. A drill point must land exactly at (X3.0, Y4.0). What is the radial distance from the origin?
24. 24. What does G43 H02 Z.1 mean in a milling program?
25. 25. A machinist uses a G28 command. What happens?

26. 26. A part diameter is measured at 0.998 with a tolerance of ±0.003. Is it within spec?
27. 27. What is the tangent of 45° and how is it used in machining?
28. 28. A bore is specified to be 1.500 ± .002. The part measures 1.502. Is this acceptable?
29. 29. A cut is made at 30° to the X-axis. If the X travel is 10 inches, how far in Z does the cutter move?
30. 30. What safety risk is associated with improper coolant flow during part cutting?