INSTRUCTIONS
Answer the questions with short, complete statements or drawings as needed.

QUESTIONS
1. List the five basic types of dimensioning and geometric tolerancing symbols.
2. Name the five types of geometric characteristic symbols.
3. Name each of the following geometric characteristic symbols.
4. Any letter of the alphabet can be used to identify a datum except for __________, __________, and __________.
5. What information is placed in the lower half of the datum target symbol?
6. What information is placed in the top half of the datum target symbol?
7. Given the following symbols, provide the meaning of each symbol in the spaces to the right.

8. Name each of the elements in the following feature control frame.

9. Label the symbol in the following application using the blank provided.

10. Completely define the term basic dimension.
11. How are basic dimensions shown on a drawing?

12. Name the following symbol and identify the proper drafting dimensions B through F and put symbol name at A.

13. Name the following symbols and identify the proper drafting dimensions and features C through G and put symbol name at A and B.

14. Name the following symbols.

15. Name the following symbol and identify the proper drafting dimensions A and B.


17. Define datum feature.

18. Describe datum feature simulators. Include the term simulated datums in your description and give at least three examples of datum feature simulators used in manufacturing.

19. Identify at least five locations where a feature control frame can be placed on a drawing.

20. Define datum plane.

21. List at least five items that can be considered as datum features on an object or part.

22. Identify the datum feature, the part, the simulated datum plane, the physical datum feature simulator, and the datum plane labeled A through F on the following illustration.

23. Identify at least three required conditions for datum feature simulators.

24. Define actual mating envelope.

25. Define tangent plane.
26. Name the three datums of a complete datum reference frame.

27. When referring to the datum reference frame in the feature control frame, the ____________ datum is given first followed by the ____________ and ____________ datums. This is known as the datum ____________.


29. Define datum targets.

30. The primary datum plane must be established by at least ________ point(s) on the primary datum surface.

31. The secondary datum plane must be established by at least ________ point(s) on the secondary datum surface.

32. The tertiary datum plane must be established by at least ________ point(s) on the tertiary datum surface.

33. How is a datum target area represented on a drawing?

34. How are datum target areas treated on a drawing when the target area is too small to draw?

35. Describe how to properly display the symbols for a circular datum target area, a square datum target area, a rectangular datum target area, and a spherical datum target area.

36. What does a movable datum target symbol indicate?

37. How are datum target lines represented on a drawing?

38. When a portion of a surface is used to establish a single datum, this is referred to as a(n) ________ datum surface.

39. Two or more surfaces that are on the same plane are referred to as ________ surfaces.

40. Depending on the functional requirements of a part, more than one datum reference frame can be established. This is referred to as a(n) ________ datum reference frame.

41. Describe the basic function of the continuous feature symbol.

42. Define perfect form boundary.

43. Define regardless of feature size (RFS).

44. How is a feature control frame connected to a related feature when surface control is intended?

45. Given the following drawing and a list of possible produced sizes, specify the geometric tolerance at each possible produced size.

<table>
<thead>
<tr>
<th>Possible Produced Sizes</th>
<th>Geometric Tolerance at Given Produced Sizes</th>
</tr>
</thead>
<tbody>
<tr>
<td>Ø6 ± 0.4</td>
<td>6.4 MMC</td>
</tr>
<tr>
<td></td>
<td>6.3</td>
</tr>
<tr>
<td></td>
<td>6.2</td>
</tr>
<tr>
<td></td>
<td>6.1</td>
</tr>
<tr>
<td></td>
<td>6.0</td>
</tr>
<tr>
<td></td>
<td>5.9</td>
</tr>
<tr>
<td></td>
<td>5.8</td>
</tr>
<tr>
<td></td>
<td>5.7</td>
</tr>
<tr>
<td></td>
<td>5.6 LMC</td>
</tr>
</tbody>
</table>

46. How is an axis geometric control specified?

47. Given the following drawing and a list of possible produced sizes, specify the geometric tolerance at each possible produced size.

<table>
<thead>
<tr>
<th>Possible Produced Sizes</th>
<th>Ø Geometric Tolerance at Given Produced Sizes</th>
</tr>
</thead>
<tbody>
<tr>
<td>Ø6 ± 0.5</td>
<td>6.5 MMC</td>
</tr>
<tr>
<td></td>
<td>6.4</td>
</tr>
<tr>
<td></td>
<td>6.2</td>
</tr>
<tr>
<td></td>
<td>6.0</td>
</tr>
<tr>
<td></td>
<td>5.8</td>
</tr>
<tr>
<td></td>
<td>5.6</td>
</tr>
<tr>
<td></td>
<td>5.5 LMC</td>
</tr>
</tbody>
</table>

48. Given the following drawing and a list of possible produced sizes, specify the geometric tolerance at each possible produced size.

<table>
<thead>
<tr>
<th>Possible Produced Sizes</th>
<th>Ø Geometric Tolerance at Given Produced Sizes</th>
</tr>
</thead>
<tbody>
<tr>
<td>Ø6 ± 0.4</td>
<td>6.5 MMC</td>
</tr>
<tr>
<td></td>
<td>6.4</td>
</tr>
<tr>
<td></td>
<td>6.2</td>
</tr>
<tr>
<td></td>
<td>6.0</td>
</tr>
<tr>
<td></td>
<td>5.8</td>
</tr>
<tr>
<td></td>
<td>5.6</td>
</tr>
<tr>
<td></td>
<td>5.5 LMC</td>
</tr>
</tbody>
</table>
### Possible Produced Sizes | Ø Geometric Tolerance at Given Produced Sizes
---|---
6.4 MMC |  
6.2 |  
6.0 |  
5.8 |  
5.6 LMC |  

49. Give the proper abbreviation and definition for regardless of material boundary.

50. Given the following drawing and a list of possible produced sizes, specify the geometric tolerance at each possible produced size.

![THE DRAWING]

| Possible Produced Sizes | Ø Geometric Tolerance at Given Produced Sizes |
---|---|
11.8 MMC |  
11.9 |  
12.0 |  
12.1 |  
12.2 |  
12.3 |  
12.4 LMC |  

51. Which of the following statements are true in regard to datum precedence and datum reference? (More than one can be true.)

A) Datum precedence is established by the order of placement in the feature control frame.

B) Datum precedence is established by alphabetical order of datum reference letters.

C) The first datum listed in the feature control frame is the primary datum reference.

D) "A" is always the primary datum.

E) The third datum listed in the feature control frame is the tertiary datum reference.

F) RMB is assumed unless otherwise specified.

52. Define form tolerances.

53. Name the geometric tolerance that specifies a zone within which the required surface element or axis must lie.

54. Explain the difference between the methods used to represent surface and axis straightness.

55. Axis straightness can be specified on an MMC basis by placing the MMC symbol after the geometric tolerance in the feature control frame. The specified geometric tolerance is held at the MMC produced size. Explain what happens to the geometric tolerance as the produced size departs from MMC.

56. What geometric tolerance establishes the distance between two parallel planes within which the surface must lie?

57. Which geometric tolerance is characterized by any given cross section taken perpendicular to the axis of a cylinder or cone or through the common center of a sphere?

58. What is the difference between the circularity geometric tolerance and the cylindricity geometric tolerance?

59. Define free state variation.

60. Define restrained condition.

61. Define parallelism.

62. What does it mean when a feature control frame with a parallelism geometric characteristic symbol is placed below a diameter dimension?

63. Define tangent plane and describe the relationship among the actual surface, the tangent plane, and the geometric tolerance.

64. Define radial element and describe how a radial element specification is applied to a drawing.

65. Define perpendicularity.

66. A ________ tolerance is established by a geometric tolerance zone made up of two parallel planes that are a basic 90° to a given datum plane or axis where the actual surface must lie.

67. A(n) ________ geometric tolerance zone is established by two parallel planes at any specified basic angle, other than 90°, to a datum plane or axis. The specified angle must be ________, and it must be dimensioned from the ________ plane.

68. Given the following drawing, a reference chart showing a range of possible produced sizes, and three optional feature control frames that can be applied to the diameter...
dimension, provide the geometric tolerance at each possible produced size for each feature control frame application.

79. Give the formulas for calculating the slot boundary when applying positional tolerancing to slotted features.
80. How is the feature control frame placed in relation to a hole and counterbore when the positional tolerance is the same for the hole and counterbore?

81. How is the feature control frame placed in relation to a hole and counterbore when the positional tolerance is different for the hole and counterbore?

82. What is the difference between the appearance of the feature control frame used for a composite positional tolerance and the one used for a positional tolerance with two single segments?

83. Explain the primary difference between the composite positional tolerance and the two single-segment positional tolerance as applied to circular patterns.

84. Define virtual condition.

85. Give the formulas for calculating virtual condition for internal and external features.
   Internal feature:
   External feature:

86. Describe how a thread note looks when a location tolerance applied to the axis of the cylinder is established by the pitch diameter, and how this compares to applications when a location tolerance applied to the axis of the cylinder is established by the major or minor diameter.

87. Give the formula used to determine the positional tolerance of a floating fastener.

88. Give the formula used to determine the positional tolerance of a fixed fastener.

89. Under which condition(s) is a projected tolerance zone recommended?

90. Identify the two ways that a projected tolerance zone can be shown on a drawing.

91. Given the following drawing and a range of possible produced sizes, provide the geometric tolerance and virtual condition at each possible produced size.

92. Define concentricity.

93. Describe the symmetry geometric tolerance.

94. Calculate the virtual condition of a hole through a part where the hole diameter is \( \varnothing 14.5 \pm 0.3 \) and the associated positional tolerance is \( \varnothing 0.1 \) at MMC. Show your calculations.

95. Calculate the virtual condition of a pin that extends 15 mm above the primary datum of a part where the pin diameter is \( \varnothing 14.5 \pm 0.3 \) and the associated perpendicularity tolerance is \( \varnothing 0.1 \) at MMC. Show your calculations.

96. Name the two types of profile geometric tolerances.

97. Given the following drawing, describe the profile geometric tolerance and related drawing specifications. Indicate whether the drawing shows preferred ASME Y14.5 use or an alternate practice.
98. Given the following drawing, describe the profile geometric tolerance and related drawing specifications. Indicate whether the drawing shows preferred ASME Y14.5 use or an alternate practice.

NOTE: ASSUME THE SHAPE BETWEEN M AND N IS DEFINED WITH BASIC DIMENSIONS RELATIVE TO DATUM B.

99. Given the following drawing, describe the profile geometric tolerance and related drawing specifications. Indicate whether the drawing shows preferred ASME Y14.5 use or an alternate practice.

NOTE: ASSUME THE SHAPE BETWEEN M AND N IS DEFINED WITH BASIC DIMENSIONS RELATIVE TO DATUM B.

100. Define runout.
CHAPTER 13 GEOMETRIC DIMENSIONING AND TOLERANCING PROBLEMS

INSTRUCTIONS

From the selected problems, determine which views and dimensions should be used to detail the part completely. Include all dimensions needed using unidirectional dimensioning.

1. Make a multiview sketch of the selected problem as close to correct proportions as possible. Be sure to indicate where you intend to place the dimension lines, extension lines, arrowheads, geometric tolerancing symbols, and hidden features to help you determine the layout for your final drawing.

2. Using your sketch as a guide, make an original multiview drawing on an adequate size ASME-standard drawing sheet with border and sheet blocks. Use an appropriate scale.

3. When using solid modeling software, create the solid model using the given geometry and confirm the accuracy of the given engineering information as you proceed. Consult with your instructor or supervisor if you discover problems with the geometry and revise the drawings as needed to make the geometry accurate. Use your completed part solid models to develop fully dimensioned 2-D detail drawings. Place a 3-D model in the upper left corner of the drawing for use as a visualization aid.

4. Include the following general notes at the lower left corner of the sheet .5 in. each way from the corner border lines:

NOTES:

1. DIMENSIONS AND TOLERANCES PER ASME Y14.5-2009.
2. REMOVE ALL BURRS AND SHARP EDGES.

Additional general notes can be required, depending on the specifications of each individual assignment. Use the following for tolerances for unspecified inch values. A tolerance block is recommended as described in Chapter 2.

<table>
<thead>
<tr>
<th>Unspecified Tolerances</th>
<th>Decimals</th>
<th>In.</th>
</tr>
</thead>
<tbody>
<tr>
<td>X</td>
<td>± .1</td>
<td></td>
</tr>
<tr>
<td>XX</td>
<td>± .01</td>
<td></td>
</tr>
<tr>
<td>XXX</td>
<td>± .005</td>
<td></td>
</tr>
<tr>
<td>ANGULAR</td>
<td>± 30</td>
<td></td>
</tr>
<tr>
<td>FINISH</td>
<td>125 µin.</td>
<td></td>
</tr>
</tbody>
</table>

For metric drawings, provide a general note that states TOLERANCES FOR UNSPECIFIED DIMENSIONS COMPLY WITH ISO 2768-m. Provide a general note that states SURFACE FINISH 3.2 µm UNLESS OTHERWISE SPECIFIED.

Each problem assignment is given as an engineer’s layout to help simulate actual drafting conditions. Dimensions and views on engineers’ layouts may not be placed in accordance with acceptable standards. You need to review the chapter material carefully when preparing the layout sketch. In some problems, the engineer's layout assumes certain information, such as the symmetry of a part or the alignment of holes. You need to place enough dimensions or draw lines between features to dimension the part fully.
**Part 1: Problems 13.1 Through 13.15**

**PROBLEM 13.1** Geometric tolerancing (metric)

Part Name: Flow Pin
Material: Bronze
Finish: Finish all over 0.20 mm.

**PROBLEM 13.2** Geometric tolerancing (metric)

Part Name: LN2 Test Pump Lock Nut
Material: AMS 5732
Additional General Notes: Mark per AS478 Class D with 1193125 and applicable dash number.
Finish all over 1.6 µm.

**PROBLEM 13.3** (in.)

Part Name: Half Coupling
Material: 1.250 6061-T6 aluminum
SPECIFIC INSTRUCTIONS: Provide MMC material condition after position tolerance except for RFS at threads.
*Problem based on original art courtesy TEMCO.*
**PROBLEM 13.4** (in.)  
Part Name: Coupling  
Material: AISI 1010, Killed  
SPECIFIC INSTRUCTIONS: Provide MMC material condition after position tolerance except for RFS at threads.  
*Problem based on original art courtesy TEMCO.*

**PROBLEM 13.5** (in.)  
Part Name: Half Coupling  
Material: Ø 1.625 6061-T6511  
*Problem based on original art courtesy TEMCO.*

**PROBLEM 13.6** (metric)  
Part Name: Spline Plate  
Material: SAE 3135

**PROBLEM 13.7** (in.)  
Part Name: Nut  
Material: No. 10 Bronze
**PROBLEM 13.8 (metric)**
Part Name: Coupling Bracket
Material: SAE 4310 Steel

**PROBLEM 13.9 (in.)**
Part Name: Thrust Washer
Material: SAE 5150

**PROBLEM 13.10 (metric)**
Part Name: Spacer
Material: SAE 4310

**PROBLEM 13.11 (metric)**
Part Name: Bearing Support
Material: SAE 1040
**PROBLEM 13.12** (metric)
Part Name: Lock Nut
Material: SAE 3130

**PROBLEM 13.13** (in.)
Part Name: Cover Plate
Material: Phosphor bronze

**PROBLEM 13.14** (in.)
Part Name: Angle Support Mounting
Material: SAE 3110

**PROBLEM 13.15** (metric)
Part Name: Hub
Material: SAE 3310
**Part 2: Problems 13.16 Through 13.23**

**PROBLEM 13.16** Geometric tolerancing (metric)

Part Name: Fixture MIBRDA—1265  
Material: SAE 4320  
Harden: Brinell 200–240  
Additional General Notes  
1. Finish all over 0.80 μm.

**PROBLEM 13.17** Geometric tolerancing (metric)

Part Name: Mounting Bracket  
Material: Stainless steel  
Additional General Notes:  
1. All fillets and rounds R24.  
2. Finish all over 1.6 μm.
**PROBLEM 13.18** Geometric tolerancing (metric)

Part Name: Oscillator Housing  
Material: Phosphor bronze  
Additional General Notes:  
1. Finish all over 0.80 μm.

**SPECIFIC INSTRUCTIONS:**
Use the following engineer’s notes to complete the geometric tolerancing of the Oscillator Housing:

- Establish datum A with three equally spaced datum target points at each end of the Ø28.1-28.0 cylinder.
- Establish datum B at the left end surface.
- Make the bottom surfaces of the 2X Ø40.2–40.0 features perpendicular to datum A by 0.06.
- Provide a cylindricity tolerance of 0.3 to the outside of the part.
- Make the 2X Ø40.2–40.0 features concentric to datum A by 0.1.
- Locate the 6X Ø6 holes with reference to datum A at MMC and datum B with a position tolerance of 0.05 at MMC.
- Locate the 4X Ø4 holes with reference to datum A at MMC and datum B with a position tolerance of 0.04 at MMC.

**PROBLEM 13.19** (in.)

Part Name: Hub  
Material: SAE 4320

**PROBLEM 13.20** (in.)

Part Name: Lock Spacer  
Material: SAE 1030
**PROBLEM 13.21** (metric)

Part Name: Mounting Plate

Material: SAE 4140

![Diagram of Mounting Plate]

**PROBLEM 13.22** Geometric tolerancing (metric)

Part Name: Side Panel Mounting Plate

Material: SAE 30308

![Diagram of Side Panel Mounting Plate]
**Problem 13.23** Advanced geometric tolerancing from actual industry drawing (metric)

Part Name: Pinion Gear Shaft

Material: CRES 15-5PH ASTM A564

Additional General Notes:

1. Finish all over 1.6 µm.
2. Heat treat per Mil-H-6875 to H1100 condition.
3. Penetrant inspect finished part per Mil-Std-271, Group III. No evidence of linear indications permitted.
4. Part to be clean and free of foreign debris.

*Problem based on original art courtesy Aerojet TechSystems Co.*
**INSTRUCTIONS**

Given the engineering layout below, design and detail the plate, angle, and yoke per ASME Y14.5-2009 standards. Use arrowless dimensioning unless otherwise specified by your instructor. Use the following information and specifications:

- Not all features are shown in all views.
- Some features that are hidden from view are not drawn with hidden lines.
- Item 4 (Pin) is a purchase part. Do not detail this part.

**DESIGN PROBLEM**

**Part 3: Problem 13.24**

**PROBLEM 13.24** *Problem courtesy of Martin Soll.*

---

**MATERIAL LIST**

<table>
<thead>
<tr>
<th>PART NO.</th>
<th>TITLE</th>
<th>QTY</th>
<th>TOLERANCES UNLESS NOTED</th>
</tr>
</thead>
<tbody>
<tr>
<td>15982-A</td>
<td>TRUNNION ASSEMBLY</td>
<td></td>
<td></td>
</tr>
</tbody>
</table>

---

**INSTRUCTIONS**

- Pin specifications: $\phi 2.0000 \pm .0001$
  Straightness within .0002 over full length.
  Clearance fit in Angle = .005/.007
  Clearance fit in Yoke = .003/.005
- Dowel pin specifications: $\phi .5000 \pm .0005$
  Interference fit in Plate = .001/.003
  Clearance fit in Angle and Yoke = .002/.005
- Be sure the pin will slide smoothly through the three holes,
  and the 3.000.020/.000 spacing is maintained.

---

Copyright © 2017 Cengage Learning®. All Rights Reserved. May not be scanned, copied, or duplicated, or posted to a publicly accessible website, in whole or in part.
MATH PROBLEMS
Part 4: Problems 13.25 Through 13.34

Round to the nearest tenth.

PROBLEM 13.25 4.849
PROBLEM 13.26 3.650
PROBLEM 13.27 .275
PROBLEM 13.28 5.249

Round to the nearest hundredth.

PROBLEM 13.29 4.849
PROBLEM 13.30 7.0574
PROBLEM 13.31 .27499

Perform the following calculations, rounding appropriately.

PROBLEM 13.32 3.7 + 4.19 + 8.00004
PROBLEM 13.33 7.8 × 6.3 × 5.29

PROBLEM 13.34 Find C using the formula C = πD for D = 6542 cm.