

Geometric Dimensioning and Tolerancing for Mechanical Design

Answer Guide

(Answers to the questions and problems at the end of each chapter)

Chapter 1

Introduction to Geometric Dimensioning and Tolerancing

Chapter Review

Page 7

1. Geometric Dimensioning and Tolerancing is a symbolic language used to specify

the <u>size</u>, <u>shape</u>, <u>form</u>, <u>orientation</u>, and <u>location</u> of features on a part.

2. Features toleranced with GD&T reflect the <u>actual relationship</u> between mating parts.

- 3. Geometric Dimensioning and Tolerancing was designed to insure the proper assembly of <u>mating parts</u>, to improve <u>quality</u>, and reduce <u>COSt</u>.
- 4. Geometric tolerancing allows the maximum available <u>tolerance</u> and, consequently, the most <u>economical</u> parts.
- 5. <u>ASME Y14.5–2009</u> is the current, authoritative reference document that specifies the proper application of GD&T.
- 6. Plus or minus tolerancing generates a <u>rectangular</u> -shaped tolerance zone.
- 7. $\underline{GD\&T}$ generates a cylindrical-shaped tolerance zone to control an axis.
- 8. If the distance across a square tolerance zone is \pm .005 or a total of .010, what is the approximate distance across the diagonal? $\pm .007 \text{ or } .014$
- 9. Bonus tolerance equals the difference between the actual mating envelope size and the

maximum material condition

10. While processing, a rectangular part usually rests against a <u>datum</u>

<u>reference frame</u> consisting of three mutually perpendicular planes.

Chapter 2

Dimensioning and Tolerancing

Fundamentals

Chapter Review Page 15

1.	Each dimension shall have a <u>tolerance</u> identified as reference, maximum, minimum, o	_except those dimensions r stock.	specifically
2.	Dimensioning and tolerancing must be	complete	so there is a full
	understanding of the charge	acteristics	of each feature.
3.	Dimensions shall not be subject to more than o	ne <u>interpretat</u>	tion
4.	The drawing should <u>define</u>	the pa	rt without specifying
	manufacturing		methods.
5.	A <u>90° angle</u> depicting features are shown on a 2D orthograp specified.	applies where co phic drawing at right angle	enter lines and lines les and no angle is
6.	<u>A basic 90° angle</u> applies where surfaces shown at right angles on a 2D orthogra dimensions and no angle is specified.	centerlines of features in aphic drawing are located	n a pattern or I or defined by basic
7.	All dimensions and tolerances are applicable at specified. Measurements made at other temperatives	<u>68°F (20°C)</u> atures may be adjusted m	unless otherwise athematically.
8.	All dimensions and tolerances apply in the	free state cond	dition
	except for non-rigid parts.		
9.	All tolerances apply for the full depth	h	
	, and, and,	full width	of the feature
10.	Dimensions and tolerances apply only at the	drawing level	
	where they are specified.		
11.	Units of linear measurement are typically expre	essed either in the <u>inc</u>	<u>h</u> system or the
Geom	etric Dimensioning and Tolerancing fo	r Mechanical Design	Answer Guide

<u>metric</u>_____system.

- 12. For decimal inch tolerances, a <u>*Zero*</u> is never placed before the decimal point for values less than one inch.
- 13. For decimal inch tolerances, a dimension is specified with the same number of decimal places as its <u>tolerance</u>.
- 14. What are the two types of direct tolerancing methods?

Limit dimensioning and plus and minus dimensioning

15. For decimal inch tolerances, where a unilateral tolerance is specified and either the plus or minus limit is zero, its zero value will have <u>the same number of decimal</u>

places

as the other limit and the appropriate ______ *plus and minus sign*_____.

- 16. For decimal inch tolerances, where bilateral tolerancing or limit dimensioning and tolerancing is used, both values <u>have the same number of decimal places</u>
- 17. Dimensional limits are used as if <u>an infinite number of zeros</u> followed the last digit after the decimal point.
- 18. Angular units of measurement are specified either in <u>degrees and decimal parts</u> of <u>a degree</u> or <u>degrees, minutes, and seconds</u>

19. What two dimensions are not placed on the field of the drawing?*The 90° angle and a zero*

20. If CAD/CAM database models are used and they do not include tolerances, tolerance values may be expressed in a <u>CAD product definition data set.</u>

Chapter 3

Symbols, Terms, and Rules

Chapter Review

Page 40

- 1. What type of geometric tolerance has no datum features? <u>Form controls</u>
- 2. Which of the form tolerances is the most common? <u>Flatness</u>
- 4. What is the name of the symbol that is used to identify physical features of a part as datum features and should not be applied to centerlines, center planes, or axes?

Datum feature symbol

- 5. Datum feature identifying letters may be any letter of the alphabet except? <u>I, O, & O</u>
- 6. If the datum feature symbol is placed in line with a dimension line or on a feature control frame associated with a feature of size, then the datum feature is what is what kind of feature?

A feature of size

- One of the 14 geometric characteristic symbols always appears in the <u>first</u> compartment of the feature control frame.
- 8. The second compartment of the feature control frame is the <u>tolerance</u> section.
- 9. The tolerance is preceded by a diameter symbol only if the tolerance zone is <u>cylindrical</u>.
- 10. Datum features are arranged in order of <u>precedence or importance</u>.
- 11. Read the feature control frame in Figure 3-35 and write it below.

The position tolerance requires that

The axis of the controlled feature

Must lie within a cylindrical tolerance zone

.010 in diameter

At maximum material condition (MMC)

Oriented and located with basic dimensions to a datum reference

frame established by datum feature A and datum features B and C at their maximum material boundaries (MMB)

- 12. The all around and between symbols are used with what control? <u>*Profile</u></u></u>*
- 13. The all over symbol consists of two small <u>concentric circles</u> placed at the joint of the leader connecting the feature control frame to the feature.
- 14. The <u>continuous feature</u> symbol specifies that a group of two or more interrupted features of size are to be considered one single feature of size.
- If no depth or remaining thickness is specified, the spotface is the <u>minimum</u> depth necessary to clean up the surface of the specified diameter.
- 16. The <u>independency</u> symbol indicates that perfect form of a feature of size at MMC or LMC is not required.
- 17. The <u>unequally disposed profile</u> symbol indicates that the profile tolerance is unilateral or unequally disposed about the true profile.
- 18. The <u>datum translation</u> symbol indicates that a datum feature simulator is not fixed and is free to translate within the specified geometric tolerance.
- 19. The <u>actual mating envelope</u>

is a similar, perfect, feature(s) counterpart of smallest size that can be contracted about an external feature(s) or largest size that can be expanded within an internal feature(s) so that it coincides with the surface(s) at the highest points.

- 20. A theoretically exact dimension is called? <u>*a basic dimension*</u>
- 21. What is the theoretically exact point, axis, line, plane, or combination thereof derived from the theoretical datum feature simulator called? <u>*a datum*</u>
- 22. A <u>datum feature</u> is a feature that is identified with either a datum feature symbol or a datum target symbol.
- 23. A <u>datum feature simulator</u> (Physical) is the physical boundary used to establish a simulated datum from a specified datum feature.
- 24. A <u>datum reference frame</u> consists of three mutually perpendicular intersecting datum planes.
- 25. What is the name of a physical portion of a part, such as a surface, pin, hole, tab, or slot?

A feature

26. A regular feature of size is a feature that is associated with a directly toleranced dimension Geometric Dimensioning and Tolerancing for Mechanical Design Answer Guide and takes one of the following forms:

- a. <u>A cylindrical surface</u>
- b. <u>A set of two opposed parallel surfaces</u>
- c. <u>A spherical surface</u>
- d. <u>A circular element</u>
- e. <u>A set of two opposed parallel elements</u>

- 27. What is a feature of size with the maximum amount of material within the stated limits of size called? <u>maximum material condition (MMC)</u> _____
- 28. What is a feature of size with the least amount of material within the stated limits of size called? <u>least material condition (LMC)</u>
- 29. What kind of feature always applies at MMC/MMB, LMC/LMB, or RFS/RMB?

a feature of size or a datum feature of size

30. The maximum material condition modifier specifies that the tolerance applies at the *maximum material condition (MMC)* size of the feature.

Pertains to	Type of Tolerance	Geometric Characteristics	Symbol
		STRAIGHTNESS	
Individual Feature		FLATNESS	
Only	Form	CIRCULARITY	0
		CYLINDRICITY	$\not\!$
Individual Feature or		PROFILE OF A LINE	\cap
Related Features	Profile	PROFILE OF A SURFACE	\Box
		ANGULARITY	~
	Orientation	PERPENDICULARITY	T
		PARALLELISM	//
Related Features		POSITION	\
	Location	CONCENTRICITY	Ô
		SYMMETRY	=
	Runout	CIRCULAR RUNOUT	A
		TOTAL RUNOUT	11

Figure 3-36 Geometric characteristic symbols.

31. Write the names and geometric characteristic symbols where indicated in Fig. 3-36.

Name	Symbol	Name	Symbol
All Around		Free State	F
Between)	Projected Tolerance Zone	P
Number of Places	Х	Tangent Plane	T
Counterbore/Spotface	\$	Radius	r
Countersink	%	Radius, Controlled	С
Depth/Deep	^	Spherical Radius	У
Diameter	Ø	Spherical Diameter	Z
Dimension, Basic	1.000	Square	&
Dimension, Reference	(60)	Statistical Tolerance	S
Dimension Origin	!	Datum Target	Ø.500 A1

Arc Length	110	Target Point	
eometric Dimensioning	and Iolerancin	g for Mechanical Design	Answer Guide

Conical Taper	0	Slope	#

Figure 3-37 Geometric tolerancing symbols.

32. Draw the indicated geometric tolerancing symbols in the spaces on Figure 3-37.

- 33. The MMC modifier indicates that the tolerance applies at the maximum material condition size of the feature and that a <u>bonus</u> tolerance is available as the size of the feature departs from MMC toward LMC.
- 34. The bonus tolerance equals the difference between the <u>actual mating envelope</u> size and MMC
- 35. The total positional tolerance equals the sum of the <u>bonus</u> tolerance and the <u>geometric tolerance</u> tolerance.
- 36. What is the term used to indicate that a specified geometric tolerance applies at each increment of size of a feature within its limits of size? <u>RFS</u>
- 37. MMB, LMB, and RMB all apply in a feature control frame for what kind of feature?

A datum feature of size

38. What is the single worst-case boundary generated by the collective effects of the LMC limit of size, the specified geometric tolerance, and the size tolerance called?

Resultant Condition

- 49. What is the theoretically exact location of a feature of size established by basic dimensions called? <u>True position</u>
- 40. What is the theoretically exact profile of a feature established by basic dimensions called?

True profile

- 41. Using the drawing in Fig. 3-38, complete Table 3-3.
- 42. Using the drawing in Fig. 3-38, complete Table 3-4.



Figure 3-38 Refer to this drawing for questions 43 through 48.

Actual Mating Envelope	MMC	Bonus	Geometric Tolerance	Total Positional Tolerance
MMC .515	.515	.000	.010	.010
.520	.515	.005	.010	.015
.525	.515	.010	.010	.020
.530	.515	.015	.010	.025
.535	.515	.020	.010	.030
LMC .540	.515	.025	.010	.035

Internal Feature (Hole)

 Table 3-3 Total positional tolerance for Holes

		Hole	Pin
43.	What is the MMC?	.515	.500
44.	What is the LMC?	.540	.495
45.	What is the geometric tolerance?	.010	.005
46.	What material condition modifier is specified?	ММС	ММС
47.	What datum feature(s) control(s) perpendicularity?	A	A

				_
48.	What datum feature(s) control(s) location?	<i>B</i> & <i>C</i>	<i>B</i> & <i>C</i>	
				_

External Feature (Pin)

Actual Feature			Geometric	Total Positional
Size	MMC	Bonus	Tolerance	Tolerance
MMC .500	.500	.000	.005	.005
.499	.500	.001	.005	.006
.498	.500	.002	.005	.007
.497	.500	.003	.005	.008
.496	.500	.004	.005	.009
LMC .495	.500	.005	.005	.010

Table 3-4 Total positional tolerance for Pins

49. The <u>virtual</u>

condition of a feature of size specified with a MMC modifier is a constant boundary generated by the collective effects of the considered feature's MMC limit of size and the specified geometric tolerance.

- 50. Where only a tolerance of size is specified, the limits of size of an individual feature prescribe the extent to which variations in its **geometric form**, as well as **size**, are allowed. This statement is the essence of Rule #l
- 51. The form tolerance increases as the actual size of the feature departs from <u>MMC</u>

toward <u>LMC</u>

52. If features on a drawing are shown coaxial, or symmetrical to each other and not controlled for

location or *orientation*, the drawing is incomplete.

53. If there is no orientation control specified for a rectangle on a drawing, the perpendicularity is controlled, not by the size *tolerance*, but by the

title block angularity tolerance tolerance.

- 54. Rule #2 states that <u>*RFS*</u> automatically applies, to individual tolerances of feature of sizes and <u>*RMB*</u> to datum features of size.
- 55. Each tolerance of orientation or position and datum reference specified for screw threads applies to the axis of the thread derived from the <u>pitch diameter</u>.
- 56. Each geometric tolerance or datum reference specified for gears and splines must designate the specific feature at which each applies such as

MAJOR DIA, PITCH DIA, or MINOR DIA

Problems

Page 46



Figure 3-39 Material condition symbols: problem 1.

- 1. Read the complete tolerance in each feature control frame in Fig. 3-39, and write them below (datum feature A is a feature of size).
 - A. <u>The position tolerance requires that</u> <u>The axis of the controlled feature</u> <u>Must lie within a cylindrical tolerance zone</u> <u>.005 in diameter</u> <u>At regardless of feature size (RFS)</u> <u>Oriented and located with basic dimensions to datum feature A at</u> <u>regardless of material boundary (RMB)</u>
 - B. <u>The position tolerance requires that</u>

The axis of the controlled feature

Must lie within a cylindrical tolerance zone

.005 in diameter

At maximum material condition (MMC)

Oriented and located with basic dimensions to datum feature A at maximum material boundary (MMB)



Figure 3-40 Definitions: problem 2.

2. Place each letter of the items on the drawing in Fig. 3-40 next to the most correct term below.

<u> </u>	_ Feature	G	_Basic Dimension	I	_Feature control frame
A	MMC	F	_ Datum Feature	<i>D</i>	_ True Position

Chapter 4

Datums

Chapter Review

Page 64

- 1. Datums are theoretically perfect ______ *points, axes, lines, and planes* ______
- 2. Datums establish the <u>*Origin*</u> from which the location or geometric characteristic of features of a part are established.
- 3. Datums exist within a structure of three mutually perpendicular intersecting datum planes known as a <u>datum reference frame</u>.
- 4. Datums are assumed to exist in and be simulated by the <u>processing equipment</u>.
- 5. A part is oriented and immobilized relative to the three mutually perpendicular intersecting datum planes of the datum reference frame in a selected order of <u>precedence</u>.
- 7. Datum features are specified in order of precedence as they appear from left to right in the *feature control frame*
- 8. Datum feature letters need not be in <u>alphabetical</u> order.
- 9. When selecting datum features, the designer should consider features that are:

Functional surfaces______.

Mating surfaces______.

Readily accessible surfaces______.

Surfaces that allow repeatable measurements

- 10. The primary datum feature controls ______ *the orientation of the part*
- 11. The datum feature symbol is used to identify <u>physical features</u> of a part as datum features.
- 12. Datum feature symbols shall **not** be applied to <u>centerlines, center planes, or</u> <u>axes</u>.
- 13. One method of tolerancing datum features at an angle to the datum reference frame is to place a datum feature symbol on the <u>inclined surface</u> and control

that surface with an angularity tolerance and a basic angle.

- 14. A cylindrical datum feature is always associated with two <u>theoretical planes</u> meeting at right angles at its datum axis.
- 15. The two kinds of features specified as datum features are:
 <u>Plane flat surfaces</u>
 <u>Features of size</u>
- 16. Datum features of sizes may apply at <u>*RMB*</u>, <u>*MMB*</u> or <u>*LMB*</u>
- 17. Where datum features of sizes are specified at RMB, the processing equipment must make

physical contact with the datum features.

18. Where features of sizes are specified at MMB, the size of the processing equipment has a *constant boundary*





- 19. The two-hole pattern is perpendicular to what datum feature? <u>Datum feature A</u>
- 20, The two-hole pattern is located to what datum feature? <u>Datum feature B</u>

21. If inspected with a gage, what is the datum feature B diameter of the gage? _	Ø6.030
Geometric Dimensioning and Tolerancing for Mechanical Design	Answer Guide

22.	If inspected with a gage.	what is the diameter of the 2	pins on the gage?	Ø.500

23. If datum feature B had been specified at RFS, explain how the gage would be different.

Datum feature B would have to have a variable diameter such as a chuck to make physical contact with the outside diameter.

24. If datum feature B had been specified as the primary datum feature at RFS, explain how the gage would be different.

Datum feature B would not only have to be a variable diameter, such as a chuck, to make physical contact with the outside diameter, but the outside diameter, datum feature B, would align with the gage as well.

25. If a datum feature symbol is in line with a dimension line, the datum feature is the

feature of size measured by the dimension.

26. Where more than one datum feature is used to establish a single datum, the _____

datum reference letters and appropriate *modifiers*

are separated by a dash and specified in one compartment of the feature control frame.

- 27. Where cylinders are specified as datum features, the entire surface of the feature is considered to be the <u>datum feature</u>
- 28. If only a part of a feature is required to be the datum feature, a <u>heavy chain line</u> is drawn adjacent to the surface profile and dimensioned with basic dimensions.
- 29. Datum targets may be used to immobilize parts with *uneven or irregular surfaces*.
- 30. Once datum targets are specified, costly manufacturing and inspection <u>tooling</u> is required to process the part.

Problems

Page 66



Figure 4-20 Datum features at MMB and RMB: problem 1.

1. Complete the feature control frames with datum features and material condition symbols to reflect the drawing in Fig. 4-20.



Figure 4-21 Specifying datum features and datum feature symbols: problem 2.

2. Provide the appropriate datum feature symbols and datum features in the feature control frames on the drawing in Fig. 4-21. (*Two solutions suggested.*)





3. Specify the appropriate datum feature symbols and datum features in the datum exercise in Fig. 4-22.

(One solution. Explore other possibilities.)



Figure 4-22 Specifying datum features and datum feature symbols: problem 4.

4. On Fig. 4-22, draw and dimension the gage used to inspect the part in problem 3.

Chapter 5

Form

Chapter Review

Page 82

- 1. Form tolerances are independent of all <u>other features</u>.
- 2. No <u>datum features</u> apply to form tolerances.
- 3. The form of individual features of size is automatically controlled by the size tolerance, rule #1
- 4. A form tolerance may be specified as a refinement where <u>the size tolerance</u> does not adequately control the form of a feature .
- 5. All form tolerances are surface controls except for <u>flatness of a median plane</u> <u>and straightness of a median line</u>.
- 6. No <u>cylindrical tolerance zones</u> or <u>material condition modifiers</u> are appropriate for surface controls.
- 7. Flatness of a surface or derived median plane is a condition where all line elements of that surface are in <u>one plane</u>
- 8. For flatness, in a view where the surface to be controlled appears as a <u>line</u>, a feature control frame is attached to the surface with a <u>leader or extension line</u>.
- 9. The feature control frame controlling flatness contains a <u>flatness symbol</u> and a <u>numerical tolerance</u>.
- 10. The surface being controlled for flatness must lie between <u>two parallel planes</u> separated by the flatness tolerance. In addition, the feature must fall within the <u>size tolerance</u>.
- 11. The flatness tolerance zone does not need to be <u>parallel</u> to any other surface.
- 12. The feature of size may not exceed the <u>boundary of perfect form at MMC</u>



Figure 5-14 Specifying flatness: question 13.

- 13. Specify the flatness of the top surface of the part in Fig. 5-14 within .006 in a feature control frame.
- 14. Draw a feature control frame with an overall flatness of .015 and a unit flatness of .001 per square inch.



- 15. When verifying flatness, the feature of size is first measured to verify that it falls within the *limits of size*
- 16. The surface is adjusted with jackscrews to remove any <u>parallelism</u> error.
- 17. Flatness verification is achieved by measuring the surface in all directions with a

<u>dial indicator</u>

- 18. Straightness is a condition where an element of a <u>surface, or derived median</u> is a straight line.
- 19. In a view where the line elements to be controlled appear as a <u>line</u>, a feature control frame is attached to the surface with a <u>leader or extension line</u>
- 20. Straightness tolerance is a refinement of the <u>size tolerance, rule #1</u>, and must be less than the <u>size tolerance</u>.

Actual Part Size	Straightness Tolerance	Controlled by
1.020	.000	
1.018	.002	Rule #1
1.016	.004	
1.014	.004	
1.010	.004	Straightness
1.005	.004	Tolerance

1.000	.004	
-------	------	--

Table 5-6 Question 21

- 21. Complete Table 5-6 specifying the straightness tolerance and what controls it for the drawing in Fig. 5-6.
- 22. The measurement of surface variation for straightness is performed similar to the measurement for

<u>flatness</u>

- 23. Each line element is <u>independent</u> of every other line element.
- 24. Where a feature control frame with a straightness tolerance is associated with a size dimension, the straightness tolerance applies to <u>the median line</u>.
- 25. While each actual local size of the feature must fall within the <u>size tolerance</u>, the feature controlled with straightness of a median line may exceed the

boundary of perfect form at maximum material condition.

- 26. A straightness control of a median line will allow the feature to violate $Rule \ #l$
- 27. If specified at MMC, the total straightness tolerance of a median line equals the tolerance in the feature control frame plus any <u>bonus tolerance</u>.

	Cylindrical Feature (Straightness of a Median Line)				
Feature Size	Ø.006	Ø.006@)			
1.020 MMC	.006	.006			
1.015	.006	.011			
1.010	.006	.016			
1.005	.006	.021			
1.000 LMC	.006	.026			

Table 5-7 Straightness of a median line at RFS and MMC: question 28.

- 28. Complete Table 5-7 specifying the appropriate tolerances for the sizes given.
- 29. Straightness verification of a feature of size specified at MMC can be achieved by

placing th	ie part	in a full	form fun	<i>ictional gage</i>		
0		0 1	-0	0-0	DEG	

30. Straightness verification of a feature of size specified at <u>*RFS*</u> cannot be achieved by placing the part in a full form functional gage.

- 31. When verifying circularity, the feature of size is first measured to verify that it falls within the <u>limits of size</u> and <u>Rule #1</u>
- 32. Circularity can be accurately inspected on a <u>circularity inspection machine</u>.
- 33. Cylindricity is a condition of the surface of a cylinder where all points of the surface are *equidistant from the axis*
- 34. The cylindricity tolerance consists of two <u>coaxial cylinders</u> in which the <u>radial distance</u> between them is equal to the tolerance specified in the <u>feature control frame</u>.
- 35. Cylindricity is a <u>composite</u> form tolerance that simultaneously controls <u>circularity</u>, <u>straightness of a surface</u>, <u>and taper</u> of cylindrical features.

				Feature of Size	Feature of Size	0	Ø
1.	For these controls, datums do not apply	X	X	X	X	X	X
2.	For these controls, rule #1 applies	X	X			X	X
3.	These are surface controls	X	X			X	X
4.	These controls may be specified with a leader	X	X			X	X
5.	These are refinements of the size tolerance	X	X			X	X
6.	These tolerances violate rule #1			X	X		
7.	These controls apply to features of size			X	X		
8.	These controls are associated with the dimension			X	X		
9.	These controls may exceed the size tolerance			X	X		
10	The Ø symbol may be used				X		
11	The MMC (circle M) symbol may be used			X	X		

 Table 5-8 Summary of form controls: Question 36.

- 36. On Table 5-8, place an X under the control that agrees with the statement.
- 37. Free-state variation is a term used to describe the distortion of a part after the removal of forces applied during the <u>manufacturing process</u>
- 38. Where a form or location tolerance is specified for a feature in the free state, the free-state symbol is placed inside the <u>feature control frame</u> following the <u>tolerance and any modifiers</u>.
- 39. A minimum of <u>four measurements</u> must be taken to insure the accuracy of an average diameter.

40. The restrained condition should simulateactual assembly conditionsGeometric Dimensioning and Tolerancing for Mechanical DesignAnswer Guide

Problems





 $ANGLES = \pm .010$

Figure 5-15 Flatness: problem 1.

1. Specify a flatness control of .005 for the top surface of the part in Fig. 5-15.

(Either a leader or an extension line can be used)

2. Below, draw a feature control frame with a unit flatness of .003 per square inch and an overall flatness of .015.









Figure 5-16 Flatness: problem 3.

3. Is the part in Fig. 5-16 an acceptable part – why or why not?

No, this part is not acceptable. It is 1.018 thick and bowed .004, a total of 1.022. The part exceeds the 1.020 boundary of perfect form



Figure 5-17 Straightness of a surface: problem 4.

4. Specify straightness of a surface of .002 on the cylinder in the drawing in Fig. 5-17.

(Either a leader or an extension line can be directed to the part surface.)



Figure 5-18 Straightness of a median line: problem 5.

5. On the cylinder in Fig. 5-18, specify straightness of a median line of .010 at MMC.



(The feature control frame must be associated with the dimension.)

Figure 5-19 Circularity: problems 6.

6. Is the part in Fig. 5-19 an acceptable part – why or why not?

<u>Undetermined, further inspection is necessary to reject this part. If</u> Geometric Dimensioning and Tolerancing for Mechanical Design Answer Guide the outer diameter of the circularity tolerance zone is within 1.508, the MMC of the cylinder, it is a good part. The cylinder may not exceed the boundary of perfect form at maximum material condition.



Figure 5-20 Circularity and cylindricity: problems 7 and 8.

- 7. Specify a circularity tolerance of .002 on the cone in the drawing in Fig. 5-20.
- 8. Specify a cylindricity tolerance of .0005 on the cylinder in the drawing in Fig. 5-20.

Chapter 6

Orientation

Chapter Review

Page 98

- Orientation is the general term used to describe the <u>angular</u> relationship between features.
- 2. Orientation controls include <u>perpendicularity, parallelism, angularity,</u> <u>and in some cases, profile</u>
- 3. All orientation controls must have <u>datum features</u>.
- 4. In a view where the surface to be controlled appears as a line, the perpendicularity feature control frame is attached to the surface with a <u>leader or extension line</u>.
- 5. The datum feature is identified with a <u>datum feature symbol</u>.
- 6. A surface being controlled with a **perpendicularity** tolerance must lie between _____

tolerance specified in the feature control frame. The tolerance zone must also be

perpendicular to the datum plane.

- 7. A Tangent Plane symbol, circle T, in the feature control frame specifies that the tolerance applies to the precision plane contacting the <u>high points</u> of the surface.
- 8. Where controlling the perpendicularity of a feature of size, the feature control frame is associated with the <u>size dimension</u> of the feature being controlled.
- 9. If the tolerance in the feature control frame applies to a feature of size and no material condition symbol is specified, <u>*RFS*</u> automatically applies.
- 10. If the tolerance applies at MMC then a possible <u>bonus</u> tolerance exists.


Figure 6-14 Specifying perpendicularity of a plane surface: question 11.

11. Supply the appropriate geometric tolerance on the drawing in Fig. 6-14 to control the 3.00-inch vertical surface of the part perpendicular to the bottom surface within .005.

(Either a leader or an extension line may be used.)



Figure 6-15 Specifying perpendicularity of a feature of size: question 12.

12. Supply the appropriate geometric tolerance on the drawing in Figure 6-15 to control the

1.00-inch diameter vertical pin perpendicular to the bottom surface of the plate within .005 Geometric Dimensioning and Tolerancing for Mechanical Design Answer Guide at RFS.



Figure 6-16 Perpendicularity specified at MMC: question 13.

- 13. If the pin in Figure 6-15 were actually produced at a diameter of 1.004 and toleranced with the feature control frame in Figure 6-16, what would the total perpendicularity tolerance be? .008
- 14. The feature control frame for parallelism of a surface must at least contain

<u>a parallelism symbol, a numerical tolerance, and at least one datum</u> feature

15. Parallelism tolerance of a flat surface is a refinement of the size tolerance and must be less than the size tolerance

16. A surface being controlled with a parallelism tolerance must lie between _____

two parallel planes _________ separated by the

parallelism tolerance specified in the feature control frame. The tolerance zone must also be

______to the datum plane.

parallel

- 17. The controlled surface may not exceed the <u>boundary of perfect form at</u> maximum material condition
- 18. Where applied to a flat surface, parallelism is the only orientation control that requires perfect orientation (Parallelism is a 0° angle.) at ______MMC



Figure 6-17 Specifying parallelism: question 19.

- 19. Supply the appropriate geometric tolerance on the drawing to control the top surface of the part in Fig. 6-17 parallel to the bottom surface within .010. (Either a leader or an extension line can be used)
- 20. When controlling the parallelism of a feature of size, the feature control frame is associated with the ______ of the feature being controlled.
- 21. If the feature of size is a cylinder, the numerical tolerance is usually preceded by a \mathcal{O} .
- 22. The numerical tolerance for angularity of a surface is specified as a linear dimension because it generates a *uniform* -shaped tolerance zone.
- 23. A plus or minus angularity tolerance is not used because it generates a _____

nonuniform, fan______-shaped tolerance zone.

- 24. When controlling the angularity of a feature of size, the feature control frame is associated
- 25. If the diameter symbol precedes the numerical tolerance, the axis is controlled with a

______zone.

26. As an alternative practice, the angularity symbol may be used to control _____ parallel and perpendicular relationships_____.

	Plane Surfaces		Axes & Ctr. Planes			
		//	۷			۷
Datum features are required	X	X	X	X	X	X
Controls flatness if flatness is not specified	X	X	X			
Circle T modifier can apply	X	X	X			
Tolerance specified with a leader or extension line	X	X	X			
Tolerance associated with a dimension				X	X	X
Material condition modifiers apply				X	X	X
A virtual condition applies				X	X	X

 Table 6-2 Orientation: question 27

27. Mark an X in the box that indicates the control applies to the statement at the left in Table 6-2.



Figure 6-18 Perpendicularity of a plane surface: problem 1.

1. Specify the 3.00-inch surface of the part in Fig. 6-18 perpendicular to the bottom and back surfaces within a tolerance of .010. Draw and dimension the tolerance zone.



Figure 6-19 Perpendicularity of a pin to a plane surface: problem 2.

2. Specify the Ø1.00-inch pin perpendicular to the top surface of the plate in Fig.6-19 within a tolerance of .015 at MMC. On the drawing, sketch and dimension a gage used to inspect this part.



Figure 6-20 Parallelism of a plane surface: problem 3.

3. Specify the top surface of the part in Fig. 6-20 parallel to the bottom surface within a tolerance of .004. Draw and dimension the tolerance zone.



Figure 6-21 Angularity of a plane surface: problem 4.

4. In Fig. 6-21, specify the top surface of the part to be at an angle of 20° to the bottom surface within a tolerance of .003. Draw and dimension the tolerance zone.



Figure 6-22 Orientation: problem 5.

5. Complete the feature control frames in Fig. 6-22 so that the two parts will always assemble, datum features A & B will meet, and the part can be produced using the most cost effective design. The pin is machined in a lathe and the hole is drilled.

(There are several possible solutions to this problem. The virtual conditions should be equal to insure assembly and to provide maximum tolerance. Typically, for this method of manufacturing, more tolerance is given to the hole.)

Chapter 7

Position, General

Chapter Review

Page 125

Position is a composite tolerance that controls both the <u>location and</u> 1.

the orientation of feature of sizes at the same time.

- 2. The tolerance of position may be viewed in either of two ways:
 - A theoretical tolerance zone located at true position of the toleranced feature within which the center point, axis, or center plane of the feature may vary from true position.
 - <u>A virtual condition boundary</u> of the toleranced feature, when specified at MMC or LMC and located at true position, which may not be violated by its surface or surfaces of the considered feature.
- 3. A feature of size has four geometric characteristics that must be controlled. These characteristics are <u>size, form, orientation, and location</u>
- Since the position tolerance only controls feature of sizes such as pins, holes, tabs, and slots, 4. the feature control frame is always associated with a <u>size dimension</u>.
- The location of true position, the theoretically perfect location of an axis, is specified with 5. *basic dimensions* from the datum features indicated.
- Once the feature control frame is assigned, an imaginary <u>tolerance zone</u> 6. is defined and located about true position.
- Datum features are identified with <u>datum feature symbols</u>. 7.
- Datum features A, B, and C identify a, <u>datum reference frame</u>; 8. consequently, they describe how the part is to be held for <u>processing</u>.
- 9. To inspect a hole, the largest pin gage to fit inside the hole is used to simulate the actual mating envelope
- 10. The measurement from the surface plate to the top of the pin gage minus half of the diameter of the pin gage equals the distance from _____

datum feature B to the actual axis of the hole

11. If no material condition symbol is specified in the feature control frame, the <u>*RFS*</u> Geometric Dimensioning and Tolerancing for Mechanical Design Answer Guide modifier automatically applies to the tolerance of the feature.

- 12. When the maximum material condition symbol is specified to modify the tolerance of a feature of size, the following two requirements apply:
 - The specified tolerance applies <u>at maximum material condition of the</u> <u>feature</u>.
 - As the size of the feature departs from maximum material condition toward least material condition, <u>a bonus tolerance is achieved in the exact amount of</u> <u>such departure</u>.
- 13. The difference between the actual mating envelope size and MMC is the <u>bonus</u>

<u>tolerance</u>

14. The bonus plus the geometric tolerance equals <u>the total positional tolerance</u>.



Figure 7-20 Geometric tolerance: questions 15 through 18.

- 15. If the tolerance in Fig. 7-20 is for a pin .525 in diameter, what is the total tolerance? ______. 035
- 16. What would be the size of the hole in a functional gage to inspect the pin above? _____.560____
- 17. If the tolerance in Fig. 7-20 is for a hole .540 in diameter, what is the total tolerance? _____ 040
- 18. What would be the size of the pin on a functional gage to inspect the hole above? <u>.500</u>
- 19. Where a datum feature of size is toleranced with a geometric tolerance and is referenced in a feature control frame at MMB, the resulting maximum material boundary for the datum feature is equal to its <u>maximum material condition</u>

or <u>virtual condition with respect to the preceding datum feature</u>.

20. The worst-case inner and outer boundaries of a feature of size are its <u>virtual</u> <u>condition and resultant condition</u>

Problems





Figure 7-21 Design a gage to inspect for shift tolerance: problem 1.

On a gage designed to control the 4-hole pattern in Fig. 7-21, what size pin must be produced to inspect the center hole (datum feature D)? <u>Ø1.000</u>
 On the same gage, what is the diameter of the four pins locating the hole pattern?
 <u>Ø.500</u>

Figure 7-22 Zero positional tolerance conversion: problem 2.

2. Convert the tolerance in Fig. 7-22 to the zero positional tolerances for the pin and the hole.

Zero tolerance is not used when the tolerance applies at	RFS	, or when no
bonus tolerance is available as in a tolerance specified for	threads or p	press fit pins

Figure 7-23 A hole specified at LMC: problem 3.

3. Calculate the minimum wall thickness between the inside diameter and datum feature B in Fig. 7-23.

Datum feature B @ LMC	Ø 2.480
I. D. @ LMC	-Ø2.020
Tolerance @ LMC	<u> </u>
	440

The wall thickness equals half of the differences in diameters or .220.

(Calculating diameters and diving the final diameter by 2 minimizes errors.)

Figure 7-24 Boundary conditions: problem 4.

4. First calculate the virtual conditions and resultant conditions for the pin and hole. Then calculate the maximum and minimum distances for dimensions X and Y in Fig. 7-24.

The Virtual Condition of the PIN.

 $VC_p = MMC + Geo. Tol.$ $VC_p = 1.000 + .004 = 1.004$ $VC_p/2 = .502$

Resultant Condition of the PIN.

 $RC_p = LMC - Geo. Tol. - Bonus$ $RC_p = .998 - .004 - .002 = .992$ $RC_p/2 = .496$

The Virtual Condition of the HOLE.

 $VC_h = MMC - Geo. Tol.$ $VC_h = 1.000 - .004 = .996$ $VC_h/2 = .498$

Resultant Condition of the HOLE.

 $RC_h = LMC + Geo. Tol. + Bonus$ $RC_h = 1.006 + .004 + .006 = 1.016$ $RC_h/2 = .508$

The maximum and minimum distances for dimension X:

$X_{Max} = Dist RC_p/2 - VC_h/2 =$	$X_{Min} = Dist. VC_p / 2 - RC_h / 2 =$
$X_{Max} = 3.000496498 =$	$X_{Min} = 3.000502508$
$X_{Max} = 2.006$	$X_{Min} = 1.990$

The maximum and minimum distances for dimension Y:

$$Y_{Max} = L @ MMC - Dist. - VC_{h}/2 = Y_{Min} = L @ MMC - Dist. - RC_{h}/2 = Y_{Max} = 6.010 - 4.500 - .498 = Y_{Min} = 5.990 - 4.500 - .508 =$$

 $Y_{Max} = 1.012$

 $Y_{Min} = .982$

Chapter 8

Position, Location

Chapter Review

Page 153

1. The floating fastener formula is:

 $\underline{T = H - F} \quad or \quad H = F + T$

2. T = <u>The clearance hole Location Tolerance at MMC</u>

H = <u>The Clearance Hole MMC diameter</u>

F = <u>The Fastener's MMC diameter, the nominal size</u>

- 3. The clearance hole LMC diameter formula is H(a) LMC = (F + F head) / 2.
- 4. The fixed fastener is fixed by one or more of the <u>members being fastened</u>.
- 5. The formula for fixed fasteners is:

 $t_1 + t_2 = H - F$ or $H = F + t_1 + t_2$

- 6. The location tolerance for both the threaded hole and the clearance hole must come from the difference between the actual diameter of the clearance hole and the <u>diameter of the fastener</u>.
- 7. It is common to assign a larger portion of the tolerance to the <u>threaded</u> hole.
- 8. A fastener fixed at its head in a countersunk hole and in a threaded hole at the other end is called what? <u>A double fastener fixed fastener</u>
- 9. Where specifying a threaded hole or a hole for a press fit pin, the orientation of the

______ determines the orientation of the mating pin.

- The most convenient way to control the orientation of the pin outside the hole is to
 <u>project</u>
 the tolerance zone into the mating part.

part ______ or tallest ______ stud or pin ______ after installation.

- 12. The dimension of the projected tolerance zone height is specified as a <u>minimum</u>.
- 13. Two or more patterns of features are considered to be one composite pattern if they

are located with basic dimensions, to the same datums features, in

the same order of precedence, and at the same material conditions

- 14. Datum features of size specified at RFS require <u>physical contact</u> between the gagging element and the datum feature.
- 15. If the patterns of features have no relationship to each other, a note such as

<u>SEP REQT</u> may be placed under each feature control frame allowing each pattern to be inspected separately.

16. Composite tolerancing allows the relationship from <u>feature-to-feature</u> to be kept to a tight tolerance and the relationship between the _____

pattern and its datum features to be controlled to a looser tolerance.

- 17. A composite positional feature control frame has one <u>position</u> symbol that applies to the two horizontal <u>segments</u> that follow.
- 18. The upper segment of a composite feature control frame, called the _____

pattern-locating control, governs the relationship between the datum features and the *pattern*.

- 19. The lower segment of a composite feature control frame is called the <u>feature-relating</u> control; it governs the relationship from <u>feature-to-feature</u>.
- 20. The primary function of the position control is to control <u>location</u>.
- 21. Datum features in the lower segment of a composite feature control frame must satisfy what two conditions:

<u>They are required to repeat the datum features in the upper segment</u> <u>and they only control orientation</u>

(Assume plane-surface datum features for question numbers 22 and 23,.)

- 22. Where the secondary datum feature is included in the lower segment of a composite feature control frame, the tolerance zone framework must remain <u>*Parallel*</u> to the secondary datum plane.
- 23. The lower segment of a multiple single-segment feature control frame acts just like any other <u>position control</u>
- 24. Counterbores that have the same location tolerance as their respective holes are specified by indicating the <u>hole callout and the counterbore callout followed</u> by the geometric tolerance for both
- 25. Counterbores that have a larger location tolerance than their respective holes are specified by

separating the hole callout from the counterbore callout _____.

26.	When tolerancing elongated holes, no	diameter symbol	prece	edes
	the tolerance in the feature control fram	ne since the tolerance zone is not a _	<u>cylinder</u>	
27	The virtual condition boundary is the	exact shape		of

27.	The virtual condition boundary is the	exact snape	01
	the elongated hole and equal in size to its _	virtual condition	

28. The <u>position control</u> is used to locate a <u>size feature feature of size</u> symmetrically at MMC to a datum feature of size specified at MMB.

Problems

Figure 8-27 Floating fastener drawing: problem 1.

1. Tolerance the clearance holes in Fig. 8-27 to be fastened with 5/16 - 18 UNC hex head bolts (.313 in diameter) and nuts with a .010 diameter positional tolerance.

$$H = F + T$$

Figure 8-28 Floating fastener drawing: problems 2 through 5.

2. Specify the MMC and LMC clearance hole sizes for #10 (Ø.190) socket head cap screws.

2 X Ø.220-.246 n]w.030m]A]B]C] 2 X Ø.200-.246 2 X Ø.190-.246 n]w.010m]A]B]C] n]w.000m]A]B]C]

3. If the actual size of the clearance holes in problem 2 is .230 in diameter, calculate the total positional tolerance for each callout.

Actual Size	.230	.230	.230
MMC	- <u>.220</u>	<u>200</u>	<u> </u>
Bonus	.010	.030	.040
Geo. Tolerance	+.030	+.010	+.000
Total Tolerance	<u>.040</u>	.040_	.040_

4. Specify the MMC and LMC clearance hole sizes for 3/8 (Ø.375) hex head bolts.

2 X Ø.400460	2 X Ø.390460	2 X Ø.375460
n]w.025m]A]B]C]	n]w.015m]A]B]C]	
	n]w.000m]A]B]C]	

5. If the clearance holes in problem 4 actually measure .440 in diameter, calculate the total positional tolerance for each callout.

Actual Size	.440	.440	.440
MMC	400	<u> </u>	<u> </u>
Bonus	.040	.050	.065
Geo. Tolerance	+.025	+.015	+.000

Figure 8-29 Fixed fastener assembly: problems 6.

6. Tolerance the clearance and threaded holes in the plates in Fig. 8-29 to be fastened with 5/8–11 UNC hex head bolts (.625 in diameter). Use a .000 positional tolerance and 60% location tolerance for the threaded hole.

Geometric Dimensioning and Tolerancing for Mechanical Design Answ

Answer Guide

.065

Figure 8-30 Fixed fastener drawing: problems 7 through 10.

7. Specify the MMC and LMC clearance hole sizes for #8 socket head cap screws.

2 X Ø.164 (#8)-32 UNF-2B	2 X Ø.164 (#8)-32 UNF-2B	2 X Ø.164 (#8)-32 UNF-2B
n]w.025m]A]B]C]	n]w.025m]A]B]C]	
	n]w.025m]A]B]C]	
<u>.199</u> – <u>.213</u>	<u></u>	<u>.189</u> – <u>.213</u>
n]w.010m]A]B]C]	n]w.005m]A]B]C]
	n]w.000m]A]B]C]

8. If the clearance holes in problem 5 actually measure .205 in diameter, calculate the total positional tolerance for each callout.

Actual Size	.205	.205	.205
MMC	199	<u>194</u>	<u> </u>
Bonus	.006	.011	.016
Geo. Tolerance	+.010	+.005	+.000
Total Tolerance	.016	.016	.016

9. Specify the MMC and LMC clearance hole sizes for the 1/2-inch hex head bolts.

2 X Ø.500-20 UNF-2B	2 X Ø.500-20 UNF-2B	2 X Ø.500-20 UNF-2B
n]w.060m]A]B]C]	n]w.060m]A]B]C]	
	n]w.060m]A]B]C]	
<u>2 X Ø.580</u> – <u>.612</u>	<u>2 X Ø.570</u> – <u>.612</u>	<u>2 X Ø.560</u> – <u>.612</u>
n]w.020m]A]B]C]	n]w.010m]A]B]C]	
	n]w.000m]A]B]C]	

10. If the clearance holes in problem 9 actually measure .585 in diameter, calculate the total positional tolerance for each callout.

Actual Size	.585	.585	.585
MMC	<u> </u>	<u> </u>	<u> </u>
Bonus	.005	.015	.025
Geo. Tolerance	+.020	+.010	+.000
Total Tolerance	.025	<u>.025</u>	<u>.025</u>

Figure 8-31 Projected tolerance zone: problem 11.

11. Complete the drawing in Fig. 8-31. Specify a .040 tolerance at MMC with the appropriate projected tolerance.

Figure 8-32 Projected tolerance zone: problem 12.

12. Complete the drawing in Fig. 8-32. Specify a .050 tolerance at MMC with the appropriate projected tolerance.

Figure 8-33 Fixed fastener assembly: problem 13.

13. The part with clearance holes in Fig. 8-33 assembles on top of the part with threaded holes and is fastened with cap screws. Allow a tolerance of at least .030 on both threaded and clearance holes, use "0" positional tolerance, and specify projected tolerance zones.

(*The threaded hole tolerance of .040 was chosen so that there would be at least .030 tolerance for the clearance hole.*)

Figure 8-34 Multiple patterns of features: problems 14 through 16.

- 14. Position the small holes with .000 tolerance at MMC and the large holes with .010 tolerance at MMC; locate them to the same datums and in the same order of precedence. Use MMC/MMB wherever possible.
- 15. Must the hole patterns be inspected in the same setup or in the same gage? Explain?

Yes, they must be inspected at the same time. The large hole and small hole patterns are tied together by their datums features.

16. Can the requirement be changed, how?

Yes, place a note, SEPT REQT, under each feature control frame.

Figure 8-35 A pattern of holes located to a datum feature of size: problem 17.

17. In Fig. 8-35, the inside diameter and the back are mating features. Select the primary datum feature. (Consider a form control.) The virtual condition of the mating shaft is 1.125 in diameter. Locate the keyway for a ¹/₄-inch key. Locate the three-hole pattern for 7/16-inch (.438 in diameter) cap screws as floating fasteners with a positional tolerance of at least .030.

Figure 8-36 Composite tolerancing: problems 18 and 19.

- 18. The pattern of clearance holes in the part in Fig. 8-36 must be located within a cylindrical tolerance zone of .060 in diameter at MMC to the datum features specified. This plate is required to assemble to the mating part with 1/4-inch hex bolts as floating fasteners. Complete the geometric tolerance.
- 19. It has been determined that the hole pattern in Fig. 8-36 is required to remain parallel, within the smaller tolerance, to datum feature B. Draw the feature control frame that will satisfy this requirement.

Figure 8-37 Counterbore tolerance: problems 20 and 21.

- 20. Tolerance the holes and counterbores in Fig. 8-37 for four ¹/₄-inch socket head cap screws. The counterbores are a diameter of $.422 \pm .010$, the depth is $.395 \pm .010$, and the geometric (Limit tolerances may also be used.) tolerance is .010 at MMC.
- 21. If the geometric tolerance for just the counterbores in Fig. 8-37 can be loosened to .020 at MMC instead of .010, draw the entire callout below.

Figure 8-38 Elongated hole: problem 22.

22 Specify a geometric tolerance of .040 at MMC in the 1/2-inch direction and .060 at MMC in the 1-inch direction for the elongated holes in Fig. 8-38.

(Don't forget to include basic dimensions.)

Figure 8-39 Symmetry: problem 23.

- 23. In Fig. 8-39, control the symmetry of the 2-inch feature with respect to the 4-inch feature within a tolerance of .020. Use MMC/MMB wherever possible.
- 23. In Fig. 8-39, control the symmetry of the 2-inch feature with respect to the 4-inch feature within a tolerance of .020. Use MMC/MMB wherever possible.

Chapter 9

Position, Coaxiality

Chapter Review

Page 171

- 1. Coaxiality is that condition where the axes of two or more surfaces of revolution are _ <u>coincident</u>.
- 2. There is a misconception that centerlines or title block tolerances control the <u>coaxiality</u> between two cylinders.
- 3. The <u>position</u> control is the appropriate tolerance for coaxial surfaces of revolution that are cylindrical and require a maximum or least material condition.
- 4. A <u>cylindrical</u> tolerance zone is used to control the axis of a feature toleranced with a position or a concentricity control.
- 5. The tolerance of position to control coaxiality may apply at <u>MMC, RFS, or LMC</u> and the datum feature(s) may apply at <u>MMB, RMB, or LMB</u>
- 6. The upper segment of a composite feature control frame controls the location of the hole pattern to the <u>datum features</u>
- The lower segment of a composite feature control frame controls the coaxiality of holes to <u>one another within the tighter tolerance</u>
- 8. The smaller tolerance zone framework of a composite feature control frame with no datum features may float <u>up and down, back and forth, and at any angle to the</u> <u>datum features</u> within the larger tolerance zone.
- 9. The position control, with no datum features, can be applied to two or more coaxial features controlling their <u>coaxiality</u> simultaneously within the specified tolerance.
- 10. A mating plug and socket will assemble every time if they are designed to their <u>virtual conditions</u>

Problems

Figure 9-8 Specify coaxiality tolerance: problems 1 through 3.

1. What controls the coaxiality of the two cylinders on the drawing in Fig. 9-8?

The way the drawing in Figure 9-8 is shown, nothing controls coaxiality.

- 2. On the drawing in Fig. 9-8, specify a coaxiality tolerance to control the 1.000-inch diameter feature within a cylindrical tolerance zone of .004 to the 2.00-inch diameter feature. Use MMC and MMB wherever possible.
- Once the feature control frame has been added to the drawing in Fig. 9-8, if the smaller diameter is produced at 1.00 in diameter, how much total coaxiality tolerance applies?

÷

Figure 9-9 Specify coaxiality: problem 4.

4. Locate the two holes in the hinge brackets within .030 at MMC to the datum features indicated, and specify coaxiality to each other. They must be able to accept a .500 diameter hinge pin. Use MMC and MMB wherever possible.

Figure 9-10 Specify coaxiality for the plug and socket: problem 5 and 6.

5. Control the coaxiality of the plug and socket in Fig. 9-10 so that they will always assemble. Specify MMC and MMB wherever possible.

	<u>Plug</u>		<u>Socket</u>
ММС	.750		.751
Geo. Tol.	<u>+.000</u>		<u>–.001</u>
Virtual Condition	.750	=	.750

6. Draw and dimension the tolerance zones at MMC in Fig.9-10.

Chapter 10

Concentricity and Symmetry Chapter Review

Page 181

1.	Both concentricity and symmetry controls are reserved for a few		
	unique tolerancing applications		
2.	Concentricity and symmetry both employ the same tolerancing <u>concept</u> ;		
	they just apply to different <u>geometries</u> .		
3.	Concentricity is that condition where the median points of all diametrically opposed elements of a surface of revolution are congruent with <u>the axis (or center point)</u>		
	of a datum feature		
4.	Concentricity is a <u>location</u> control. It has a <u>cylindrical shaped</u>		
	tolerance zone that is coaxial with the <u>datum axis</u> .		
5.	Concentricity tolerance only applies on a <u><i>RFS</i></u> basis.		
	It must have at least one datum feature that only applies at		
	regardless of material boundary		
6.	For concentricity, the aggregate of all <u>median points</u>		
	must lie within a <i>cylindrical (or spherical) shaped</i> tolerance zone		
	whose axis is coincident with the axis of <i>the datum feature</i>		
7.	Concentricity can be inspected, for acceptance only, by placing a <u>dial indicator</u> on		
	the toleranced surface of revolution and rotating the part about its <u>datum axis</u> .		
8.	To reject parts and to inspect features such as regular polygons and ellipses, the traditional		
	method of <u>differential measurements</u> is employed.		
9.	The concentricity tolerance is often used to accurately control <u>balance</u>		
	for high speed rotating parts.		

 Parts toleranced with concentricity are time consuming and expensive, to <u>inspect</u>, but less expensive to <u>manufacture</u> than with the runout tolerance.

11.	Symmetry is that condition where the	median points	of all opposed
	or correspondingly located points of tw	<u>congruent</u>	
	with the axis or center pla	ane	of a datum feature.

- 12. Symmetry is a *location* control.
- 13. Symmetry has a tolerance zone that consists of <u>two parallel planes</u> evenly disposed about the <u>center plane or axis</u> of the datum feature.
- 14. Symmetry tolerance only applies at <u>*RFS*</u>.
- 15. Symmetry must have at least one <u>datum feature</u> that may only apply at <u>regardless of material boundary</u>.
- 17. The symmetry tolerance is independent of both <u>size and form</u>.
- 18. Differential measurement excludes <u>size, shape, and form</u> while controlling the median points of the feature.
- The symmetry tolerance is often used to accurately control <u>balance</u>
 for rotating parts or to insure equal <u>wall thickness</u>.
- 20. Specify symmetry only when it is necessary because it is <u>time consuming and</u> <u>expensive</u>

to manufacture and inspect.
Problems



Figure 10-7 Coaxiality of a cylinder: problem 1.

1. The mass of this high speed rotating part in Fig.10-7 must be accurately balanced. The form of the surface is sufficiently controlled by the size tolerance. Specify a coaxiality control for the axis of the 4.000-inch diameter within a tolerance of .001 at RFS to datum feature A at RMB.



Figure 10-8 Coaxiality of an ellipse: problem 2.

2. The mass of the ellipse shown in Fig. 10-8 must be balanced accurately. Specify a coaxiality control that will locate the median points of the ellipse within a tolerance of .004 at RFS to

datum feature A at RMB.



Figure 10-9 Coaxiality of the hexagon: problem 3.

3. The mass of the hexagon shown in Fig. 10-9 must be accurately balanced. Specify a coaxiality control for the median points of the hexagon within a tolerance of .005 at RFS to datum feature A at RMB.



Figure 10-10 Symmetry of the slot: problem 4.

4. The part in Fig. 10-10 rotates at a high speed and the mass must be accurately balanced. Specify a geometric tolerance that will centrally locate the slot in this part within a tolerance of .005 at RFS to datum feature A at RMB.

Answer Guide

Chapter 11

Runout

Chapter Review

Page 191

- 1. Circular runout applies independently to each <u>circular element</u> on the surface of a part either constructed around a datum axis or perpendicular to a datum axis as the part is rotated <u> 360° </u> about its datum axis.
- Where applied to surfaces of revolution, circular runout controls a combination of variations in <u>circularity and coaxiality</u>.
- 3. Total runout is a compound tolerance that provides control of all <u>surface elements</u> of a feature.
- 4. Total runout tolerance is applied simultaneously to all circular and profile measuring positions either *around* or *perpendicular*

to its datum axis as the part is rotated 360° about that datum axis.

5. Where applied to surfaces constructed around a datum axis, total runout controls a combination of surface variations such as

circularity, straightness, coaxiality, angularity, taper, and profile____.

- 6. Where applied to surfaces at a 90° angle to the datum axis, total runout controls a combination of variations of *perpendicularity to the datum axis and flatness*
- 7. The runout feature control frame consists of <u>a runout symbol, the numerical</u> <u>tolerance, and at least one datum feature that applies at RMB</u>.
- 8. In many cases, two functional <u>datum features</u> are used to support a rotating part.
- 9. The datum reference frame for the face and diameter is quite a different requirement than the datum reference frame for the <u>multiple datum feature</u>.
- 10. Design requirements may make it necessary to restrict datum surface variations with respect to (other geometric controls) <u>straightness, flatness, circularity, cylindricity,</u> <u>and parallelism</u>

- It may be necessary to include a runout control for individual datum features on a <u>multiple datum feature reference</u>.
- 12. If two or more surfaces are controlled with a runout tolerance to a common datum reference, the worst-case runout between two surfaces is the <u>sum of the two tolerances</u>.
- 13. If two features have a specific relationship between them, one should be <u>toleranced</u> <u>directly to the other and not through a common datum axis</u>.
- 14. Multiple leaders directed from a runout feature control frame may be specified without *affecting the runout tolerance*

Problems

Page 192



Figure 11-8 Runout control: problem 1.

On the part in Fig. 11-8, control the 4-inch diameter with a total runout tolerance of .002 to 1. both 1-inch diameters.



Figure 11-9 Partial runout: problem 2.

2. On the drawing in Fig. 11-9, specify a circular runout tolerance of .002 controlling the 2-inch diameter to both of the 1-inch diameters. This control is a partial runout tolerance one inch long, starting from the left end of the 2-inch diameter. Specify a circular runout of .001 for each of the 1-inch diameters.



Figure 11-10 Datum features toleranced with a cylindricity tolerance: problem 3.

3. Tolerance the 2-inch diameter in Fig. 11-10 with a total runout tolerance of .001 to both of the 1-inch diameter shafts. Tolerance each 1-inch diameter shaft with a cylindricity tolerance of .0005.



Figure 11-11 Multiple features tolerance with one feature control frame: problem 4 and 5.

4. In Fig. 11-11, which datum feature, A or B, takes precedence?

Datum feature A is no more important than datum feature B, and datum feature B is no more important than datum feature A.

5. What is the worst possible runout tolerance between the two largest diameters in Fig. 11-11?

Chapter 12

Profile

Chapter Review

Page 209

1. Profile of a line is the *outline*

of an object in a plane as the plane passes through the object.

- 2. Profile of a surface is the result of *projecting the profile of an object on a plane* or taking cross sections through the object at various intervals.
- 3. The true profile may be dimensioned with what kind of dimensions?

With basic size dimensions, basic coordinate dimensions, basic radii, basic angular dimensions, formulas, mathematical data, or, undimensioned drawings .

The feature control frame is directed to the profile surface with a 4.

leader or an extension line

- 5. What symbols do not apply in the tolerance section of profile feature control frames? The diameter symbol and material condition modifiers
- If the leader from a profile feature control frame points directly to the true profile, the 6. tolerance specified is <u>equally disposed about the true profile</u>.
- 7. If the leader from a profile tolerance points directly to a segment of a phantom line extending, outside or inside, parallel to the profile, then _____ all the tolerance is outside or inside the true profile

Where a profile tolerance applies all around the profile of a part, the _____ 8.

<u>"all around" symbol</u> is specified.

- Draw the "all around" symbol. 9.
- 10. If the profile is to extend between two points, the points are <u>labeled</u> and a note using the <u>between symbol</u> is placed beneath the feature control frame.
- 11. Draw the between symbol.

12.	. If a part is to be controlled with a profile tolerance over the entire surface of the par	t, the
	<i>"ALL OVER "</i> symbol is	specified.
13.	. Profile tolerances <u>may or may not</u> have datum	features.
14.	. The profile of a surface control usually requires a datum feature(s) to properly	
	orient and locate the surface	
15.	. Datum features are generally <u>not used</u> for profile of a line tolerand	ces where
	only	ontrolled.
16.	If the design requires a smaller radius than the radius allowed by the profile tolerand local note such as, <i>"ALL CORNERS R.015 MAX," or "R.015 M</i>	ce, a AX"
	is directed to the radius with a <u>leader</u>	
17.	The profile tolerance may be combined with other <u>geometric tolerance</u>	<u>S</u>
	to <u>refine</u> certain aspects of	a surface.
18.	Coplanarity is the condition of <u>of two or more</u>	
	surfaces having all elements in one plane	
19.	 Coplanarity is toleranced with the profile of a surface feature control frame, connecting the <u>leader</u>, to a <u>phantom line</u> connecting the 	ted with surfaces.
20.	The number of coplanar surfaces followed by an X precedes the <u>feature c</u> <u>frame</u> .	ontrol_
21.	. Where coplanar surfaces are used as a datum feature, it is best to attach the datum feature symbol to <i>the profile feature control frame</i>	eature
22.	. Conicity may be controlled with a <i>profile of a surface tolerance</i>	
23.	. Composite profile tolerancing is very similar to <i>composite positional toler</i>	ancing
24.	. The upper segment of a composite profile feature control frame is called the	
	profile locating control ; it go	overns the
	location relationship between the datum features and the profi	<u>le</u> .
25.	The lower segment, referred to as the <u>profile refinement control</u>	?
	is a smaller tolerance than the profile locating control and governs	
	the size, form, and orientation relationship of the	e profile.
Geom	netric Dimensioning and Tolerancing for Mechanical Design Answ	wer Guide

- 26. The feature profile must fall inside <u>both profile tolerance zones</u>.
- 27. Datum features in the lower segment of a composite feature control frame must satisfy two conditions:
- <u>Datum features in the lower segment must repeat the datum features</u> in the upper segment of the feature control frame.
- Datum features in the lower segment only control orientation.
- 28. A second datum feature may be repeated in the lower segment of the composite feature control frame. Both datum features only control <u>orientation</u>
- 29. The lower segment of a multiple single-segment profile feature control frame acts just like *any other profile control*
- 30. The upper segment of a multiple single-segment profile feature control frame allows the smaller tolerance zone to <u>translate</u>

relative to the datum feature not repeated in the lower segment within the larger tolerance.

Problems

Page 210



Figure 12-18 Profile of a surface: problem 1.

1. Specify a profile of a surface tolerance of .020, perpendicular to datum feature A, and all around the part in Fig. 12-18.



Figure 12-19 Profile of a surface between two points: problem 2.

2. Control the top surface between points X and Y in Fig. 12-19, specify a profile tolerance of . 030, located to datum features A, B, and C.



Figure 12-20 Locating a profile of a surface: problem 3.

3. Control the entire surface of the die cavity in Fig. 12-20 to the datum features indicated within a tolerance of .015 outside the true profile. (Outside the profile is external to the true profile line. Inside the profile is within the profile loop.)



Figure 12-21 Locating a mating profile of a surface: problem 4.

4. Control the entire surface of the punch in Fig. 12-21 to the datum features indicated within a tolerance of .015 inside the true profile. (Outside the profile is external to the true profile line. Inside the profile is within the profile loop.)



Figure 12-22 Coplanarity: problem 5.

5. The primary datum feature is the two lower coplanar surfaces in Fig. 12-22. Specify the primary datum feature to be coplanar within .004.



Figure 12-23 Profile controlled to datum features of size: problem 6.

6. Tolerance the drawing in Fig. 12-23. Specify controls locating the hole-patterns to each other and perpendicular to the back of the part. The holes are for ½-inch and 5/8-inch bolts respectively. Specify a control locating the profile of the part to the hole-patterns and perpendicular to the back of the part within a tolerance of .060.

(Explore other solutions to this problem)



Figure 12-24 Composite profile tolerancing: problems 7 through 9.

- 7. In Fig. 12-24, specify a profile tolerance for the center cutout that will control the size, form, and orientation to datum feature A within .010 and locate it to the datum features indicated within .060. Complete the drawing.
- 8. Draw a profile tolerance below that will satisfy the requirements for problem 7 and **orient** the cutout parallel to datum feature B within .010.

\bigcirc	.060	A	В	С
	.010	A	В	

9. Draw a profile tolerance below that will satisfy the requirements for problem 7 and locate the cutout to datum feature B within .010.





Figure 12-25 Profile of a sheet metal part: problem 10.

10. Specify the lower two surfaces of the bottom of the sheet metal part in Fig. 12-25 coplanar within .020. Tolerance the holes with geometric tolerancing. The MMC for each hole is the virtual condition for the mating part. Specify the profile of the top surface of the sheet metal part within .040.

Chapter 13

Graphic Analysis

Chapter Review

Page 225

- 1. List the advantages of graphic analysis.
 - 1. <u>Provides functional acceptance</u>
 - 2. <u>Reduces cost and time</u>
 - 3. <u>Eliminates gage tolerance or wear allowance</u>
 - 4. <u>Allows function verification of MMCMMB, RFSRMB, and LMCLMB</u>
 - 5. <u>Allows verification of any shape tolerance zone</u>
 - 6. <u>Provides a visual record for the material review board</u>
 - 7. <u>Minimizes storage required</u>
- 2. List the factors that affect the accuracy of graphic analysis.
 - 1. <u>The accuracy of the inspection data</u>
 - 2. <u>The completeness of the inspection process</u>
 - 3. <u>The drawing's ability to provide common drawing interpretations</u>



Figure 13-12 Refer to this feature control frame for questions 3 through 7.

- 3. A piece of graph paper with datum features, true positions, tolerance zones, and actual feature locations drawn on it (see Fig. 13-12) is called a <u>data graph</u>
- 4. A piece of tracing paper with datum features, true positions, tolerance zones, and actual feature locations traced or drawn is called a <u>tolerance zone overlay gage</u>
- 5. The upper segment of the composite feature control frame, the drawing, and the inspection data dictates the configuration of the <u>data graph</u>

6. The lower segment of the feature control frame, the drawing, and the inspection data dictate Geometric Dimensioning and Tolerancing for Mechanical Design Answer Guide

the configuration of the <u>tolerance zone overlav gage</u>.

If the tracing paper can be adjusted to include all feature axes within the _____ 7.

tolerance zones on the tracing paper, the feature–to–feature relationships are in tolerance.



Figure 13-13 Refer to this feature control frame for questions 8 through 11.

- 8. To inspect a datum feature of size, the feature control frame (Fig. 13-13), the drawing and the inspection data dictate the configuration of the <u>data graph</u>.
- 9. Draw the actual location of each feature on the data graph. If each feature axis falls inside its respective tolerance zone, the part is <u>in tolerance</u>
- 10. If any feature axis falls outside of its tolerance zone, <u>further analysis may be</u>

required to reject the part

11. If the tracing paper can be adjusted to include all feature axes on the overlay gage within their respective tolerance zones on the data graph and datum axis D contained within its shift tolerance zone while orienting datum feature B on the overlay gage parallel to datum feature B on the data graph, the pattern of features is <u>acceptable (in tolerance)</u>.

Problems

Page 226



Figure 13-14 A pattern of features controlled with a composite tolerance: problem 1.

Feature Number	Feature Location from Datum feature C X-Axis	Feature Location from Datum feature B Y-Axis	Feature Size	Departure from MMC (Bonus)	Datum-to- Pattern Tolerance Zone Size	Feature-to- Feature Tolerance Zone Size
1	1.002	1.003	Ø .200	.010	Ø.020	Ø.010
2	1.005	3.006	Ø.198	.008	Ø.018	Ø.008
3	3.005	3.002	Ø.198	.008	Ø.018	Ø.008
4	3.003	.998	Ø.196	.006	Ø.016	Ø.006

Table 13-3 Inspection data for the graphic analysis problem: problem 1.

1. A part was made from the drawing in Figure 13-14, and the inspection data was tabulated in Table 13-3. Perform a graphic analysis of the part. Is the pattern within tolerance? <u>Yes</u>



Figure 13-15 A pattern of features controlled with a composite tolerance: problem 2.

Feature Number	Feature Location from Datum feature C X-Axis	Feature Location from Datum feature B Y-Axis	Feature Size	Departure from MMC (Bonus)	Datum-to- Pattern Tolerance Zone Size	Feature-to- Feature Tolerance Zone Size
1	1.004	.998	Ø.174	.008	Ø.018	Ø.010
2	.995	3.004	Ø.174	.008	Ø.018	Ø.010.
3	3.000	3.006	Ø.172	.006	Ø.016	Ø.008
4	3.006	1.002	Ø.176	.010	Ø.020	Ø.012

Table 13-4 Inspection data for the graphic analysis problem: problem 2.

2. A part was made from the drawing in Fig. 13-15, and the inspection data was tabulated in

Table 13-4. Perform a graphic analysis of the part. Is the pattern within tolerance? <u>No</u>

(The pattern must remain parallel to datum feature B because datum feature B has been repeated in the lower segment of the feature control frame.)

If it is not in tolerance, can it be reworked, if so, how? <u>The pattern will be in</u> tolerance if holes numbered 2 and 3 are enlarged by about .004.



Figure 13-16 A pattern of features controlled to a feature of size: problem 3.

Feature Number	Feature Location from Datum feature D X-Axis	Feature Location From Datum feature D Y-Axis	Actual Feature Size	Departure from MMC (Bonus)	Total Geometric Tolerance
1	-1.992	-1.493	Ø .278	.008	Ø.008
2	-1.993	1.509	Ø .280	.010	Ø.010
3	2.010	1.504	Ø .280	.010	Ø.010
4	2.010	-1.490	Ø .282	.012	Ø.012
Datum feature			Ø .520	Shift Tolerance =	Ø.020

 Table 13-5 Inspection data for the graphic analysis problem: problem 3.

3. A part was made from the drawing in Fig. 13-16, and the inspection data was tabulated in Table 13-5. Perform a graphic analysis of the part. Is the pattern within tolerance? *Yes*



Figure 13-17 A pattern of features controlled to a feature of size: problem 4.

Feature Number	Feature Location from Datum feature D X-Axis	Feature Location From Datum feature D Y-Axis	Actual Feature Size	Departure from MMC (Bonus)	Total Geometric Tolerance
1	-1.995	-1.495	Ø .224	.010	Ø.014
2	-1.996	1.503	Ø .218	.004	Ø.008
3	2.005	1.497	Ø .220	.006	Ø.010
4	1.997	-1.506	Ø .222	.008	Ø.012
Datum feature			Ø.380	Shift Tolerance =	Ø.005

 Table 13-6 Inspection data for the graphic analysis problem: problem 4.

4. A part was made from the drawing in Figure 13-17, and the inspection data was tabulated in Table 13-6. Perform a graphic analysis of the part. Is the pattern within tolerance? <u>No</u>

If it is not in tolerance, can it be reworked, if so, how? <u>The pattern will be in</u> <u>tolerance if all holes are enlarged to their LMC size</u>.

A Strategy for Tolerancing Parts



Figure 14-18 A hole located and oriented to datum features A, B, and C for questions 1 through 6.

- 1. What type of geometric tolerances applies to the primary datum feature in a drawing like the drawing in Figure 14-18? *Form tolerance*
- 2. What geometric tolerance applies to the primary datum feature in the drawing in Fig.14-18? <u>Flatness</u>
- 3. The primary datum feature controls <u>the orientation</u> of the feature being controlled.



Figure 14-19 A feature control frame with two location datum features: question 4.

4. If the feature control frame for the hole in Fig. 14-18 happened to be the one shown in Fig. 14-19, what relationship would the 1-inch diameter hole have to datum features B & C?

The tolerance zone of the 1-inch hole would be parallel to datum feature B and parallel to datum feature C and at the same time, located with basic dimensions up from datum feature B and over from datum feature C.

Figure 14-20 A feature control frame with three position datum features: question 5.

5. If the feature control frame for the hole in Figure 14-18 happened to be the one shown in Figure 14-20, what relationship would the 1-inch diameter hole have to datum features A, B & C?

<u>The tolerance zone of the 1-inch hole would be perpendicular to</u> <u>datum feature A, located with basic dimensions up from datum feature</u> <u>B and over from datum feature C.</u>



Figure 14-21 A position feature control frame with a refinement: question 6.

- 6. Complete the feature control frame in Figure 14-21 so that it will refine the orientation of the hole in Figure 14-18 within a cylindrical tolerance zone of .000 at MMC.
- 7. Draw a feature control frame to control a pattern of holes within a cylindrical tolerance zones .125 in diameter at MMC to their datum features, datum features A, B, and C. Refine the tolerance of the feature-to-feature relationship to cylindrical tolerance zones .000 in diameter at MMC.



- 8. What is the orientation tolerance for the pattern of holes specified in the answer for question number 7? <u>Perpendicular to datum feature A within a cylindrical</u> tolerance zone of .000 in diameter at MMC.
- Keeping in mind that the primary datum feature controls orientation, explain how you would select a primary datum feature on a part. <u>Key points in selecting a primary</u> <u>datum feature are:</u>

Select a functional surface

Select a mating surface

<u>Select a sufficiently large, accessible surface that will provide</u> <u>repeatable positioning in a datum reference frame while processing</u> <u>and ultimately in assembly</u>

10. How would you determine which datum feature should be secondary and which should be tertiary?

The secondary datum feature may be more important because it is larger than the tertiary datum feature or because it is a mating surface.



Figure 14-22 Pattern of features: questions 11 through 17.

- 11. Select a primary datum feature and specify a form control for it.
- 12. Select a secondary datum feature and specify an orientation control for it. The virtual condition of the mating part is a diameter of 4.255.
- 13. Tolerance the keyseat for a $\frac{1}{2}$ -inch key.

- 14. Tolerance the $\frac{1}{2}$ -inch clearance holes for $\frac{1}{2}$ -inch floating fasteners.
- 15. Are there other ways this part can be toleranced? How

Yes, the hole pattern could be datum feature C, or datum feature C could be left off entirely.

16. If the outside diameter is actually produced at 4.240, how much shift tolerance is available?

If datum feature B were perfectly perpendicular to datum feature A, there would be a cylindrical tolerance zone shift of .015 in diameter.

17. If the outside diameter is actually produced at 4.240 and the keyseat is actually produced at . 505, how much can this part actually shift? Sketch a gage about the part._____

If datum feature B were perfectly perpendicular to datum feature A, The outside diameter could shift back and forth .015 and up and down .005. The part could rotate some.



Figure 14-23 Two patterns of features: questions 18 through 21.

- 18. Locate the two-hole pattern to the surface datum features with a positional tolerance of .085 in diameter at MMC. Locate the two holes to each other and orient them to datum feature A within a cylindrical tolerance of .010 in diameter at MMC.
- 19. Locate the three-hole pattern to the two-hole pattern within a .000 positional tolerance.
- 20. The two-hole pattern is specified as a datum feature at MMB, at what size does each of the two holes apply?

Virtual condition – .500 in diameter

21. If the 1/2-inch holes are actually produced at a diameter of .535, what is the shift tolerance allowed for the three-hole pattern?

If the holes were perfectly oriented and located and produced at a diameter of .535, there would be cylindrical tolerance zone shift of . 035 in diameter.







Figure 14-24 Locating a hole pattern to plane surface datum features: problem 1.

- 1. Dimension and tolerance the four-hole pattern in Fig. 14-24 for # 10 cap screws as fixed fasteners. Allow maximum tolerance for the clearance holes and 60% of the total tolerance for the threaded holes in the mating part. (*The .223 MMC clearance hole diameter is subject to good engineering judgment. It might very well have been rounded off to .220.*)
- How flat is datum surface A? <u>Within .060 – in reality probably within .015</u>
- How perpendicular are datum features B and C to datum feature A and to each other? $\pm 1^{\circ}$



Figure 14-25 A 4-hole pattern located to a datum feature of size: problem 2.

- 2. In Fig. 14-25, tolerance the center hole to the outside edges with a tolerance of .060. Refine the orientation of the 1/2-inch hole to the back of the part within .005. Locate the four-hole pattern to the center hole. Clock the pattern to a surface. The four-hole pattern mates with a part having four pins with a virtual condition of .250. Give each feature the maximum tolerance possible.
 - At what size does the center hole apply for the purposes of positioning the four-hole pattern? <u>Virtual condition with respect to orientation .500 in</u> <u>diameter</u>
 - If the center hole is produced at a diameter of .535, how much shift of the four-hole pattern is possible? <u>A cylindrical tolerance zone .035 in diameter if</u> <u>datum feature D is perfectly perpendicular to datum feature A</u>



Figure 14-26 Tolerancing: problem 3.

In Fig. 14-26, the location of the hole patterns to the outside edges is not critical; a tolerance of . 060 at MMC is adequate. The location between the two 1/2-inch holes and their orientation to datum feature A must be within .010 at MMC. Control the six-hole pattern to the two-hole pattern within .000 at MMC. The mating part has virtual condition pins of .500 and .250 in diameter. Complete the drawing.

- At what size does the two-hole pattern apply for the purposes of positioning the six-hole pattern? *Virtual condition with respect to orientation* .500 in *diameter*
- If the two large holes are produced at a diameter of .520, how much shift of the four-hole pattern is possible? <u>As much as .020 in diameter</u>