TITAN-502M RC Blueprint Reading & GD&T; Quiz – Answer Key & Explanations

Section 1 – General Blueprint Reading Answers

1. Material: 6061-T6 Aluminum – a lightweight, strong, heat-treated alloy used for machining.

2. One-place decimal tolerance (X): ±.030.

3. Three-place decimal tolerance (XXX): ±.005.

4. Angular tolerance: ±1°.

5. Surface finish: 63 microinches (Ra).

6. All edges should be broken by .010 to remove burrs.

7. Maximum fillet radius: .020 unless otherwise specified.

8. Scale of main drawing (Sheet 1): 2:3.

9. Drafter: W. Boyce.

10. Drawing Number: T502M; Revision: RC.

11. Overall length: 16.006 inches.

12. Maximum width: approximately 5.045 inches.

13. Hole diameter: .6250 ± .0005.

14. Two holes dimensioned .742 +0/-.003.

15. “2X” means there are two identical features of that dimension.

16. .002 M A B controls the position of the feature relative to datums A and B under MMC.

17. Thread callout: 1/2-20 UNF through one side.

18. Section B-B shows the 1/2-20 UNF threaded hole.

19. The CAD model is the master reference—use it for any dimensions not shown on the print.

20. The 'T6' temper indicates solution heat-treated and artificially aged for strength.

Section 2 – Advanced GD&T; Interpretation Answers

21. This is a **positional tolerance**. The 'M' stands for Maximum Material Condition (MMC), meaning the tolerance zone may increase as the feature departs from MMC.

22. Datums A, B, and C indicate the setup order: A (primary) establishes a plane, B (secondary) aligns one edge, and C (tertiary) locates the final orientation or rotation.

23. Without 'M', the tolerance applies at Regardless of Feature Size (RFS), meaning no bonus tolerance; the feature must always stay within the stated .002 zone.

24. The part sits flat on Datum A, aligns edge B parallel/perpendicular to the coordinate axis, and uses the hole (C) to lock rotation or final position.

25. MMC allows bonus tolerance—this ensures assembly fit while making inspection easier and reducing scrap from minor hole-size variations.

26. This means that unless a tighter geometric control exists, all surfaces must lie within .030 relative to the A, B, and C datum reference frame.

27. Most likely a **position tolerance** controlling the location of a hole feature because of the .002 M A B callout and its association with cylindrical geometry.

28. Use a CMM or functional gage referencing datums A and B. Both holes must be within .002 true position relative to those datums at MMC.

29. Datum C acts as the tertiary control (rotation). If it's slightly off, the part's rotation may deviate, affecting hole alignment but not height or width alignment.

30. Yes, the part passes if the hole is at its least material condition since bonus tolerance adds to the .002 positional zone. The actual value (.0022) can still fall within the total allowed tolerance.