LORD Corporation

TS-026
STANDARD MEASURING METHODS

INDEX

INDEX

Tables

Appendices

1.0 SCOPE

1.1 This specification establishes standard measuring methods in order to achieve consistency of inspection results by LORD Corporation and LORD Corporation supplier inspection personnel. The methods and procedures presented in this document shall be used as indicated unless otherwise specified. Part geometry will determine the specific inspection method/procedure to be used. A particular method of inspection used on a dimension for one specific part is not necessarily the best method to be used on a different part. New or more effective methods are acceptable once the method has been proven to show and assure conformance to the drawing or specification requirements. Alternate methods of inspection and gaging not identified in this specification must be reviewed, documented, and approved by LORD Quality prior to implementation.

2.0 APPLICABLE DOCUMENTS

Unless otherwise indicated, the current issue of the below listed documents shall be considered a part of this specification to the extent herein indicated.

2.1 Military Specifications/Standards

MIL-S-7742	Screw Threads, Standard, Optimum Selected Series: General Specification for
MIL-S-8879	Screw Threads, Controlled Radius Root with Increased Minor Diameter, General Specification for

2.2 Industry Documents

ANSI B89.3.1	Measurement of Out-Of-Roundness
ANSI B92.1	Involute Splines and Inspection
ASME B1.1	Unified Inch Screw Threads (UN and UNR Thread Form)
ASME B1.2	Gages and Gaging for Unified Inch Screw Threads
ASME B1.3	Screw Thread Gaging Systems for Acceptability - Inch and Metric Screw Threads (UN, UNR, UNJ, M, and MJ)
ASME B1.7	Screw Threads: Nomenclature, Definitions, and Letter Symbols
ASME B1.13M	Metric Screw Threads: M Profile
ASME B1.21M	Metric Screw Threads: MJ Profile
ASME B46.1	Surface Texture (Surface Roughness, Waviness and Lay)
ASME Y14.5	Dimensioning and Tolerancing

ASTM SI 10	American National Standard for Metric Practice
AS8879	Screw Threads – UNJ Profile, Inch, Controlled Radius Root with Increased Minor Diameter
BS 3643-1	ISO Metric Screw Thread
FED-STD-H28/2	Unified Inch Screw Threads – UN and UNR Thread Forms
FED-STD-H28/6	Gages and Gaging for Unified Screw Threads – UN and UNR Thread Forms
FED-STD-H28/20	Screw-Thread Standards for Federal Services, Section 20, Inspection Methods for Acceptability of UN, UNR, UNJ, M, and MJ Screw Threads

2.3 LORD Documents

PRC-T-0010	Requirements for Visual Inspection of Aerospace Parts
LOP-396	Calibration and Control of Measuring and Test Equipment and Measurement Standards

3.0 DEFINITIONS

3.1 Basic Dimension - A numerical value used to describe the theoretically exact size, profile, orientation, or location of a feature or datum target. It is the basis from which permissible variations are established by tolerances in feature control frames or on other dimensions or notes.

3.2 Datum - A theoretically exact point, axis, line, plane, or combination thereof derived from theoretical datum feature simulator.

3.3 Datum Axis - The axis of a datum feature simulator established from the datum feature.

3.4 Datum Center Plane - The center plane of a datum feature simulator established from the datum feature.

3.5 Datum Feature - A feature that is identified with either a datum symbol or a datum target symbol.

3.6 Dimension - A numerical value(s) or mathematical expression in appropriate units of measure used to define the form, size, orientation or location, of a part or feature.

3.7 Dimension Reference - A dimension, usually without a tolerance, that is for information purposes only.

3.8 Feature - A physical portion of a part such as a surface, pin, hole, or slot or its representation on a drawings, models, or digital data files.

3.9 Feature Axis - The axis of the unrelated actual mating envelope of a feature.

3.10 Free State - The condition of a part free of applied forces.

3.11 Full Indicator Movement (FIM) - The total movement of an indicator where appropriately applied to a surface to measure its variations.

3.12 Least Material Condition (LMC) - The condition in which a feature of size contains the least amount of material within the stated limits of size (e.g., maximum hole diameter, minimum shaft diameter).

3.13 Least Material Boundary (LMB) - The limit defined by a tolerance or combination of tolerances that exists on or inside the material of a feature(s).

3.14 Maximum Material Condition (MMC) - The condition in which a feature of size contains the maximum amount of material within the stated limits of size (e.g., minimum hole diameter, maximum shaft diameter, etc.)

3.15 Maximum Material Boundary (MMB) - The limit defined by a tolerance or combination of tolerances that exists on or outside the material of a feature(s).

3.16 Regardless of Feature Size (RFS) - Indicates a geometric tolerance applies to any increment of size of the actual mating envelope of the feature of size.

3.17 Regardless of Material Boundary (RMB) - Indicates that a datum feature simulator progresses from MMB towards LMB until it makes maximum contact with extremities of a feature.

3.18 Restraint - The application of force(s) to a part to simulate its assembly or functional condition resulting in possible distortion of a part from its free-state condition.

3.19 Size, Actual Size - The measured value of any individual distance at any cross section of a feature of size.

3.20 Size, Limits of - The specified maximum and minimum size. Unless otherwise specified, the limits of size of a feature prescribe the extent within which variations of geometric form, as well as size, are allowed.

3.21 Size, Nominal - The designation used for purpose of general identification.

3.22 Tolerance - The total amount a specific dimension is permitted to vary. The tolerance is the difference between maximum and minimum limits.

3.23 Tolerance, Bilateral - A tolerance in which variations is permitted in both directions from the specified dimension.

3.24 Tolerance, Unilateral - A tolerance in which variation is permitted in one direction from specified dimension.

3.25 True Position - The theoretically exact location of a feature of size, as established by basic dimensions.

3.26 Virtual Condition - A constant boundary generated by the collective effects of a considered feature of the size's specified MMC and LMC and the geometric tolerance for the material condition.

4.0 Measurement Guidelines

4.1 Measurement Accuracy and Precision. Accuracy of a measuring device is its ability to measure the true value of a part dimension while precision is the ability to obtain consistent results repeatedly. The measuring equipment used to evaluate a drawing dimensional characteristic shall be 10 times more accurate than the tolerance to be measured. For example, a part feature that has a total tolerance of .001" should be measured with an instrument that discriminates to at least .0001".

4.2 Measurement Errors. The accuracy and precision of any measurement is largely dependent on the following several types of measurement error that need to be avoided:

a. Observational
- misreading the gage
- parallax error

b. Manipulative
- holding/using the gage incorrectly
- not locating on datums properly
- measurement pressure (feel) excessive or too light
- cosine error (see 4.2.1)

c. Gage inaccuracy
- out-of-tolerance or incorrectly calibrated gage

d. Part condition
- dirt and burrs
- surface finish

4.2.1 Cosine Error. When using a dial indicator (with a ball contact tip), the arm of the indicator should be parallel to a flat surface or tangent to a curved surface so that the contact tip moves perpendicular to the part surface. However, there may be circumstances where this is not possible due part geometry or some type of interference. When this is the case, then cosine error occurs and a correction factor is needed to account for the cosine error. Table 1 lists the correction factors to use for various angles of the dial indicator arm in relation to the measurement surface. Visually determine the angle of the indicator arm to the surface then multiply the indicator reading by the applicable Table 1 correction factor to arrive at the true measurement. The larger the angle, the larger the error becomes. Cosine error needs to be considered for CMM measurements and calibration as well. Optical inspection methods are free of cosine error making this the preferred method for inspection of profile and angular characteristics (when possible).

Table 1 – Cosine Error Correction Factors

Angle	Correction Factor
10°	0.985
20°	0.940
30°	0.866
40°	0.766
50°	0.643
60°	0.500

4.3 Environmental Control (Temperature): As a general practice, all inspection should be performed in a climate-controlled area at 68° +/- 2° F (20° +/- 1° C). For inspection of close tolerance dimensions (i.e., .0005” or less on aluminum and brass and .0003” or less on all other materials), the part shall be inspected in a climate-controlled area at 68° +/- 2° F (20° +/- 1° C) and be stabilized for a minimum of four hours with the measuring equipment and masters to be used in the inspection process. All gage calibration shall be performed in a climate-controlled area at 68° +/- 2° F (20° +/- 1° C), which is considered to be the zero point for measuring.

As the temperature deviates from 68° F (20° C), materials expand at different rates (known as the “coefficient of linear expansion”). Temperature variations and material differences between the part and inspection equipment will have an effect on the measurements obtained. The variance resulting from the difference in expansion rates has the potential to become quite significant and must be compensated for when measuring large dimensions or dimensions with a small tolerance. Refer to industry standard publications for the coefficient of linear expansion rates for the part material and inspection equipment material.

4.3.1 Temperature Compensation Correction Factor. Where the part and measuring equipment cannot be stabilized and inspected in a climate controlled area (68° +/- 2° F (20° +/- 1° C)) and it is suspected that a temperature correction factor is needed, then the part measurement shall be corrected for temperature per the following equation:

$$M_1 - ((\alpha_P - \alpha_G) \times \Delta T \times M_1) = AM_1$$

Where:

- M_1 = Actual measurement of the part once the part and inspection equipment have been stabilized in the environment to be measured for a minimum of four hours.
- α_P = Coefficient of Linear Expansion for the part material
- α_G = Coefficient of Linear Expansion for the gage material
- ΔT = Difference in inspection temperature from 68° F (20° C)
- AM_1 = Adjusted (corrected) Measurement

4.4 General Rules. A number of general rules shall be applied in inspection of geometric tolerances.

4.4.1 Geometric Tolerance Rules. Rules shall be in accordance with the ASME Y14.5 convention stated on the drawing. If a convention version is not specified, the date of the latest drawing revision shall be used to determine which ASME Y14.5 convention to use (i.e. if a drawing revision is dated 2011, then use ASME Y14.5-2009 as that was the version/convention in place at the time of the drawing revision).

NOTE: Since multiple conventions are used, previous references that are not part of the current revision (such as RFS) will remain within this document.

4.4.2 Limits of Size Rule (Rule No. 1). Where only a tolerance of size is specified, the limits of size of an individual feature prescribed the extent to which variations in its geometric form, as well as size, are allowed. The actual size of an individual feature at each cross section shall be within the specified tolerance of size.

a. The surface or surfaces of a feature shall not extend beyond a boundary (envelope) of perfect form at MMC. This boundary is the true geometric form represented by the drawing. No variation in form is permitted at its MMC limit of size.
b. Where the actual local size of a feature has departed from MMC toward LMC, variation in form is allowed equal to the amount of such departure.
c. There is no requirement for a boundary of perfect form at LMC. Thus, a feature produced at its LMC limit of size is permitted to vary from true form to the maximum variation allowed by the boundary of perfect form at MMC. The control of geometric form prescribed by limits of size does not apply to the following:
    i. Stock such as bar, sheet, tubing, etc. that is produced to industry or government specifications/standards that prescribe limits for straightness, flatness or other geometric characteristics. Unless geometric tolerances are specified on the drawing of a part made from these items, the specifications/standards for these parts govern the surfaces that remain in the as-furnished condition on the finished part.
    ii. Parts subject to free-state variation in the unrestrained condition (e.g. average diameter, average wall, non-rigid parts, etc.).

A surface (or surfaces) of a feature is not permitted to exceed the boundary of perfect form at MMC unless a note stating “PERFECT FORM AT MMC NOT REQUIRED” is specified on the drawing exempting the size dimension from the requirements of Rule No. 1.

4.4.3 Pitch Diameter Rule. Each tolerance of orientation or position and datum reference specified for a screw thread applies to the axis of the thread derived from the pitch cylinder. Where an exception to this practice is necessary, the specific feature of the screw thread (such as MAJOR ø or MINOR ø) shall be stated beneath the feature control frame or beneath the datum feature symbol as applicable.

4.4.4 Virtual Condition Rule. A constant boundary generated by the collective effects of a considered feature of the size’s specified MMC or LMC and the geometric tolerance for that material condition.

4.4.5 Gears and Splines. For a tolerance of orientation or position and datum reference specified for features other that threads, such as gears or splines, the drawing must designate the specific feature of the gear or spline to which each applies (such as MAJOR DIA, PITCH DIA, or MINOR).

4.5 Free-State Variation. Amount a part distorts after removal of external forces applied during manufacturing. For non-rigid parts such as thin-walled tubes, etc., free state variation is applicable unless the drawing indicates that the feature is to be restrained. Where free state variation is required, the feature tolerance shall be inspected in the free-state and be within the stated amount of tolerance. On a circular or cylindrical feature, an average diameter shall not be used unless the feature is qualified with the word “AVERAGE” or “AVG” on the drawing.

4.6 Average Diameter Measurement. An average diameter is the mean of several diameters (not less than four) across a circular or spherical part used to determine conformance to diameter tolerance only. If practicable, the average diameter may be determined by using a periphery tape. Only when a diameter is allowed a maximum circularity tolerance in the free state should it be specified as average diameter.

4.7 Average Wall Thickness. Where an average wall thickness is specified, it shall be determined by taking the average of no less than four measurements taken on the same cross sectional plane, perpendicular to a part axis.

4.8 Restrained Features. Where a non-rigid part feature is to be inspected in the restrained condition, the drawing shall have a note that indicates that the part is to be inspected in the restrained condition. The drawing shall also indicate the restraining force or dimension/size to which it is to be restrained.

4.9 Part Alignment and Orientation for Inspection. The part shall be properly aligned and oriented for inspection as required by the drawing. In the absence of specified datums, the inspector shall orient the part from implied datums selected on the basis of functional use of the part or the feature to be inspected. Contact LORD Quality Engineering to assist in defining implied datum requirements.

4.10 Plated and Coated Parts. Unless otherwise specified on the LORD engineering drawing, dimensions and tolerances apply *after* the application of specified inorganic coatings (plating or chemical treatments) but *before* the application of specified organic coatings (paint, primers, solid film lubricants, corrosion preventative compounds, etc.).

4.11 Thread Inspection. The methods of inspecting the dimensional characteristics of thread forms specified in this specification are in compliance with FED-STD-H28/20 and ASME B1.3. New or more effective methods are acceptable so long as they have been demonstrated to show and assure conformance of the threads to the drawing or specification requirements. Alternate methods of inspection and gaging not identified in this specification must be reviewed, documented, and approved by LORD Quality prior to implementation. Definitions of terms for characteristics of threads are as defined in FED-STD-H28/2 and ASME B1.7.

4.11.1 Thread Gaging. Thread inspection gages shall meet the requirements ASME B1.2, FED-STD-H28/6, and FED-STD-H28/20. The “3 finger” method is to be used when turning the thread plug/ring inspection gaging into or onto the part. The “Go” thread plug/ring gage should go on or in the part easily without force for the length of the threads. A light lubricant shall be used to prevent galling of the threads. For a “No-Go” gage, it is permissible for the thread gage to engage the part 3 full turns.

4.11.2 Depth or Length of Full Threads. The “turn method” (which utilizes thread lead) shall be used to measure the depth or length of full threads. The thread lead is the distance that a thread gage will travel in one full turn. To determine the minimum number of thread turns, perform the following calculation:

a. For English threads only: 1 ÷ Thread Pitch = Thread Lead
    i. Example: 1/4-20UNC-3B 1 ÷ 20 = .05”
b. Full Thread Requirement (from the drawing) ÷ Thread Lead = # of Full Turns
    i. English Example: 1/4-20UNC-3B x .50 Minimum Full Thread
       .50 ÷ .05 = 10 turns
    ii. Metric Example: M8x1.25-6H x 25mm Full Thread
       25 ÷ 1.25 = 20 turns
c. Note: Unless otherwise specified by the engineering drawing, the size of the lead-in chamfer of the thread must be accounted for when performing the turns count. The actual size of the chamfer shall be added to the total turns depth or length calculation to determine the full thread depth/length.

4.11.3 Plated Threads. Unless otherwise specified on the engineering drawing or by the applicable drawing specified thread specification, threads that are to be plated shall be inspected prior to plating in accordance with the specified inspection system to the adjusted dimensional requirements (i.e., before plating dimensions). After plating, a “Go” thread ring/plug gage (as applicable) shall be used to check the “Go” functional diameter condition and a “No-Go” thread ring/plug gage (as applicable) shall be used to check the minimum material pitch diameter. No other additional inspection is required.

4.11.4 Coated Threads. Unless otherwise specified on the engineering drawing or by the applicable drawing specified thread specification, coated threads shall not be inspected after the application of the coating. Thread dimensional limits normally apply *before* the application of the coatings and any inspection of the threads after coating will likely damage or remove the coating. If the drawing or specification indicates the dimensional limits apply after coating, follow the requirements of Plated Threads (paragraph 4.11.3), however inspection of the functional diameters shall be done with indicating gages. When inspecting these features, the indicating gages must be placed in or on the threads in various segments and not turned into or onto the part (to avoid damaging the coating).

4.12 Coordinate Measuring Machine (CMM). The term "CMM" refers to a Direct Computer Controlled (DCC) CMM or a manual programable CMM whenever used in this document. The use of a scanning CMM, a laser CMM, or Automated Optical Inspection (AOI) may be used as alternate equipment. Contour mapping devices may also be used as an alternate wherever applicable.

4.12.1 Centerline of Cylindrical Features. The centerline of cylindrical features shall be established using the following guidelines:

a. Number of measurements to be taken:
   - Feature tolerance less than or equal to .005" - Generate a minimum of eight single point measurements in a minimum of two planes.
   - Feature tolerance greater than .005" – Generate a minimum of six single point measurements in a minimum of two planes.
   - Depending on the length of the feature, measurements in multiple planes may be required to accurately locate the axis of the feature.

b. Using the results of the above coordinate measurements, the center of the cylindrical form shall be determined as follows:
   - Use the "maximum inscribed circle" method for internal cylindrical forms (i.e. holes).
   - Use the "minimum circumscribed circle" method for external cylindrical forms.

4.12.2 Median Plane (Centerline of Non-Cylindrical Features). The median plane shall be established using the following guidelines to determine the number of measurements to be taken.

- Feature tolerance less than or equal to .005" – Generate a minimum of eight single point measurements on each opposing surface.
- Feature tolerance greater than .005" - Generate a minimum of six single point measurements on each opposing surface.

4.13 Surface Finish (Roughness). The roughness of a surface can be measured by a number of different statistical descriptors. Among these descriptors, the Ra measure (or Arithmetic Average Roughness) is one of the most common surface roughness measures used. The Ra descriptor (measured in microinches or micrometers) basically reflects the average height of roughness irregularities (i.e., peaks and valleys) from a mean line. Unless otherwise specified by the LORD engineering drawing, surface roughness (finish) shall be measured by the Ra descriptor.

4.14 Recommended Conventions. The recommended best practices and general guidelines for the number of readings (measurement points) needed to obtain measurement results are listed in Appendix A.

4.15 Measurement Conversion Factors. Unless a specific metric value is stated on the engineering drawing, conversion of English units (specified in this document) to metric units shall be in accordance with Table 2. For more information on conversions and rounding, see ASTM SI 10 or contact LORD Quality Engineering for further direction. NOTE: Rounding may be prohibited and/or limited by contract detail requirements.

Table 2 – English to Metric Conversion Factors

English Value (inches)	Metric Value (mm)
.0005	0.013
.001	0.025
.002	0.050
.004	0.10
.005	0.13
.010	0.25
.020	0.50

4.16 Visual Inspection. For all LORD aerospace product, visual inspection acceptance criteria shall be in accordance with LORD specification PRC-T-0010. For all other product, refer to industry standard acceptance criteria or contact LORD Quality for additional information.

4.17 Inspection Equipment Calibration and Control. At LORD, all calibration and control of measuring equipment shall be in accordance with LORD Procedure LOP-396. LORD suppliers shall have a procedure for calibration and control of their measuring equipment in accordance with their Quality Management System requirements.

5.0 Measurement Methods

This section presents the recommended measuring methods for features of size as well as for geometric dimensioning and tolerance requirements. A preferred and comparable measuring method is listed for each dimensioning and tolerance callout listed. The stated preferred method(s) shall be used unless the stated equipment is not available. In these cases, a specified comparable method can be used (contact LORD Quality Engineering if support is needed). See ASME Y14.5 for additional definitions and interpretation of geometric dimensioning and tolerancing requirements.

5.1 External Diameters

Measurement Methods [1]:

	Type of Measuring Equipment (P = Preferred / C = Comparable)									
Dimensional Inspection										
Total Tolerance [2] (All Sizes)	Indicator & Standard	Micrometer (.0001")	Micrometer (.001")	Air Gage & Standard [3]	Shallow Bore Gage or Light Gage	Optical Comparator or Vision System [4]	Snap Gages	Ring Gage	Caliper (All Types)	CMM [5]
Less than .001"	**P**	C		C	C					
Equal to or greater than .001" and less than .002"	C	**P**		C [6]	C	C				C
Equal to or greater than .002" and less than .010"	C	**P**		C [6]	C	C	C			C
Equal to or greater than .010"	C	C	**P**	C [6]	C	C	C	C	C	C
Form Inspection [7]										
All Sizes and Tolerances								**P** [8]		C [9]

Notes:
1. See section 5.4 "Special Configurations" for measurement of Tubular or Cylindrical (non-rigid) Shims.
2. See paragraph 4.14 for metric equivalents.
3. The air gage system ***must*** be sized according to the overall allowable tolerance range while utilizing a two-nozzle air ring.
4. The vison system is only to be used on parts less than or equal to .125" nominal thickness (typically stamped or formed parts).
5. The CMM should only be used when it has been determined that the feature cannot be measured by another acceptable method or other methods would cause deflection in the part that would influence the measurement. Refer to paragraph 4.12.1 for additional requirements.
6. For internal inspection at LORD, contact the Quality Engineer for approval to use this system (see Note 3).
7. Form Inspection (reference paragraph 4.4.2) shall be required otherwise determined by LORD Quality during the APQP stage and documented accordingly in the APQP process.
8. The ring gage must be equal to or longer than the feature length being inspected, otherwise use the comparable method.
9. Use the "Minimum Circumscribed Cylinder" output on the CMM.

5.2 Internal Diameters and Holes

Measurement Methods:

	Type of Measuring Equipment (P = Preferred / C = Comparable)								
Dimensional Inspection									
Total Tolerance [1] (All Sizes)	Air Gage & Standard [2]	Dial Bore Gage [3]	Plug Gages [4,5,6]	Indicating Plug Gage	Hole Gage [3,7]	Micrometer, Inside	Caliper (All Types)	CMM [8]	Vison System [9]
Less than .0005”	**P**	C							
Equal to or greater than .0005” and less than .001”	**P**	C			C				
Equal to or greater than .001” and less than .004”	C [10]	**P**	C	C	C			C	C
Equal to or greater than .004” and less than .010”	C [10]	**P**	C	C	C			C	C
Equal to or greater than .010”	C [10]	C	**P**	C	C	C	C	C	C
Form Inspection [11]									
All Sizes and Tolerances			**P [12]**					C [13]	

Notes:

1. See paragraph 4.15 for metric equivalents.
2. The air gage system ***must*** be sized according to the overall allowable tolerance range while utilizing a two nozzle air plug.
3. Only two-point contact measurement devices shall be used to determine the actual size of a hole unless an “Average Diameter” is specified. The inspection equipment shall be set using a ring gage when possible.
4. Plug gages are functional in nature and include wear allowance and gage maker’s tolerance. If the plug gage rejects the part, measure with a listed measuring device (e.g., dial bore) to determine the actual size of the feature.
5. A limitation of plug gages is that they cannot detect certain hole conditions such as taper, out-of-round, hourglass-shaped and bell-mouth conditions. If these conditions are suspected and are of concern, use a listed variable measurement device to determine the actual geometric shape of the hole.
6. When verifying holes that are plated or coated, use the “Go” pin first, followed by the “No-Go”. This is to eliminate the possibility of a build-up on the ends of the holes that would affect the “No-Go” gage. Gages are to be cleaned during and after inspection of the plated/coated holes to eliminate potential build up.
7. Hole Gages include Holtest Gages, Mahr Gages, and split ball gages.
8. The CMM should only be used when it has been determined that the feature cannot be measured by another acceptable method or other methods would cause deflection in the part that would influence the measurement. Refer to paragraph 4.12.1 for additional requirements.
9. The vison system is only to be used on parts less than or equal to .125 nominal thickness (typically stamped or formed parts).
10. For internal inspection at LORD, contact the Quality Engineer for approval to use this system (see Note 2).
11. Form Inspection (reference paragraph 4.4.2) shall be required unless otherwise determined by LORD Quality during the APQP stage and documented accordingly in the APQP process.
12. The plug gage must be equal to or longer than the feature length being inspected, otherwise use the comparable method.
13. Use the “Maximum Inscribed Cylinder” output on the CMM.

5.3 Standard Features

Measurement Methods:

		Type of Measuring Equipment (P = Preferred / C = Comparable)							
Dimensional Inspection									
Feature Type	Total Tolerance [1] (All Sizes)	Micrometer	Caliper (All Types)	Indicator	Optical Comparator	CMM [2]	Height Gage [3]	Size Blocks & Plugs	Indicating Snap Gage
Height, Length or Width (outside dimensions)	Less than .005"	C		**P**	C	C	C		
	Equal to or greater than .005" and less than .020"	**P**		C	C	C	C		
	Equal to or greater than .020"	**P**	**P**	C	C	C	C		C
Width of Slots, Grooves, Recesses, etc.	Less than .005"		C	**P**	C	C		C	
	Equal to or greater than .005" and less than .020"		**P**		C	C	C	C	
	Equal to or greater than .020"		**P**		C	C	C	C	
Form Inspection [4]									
Height, Length or Width (outside dimensions)	All Sizes and Tolerances			**P**	C	C [5]	C		
Width of Slots, Grooves, Recesses, etc.	All Sizes and Tolerances					C [6]		**P**	

Notes:
1. See paragraph 4.15 for metric equivalents.
2. The CMM should only be used when it has been determined the feature cannot be measured by another acceptable method or other methods would cause deflection in the part that would influence the measurement. Dimensional (size) inspection shall be by opposing points. Refer to paragraph 4.12.2 for additional requirements.
3. Includes Micro-Hite inspection device.
4. Form Inspection (reference paragraph 4.4.2) shall be required unless otherwise determined by LORD Quality during the APQP stage and documented accordingly in the APQP process.
5. Use the "Parallel Planes (Maximum)" output on the CMM. Note: Camio CMM software identifies this algorithm as "Parallel Planes"; other CMM software manufacturers have similar algorithms but may name them differently.
6. Use the "Parallel Planes (Minimum)" output on the CMM. Note: Camio CMM software identifies this algorithm as "Parallel Planes"; other CMM software manufacturers have similar algorithms but may name them differently.

5.3 Standard Features (continued)

Measurement Methods:

		Type of Measuring Equipment (P = Preferred / C = Comparable)						
Feature Type	Total Tolerance [1] (All Sizes)	Micrometer	Caliper (All Types)	Ultrasonic Gage	Indicator	CMM [3]	Height Gage [4]	Ruler (Steel Rule)
Wall Thickness [2]	Less than .005"	**P**		C		C		
	Equal to or greater than .005" and less than .020"	**P**	C	C				
	Equal to or greater than .020"	**P**	C	C				
Depth of Features (Slot, Hole, Surfaces)	Less than .005"				**P**	C		
	Equal to or greater than .005" and less than .020"	**P**			C	C		
	Equal to or greater than .020"	**P**	C		C	C	C	C

Notes:

1. See paragraph 4.15 for metric equivalents.
2. See section 5.4 "Special Configurations" for inspection of thin wall shims.
3. The CMM should only be used to inspect these features when it has been determined they cannot be measured by another acceptable method or other methods would cause deflection in the part that would influence the measurement. Refer to paragraph 4.12.2 for additional requirements.
4. Includes Micro-Hite inspection device.

5.3 Standard Features (continued)

Measurement Methods:

		Type of Measuring Equipment (P = Preferred / C = Comparable)							
Feature Type	Total Tolerance [1] (All Sizes)	Optical Comparator/ Visual System	Radius Gages	Protractor	Sine Bar	CMM	Surface Tracer	Chamfer Micrometer	Scale (Steel Rule)
Radii	Less than .002”	**P** [2]					C		
	Equal to or greater than .002” and less than .020”	**P** [2]				C [3,4]	C		
	Equal to or greater than .020”	C [2]	**P**			C [3,4]	C		
Angles	Less than 1°	C			**P**	C	C		
	Equal to or greater than 1°	C		**P**	C	C	C		
Chamfers (internal and external)	Less than .010”	**P**						C	
	Equal to or greater than .010”							C	**P**

Notes:
1. See paragraph 4.14 for metric equivalents.
2. The shadow image of the radius which is observed on the comparator screen will be the extreme radius and may not represent the entire surface. Focus adjustments and surface illumination methods may be able to account for the entire surface. For radii with length, further inspection may be required to account for surface irregularities which are below the extreme radius or not made visible by the method used.
3. 120° of arc length is needed to accurately determine the radius.
4. Radius to be determined using the “least mean square” method.

5.4 Special Configurations

Measurement Methods:

		Type of Measuring Equipment (P = Preferred / C = Comparable)							
Feature Type	Total Tolerance (All Sizes)	Optical Comparator	CMM	Micrometer	Functional Gage/Nest	Caliper (All Types)	Indicator	Ultrasonic Gage	
Conical and Spherical Thin Wall Shims - Surface Profile	All	**P**	C						
Tubular or Cylindrical (non-rigid) Shims - Outside Diameter	All		C	C	**P**	C	C		
Thin Shim Wall Thickness	All			**P**				**P**	
Gage Diameter on a Conical Taper	All		**P** [1]						

Notes:

1. A minimum of four readings shall be taken along the conical surface in a minimum of two planes. It may be necessary to take more readings after giving consideration to surface finish, the size of the plane, and the form of the surface. Refer to paragraph 4.12.1 for additional requirements.

5.5 Thread Inspection

Unless otherwise specified on the LORD Engineering drawing or by the applicable drawing specified thread specification, all thread inspections shall be performed using the inspection system as defined in ASME B1.3 per the following thread matrix. For all System 23 thread inspections, contact the LORD Supplier Quality Engineer to review, determine, and document the specific inspection and gaging requirements. For legacy parts not inspected to this matrix, contact the LORD Supplier Quality Engineer to document the approved inspection and gaging methods.

5.5.1 Thread Matrix

Thread Standard or Type	System 21	System 22	System 23	Specialty Threads
ASME B1.1 / ASME B1.13M		X		
ASME B1.21M		X		
AS8879 / MIL-S-8879 – Category 1		X		
AS8879 / MIL-S-8879 – Category 2			X	
BS 3643-1		X		
MA1370		X		
MIL-S-7742 - “Other” Thread Category	X			
MIL-S-7742 - “Safety Critical” Thread Category			X	
“STI” Threads				X
Self Locking Thread Systems				X
Thread Standard not defined on Drawing	X			
All other Thread Standards	Contact the LORD SQE and follow the recommended inspection practices per the thread standard			

5.5.2 System 21 Thread Inspection

Measurement Methods:

		Type of Measuring Equipment (P = Preferred / C = Comparable)						
Feature Type	Thread Characteristics [1] (All Sizes)	Indicating Thread Gage Systems [2]	Thread Ring/Plug Gage [3]	Plain Ring/Plug Gages	Wires	Micrometer (.0001" Minimum Resolution)		
External Thread [5]	GO Functional Diameter	**P**	C					
	Pitch Diameter Size	**P**	C		C			
	Major Diameter	**P**		C		C		
	Full Thread Length [4]		**P**					
Internal Thread [5]	GO Functional Diameter	**P**	C					
	Pitch Diameter Size	**P**	C					
	Minor Diameter	C		**P**				
	Full Thread Depth [4]		**P**					

Notes:
1. For plated or coated threads, see paragraph 4.11.3 or 4.11.4.
2. Thread inspections systems manufactured by Johnson Gage and Tri-Star Gage satisfy the requirements for this equipment.
3. Refer to paragraph 4.11.1 for use of this gaging.
4. Refer to paragraph 4.11.2 for thread depth/length calculations.
5. For a Bell Helicopter Classified Part (i.e., Primary, Critical or Flight Safety), refer to Bell Document SQRM-001 for additional thread inspection requirements.

5.5.3 System 22 Thread Inspection

Measurement Methods:

Feature Type	Thread Characteristic [1] (All Sizes)	Type of Measuring Equipment (P = Preferred / C = Comparable): Indicating Thread Gage Systems [2]	Thread Ring/Plug Gage [3]	Plain Ring/Plug Gages	Wires	Micrometer (.0001" Minimum Resolution)	Optical Comparator (50X Minimum) [4]	AOI [5]
External Thread [7]	Maximum Material - Functional Diameter	**P**	C				C	C
	Minimum Material Pitch Diameter	**P**			C		C	C
	Minor Diameter	**P**					C	C
	Major Diameter	**P**		C		C	C	C
	Root radius						**P**	C
	Full Thread Length [6]		**P**				C	C
Internal Thread [7]	Maximum Material - Functional Diameter	**P**	C					
	Minimum Material Pitch Diameter	**P**						
	Minor Diameter	C		**P**				
	Full Thread Depth [6]		**P**					

Notes:
1. For plated or coated threads, see paragraph 4.11.3 or 4.11.4.
2. Thread inspections systems manufactured by Johnson Gage and Tri-Star Gage satisfy the requirements for this equipment.
3. Refer to paragraph 4.11.1 for use of this gaging.
4. An Optical Comparator may be considered an alternate preferred method for all characteristics when threads are inspected with a certified overlay and the threads are aligned to the helix angle specified on the overlay.
5. An Automated Optical Inspection (AOI) equipment may be considered an alternate preferred method for all indicated characteristics.
6. Refer to paragraph 4.11.2 for thread depth/length calculations.
7. For a Bell Helicopter Classified Part (i.e., Primary, Critical or Flight Safety), refer to Bell Document SQRM-001 for additional thread inspection requirements.

5.5.4 Specialty Thread Inspection

Measurement Methods:

		Type of Measuring Equipment (P = Preferred / C = Comparable)			
Feature Type	Thread Characteristics (All Sizes)	Thread Ring/Plug Gage [1]	Plain Plug Gage		
"STI" Internal Thread (i.e. Heli-coil, Helical, etc.)	Functional and Pitch Diameter	**P**			
	Minor Diameter		**P**		
	Full Thread Depth [2]	**P**			
Self Locking Internal Thread (i.e. Spiralok) [3]	Functional and Pitch Diameter	**P**			
	Full Thread Depth [2]	**P**			

Notes:

1. Refer to paragraph 4.11.1 for use of this gaging.
2. Refer to paragraph 4.11.2 for thread depth/length calculations.
3. Tooling and inspection gaging must be per the brand name as specified on the LORD Engineering drawing. As an example, if the LORD drawing specifies a "Spiralok" thread, then only Spiralok tooling and inspection gaging may be used to produce and inspect the thread. If the LORD drawing specifies "or equivalent", then the tooling and inspection gaging must be from the same manufacturer.

5.6 Spline Inspection

Measurement Methods:

Feature Type	Spline Feature (All Sizes)	Type of Measuring Equipment (P = Preferred / C = Comparable) Go/No-Go Spline Gages [1,2]	Plain Ring/Plug Gages	Indicator and/or Gage Blocks	Micrometer (.0001" Minimum Resolution)	CMM
Internal Spline	Functional Size	**P**				
	Minor Diameter		**P**			
	Measurement between Pins			**P**		C
External Spline	Functional Size	**P**				
	Major Diameter				**P**	
	Measurement over Pins				**P**	C

Notes:

1. For parts rejected by the spline gage(s), inspect the measurement over pins. The over pin measurement will not determine the functional fit of the mating part, but can be used to evaluate the effective space width (internal spline) or tooth thickness (external spline). See ANSI B92.1 for a description of analytical inspection methods for measuring other spline variations such as index, profile, lead, eccentricity, etc.
2. Unless otherwise specified, a composite "Go" spline gage and sector "No Go" gage should be used. See ANSI B92.1-1970 for description of the types of "Go" and "No Go" gages used for inspection of splines.

5.7 Form Features

Form tolerances are applicable to single (individual) features or elements of single features; therefore form tolerances are not related to datums. The following subparagraphs cover inspection methods for flatness, straightness, circularity and cylindricity form tolerances.

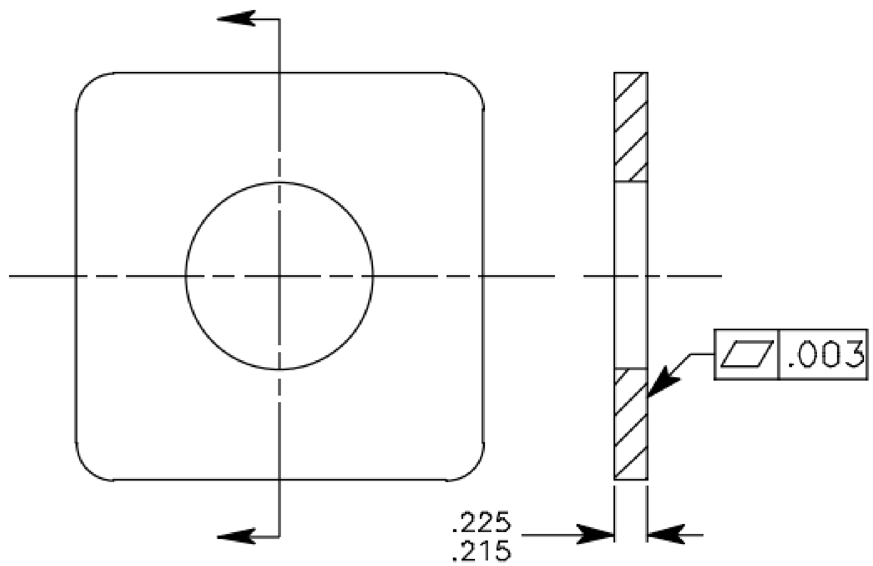
5.7.1 Flatness

Definition: Flatness is the condition of a surface or derived median plane having all elements in one plane.

Tolerance: A flatness tolerance specifies a tolerance zone defined by two parallel planes within which the surface or derived median plane must lie.

Key Points:
1. Datum references and feature modifiers are not applicable to flatness. Flatness is to itself.
2. Flatness must be measured and cannot be gaged.

Example:



Meaning: The indicated surface is to be flat within a .003" wide tolerance zone over the entire surface and within the size tolerance limits. If "PERFECT FORM AT MMC NOT REQUIRED" is identified for the size requirement, then the flatness tolerance zone may exceed the size tolerance limits.

5.7.1 Flatness (continued)

Measurement Methods:

Total Tolerance [1,2]	Measurement Resolution [1]	Preferred Measurement Methods (see "Method" below)	Comparable Measurement Methods
Equal to or less than .0005"	.000025" Max	Method 1	CMM [3,4]
Less than or equal to .001" and greater than .0005"	.00005" or 10:1	Method 1	CMM [3,4]
Less than or equal to .005" and greater than .001"	.0001" or 10:1	Method 1	CMM [3]
Less than .010" and greater than .005"	.0005" or 10:1	Methods 1 or 3	CMM [3]
Equal to or greater than .010"	.001" or 10:1	Methods 1, 2, or 3	CMM [3]

Notes:
1. See paragraph 4.14 for metric equivalents.
2. For flatness tolerances of .0001" or less, check the flatness of the feature using optical flats.
3. The CMM is not the preferred method to inspect form tolerances and should only be used when another method is not available.
4. Contact LORD Quality Engineering if a CMM will be used to inspect a form tolerance ≤ .001".

Procedure: To inspect flatness, an optimum plane must be established and a FIM measurement made of the controlled surface with reference to this plane. The open set-up methods listed above to measure flatness are performed as follows:

Method 1: Mount the part on three jackscrews or gage block stacks or pins with the surface to be controlled facing down and sweep the surface with an indicator to find the resultant full indicator movement (FIM) of flatness.

Method 2: Place the part on a surface plate with surface to be controlled facing down and check with feeler gages. This method can be used to verify non-conformity only.

Method 3: Place the part on a surface plate with surface to be controlled facing up and sweep the surface with an indicator to find the resultant FIM of parallelism. If the resultant FIM is within the flatness specification the part is conforming. If the resultant FIM is greater than the flatness specification, the part must be inspected using method 1.

5.7.2 Straightness - Cylindrical Parts (Surface Element Control)

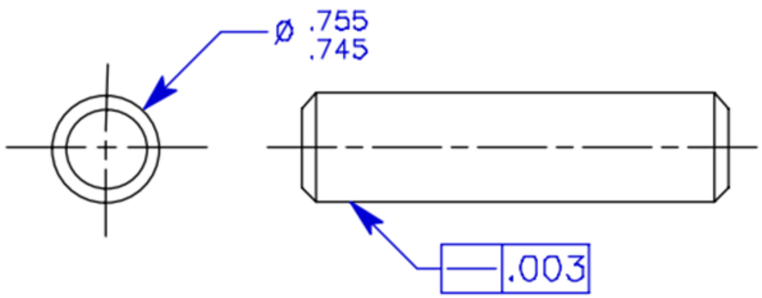
Definition: Straightness is a condition where an element of a surface or a derived median line is a straight line.

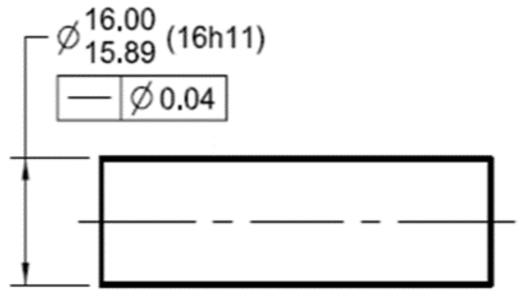
Tolerance: A straightness tolerance specifies a tolerance zone within which the considered line element or derived median line must lie.

Key Points:

1. Straightness of a line element tolerance is applied in the view where the elements to be controlled are represented by a straight line.
2. Datum references are not applicable to any straightness tolerances.
3. Straightness must be measured and cannot be gaged.
4. Perfect form at MMC does not apply when a straightness of a derived median line is applied to a feature.

Examples:





Meaning:

The feature must be within the specified .755/.745 Diameter tolerance of size and the boundary of perfect form at MMC (.755). Each longitudinal element of the surface must lie between two parallel lines .003" apart, where the two lines and the axis of the unrelated actual mating envelope of the feature share a common plane. If "PERFECT FORM AT MMC NOT REQUIRED" is identified for the straightness requirement, then the straightness tolerance zone may exceed the size tolerance limits

The derived median line of the feature's actual local size must lie within a cylindrical tolerance zone of 0.04 mm diameter, regardless of the feature size. Each circular element of the surface must be within the specified limits of size.

5.7.2 Straightness - Cylindrical Parts (Surface Element Control) (continued)

Measurement Methods:

Total Tolerance [1]	Measurement Resolution [1]	Preferred Measurement Methods	Comparable Measurement Methods
Equal to or less than .0005”	.000025” Max	Amplified indicator and centers or V-blocks	CMM [2,3]
Less than or equal to .001” and greater than .0005”	.00005” or 10:1	Amplified indicator and precision collet, centers, or V-blocks	CMM [2,3]
Less than or equal to .005” and greater than .001”	.0001” or 10:1	.0001” indicator and precision collet, centers, or V-blocks	CMM [2]
Less than .010” and greater than .005”	.0005” or 10:1	.0005” indicator and collet, centers, or V-blocks	CMM [2]
Equal to or greater than .010”	.001” or 10:1	.001” indicator and collet, centers, or V-blocks	Surface plate and feeler gage or indicator or a CMM [2]

Notes:
1. See paragraph 4.14 for metric equivalents.
2. The CMM is not the preferred method to inspect form tolerances and should only be used when another method is not available.
3. Contact LORD Quality Engineering if a CMM will be used to inspect a form tolerance ≤ .001”.

Procedure: Inspection of straightness for a cylindrical part requires that the surface elements of the part be compared to an imaginary optimum line. To inspect the part it must be mounted in a collet, center, or in a V-block. The part surface should be leveled to the surface plate and then traverse a dial indicator longitudinally across the part and measure the resultant FIM. Then rotate the part and repeat on another line element of the surface. At least four-line elements 90° apart should be inspected.

The part may also be placed on top of two equal height blocks and an indicator traversed longitudinally beneath the part and measure the resultant full indicator movement (FIM). Then rotate the part and repeat on another line element of the surface. At least four-line elements 90° apart should be inspected.

5.7.3 Straightness - Flat Surface

Definition: Straightness is a condition where an element of a surface or an axis is a straight line.

Tolerance: The individual straightness elements must lie within both the size tolerance and the straightness tolerance zone.

Key Points:

1. The tolerance is applied in the view in which the elements to be controlled appear as a straight line.
2. Straightness tolerance applied to a flat surface provides surface element control in a specific direction as a refinement of size tolerance or other form tolerances.
3. The straightness tolerance must be less than the size tolerance when used as a refinement of the size tolerance.
4. Straightness must be measured and cannot be gaged.

Example:

Meaning: The individual straightness elements must lie within both the size tolerance and the straightness tolerance zone of .003", whereas element-to-element variation in the side view, may vary within the size tolerance. If "PERFECT FORM AT MMC NOT REQUIRED" is identified for the straightness requirement, then the straightness tolerance zone may exceed the size tolerance limits.

5.7.3 Straightness - Flat Surface (continued)

Measurement Methods:

Total Tolerance [1]	Measurement Resolution [1]	Preferred Measurement Methods (see "Method" below)	Comparable Measurement Methods
Equal to or less than .0005"	.000025" Max	Method 1	CMM [2,3]
Less than or equal to .001" and greater than .0005"	.00005" or 10:1	Method 1	CMM [2,3]
Less than or equal to .005" and greater than .001"	.0001" or 10:1	Method 1	CMM [2]
Less than .010" and greater than .005"	.0005" or 10:1	Method 1	CMM [2]
Equal to or greater than .010"	.001" or 10:1	Method 1 or 2	CMM [2]

Notes:
1. See paragraph 4.14 for metric equivalents.
2. The CMM is not the preferred method to inspect form tolerances and should only be used when another method is not available.
3. Contact LORD Quality Engineering if a CMM will be used to inspect a form tolerance ≤ .001".

Procedure: To inspect straightness of a flat surface, an optimum plane must be established and a FIM measurement made of the controlled surface with reference to this plane. The open set-up methods listed above to measure straightness of a flat surface are performed as follows:

Method 1: Mount the part on three jackscrews or gage block stacks or pins with the surface to be controlled facing down and sweep the surface longitudinally in a straight line with an indicator to find the resultant full indicator movement (FIM) of flatness. Repeat on at least three other line elements on the surface.

Method 2: Place the part on a surface plate with surface to be controlled facing down and check with feeler gages. This method can be used to verify non-conformity only.

5.7.4 Circularity - Cylinder

Definition: Circularity is a condition of a surface where:

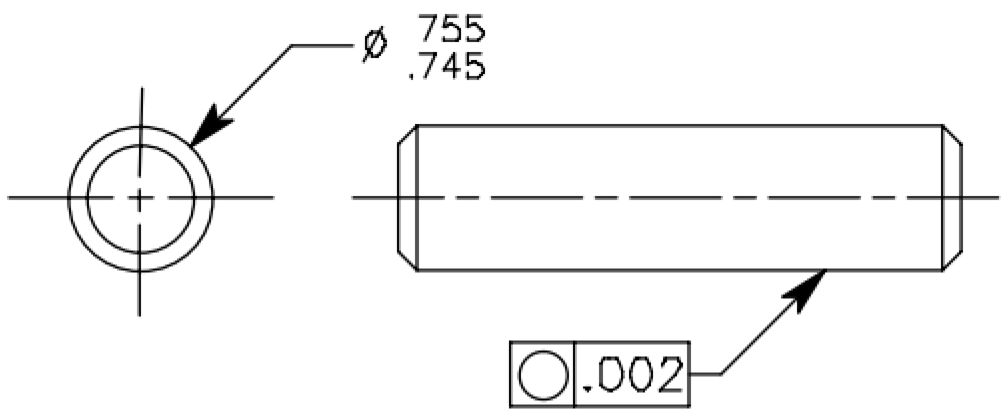
a. for a feature other than a sphere, all points of the surface intersected by any plane perpendicular to an axis are equidistant from that axis;
b. for a sphere, all points of the surface intersected by any plane passing through a common center are equidistant from that center.

Tolerance: A circularity tolerance specifies a tolerance zone bounded by two concentric circles within which each circular element of the surface must lie, and applies independently at any plane.

Key Points:

1. Circularity must be measured and cannot be gaged.
2. Circularity is regardless of feature size.
3. Datum references are not applicable to circularity.

Example:



Meaning: The surface periphery at any cross section perpendicular to the feature axis must be within the specified tolerance of size (.755/.745) and must lie between two concentric circles, one having a radius .002" larger than the other.

5.7.4 Circularity - Cylinder (continued)

Measurement Methods:

Total Tolerance [1]	Measurement Resolution [1]	Preferred Measurement Methods [2] (see "Method" below)	Comparable Measurement Methods
Equal to or less than .0005"	.000025" Max	Method 1	Circular geometry machine or CCM [3,4]
Less than or equal to .001" and greater than .0005"	.00005" or 10:1	Method 1	Circular geometry machine or CCM [3,4]
Less than or equal to .005" and greater than .001"	.0001" or 10:1	Method 1	Circular geometry machine or CCM [3]
Less than .010" and greater than .005"	.0005" or 10:1	Method 1	Circular geometry machine or CCM [3]
Equal to or greater than .010"	.001" or 10:1	Method 1 or 2	Circular geometry machine or CCM [3]

Notes:
1. See paragraph 4.14 for metric equivalents.
2. See ANSI B89.3.1-1972 for a detailed explanation of measuring out-of-roundness (circularity) of a surface of revolution.
3. The CMM is not the preferred method to inspect form tolerances and should only be used when another method is not available.
4. Contact LORD Quality Engineering if the CMM will be used to inspect a form tolerance ≤ .001".

Procedure: Method 1: Place the part in the V-blocks, centers, or a collet. Place the indicator on a section of the diameter and rotate the part 360°. Determine the FIM value, move the indicator to another circular section of the part, and repeat. The part is acceptable if the FIM, at all circular sections, is within the circularity requirement.

Method 2: Measure a circular section of the diameter in four places 45° apart and find the difference between the largest and smallest measurement. Divide the measurement by 2 to establish the radial out of roundness and repeat the measurement on other circular sections of the diameter. The part is acceptable if measurement (divided by 2) at all circular sections is within the circularity requirement.

5.7.5 Circularity - Cone

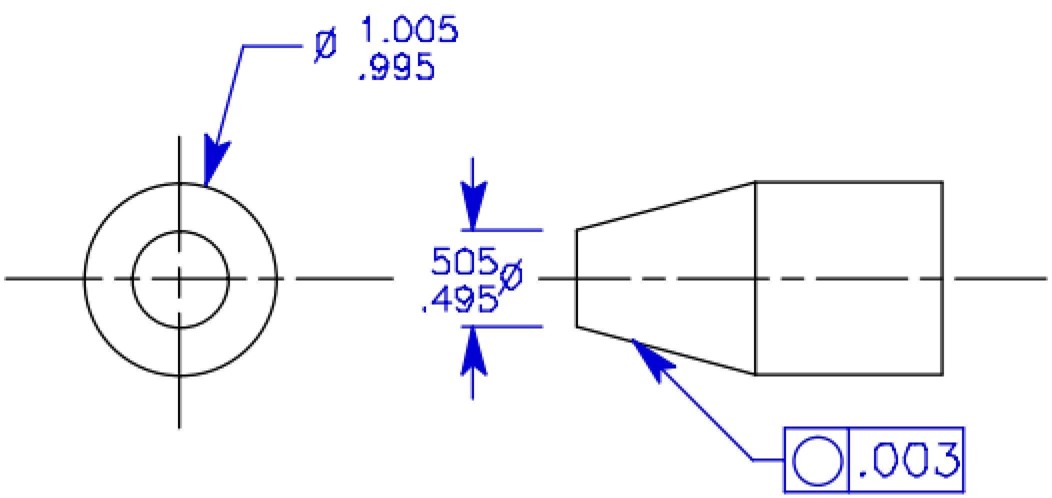
Definition: Circularity is a condition of a surface where:

a. for a feature other than a sphere, all points of the surface intersected by any plane perpendicular to an axis are equidistant from that axis;
b. for a sphere, all points of the surface intersected by any plane passing through a common center are equidistant from that center.

Tolerance: A circularity tolerance specifies a tolerance zone bounded by two concentric circles within which each circular element of the surface must lie and applies independently at any plane.

Key Points:
1. Circularity must be measured and cannot be gaged.
2. Circularity is regardless of feature size.
3. Datum references are not applicable to circularity.

Example:



Meaning: The periphery at any cross-section perpendicular to the feature axis must be within the specified tolerance of size and must lie between two concentric circles, one having a radius .003" larger than the other.

5.7.5 Circularity - Cone (continued)

Measurement Methods:

Total Tolerance [1]	Measurement Resolution [1]	Preferred Measurement Methods	Comparable Measurement Methods
Equal to or less than .0005”	.000025” Max	Amplified indicator and a precision collet	Circular geometry machine or CCM [2,3]
Less than or equal to .001” and greater than .0005”	.00005” or 10:1	Amplified indicator and a precision collet	Circular geometry machine or CCM [2,3]
Less than or equal to .005” and greater than .001”	.0001” or 10:1	.0001” indicator and a collet or centers	Circular geometry machine or CCM [2]
Less than .010” and greater than .005”	.0005” or 10:1	.0005” indicator and a collet or centers	Circular geometry machine or CCM [2]
Equal to or greater than .010”	.001” or 10:1	.001” indicator and a collet or centers	Circular geometry machine or CCM [2]

Notes:
1. See paragraph 4.14 for metric equivalents.
2. The CMM is not the preferred method to inspect form tolerances and should only be used when another method is not available.
3. Contact LORD Quality Engineering if the CMM will be used to inspect a form tolerance ≤ .001”.

5.7.6 Circularity – Sphere

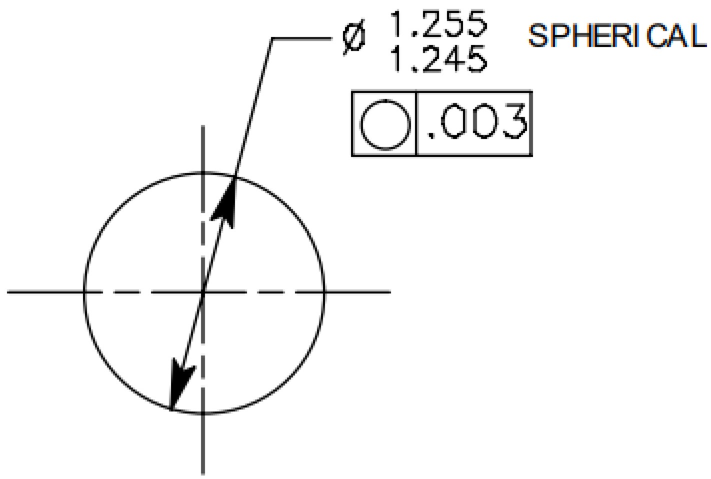
Definition: Circularity is a condition of a surface where:

a. for a feature other than a sphere, all points of the surface intersected by any plane perpendicular to an axis are equidistant from that axis;
b. for a sphere, all points of the surface intersected by any plane passing through a common center are equidistant from that center.

Tolerance: A circularity tolerance specifies a tolerance zone bounded by two concentric circles within which each circular element of the surface must lie and applies independently at any plane.

Key Points:
1. Circularity must be measured and cannot be gaged.
2. Circularity is regardless of feature size.
3. Datum references are not applicable to circularity.

Example:



Meaning: The periphery at any cross section passing through a common center must be within the specified tolerance of size (1.255/1.245) and must be between two concentric circles, one having a radius .003" larger than the other.

5.7.6 Circularity - Sphere (continued)

Measurement Methods:

Total Tolerance [1]	Measurement Resolution [1]	Preferred Measurement Methods	Comparable Measurement Methods
Equal to or less than .0005”	.000025” Max	Surface plate, holding device, and amplified indicator	Circular geometry machine or CCM [2,3]
Less than or equal to .001” and greater than .0005”	.00005” or 10:1	Surface plate, holding device, and amplified indicator	Circular geometry machine or CCM [2,3]
Less than or equal to .005” and greater than .001”	.0001” or 10:1	Surface plate, holding device, and .0001” indicator	Circular geometry machine or CCM [2]
Less than .010” and greater than .005”	.0005” or 10:1	Surface plate, holding device, and .0005” indicator	Circular geometry machine or CCM [2]
Equal to or greater than .010”	.001” or 10:1	Surface plate, holding device, and .001” indicator	Circular geometry machine or CCM [2]

Notes:
1. See paragraph 4.14 for metric equivalents.
2. The CMM is not the preferred method to inspect form tolerances and should only be used when another method is not available.
3. Contact LORD Quality Engineering if the CMM will be used to inspect a form tolerance ≤ .001”.

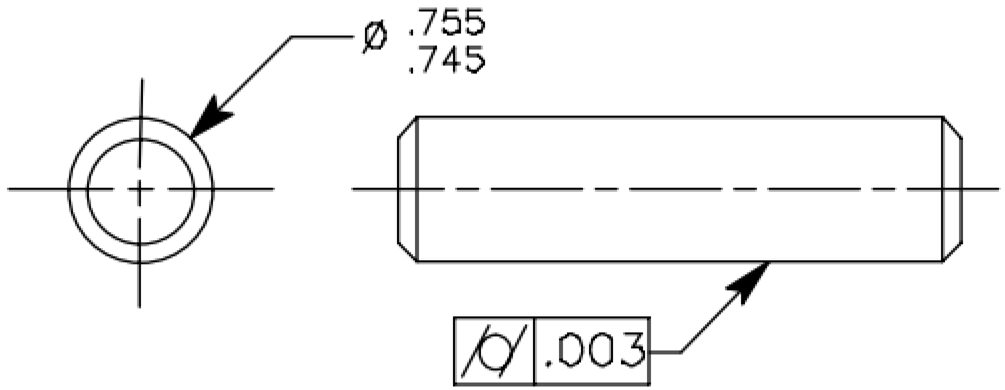
5.7.7 Cylindricity

Definition: Cylindricity is a condition of a surface of revolution in which all points of the surface are equidistant from a common axis.

Tolerance: A cylindricity tolerance specifies a tolerance zone bounded by two concentric cylinders within which the surface must lie.

Key Points:
1. Datum references and feature modifiers are not applicable to cylindricity.
2. Cylindricity simultaneously controls roundness at each circularity, straightness, and parallelism of the elements of the cylindrical surface.
3. The cylindricity tolerance must always be contained within the part size tolerance range.

Example:



Meaning: The feature must be within the specified tolerance of size (.755/.745) and must lie between two concentric cylinders, one having a radius .003” larger than the other.

5.7.7 Cylindricity (continued)

Measurement Methods:

Total Tolerance [1]	Measurement Resolution [1]	Preferred Measurement Methods	Comparable Measurement Methods
Equal to or less than .0005”	.000025” Max	CMM [2] or circular geometry machine	Surface plate, amplified indicator, and collet or centers
Less than or equal to .001” and greater than .0005”	.00005” or 10:1	CMM [2] or circular geometry machine	Surface plate, amplified indicator, and collet or centers
Less than or equal to .005” and greater than .001”	.0001” or 10:1	CMM or circular geometry machine	Surface plate, amplified indicator, and collet or centers
Less than .010” and greater than .005”	.0005” or 10:1	CMM or circular geometry machine	Surface plate, .0001” indicator, and collet or centers
Equal to or greater than .010”	.001” or 10:1	CMM or circular geometry machine	Surface plate, .001” indicator, and collet or centers

Notes:
1. See paragraph 4.14 for metric equivalents.
2. Contact LORD Quality Engineering if the CMM will be used to inspect a form tolerance ≤ .001”.

Procedure: The preferred open set-up inspection methods listed do not evaluate cylindricity precisely. The between centers method is a check of total runout which is used to verify the cylindricity is within specification since when applied to a part diameter it controls many different types of geometric errors. As such, if the composite of all these geometric errors does not exceed the stated cylindricity tolerance, then the cylindricity must be within tolerance.

5.8 Profile Control

A profile tolerance may be applied to an entire surface or to individual profiles taken at various cross sections through the part. The following subparagraph covers inspection methods for profile of a line and profile of a surface.

NOTE: ASME Y14.5-2009 has placed a stronger emphasis on profile controls replacing feature controls such as perpendicularity, parallelism, straightness, etc. Consideration must be made to understand that the profile control now strongly implies additional orientation requirements and is no longer limited to historical form characteristics.

5.8.1 Profile of a Line/Surface

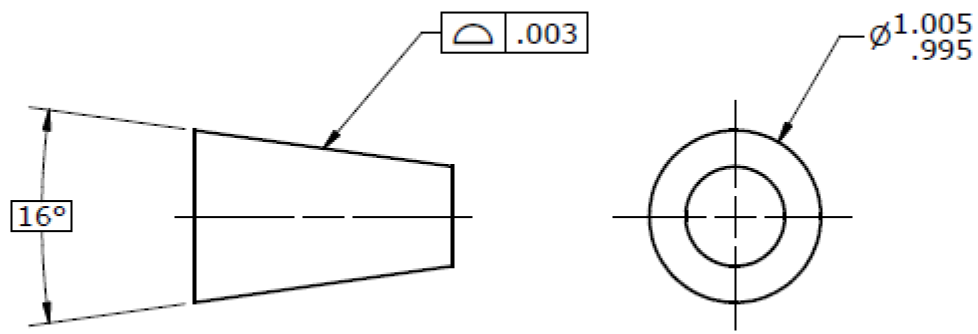
Definition: Profile tolerancing specifies a uniform or non-uniform boundary along the true profile within which the elements of the surface must lie.

Tolerance: Profile of a Line - The tolerance zone established by profile of a line tolerance is a two-dimensional zone extending along the length of the considered feature; it may apply to the profiles of parts having a varying cross section or to random cross sections of parts where it is not desired to control the entire surface as a single entity.

Profile of a Surface - The tolerance zone established by profile of a surface tolerance is a three-dimensional zone or total control across the entire length and width or circumference of the feature; it may be applied to parts having a constant cross section or to parts having a surface of revolution.

Key Points:
1. The tolerance zones for profile automatically take the shape of the surface being controlled and surround the basic profile bilaterally, unless phantom lines are used to show unilateral direction or the unequally disposed symbol (Ⓤ) is used in the feature control frame to show unequal or unilateral direction.
2. Datum references may or may not be used in the profile feature.
3. Feature modifiers (LMC, MMC, and RFS) cannot be applied to the profile tolerance; however, MMB and LMB may be applied to the datum references in certain applications.

Example:



Meaning: The profile of a surface in this example is used as a form control of the 16° cone and as a refinement within the size control. The surface must lie within the specified tolerance of size (1.005/.995) and must lie between two 16° coaxial conical boundaries .003” apart.

5.8.1 Profile of a Line/Surface (continued)

Measurement Methods:

Total Tolerance [1]	Measurement Resolution [1]	Preferred Measurement Methods	Comparable Measurement Methods
Equal to or less than .0005"	.000025" Max	CMM [2,3]	Not Applicable
Less than or equal to .001" and greater than .0005"	.00005" or 10:1	CMM [2,3]	Comparator (32X min) [4]
Less than or equal to .005" and greater than .001"	.0001" or 10:1	CMM [2,3]	Comparator (20X min) [4]
Less than .010" and greater than .005"	.0005" or 10:1	CMM [2,3]	Comparator (10X min) [4]
Equal to or greater than .010"	.001" or 10:1	CMM [2,3]	Comparator (10X min) [4]

Notes:

1. See paragraph 4.14 for metric equivalents.
2. For inspection of a line/surface profile on an internal feature, CMM shall be the primary inspection method.
3. A sufficient number of measurement points must be obtained over the controlled surface to adequately define the geometry (profile) of that surface.
4. The shadow image of the part profile which is seen on the comparator screen will be the extreme profile and may not represent the entire surface. Focus adjustments and surface illumination methods may be able to account for the entire surface. If necessary, further inspection may be required to account for surface irregularities which are below the extreme profile or not made visible by the method used.

Procedure: Where overall size permits, use of an optical comparator is usually the most economical and effective method for checking profile tolerances on contours. Where a datum is not specified in the profile callout, the part can be rotated or moved to see if the line element being viewed will lie between the tolerance lines on the profile chart. If a datum reference is specified, both the shape and location are simultaneously controlled. In this case the part cannot be moved or rotated for best fit on the comparator chart. One drawback of visual methods on an optical comparator is that the comparator only sees the outline of the part (see Note 4 above). For this reason, visual methods work well only with parts with thin cross sections.

5.9 Orientation Features

Orientation tolerances control the orientation of features to one another. The following subparagraphs cover inspection methods for perpendicularity, angularity, and parallelism orientation tolerances.

5.9.1 Perpendicularity - Surface to a Datum Plane or Axis

Definition: Perpendicularity is the condition of a surface, center plane, or axis at a right angle (90°) to a datum plane or axis.

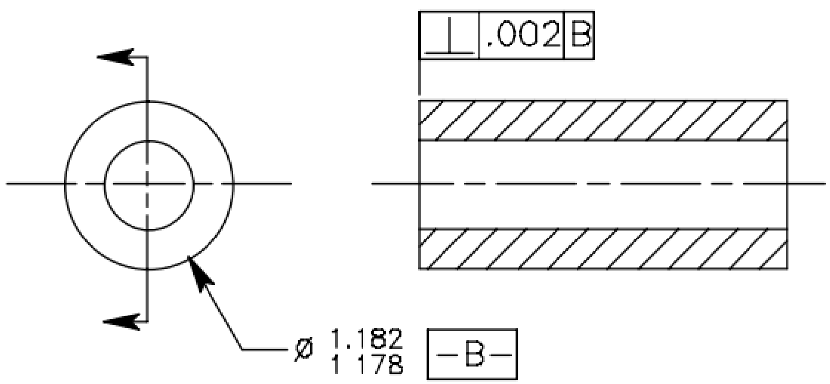
Tolerance: A perpendicularity tolerance specifies one of the following:

a. a tolerance zone defined by two parallel planes perpendicular to a datum plane or axis within which the surface of a feature must lie or the center plane of a feature must lie.
b. a tolerance zone defined by two parallel planes perpendicular to a datum axis within which the axis of the considered feature must lie.
c. a cylindrical tolerance zone perpendicular to a datum plane within which the axis of the considered feature must lie.
d. a tolerance zone defined by two parallel lines perpendicular to a datum plane or axis within which a line element of the surface must lie.

Key Points:

1. Perpendicularity tolerances must be related to a specific datum reference.
2. Perpendicularity of size features at MMC can be gaged, but other applications (e.g., planes or RFS features) must be measured.

Example:



Meaning: The surface must be within the specified tolerance of size and must lie between two parallel planes (.002" apart) which are perpendicular to the datum B axis.

5.9.1 Perpendicularity - Surface to a Datum Plane or Axis (continued)

Measurement Methods:

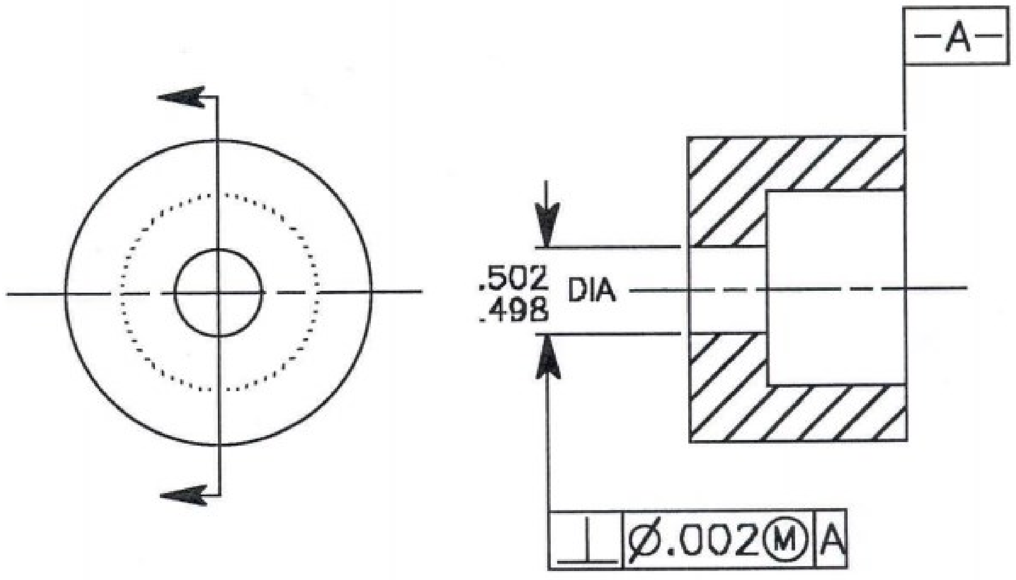
Total Tolerance [1]	Measurement Resolution [1]	Preferred Measurement Methods	Comparable Measurement Methods
Equal to or less than .0005”	.000025” Max	Surface plate, amplified indicator, and a right angle plate or precision parallel	CMM
Less than or equal to .001” and greater than .0005”	.00005” or 10:1	Surface plate, amplified indicator, and a right angle plate or precision parallel	CMM
Less than or equal to .005” and greater than .001”	.0001” or 10:1	Surface plate, .0001” indicator, and a right angle plate, precision parallel, or V-blocks	CMM
Less than .010” and greater than .005”	.0005” or 10:1	Surface plate, .0005” indicator, and a right angle plate, precision parallel, or V-blocks	CMM
Equal to or greater than .010”	.001” or 10:1	Surface plate, .001” indicator, and a right angle plate, precision parallel, or V-blocks	CMM

Notes:
1. See paragraph 4.14 for metric equivalents.

Procedure: Inspection involves staging the part on the primary datum and then aligning with the secondary datum (if specified) or another part surface that is perpendicular to the controlled surface (if no secondary datum is specified). A sweep of the controlled surface is made with a dial indicator to obtain a FIM which is the perpendicularity error.

5.9.2 Perpendicularity - Cylindrical Feature (Axis) at Maximum Material Condition to a Datum Plane

Example:



Meaning: The .502/.498 Diameter feature axis must be perpendicular within a .002” tolerance zone at MMC size (.498”) to datum plane A. As the feature size departs from MMC (gets bigger), an increase in tolerance is permitted equal to that amount of departure. When the diameter feature is at LMC size (.502”), a .006” Diameter tolerance zone is allowed (.002” position tolerance and .004” size tolerance).

5.9.2 Perpendicularity - Cylindrical Feature (Axis) at Maximum Material Condition to a Datum Plane (continued)

Measurement Methods:

Total Tolerance [1]	Measurement Resolution [1]	Preferred Measurement Methods [2]	Comparable Measurement Methods
Equal to or less than .0005”	.000025” Max	CMM	Not Applicable
Less than or equal to .001” and greater than .0005”	.00005” or 10:1	CMM	Not Applicable
Less than or equal to .005” and greater than .001”	.0001” or 10:1	Surface plate, .0001” indicator, and a right angle plate and/or V-blocks (and for holes, the largest pin/plug that will fit inside the hole)	CMM or Functional Gage [3]
Less than .010” and greater than .005”	.0005” or 10:1	Surface plate, .0005” indicator, and a right angle plate and/or V-blocks (and for holes, the largest pin/plug that will fit inside the hole)	CMM or Functional Gage [3]
Equal to or greater than .010”	.001” or 10:1	Surface plate, .001” indicator, and a right angle plate and/or V-blocks (and for holes, the largest pin/plug that will fit inside the hole)	CMM or Functional Gage [3]

Notes:
1. See paragraph 4.14 for metric equivalents.
2. Vertical measuring systems (where available) can be used as a Preferred Method.
3. A functional gage cannot be used to inspect Perpendicularity of a cylindrical feature at RFS

Procedure: Inspection involves making a setup that proves the axis of the cylindrical feature is within the stated tolerance. The perpendicularity of the axis of a hole to a datum plane can be inspected by mounting/clamping the part on the datum surface and inserting the largest pin that will fit inside the hole to be controlled and is twice the length of the part. Measure the lean of the pin on a length equal to the length of the part using an indicator (record the FIM). Rotate the part 90° and repeat the measurement on the pin. For the final perpendicularity calculation: square each individual indicator reading, add the two squares together, and then take the square root of the sum. If the resultant number is less than or equal to the perpendicularity requirement, the part is acceptable.

5.9.3 Angularity

Definition: Angularity is the condition of a surface, axis, or median plane, which is at a specified angle (other than 90°) from a datum plane or axis.

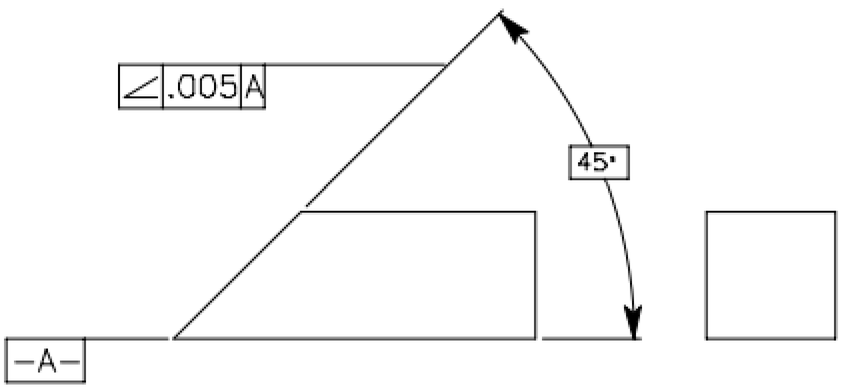
Tolerance: An angularity tolerance specifies one of the following:

a. a tolerance zone defined by two parallel planes at the specified basic angle from one or more datum planes, or a datum axis, within which the surface or center plane of the considered feature must lie.
b. a tolerance zone defined by two parallel planes at the specified basic angle from one or more datum planes, or a datum axis, within which the axis of the considered feature must lie.
c. a cylindrical tolerance zone whose axis is at the specified basic angle from or more datum planes, or a datum axis, within which the axis of the considered feature must lie.
d. a tolerance zone defined by two parallel planes at the specified basic angle from a datum plane, or axis, within which a line element of the surface must lie.

Key Points:

1. Angularity tolerances must be specified with respect to at least one datum reference.
2. Size features controlled with angularity tolerances are automatically RFS applications per rule #2 (unless MMC or LMC is stated in the feature control frame). This only applies to ASME Y14.5M-1994 and later conventions.
3. Angularity does not control size or location of features.

Example:



Meaning: Indicated surface must be at 45° within a .005" wide tolerance zone in relation to datum plane A.

5.9.3 Angularity (continued)

Measurement Methods:

Total Tolerance [1]	Measurement Resolution [1]	Preferred Measurement Methods	Comparable Measurement Methods
Equal to or less than .0005”	.000025” Max	CMM	Not Applicable
Less than or equal to .001” and greater than .0005”	.00005” or 10:1	CMM	Not Applicable
Less than or equal to .005” and greater than .001”	.0001” or 10:1	Surface plate, .0001” indicator, and sine plate or bar.	CMM
Less than .010” and greater than .005”	.0005” or 10:1	Surface plate, .0005” indicator, and angle plate or sine bar/plate	CMM or Comparator
Equal to or greater than .010”	.001” or 10:1	Surface plate, .001” indicator, and angle plate or sine bar/plate	CMM or Comparator

Notes:
1. See paragraph 4.14 for metric equivalents.

Procedure: Inspection of angularity involves making measurements with respect to the basic angle from the indicated datum. Angularity can be checked by the following methods:

Angle/Sine Plate - Mount the part to an angle plate attached to sine plate which is set at the basic angle and place on a surface plate. The part is to be mounted so that the surface to be controlled is parallel to the surface plate. Then traverse an indicator over the surface to find the FIM value.

Comparator - Stage the part on a rotary table on a comparator and check the projected image against a comparator chart with the specified angular tolerance zone drawn on it.

5.9.4 Parallelism - Surface to a Datum Plane

Definition: Parallelism is the condition of a surface or center plane, equidistant at all points from a datum plane; or axis, equidistant along its length from one or more datum planes or a datum axis.

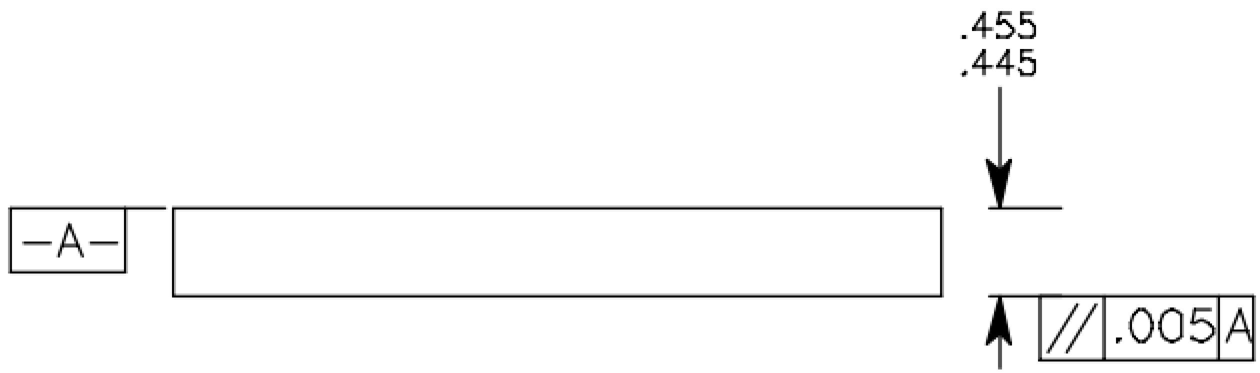
Tolerance: A parallelism tolerance specifies one of the following:

a. a tolerance zone defined by two parallel planes parallel to a datum plane or axis within which the surface or center plane of the considered feature must lie.
b. a tolerance zone defined by two parallel planes parallel to a datum plane or axis, within which the axis of the considered feature must lie.
c. a cylindrical tolerance zone parallel to one or more datum planes or a datum axis, within which the axis of the feature must lie.
d. a tolerance zone defined by two parallel lines parallel to a datum plane or axis, within which the line element of the surface must lie.

Key Points:

1. Parallelism tolerances must be specified with respect to at least one datum reference.
2. Parallelism (surface to datum plane) cannot be functional gaged; it must be measured.

Example:



Meaning: The indicated surface must be within the tolerance of size (.455/.445) and must lie between two planes .005” apart which are parallel to the datum plane A.

5.9.4 Parallelism - Surface to a Datum Plane (continued)

Measurement Methods:

Total Tolerance [1]	Measurement Resolution [1]	Preferred Measurement Methods	Comparable Measurement Methods
Equal to or less than .0005”	.000025” Max	Surface plate and amplified indicator	CMM
Less than or equal to .001” and greater than .0005”	.00005” or 10:1	Surface plate and amplified indicator	CMM
Less than or equal to .005” and greater than .001”	.0001” or 10:1	Surface plate and .0001” indicator	CMM
Less than .010” and greater than .005”	.0005” or 10:1	Surface plate and .0005” indicator	CMM
Equal to or greater than .010”	.001” or 10:1	Surface plate and .001” indicator	CMM

Notes:

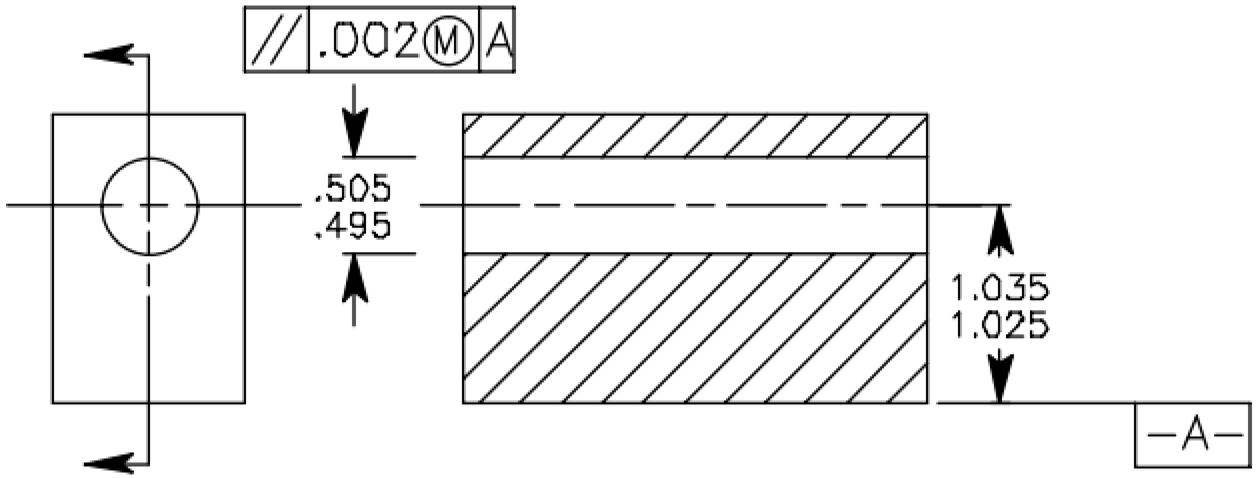
1. See paragraph 4.14 for metric equivalents.

Procedure: Parallelism is inspected by locating the part onto a surface plate so that the datum surface is resting on the surface plate (i.e. with the surface to be controlled facing up) and the entire surface swept with an indicator to find the resultant FIM.

5.9.5 Parallelism - Axis of Hole at MMC to a Datum Plane

Key Points:
1. Parallelism tolerances must be specified with respect to at least one datum reference.
2. Parallelism of size features at MMC can be functional gaged.

Example:



Meaning: The feature must be within the specified tolerance of location (1.035/1.025). The hole (at MMC) axis must lie between two parallel planes (.002" apart) which are parallel to datum plane A. As the feature size departs from MMC (gets bigger), an increase in tolerance is permitted equal to that amount of departure. When the diameter feature is at LMC size (.505"), a .012" tolerance zone is allowed (.002" parallelism and .010" size tolerance).

5.9.5 Parallelism - Axis of Hole at MMC to a Datum Plane (continued)

Measurement Methods:

Total Tolerance [1]	Measurement Resolution [1]	Preferred Measurement Methods	Comparable Measurement Methods
Equal to or less than .0005”	.000025” Max	(1) Surface plate, amplified indicator, and the largest gage pin that will fit the hole or (2) a functional gage	CMM
Less than or equal to .001” and greater than .0005”	.00005” or 10:1	(1) Surface plate, amplified indicator, and the largest gage pin that will fit the hole or (2) a functional gage	CMM
Less than or equal to .005” and greater than .001”	.0001” or 10:1	(1) Surface plate, amplified indicator, and the largest gage pin that will fit the hole or (2) a functional gage	CMM
Less than .010” and greater than .005”	.0005” or 10:1	(1) Surface plate, .0005” indicator, and the largest gage pin that will fit the hole or (2) a functional gage	CMM
Equal to or greater than .010”	.001” or 10:1	(1) Surface plate, .001” indicator, and the largest gage pin that will fit the hole or (2) a functional gage	CMM

Notes:
1. See paragraph 4.14 for metric equivalents.

Procedure: Select the largest pin that will fit the controlled hole (and is twice the length of the hole) and insert into hole so that it extends out each side of the hole. Locate the part datum on the surface plate, and using the indicator, find the top center of the pin immediately adjacent to the hole entrance on one side and set zero point on indicator. Move the indicator to the opposite entrance of the hole and take a FIM reading at the top center of the pin. The FIM value must be equal to or less than the parallelism tolerance.

5.10 Runout

The following subparagraphs cover inspection methods for circular and total runout.

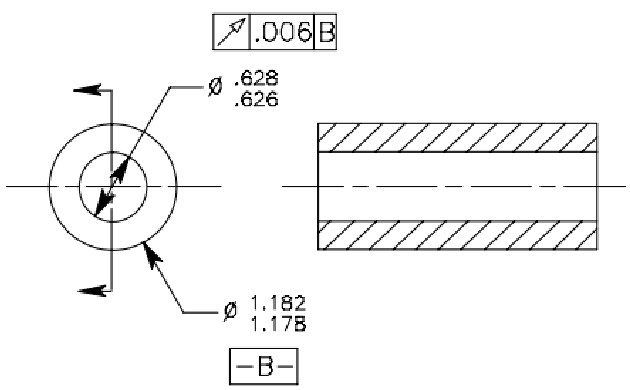
5.10.1 Circular Runout

Definition: Runout is a composite deviation from the desired form and orientation of a part surface of revolution during full rotation (360°) of the part on a datum axis. Runout is used to control the functional relationship of one or more features to a datum axis established from a datum feature specified at RMB. The types of features controlled by runout tolerances include those surfaces constructed around a datum axis and those constructed at right angles to a datum axis.

Tolerance: The tolerance zone for circular runout is two concentric circles when the surface of the controlled feature is constructed around the datum axis and is two parallel circular elements when the surface of the controlled feature is perpendicular to the datum axis.

Key Points:
1. Circular runout tolerances must be specified with respect to at least one datum reference. The datum feature(s) establish a datum axis of rotation.
2. Runout tolerances are always RFS and are evaluated on a FIM basis.
3. Circular runout inherently controls roundness and concentricity because the cumulative variations of these two geometric errors are evident in the surface-to-axis relationship.
4. When verifying runout, the indicator is fixed in orientation normal to the toleranced surface.

Example:



Meaning: Each circular element of the .628/.626 Diameter must be within a .006” wide tolerance zone (FIM) in relation to datum axis B.

5.10.1 Circular Runout (continued)

Measurement Methods:

Total Tolerance [1]	Measurement Resolution [1]	Preferred Measurement Methods	Comparable Measurement Methods
Equal to or less than .0005”	.000025” Max	Surface plate, amplified indicator, and V-block, collet, or centers	Circular Geometry Machine or CMM
Less than or equal to .001” and greater than .0005”	.00005” or 10:1	Surface plate, amplified indicator, and V-block, collet, or centers	Circular Geometry Machine or CMM
Less than or equal to .005” and greater than .001”	.0001” or 10:1	Surface plate, .0001” indicator, and V-block, collet, or centers	Circular Geometry Machine or CMM
Less than .010” and greater than .005”	.0005” or 10:1	Surface plate, .0005” indicator, and V-block, collet, or centers	Circular Geometry Machine or CMM or .001” micrometer
Equal to or greater than .010”	.001” or 10:1	Surface plate, .001” indicator, and V-block, collet, or centers	Circular Geometry Machine or CMM or .001” micrometer

Notes:
1. See paragraph 4.14 for metric equivalents.

Procedure: Locate the part in a V-block (or centers or a collet) on the datum surface and position the indicator on one end of the controlled diameter. Rotate the part on the datum surface in a constant 360° motion and monitor the FIM value. The FIM value should not exceed the tolerance value specified. Re-position the indicator along the axis of the controlled diameter and repeat on a number of circular elements along the length of the controlled diameter (Note: the number of circular elements checks will depend on the length of the diameter).

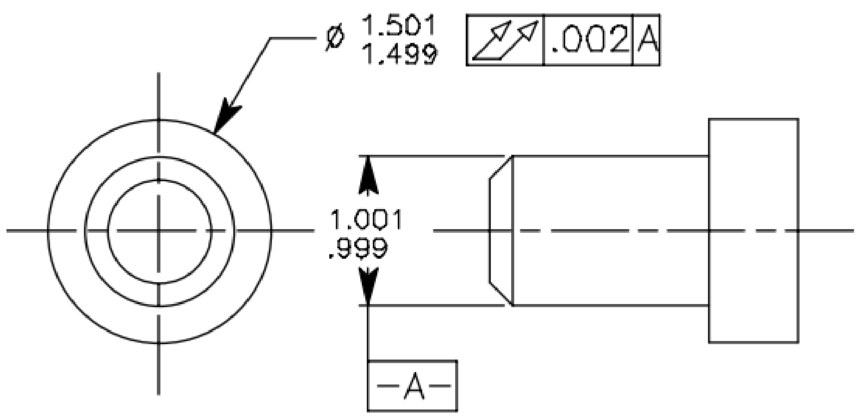
5.10.2 Total Runout

Definition: Total runout provides control of all surface elements and is used to control the functional relationship of one or more features to a datum axis established from a datum feature specified at RMB. The types of features controlled by runout tolerances include those surfaces constructed around a datum axis and those constructed at right angles to a datum axis. The tolerance is applied simultaneously to all circular and profile measuring positions as the part is rotated 360 º about the datum axis.

Tolerance: The tolerance zone for total runout applied to a surface around an axis is two coaxial similar perfect feature counterparts, concentric cylinders in the example below, constructed about the datum axis. When applied to a surface normal to the datum axis, the zone is two parallel planes constructed at right angles to the datum axis.

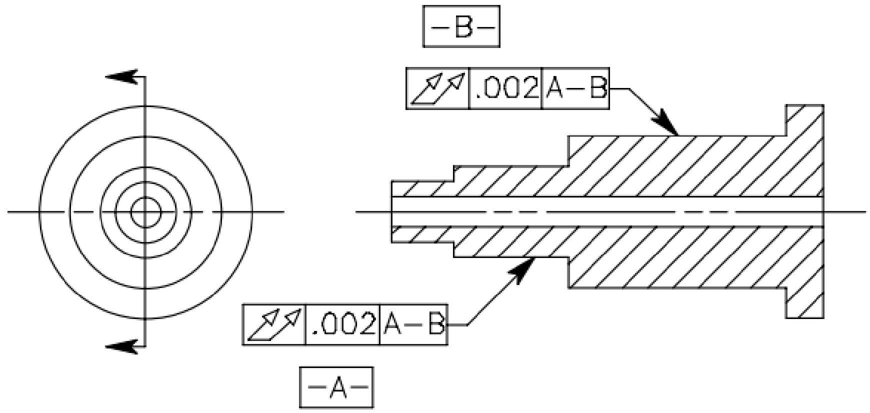
Key Facts:
1. Total runout tolerances must be specified with respect to at least one datum reference. The datum feature(s) establish a datum axis of rotation.
2. Runout tolerances are always RFS.
3. Total runout of a surface around an axis inherently controls roundness, concentricity and straightness of surface elements, and taper because the cumulative variations of these two geometric errors are evident in this composite surface-to-axis relationship
4. Total runout of a surface normal to an axis controls the cumulative variations of perpendicularity (to detect wobble) and flatness (to detect concavity or convexity).
5. When verifying total runout, the indicator is fixed in orientation normal to and translates along the toleranced surface.

Example 1:



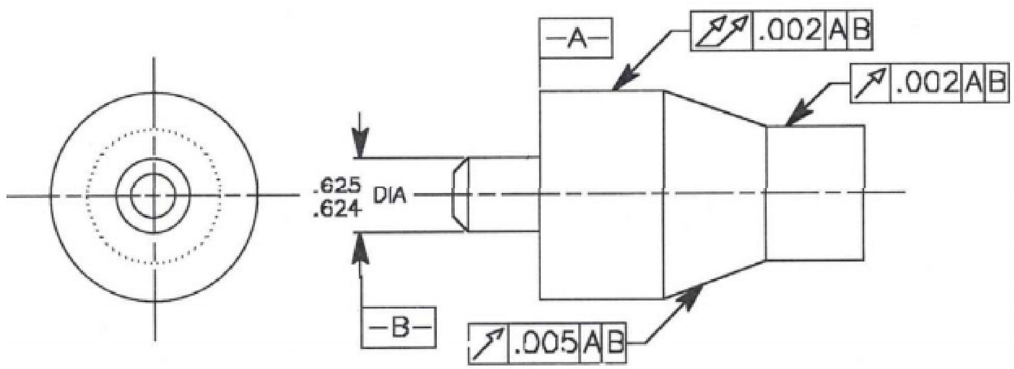
Meaning 1: All surface elements of the 1.501/1.499 Diameter (across the entire surface) must be within the .002" tolerance zone (FIM) in relation to datum axis A.

Example 2:



Meaning 2: All surface elements of each of the controlled diameters (across the entire surface) must be within the .002" runout tolerance in relation to datum axis A-B.

Example 3:



Meaning 3: The face surface perpendicular to the axis of rotation is the primary datum A, while the .625/.624 diameter is the secondary datum B. With the part in this orientation and located functionally on its datums, all circular and total runout callouts must meet their individual tolerances when the part is rotated about datum axis B.

5.10.2 Total Runout (continued)

Measurement Methods:

Total Tolerance [1]	Measurement Resolution [1]	Preferred Measurement Methods	Comparable Measurement Methods (as applicable to application)
Equal to or less than .0005”	.000025” Max	Surface plate, amplified indicator, and V-block, collet, or centers	Circular Geometry Machine or CMM
Less than or equal to .001” and greater than .0005”	.00005” or 10:1	Surface plate, amplified indicator, and V-block, collet, or centers	Circular Geometry Machine or CMM
Less than or equal to .005” and greater than .001”	.0001” or 10:1	Surface plate, .0001” indicator, and V-block, collet, or centers	Circular Geometry Machine or CMM
Less than .010” and greater than .005”	.0005” or 10:1	Surface plate, .0005” indicator, and V-block, collet, or centers	Circular Geometry Machine or CMM
Equal to or greater than .010”	.001” or 10:1	Surface plate, .001” indicator, and V-block, collet, or centers	Circular Geometry Machine or CMM

Notes:
1. See paragraph 4.14 for metric equivalents.

Procedure: For Example 1 only - Locate the part in a V-block (or centers or a collet) on the datum surface and position the indicator on one end of the controlled diameter. Rotate the part on the datum surface in a constant 360° motion while guiding the indicator along the top of the controlled diameter until it reaches the opposite end - monitor the FIM value that results during the traversing of the diameter. The composite FIM value must be equal to or less than the total runout tolerance for the part to be acceptable.

Procedure: For Example 2 only - The datum features are normally centered or positioned in an appropriate inspection device (e.g., between centers, collet, V-blocks, etc.) to establish the datum axis from which the total runout relationships are to occur. Position the indicator on one end of the controlled diameter. Rotate the part on the datum surfaces in a constant 360° motion while guiding the indicator along the top of the controlled diameter until it reaches the opposite end - monitor the FIM value that results during the traversing of the diameter. The composite FIM value must be equal to or less than the total runout tolerance for the part to be acceptable.

5.11 Location Features

The following subparagraphs cover inspection methods for position, concentricity, and symmetry location tolerances.

5.11.1 Position between Holes and Hole Pattern to Datum - Holes at MMC

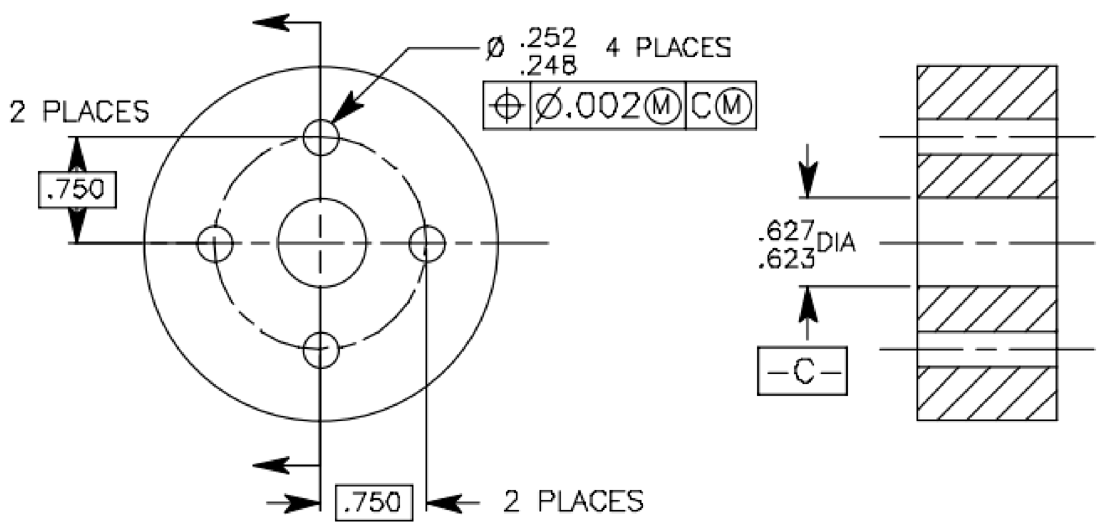
Definition: The theoretically exact location of a point, line, or plane of a feature in relationship with a datum reference or other feature.

Tolerance: A positional tolerance defines:

a. a zone within which the center, axis, or center plane of a feature of size is permitted to vary from a true (theoretically exact) position; or
b. where specified on a MMC or LMC basis, a boundary, defined as the virtual condition, located at the true (theoretically exact) position, that may not be violated by the surface or surfaces of the considered feature.

Key Points:
1. Position tolerances for location require basic dimensions to define the true position.
2. Position tolerances specified at RFS cannot be functionally gaged, nor can bonus tolerances be used.

Example:



5.11.1 Position between Holes and Hole Pattern to Datum - Holes at MMC (continued)

Measurement Methods:

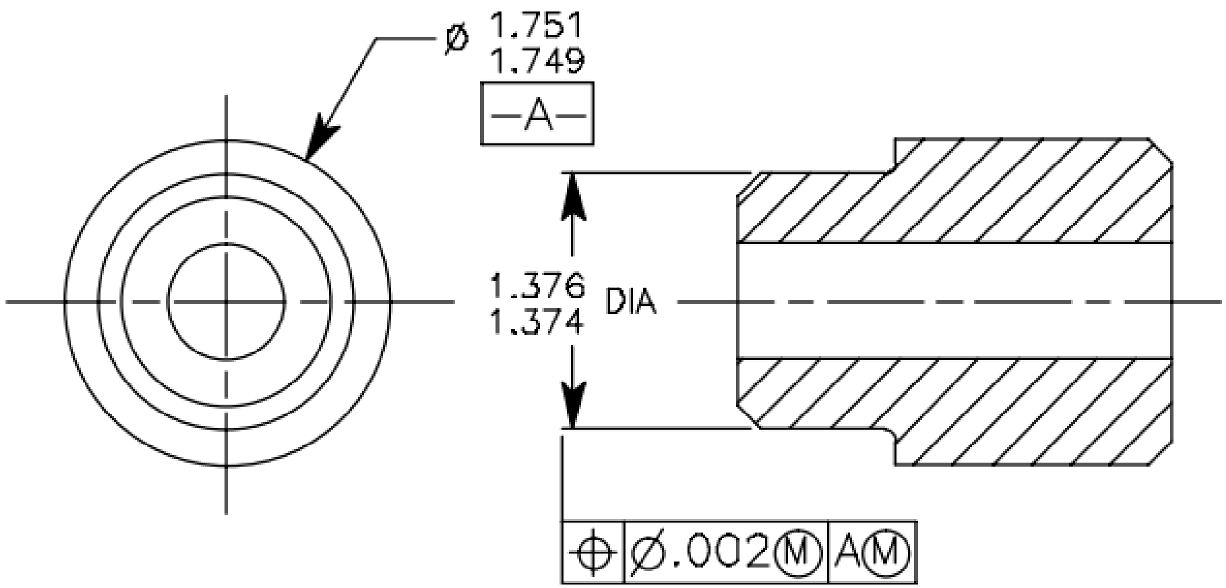
Total Tolerance [1]	Measurement Resolution [1]	Preferred Measurement Methods	Comparable Measurement Methods
Equal to or less than .0005”	.000025” Max	CMM [2] or Functional Gage [3]	Layout Inspection (surface plate, dial indicator, gage pins, etc.)
Less than or equal to .001” and greater than .0005”	.00005” or 10:1	CMM [2] or Functional Gage [3]	Layout Inspection (surface plate, dial indicator, gage pins, etc.)
Less than or equal to .005” and greater than .001”	.0001” or 10:1	CMM [2] or Functional Gage [3]	Layout Inspection (surface plate, dial indicator, gage pins, etc.)
Less than .010” and greater than .005”	.0005” or 10:1	CMM [2] or Functional Gage [3]	Layout Inspection (surface plate, dial indicator, gage pins, etc.)
Equal to or greater than .010”	.001” or 10:1	CMM [2] or Functional Gage [3]	Layout Inspection (surface plate, dial indicator, gage pins, etc.)

Notes:
1. See paragraph 4.14 for metric equivalents.
2. CMM inspection of threaded holes is to be done using flex plugs/inserts.
3. A functional gage cannot be used to inspect true position of hole(s) at RFS.

Procedure: CMM: Place the part on the CMM inspection surface, orient in the proper position, and level. Establish the datum axis by taking measurements and then take measurements in each hole. Compare the location of the axis of each hole relative to the datum axis, and if needed, determine the available bonus tolerance from the feature(s) and any available datum shift.

5.11.2 Position of Coaxial Features at MMC

Example:



Meaning: The axis of the 1.376/1.374 diameter feature is to be located coaxially within a .002” cylindrical tolerance zone relative to datum A when the feature is at MMC size (1.376 dia.). As the feature departs from its MMC toward LMC (1.374 dia.), the tolerance zone increases by the amount of departure. This is generally referred to as bonus tolerance and is calculated by taking the diameter of the feature, determined by the unrelated actual mating envelope, minus 1.376 diameter. If the feature was produced at its LMC (1.374 dia.) the tolerance zone would be .004” diameter. In addition to bonus tolerance, the MMB modifier applied to datum A allows the datum to shift. The amount of shift is determined by the difference between the maximum material boundary and the actual size (determined by the actual mating envelope) of the datum feature. It is important to note that the application of datum shift does NOT increase the tolerance zone.

5.11.2 Position of Coaxial Features at MMC (continued)

Measurement Methods:

Total Tolerance [1]	Measurement Resolution [1]	Preferred Measurement Methods	Comparable Measurement Methods
Equal to or less than .0005”	.000025” Max	CMM [2] or Functional Gage [3]	Layout Inspection (surface plate, dial indicator, gage pins, etc.)
Less than or equal to .001” and greater than .0005”	.00005” or 10:1	CMM [2] or Functional Gage [3]	Layout Inspection (surface plate, dial indicator, gage pins, etc.)
Less than or equal to .005” and greater than .001”	.0001” or 10:1	CMM [2] or Functional Gage [3]	Layout Inspection (surface plate, dial indicator, gage pins, etc.)
Less than .010” and greater than .005”	.0005” or 10:1	CMM [2] or Functional Gage [3]	Layout Inspection (surface plate, dial indicator, gage pins, etc.)
Equal to or greater than .010”	.001” or 10:1	CMM [2] or Functional Gage [3]	Layout Inspection (surface plate, dial indicator, gage pins, etc.)

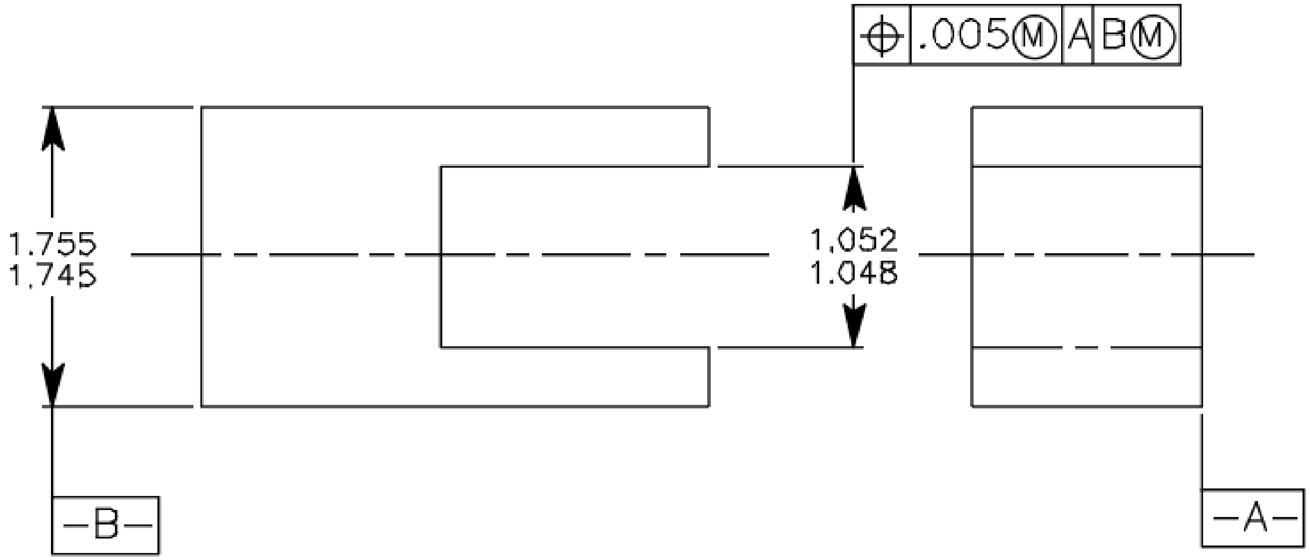
Notes:
1. See paragraph 4.14 for metric equivalents.
2. CMM inspection of threaded holes is to be done using flex plugs/inserts.
3. A functional gage cannot be used to inspect true position at RFS.

Procedure: CMM: Place the part on the CMM inspection surface, orient in the proper position, and level. Establish the datum axis by taking measurements and then take measurements to establish the feature centerline. Compare the location of the feature axis to the specified datum axis, and if needed, determine the available bonus tolerance from the feature and if datum shift is available

.

5.11.3 Position of Noncylindrical Features

Example:



Meaning: The center plane of the 1.052/1.048 slot feature is to co-planer to the center plane of datum B within a .005" wide tolerance zone with respect to datums A and B when the feature is at its maximum material condition (1.048"). As the feature departs from its MMC toward LMC (1.052"), the tolerance zone increases by the amount of departure. This is generally referred to as bonus tolerance and is calculated by taking the width (determined by the unrelated actual mating envelope) of the slot minus 1.048". If the slot was produced at its LMC (1.052"), the tolerance zone would be .009" wide. In addition to bonus tolerance, the MMB modifier applied to datum B allows the datum to shift. The amount of shift is determined by the difference between the maximum material boundary and the actual size (determined by the actual mating envelope) of the datum feature. It is important to note that the application of datum shift does NOT increase the tolerance zone.

5.11.3 Position of Noncylindrical Features (continued)

Measurement Methods:

Total Tolerance [1]	Measurement Resolution [1]	Preferred Measurement Methods	Comparable Measurement Methods
Equal to or less than .0005”	.000025” Max	CMM or Functional Gage [2]	Layout Inspection (surface plate, dial indicator, gage pins, etc.)
Less than or equal to .001” and greater than .0005”	.00005” or 10:1	CMM or Functional Gage [2]	Layout Inspection (surface plate, dial indicator, gage pins, etc.)
Less than or equal to .005” and greater than .001”	.0001” or 10:1	CMM or Functional Gage [2]	Layout Inspection (surface plate, dial indicator, gage pins, etc.)
Less than .010” and greater than .005”	.0005” or 10:1	CMM or Functional Gage [2]	Layout Inspection (surface plate, dial indicator, gage pins, etc.)
Equal to or greater than .010”	.001” or 10:1	CMM or Functional Gage [2]	Layout Inspection (surface plate, dial indicator, gage pins, etc.)

Notes:
1. See paragraph 4.14 for metric equivalents.
2. A functional gage cannot be used to inspect true position at RFS.

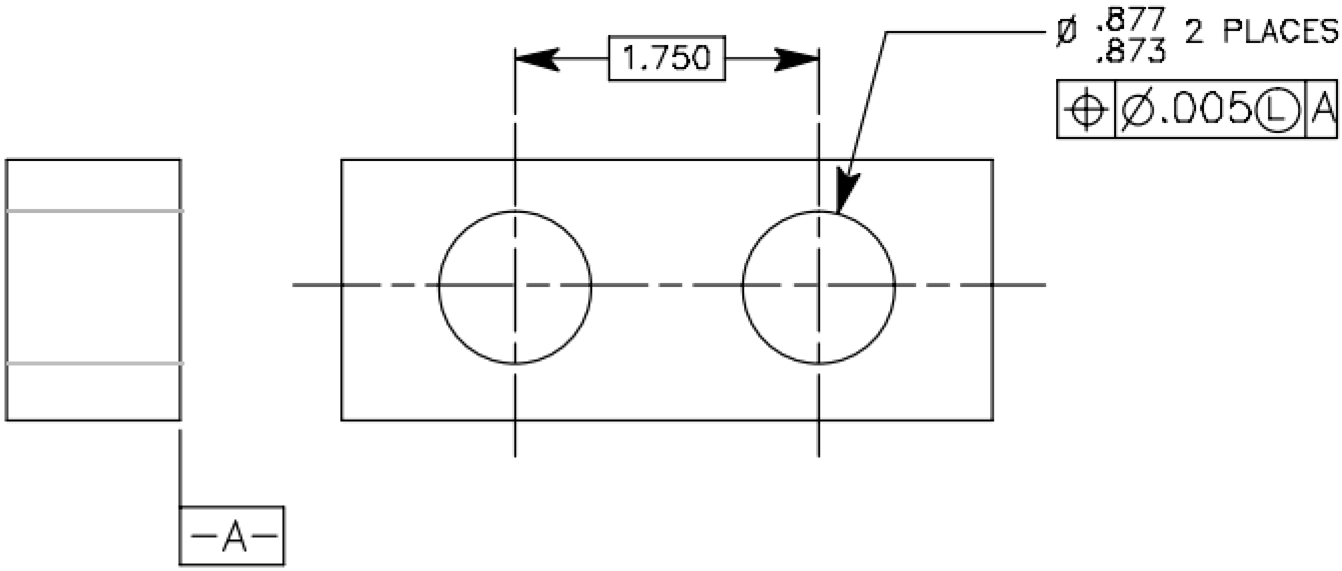
5.11.4 Position at Least Material Condition

Definition: Least Material Condition (LMC) is the condition in which a feature of size contains the least amount of material within which the stated limits of size.

Tolerance: Where positional tolerancing at LMC is specified, the stated positional tolerance applies where the feature contains the least amount of material permitted by its toleranced size dimension.

Key Points:
1. Specification of LMC requires perfect form at LMC. Perfect form at MMC is not required. Where the feature departs from its LMC limit of size, an increase in positional tolerance is allowed equal to the amount of such departure.
2. LMC is used to maintain a desired relationship between the surface of a feature and its true position at tolerance extremes.

Example:



Meaning: The axis of the .877/.873 Diameter features are to be located at a true position within a .005" tolerance zone with the feature at LMC size (.877") with respect to datum A axis. When the diameter feature is at LMC size, a .005" Diameter tolerance zone is permitted. When the diameter feature is at MMC size (.873"), a .009" Diameter tolerance zone is permissible (.005" position tolerance and .004" size tolerance).

5.11.4 Position at Least Material Condition (continued)

Measurement Methods:

Total Tolerance [1]	Measurement Resolution [1]	Preferred Measurement Methods	Comparable Measurement Methods
Equal to or less than .0005”	.000025” Max	CMM	Layout Inspection (surface plate, dial indicator, gage pins, etc.)
Less than or equal to .001” and greater than .0005”	.00005” or 10:1	CMM	Layout Inspection (surface plate, dial indicator, gage pins, etc.)
Less than or equal to .005” and greater than .001”	.0001” or 10:1	CMM	Layout Inspection (surface plate, dial indicator, gage pins, etc.)
Less than .010” and greater than .005”	.0005” or 10:1	CMM	Layout Inspection (surface plate, dial indicator, gage pins, etc.)
Equal to or greater than .010”	.001” or 10:1	CMM	Layout Inspection (surface plate, dial indicator, gage pins, etc.)

Notes:

1. See paragraph 4.14 for metric equivalents.

5.11.5 Concentricity

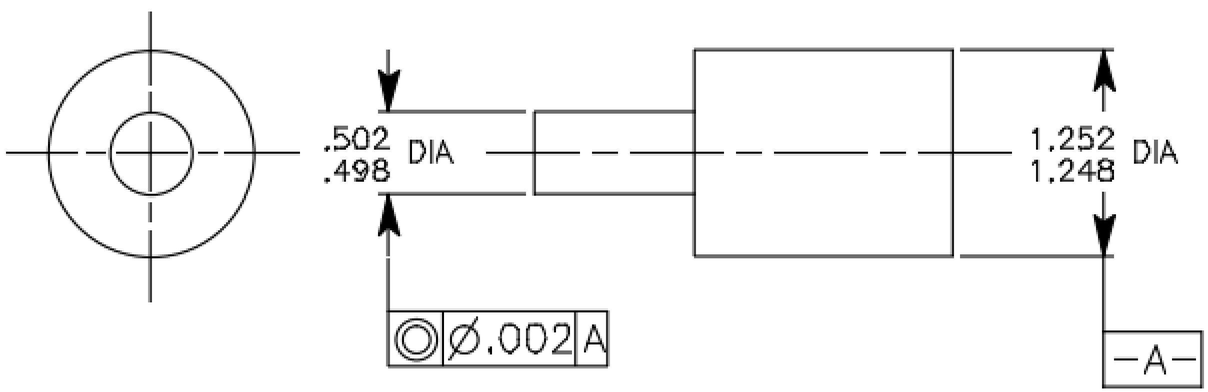
Definition: Concentricity is that condition where the median points of all diametrically opposed elements of a figure of revolution (or correspondingly-located elements of two or more radially-disposed features) are congruent with the axis (or center point) of a datum.

Tolerance: A concentricity tolerance is a cylindrical (or spherical) tolerance zone whose axis (or center point) coincides with the axis (or center point) of the datum feature(s). The median points of all correspondingly-located elements of the feature(s) being controlled, regardless of feature size, must lie within the cylindrical (or spherical) tolerance zones.

Key Points:
1. Concentricity is an axis-to-axis relationship and can only apply on an RFS basis.
2. Concentricity tolerances always require a datum reference and that the part be rotated 360° during measurement.
3. Concentricity requires that the inspector, during measurement, locate the datum axis and determine if all median points of the controlled feature fall within the cylindrical tolerance zone. Concentricity is not a FIM application. Differential measurements are usually required to inspect concentricity.

Note: Concentricity is a three dimensional control. It controls opposed median points to an axis. Concentricity will control location and can have some effect on the form and orientation of a feature. It will not control the form of perfectly oval parts but may have an impact on irregular or "D" shaped features. The application of concentricity is complex and rare.

Example:



Meaning: The .502/.498 Diameter feature shall be concentric within a .002" diameter tolerance zone to datum A regardless of the datum or feature size.

5.11.5 Concentricity (continued)

Measurement Methods:

Total Tolerance [1]	Measurement Resolution [1]	Preferred Measurement Methods	Comparable Measurement Methods
Equal to or less than .0005"	.000025" Max	CMM	Circular Geometry Machine (Precision Spindle)
Less than or equal to .001" and greater than .0005"	.00005" or 10:1	CMM	Circular Geometry Machine (Precision Spindle)
Less than or equal to .005" and greater than .001"	.0001" or 10:1	CMM	Circular Geometry Machine (Precision Spindle)
Less than .010" and greater than .005"	.0005" or 10:1	Surface plate, .0005" indicators, and a V-block, zero spindle, or expanding mandrel	CMM or Circular Geometry Machine (Precision Spindle)
Equal to or greater than .010"	.001" or 10:1	Surface plate, .001" indicators, and a V-block, zero spindle, or expanding mandrel	CMM or Circular Geometry Machine (Precision Spindle)

Notes:
1. See paragraph 4.14 for metric equivalents.

Procedure: Surface Plate, Indicator, and V-block (OD as datum): Locate the part in the V-block so that it contacts the datum diameter. Position the indicator at the top center on one end of the controlled diameter and rotate the part in a constant 360° motion while guiding the indicator along the top of the controlled diameter until it reaches the opposite end - monitor the FIM value that results during the traversing of the diameter. The composite FIM value must be equal to or less than the concentricity tolerance for the part to be acceptable.
Note: This method verified concentricity by checking total runout.

Expanding Mandrel (Hole as datum): Mount the part datum on an expanding mandrel between precision centers. Position the indicator at the top center on one end of the controlled diameter and rotate the part in a constant 360° motion while guiding the indicator along the top of the controlled diameter until it reaches the opposite end - monitor the FIM value that results during the traversing of the diameter. The composite FIM value must be equal to or less than the concentricity tolerance for the part to be acceptable.
Note: This method verified concentricity by checking total runout.

CMM: Suitably stage the part on the CMM inspection surface and take measurements around the datum diameter and establish the datum axis. Take measurements around the feature to be controlled and establish its centerline, then compare the feature centerline to that of the datum to see if within the specified diameter tolerance zone.
Note: This method verified concentricity by checking position.

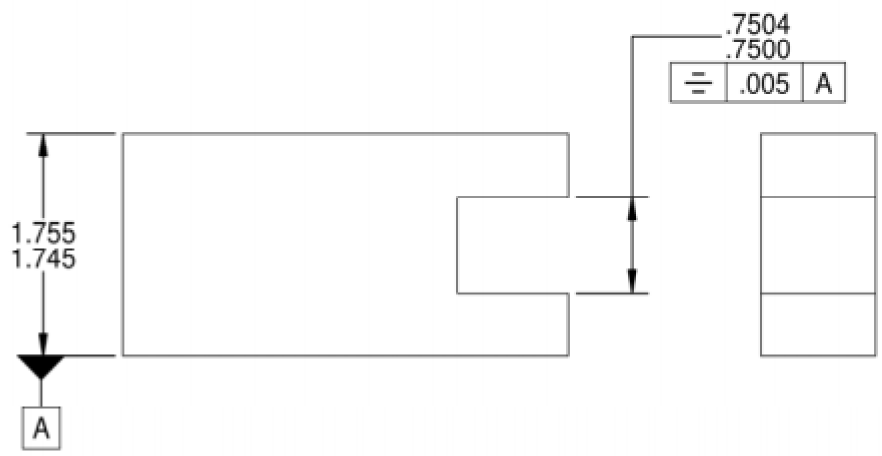
5.11.6 Symmetry

Definition: The condition where the median points of all exposed or correspondingly-located elements of two or more feature surfaces are congruent with the axis or center plane of a datum feature.

Tolerance: A symmetry tolerance is a tolerance zone whose center plane coincides with the center plane of the datum feature. The median points of all correspondingly-located elements of the feature being controlled, regardless of feature size, must lie within the tolerance zone.

Key Points:
1. The material condition RFS only is to apply and cannot be functionally gaged: it must be measured.
2. The feature and its datum must be symmetrically configured to each other.
3. The datum feature is usually noncylindrical but may be cylindrical if appropriate to the part.

Example:



Meaning: All median points of opposed elements of the .7504/.7500 slot must lie within a .005" wide tolerance zone, regardless of feature size. The tolerance zone being established by two parallel planes equally disposed about datum center plane A, regardless of feature size.

5.11.6 Symmetry (continued)

Measurement Methods:

Total Tolerance [1]	Measurement Resolution [1]	Preferred Measurement Methods	Comparable Measurement Methods
Equal to or less than .0005”	.000025” Max	CMM	Layout Inspection (surface plate, dial indicator, gage pins, etc.)
Less than or equal to .001” and greater than .0005”	.00005” or 10:1	CMM	Layout Inspection (surface plate, dial indicator, gage pins, etc.)
Less than or equal to .005” and greater than .001”	.0001” or 10:1	CMM	Layout Inspection (surface plate, dial indicator, gage pins, etc.)
Less than .010” and greater than .005”	.0005” or 10:1	CMM	Layout Inspection (surface plate, dial indicator, gage pins, etc.)
Equal to or greater than .010”	.001” or 10:1	CMM	Layout Inspection (surface plate, dial indicator, gage pins, etc.)

Notes:

1. See paragraph 4.14 for metric equivalents.

5.12 Surface Finish (Roughness)

Definition: Roughness is a measure of the texture of a surface. It is quantifiable by the vertical deviations of surface from its ideal form.

Tolerance: The surface finish requirement specified by the drawing is the maximum roughness that is allowed.

Key Points:

1. Surface roughness is to be measured by the Ra (or Arithmetic Average Roughness) parameter unless otherwise specified on the drawing. The Ra parameter (expressed in microinches or micrometers) is the average of peak and valley distances measured along the centerline of one cutoff length (usually 0.030" (0.8 mm)). The Ra (Roughness Average) is also known as AA (arithmetic average) or CLA (centerline average).
2. Cutoff is the electrical response characteristic of an instrument that is selected to limit the spacing of the surface irregularities to be included in the roughness measurement. When not specified by the drawing, a cutoff of 0.030" (0.8 mm) is to be used.
3. Inspection and assessment of surface roughness can be performed using the following methods:
   - Direct measurement methods. Direct measurement methods assess surface finish by means of stylus type devices. Measurements are obtained using a stylus drawn along the surface to be measured; the stylus motion being perpendicular to the surface.
   - Comparison techniques. Comparison techniques use specimens of known surface roughness (i.e., comparison standards). The part surface to be assessed is visually and tactilely (i.e., by touch using fingernail) compared to the surface of the known surface roughness standards.

Example:

Meaning: The .755/.745 diameter feature has a surface finish (roughness) requirement of 63 microinches (1.6 micrometers) Ra maximum.

5.12 Surface Finish (Roughness) (continued)

Measurement Methods:

Drawing Requirement (Ra)	Preferred Measurement Methods	Comparable Measurement Methods
8 microinches (0.2 micrometers) or less	See Note 1	See Note 1
Less than or equal to 16 microinches (0.4 micrometers) and greater than 8 microinches (0.2 micrometers)	Surface Profilometer [2]	Contour Tracer
Less than or equal to 32 microinches (0.8 micrometers) and greater than 16 microinches (0.4 micrometers)	Surface Profilometer [2]	Contour Tracer
Less than or equal to 63 microinches (1.6 micrometers) and greater than 32 microinches (0.8 micrometers)	Surface Profilometer [2]	Contour Tracer
Less than or equal to 125 microinches (3.2 micrometers) and greater than 63 microinches (1.6 micrometers)	Surface Profilometer [2]	Roughness Comparison Standard
Greater than 125 microinches (3.2 micrometers)	Roughness Comparison Standard	Surface Profilometer [2]

Notes:

1. Contact LORD Quality Engineering for the specific measurement method to be used.
2. Unless otherwise specified, a cut-off (or sampling length) of 0.03” (0.8 mm) shall be used. Refer to the applicable tables in ASME B46.1 to provide the recommended cutoff lengths of the instrument’s measuring filter and recommended sampling lengths for the various finishes.

Appendix A - Best Practices

COORDINATE MEASURING MACHINE (CMM)

Measurement Practices:

- The CMM is not the preferred method for inspecting certain geometric features ≤ .001". If it becomes necessary to program a position, concentricity, symmetry profile, or angularity ≤ .001", review that particular part feature with a LORD Quality Engineer.
- The CMM is not the preferred method for inspecting features of size such as diameter, length, width, and height, however, it is a comparable method. When the CMM is used to inspect features of size, it must be specifically programed to check both local size and mating size. Features should be scanned when applicable. Proper algorithms must be applied as ASME standards do not support the use of least squares fitting.
- For rough surface finishes, use a larger ball size and take more readings. The rougher the surface, the more readings should be taken.
- The larger the feature, the more readings should be taken.
- The tighter the tolerance for a feature, the more readings are required to establish the feature.
- For a 250 microinch (6.4 micrometer) surface finish, take double the amount of readings (8 readings)

Below are minimum measurement requirements to establish features:

- Straight line – minimum of 2 points
- Plane – minimum of 4 points
- Circle – minimum of 8 points for diameters > .175"; minimum of 6 points for diameters ≤ .175"
- Sphere – minimum of 9 points
- Cone – minimum of 8 points
- Cylinder – minimum of 12 points (6 points in two planes, add a minimum of 6 more points for every .375" of an inch in length).

Factors to consider when choosing the proper amount of points necessary for the best representation of the feature:

- Tolerance – The tighter the tolerance the more points necessary to ensure conformance.
- Size of Feature – The larger the feature the more points necessary to accurately represent the feature.
- Surface Finish – Rougher surfaces require more points to accurately establish the feature.
- Use of Feature – When a feature is going to be used as a datum more points may be required to accurately establish the datum simulator.

Algorithms:

- Internal maximum inscribed algorithm must be applied.
- External minimum circumscribed algorithm must be applied.

Software/Program Verification and Validation:

Verification and validation of CMM software/program should be performed to provide objective evidence that the software performs its required function. The CMM software/program should have an independent method of validation performed for each part characteristic/feature that the software/program inspects and a correlation of the two sets of results. Some acceptable independent methods of validation are:

- Layout inspection
- Fixture check
- Comparison with another CMM program previously verified by an independent method

Acceptable correlation requires the difference to be within 10% of the tolerances for each characteristic/feature. Differences greater than 10% but not exceeding 25% may be acceptable. Differences greater than 25% are not acceptable. The variable data from the independent validation should be recorded and retained.

Each version of the software should be uniquely identified and be traceable to a specific part drawing revision.

CALCULATING MMC/LMC WITHIN CMM PROGRAM

Features should be scanned when applicable. Proper algorithms must be applied. If scanning is not applied, the number of points measured should be drastically increased to ensure accurate representation of mating envelope.

Simultaneous Requirements

Per ASME Y14.5, all features with the exact same datum reference frame are considered simultaneous requirements and must be outputted as so.

POSITION/CONCENTRICITY MEASUREMENT

Establish Centerline of Feature

- Take eight readings on each hole or surface.
- Six readings can be used for a position requirement of .010” or greater.
- It is necessary to have 120° of an arc to establish a centerline.
- Rectangular parts require two readings on each of the four sides.
- Bolt patterns (three or more holes equally spaced) with no datum require eight readings on each hole. Recall lines and construct the bolt circle pattern which is the mean line.
- Bolt patterns with datum require the additional step of establishing the line of the datum.
- Rectangular bolt patterns still require the use of the mean line bolt circle pattern.
- For splines and threads, use the pitch diameter as the centerline unless otherwise specified on the print or inspection procedure.
- Use of a functional gage where possible for bolt patterns not equally spaced. If the CMM must be used, try to determine the best centerline of the entire pattern.
- Newer software may have an option to select/define a bolt hole pattern in this case the software would create the centerline which is acceptable with the assumption that the program has been verified.

Gage Diameter Location

- Measure a minimum of two planes on conical surface.
- Each plane requires a minimum of four readings, equally distanced apart. Depending on surface finish, size of plane, or flatness, take more readings; refer to the CMM best practices.

PROFILE/FLATNESS MEASURMENT

Conical/Spherical/Straight Surfaces

- Take a minimum of 12 readings. Nine for parts that are less than 1” of profile surface (Flatness).
- Take a minimum of three readings in each line segment.
- Take a minimum of three places (line segments) around, 120° apart.
- Take a minimum of four readings for each 1” of length.
- The above states the minimum requirements. The larger the surface, the more readings required; refer to the CMM best practices.

Turned Parts

- Take a minimum of three readings in each line segment.
- Take a minimum of three places (line segments) around, 120° apart.
- Take a minimum of four readings for each one inch of length.
- The above states the minimum requirements. The larger the surface, the more readings required; refer to the CMM best practices.

Flexible parts require more readings.

Note: If the surface is not machined in the same operation as the datum was machined, take readings more than four places around, 90° apart.

ANGULARITY MEASUREMENT

- Take a minimum of three readings.
- Take a minimum of three places around, 120° apart.
- Take a minimum of four readings for each one inch in length.
- The above states the minimum requirements. The larger the surface, the more readings required; refer to the CMM best practices.