TITAN-502M RC Advanced Blueprint Reading & GD&T; Quiz

Name: _____ Date: _____

Section 1 – General Blueprint Reading (20 Questions)

1. What material is specified for the TITAN-502M part?

2. What is the general dimensional tolerance for one-place decimals (X = ___)?

3. What is the general dimensional tolerance for three-place decimals (XXX = ___)?

4. What is the angular tolerance listed on the drawing?

5. What surface-finish requirement is specified in micro-inches?

6. According to the notes, what should be done to all sharp edges?

7. What is the maximum allowable fillet radius unless otherwise specified?

8. What is the scale of the main drawing on Sheet 1?

9. Who is listed as the drafter of this part?

10. What is the drawing number and revision?

11. What is the overall length of the part?

12. How wide is the part at its largest width?

13. Identify the nominal diameter of the two holes called out as .6250 ± .0005.

14. How many holes are dimensioned .742 +0/-.003 in diameter?

15. What does the “2X” prefix before a dimension indicate?

16. In Section A-A, what is the tolerance zone of the feature controlled by .002 M A B?

17. What type of thread is specified on one of the holes (include size and thread form)?

18. Which section view (A-A or B-B) shows the 1/2-20 UNF threaded hole?

19. What does the note “THIS IS A MINIMAL DIMENSION DRAWING. THE CAD MODEL IS THE MASTER…” mean for manufacturing or inspection?

20. What alloy temper designation follows the base material 6061, and what does it represent?

Section 2 – Advanced GD&T; Interpretation (10 Questions)

21. The callout .002 M A B appears on the drawing. (a) What type of geometric tolerance is this? (b) What does the M modifier signify in this context?

22. When a feature control frame references A, B, and C, what does that tell you about the order of part setup during inspection?

23. The note .002 A B C appears on another section. How does this differ from .002 M A B, and what does the absence of the “M” modifier mean for measurement?

24. If the primary datum A is a large flat surface, the secondary B is an edge, and C is a hole centerline, describe how the part should be oriented and constrained in a CMM or inspection setup.

25. Why might Titan of CNC specify .002 M A B C instead of simply .002 A B C for this part? What advantage does MMC give for functional assembly?

26. The note 'ALL SURFACES APPLY .030 A B C UNLESS OTHERWISE TOLERANCED' appears in the drawing. Explain what this means for parts that do not have a specific geometric tolerance applied.

27. Given that .002 M A B controls a cylindrical feature, what type of geometric tolerance symbol (e.g., position, concentricity, runout) most likely applies here — and why?

28. The section view includes several holes of different diameters. If two holes share the same positional tolerance .002 M A B, how should they be measured to ensure compliance?

29. The drawing includes datums A, B, and C. Explain how datum C might function as a tertiary reference in the part’s coordinate system, and what happens if datum C’s hole is slightly out of location.

30. Suppose a machinist measures a hole’s true position at .0022 relative to A, B, and C. If the tolerance is .002 M A B C, and the hole is at its least material condition, does the part pass inspection? Explain your reasoning.