Section 8

Tolerances

ALUMINUM EXTRUDERS COUNCIL

Aluminum Extrusion Manual

4th Edition

Section 8

Tolerances

How straight is straight enough? How flat is flat enough? How uniform must a wall thickness be in order to be acceptable? These are not abstract questions. Many products must be manufactured to exacting standards. The specified, acceptable range of deviation from a given dimension is known as a tolerance.

Tolerances are measurable, so they can be specified and mutually agreed upon by manufacturers and purchasers, by extruders and their customers. Aluminum profiles can be extruded to very precise special tolerances or to accepted standard dimensional tolerances.

The first portion of this section addresses standard dimensional tolerances by referencing selected tables from The Aluminum Association's 2009 Aluminum Standards & Data.

The Tables that pertain to Standard Dimensional Tolerances are linked below:

Tables 11.5 through 11.14
Tables 12.2 through 12.5
Tables 12.10 through 12.14

The following portion of this section is an introduction to geometric tolerancing.

Geometric tolerancing has been likened to a modern technical language that enables designers and engineers to communicate their requirements to the people who produce the components of an assembly.

When tolerances are met, parts fit together well, perform as intended, and do not require unnecessary machining. The aluminum extrusion process puts the metal where it is needed and offers the precision necessary to meet specified tolerances.

Introduction to Geometric Dimensioning and Tolerancing

Taken together, geometric dimensioning and tolerancing (GD&T) can be used to specify the geometry or shape of an extrusion on an engineering drawing. It can be described as a modem technical language, which has uniform meaning to all. It can vastly improve communication in the cycle from design to manufacture. Terminology, however, varies in meaning according to the Geometric Standard being used; this must be taken into account in each case.

Geometric dimensioning and tolerancing, also often referred to in colloquial terms as geometrics, is based upon sound engineering and manufacturing principles. It more readily captures the design intent by providing designers and drafters better tools with which to “say what they mean.” Hence, the people involved in manufacturing or production can more clearly understand the design requirements. In practice, it becomes quite evident that the basic “engineering” (in terms of extruding, fixturing, inspecting, etc.) is more logically consistent with the design intent when geometric dimensioning and tolerancing is used. As one example, functional gauging can be used to facilitate the verification process and, at the same time, protect design intent. Geometric dimensioning and tolerancing is also rapidly becoming a universal engineering drawing language and technique that companies, industries, and government are finding essential to their operational well-being. Over the past 40 years, this subject has matured to become an indispensable management tool; it assists productivity, quality, and economics in producing and marketing products around the world.

Rationale of Geometric Dimensioning and Tolerancing

Geometric dimensioning and tolerancing builds upon previously established drawing practices. It adds, however, a new dimension to drawing skills in defining the part and its features, beyond the capabilities of the older methods.

It is sometimes effective to consider the technical benefits of geometric dimensioning and tolerancing by examining and analyzing a drawing without such techniques used, putting the interpretation of such a drawing to the test of clarity. Have the requirements of such a part been adequately stated? Can it be produced with the clearest understanding? Geometric dimensioning and tolerancing offers that clarity.

Often an engineer is concerned about fit and function. With many standard tolerances this may become a concern. Geometric tolerancing is structured to better control parts in a fit-and-function relationship.

The Symbols

Effective implementation of geometrics first requires a good grasp of the many different symbols and their functional meaning. The following symbols are those that are most commonly used within the extrusion industry.

The current standard, as of this writing, is from the American Society of Mechanical Engineers (ASME) through the American National Standards Institute (ANSI) in publication Y14.5 - 2009, Dimensioning and Tolerancing and is considered to be the authoritative guideline for GD&T.

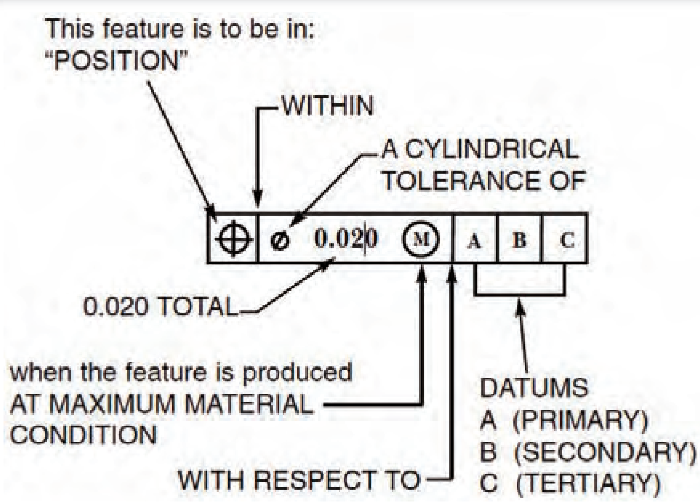
For definitions of basic terms used in geometric tolerancing, refer to the appendix at the end of this section.

Note: Tolerances used within the following examples are purely illustrative and may not reflect the standard tolerances used by the aluminum extrusion industry

Type	Designation
STRAIGHTNESS	⏤
FLATNESS	▱
ANGULARITY	∠
PERPENDICULARITY	⊥
PARALLELISM	//
CONCENTRICITY	◎
POSITION	⌖
CIRCULARITY	○
PROFILE OF A LINE	⌒
PROFILE OF A SURFACE	⌓
CYLINDRICITY	⌭
DIAMETER	∅
DATUM FEATURE	−A− or A
MAXIMUM MATERIAL CONDITION (MMC)	Ⓜ
REGARDLESS OF FEATURE SIZE (RFS)	Ⓢ
LEAST MATERIAL CONDITION (LMC)	Ⓛ
TANGENT PLANE	Ⓣ

The Feature Control Frame

The feature control frame is a rectangular box containing the geometric characteristics symbol and the form, orientation, profile, runout, or location tolerance. If necessary, datum references and modifiers applicable to the feature of the datum are also contained in the frame.



Material Conditions

Maximum Material Condition

The abbreviation for maximum material condition is MMC and the symbol is the capital letter M with a circle around it. The maximum material condition occurs when a feature contains the most material allowed by the size tolerance. It is the condition that will cause the feature to weigh the most. MMC is often considered when the designer's concern is assembly. The minimum clearance or maximum interference between mating parts will occur when the part features are at MMC.

The most critical assembly condition is when External (Male) features are their largest and Internal (Female) features are their smallest.

The maximum material condition for external features occurs when the size dimension is at its largest.

The maximum material condition for internal features occurs when the size dimension is at its smallest.

MMC - abbreviation

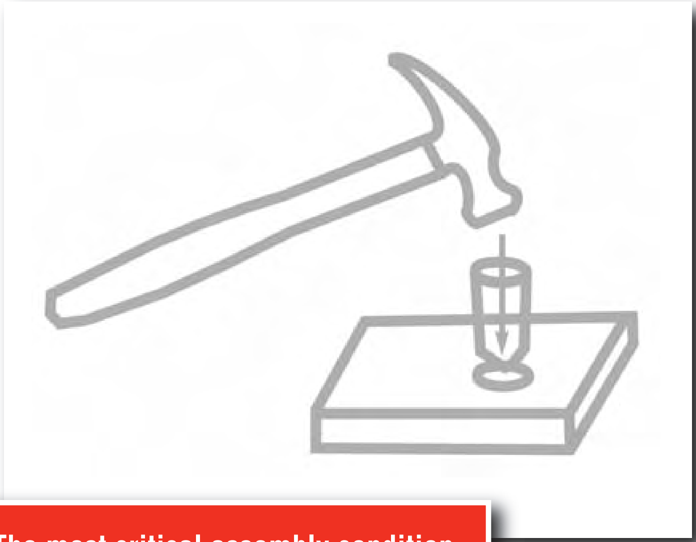
M **- symbol**

Least Material Condition

The abbreviation for least material condition is LMC and the symbol is L within a circle. Least material condition is the opposite of maximum material condition. In other words, it is a condition of a feature where it contains the least amount of material. For external parts, that occurs when the overall dimension is at a maximum. It is the maximum size of an internal feature.

LMC - abbreviation

L **- symbol**



The most critical assembly condition is when External (Male) features are their largest and Internal (Female) features are their smallest.

Regardless of Feature Size

The abbreviation for regardless of feature size is RFS, and the symbol is S within a circle. Regardless of feature size is a condition that is used when the importance of location and/or shape of a feature is independent of the feature's size and forces anyone checking the part to use open set-up inspection.

RFS - abbreviation

S **- symbol**

Rule #1 – "Where only a tolerance of size is specified, the limits of size of an individual feature prescribe the extent to which variations in its geometric form, as well as size, are allowed."

Rule #2 – "For all applicable geometric tolerances, RFS applies with respect to individual tolerance, datum reference, or both, where no modifying symbol is specified. MMC, or LMC, must be specified on the drawing where it is required."

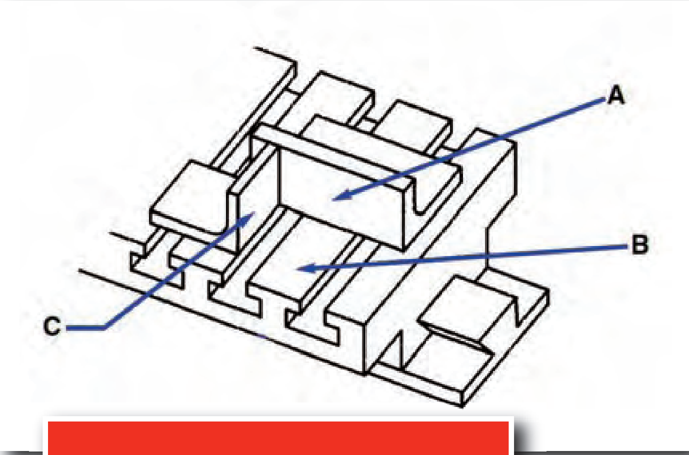
Datum

A datum is a theoretically exact point, axis, or plane that is derived from the true geometric counterpart of a specified datum feature. The datum is the origin from which the location or orientation of part features is established.

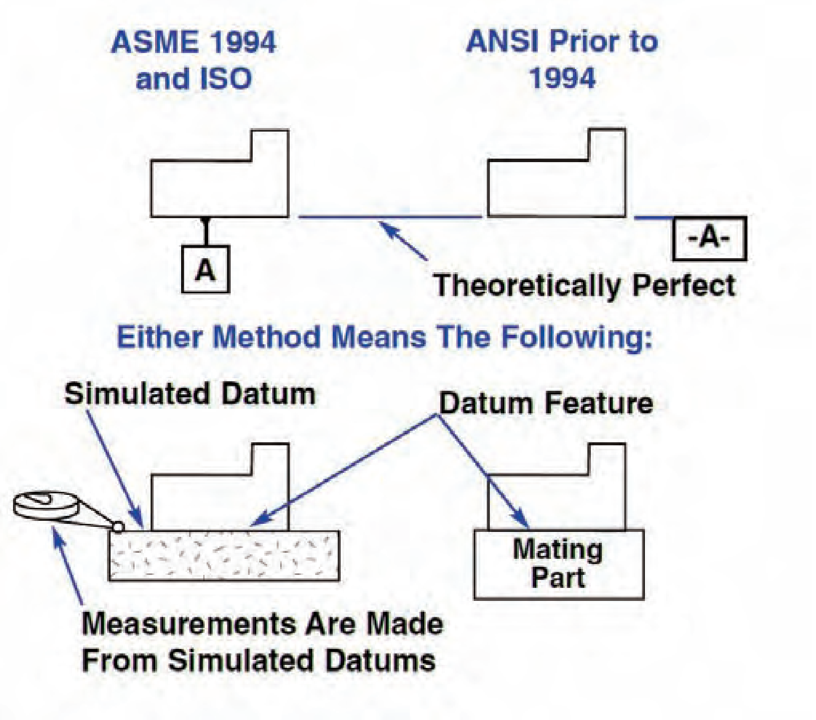
Confusion can arise if the drawing does not specify how a part is to be located. This is done by specifying datums on the drawing.

A drawing of a ball bearing would not require a datum because it is a single feature part. If a hole were drilled in the ball bearing, different measurements would result if the tolerance of the part were held to be on the feature of the ball or the hole. Adding a datum designation to one of these features and referencing to it would eliminate any confusion.

The datum feature is defined as the actual feature of a part that is used to establish the datum. Since it is not possible to establish a theoretically exact datum, they must be simulated. Typical ways to simulate a datum are to use surface plates, angle plates, gauge pins, collets, machine tool beds, etc. The intent of the standard is to hold or fixture the part with something that is as close to the true geometric counterpart as possible. The further the fixture deviates from the true geometric counterpart, the greater the set-up error and, therefore, the less reliable the measurement.



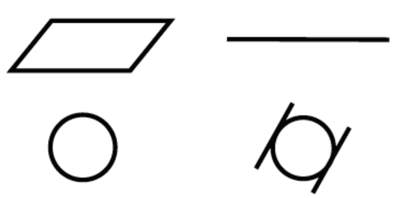
Simulated datums are what hold the parts in production, inspection, and their assembly.



The datums can be thought of as a navigation system for dimensions of the part. They might also be thought of as a “trap” for the part. On the lower drawing on the opposite page, the datum, in this case datum A, refers to a theoretically perfect datum plane. A surface plate in an inspection area would serve as a simulated datum and would make contact on the high points or extremities of the surface.

These high points are the same points that will make contact with the mating part in the final assembly. Measurements made from the surface plate to other features on the part will be the best method to predict whether the part will perform its intended function.

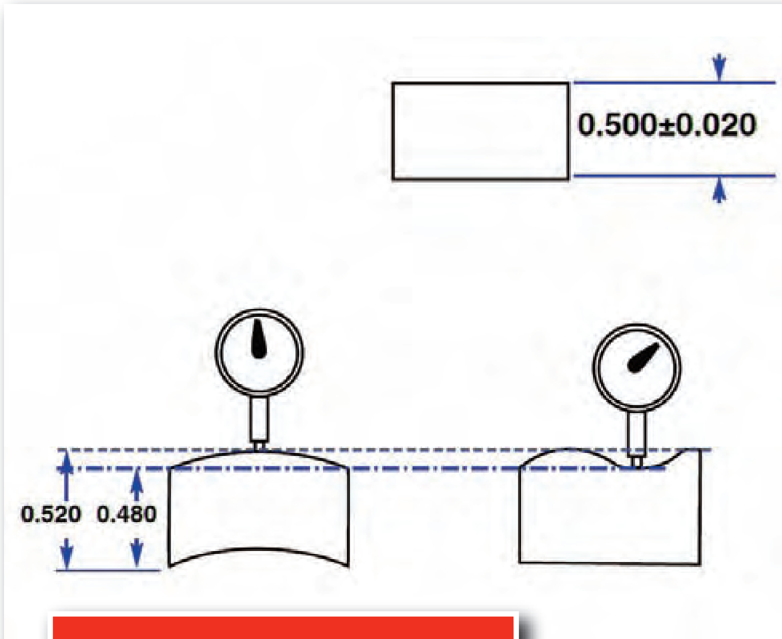
Tolerances of Form
(Unrelated)



The geometric form of a feature is controlled first by a size dimension. Prior to the use of geometric dimensioning and tolerancing, size dimension was the primary control of form and did not prove to be sufficient. In some cases, it is too restrictive and in others, the meaning is unclear. Rule #1 (see page 3) clearly states the degree to which size controls form.

In this example, the 0.500 dimension established two parallel lines. One pair is 0.520 apart (the high limit) and the other pair is 0.480 apart (the low limit). The 0.480 can float within the 0.520. If the lower surface was perfectly flat (right-hand figure), the upper surface could be anywhere within a 0.040 tolerance zone.

In this extreme case, it can be said that the top surface must be flat within 0.040.



If the part is manufactured at MMC, both surfaces would have to be perfectly flat.

Flatness

Flatness is the condition of a surface having all elements in one plane.

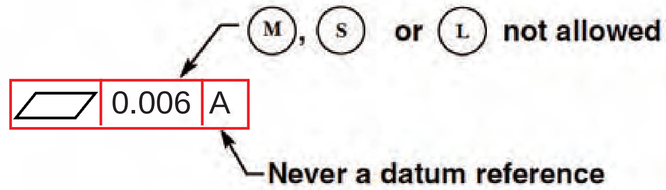
Flatness usually applies to a surface being used as a primary datum feature.

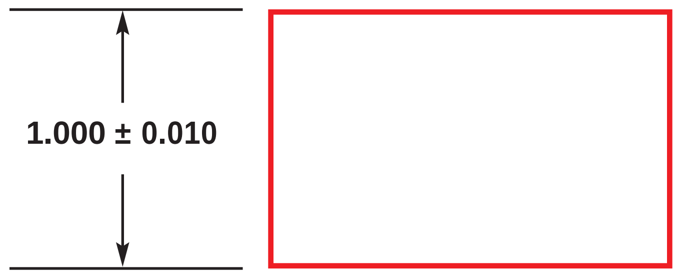
Other tolerances that provide flatness control include:

- Any size tolerance on a feature comprised of two internal or external parallel opposed planes.
- Any flat surface being controlled by:

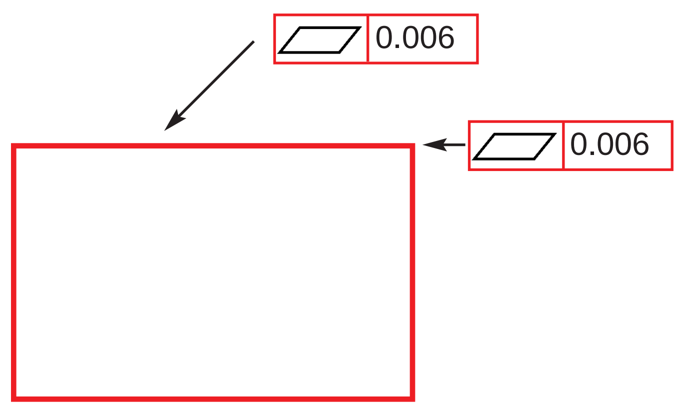
Perpendicularity	⊥	0.008	A	
Parallelism	//	0.008	A	
Angularity	∠	0.008	A	
Profile of a Surface	⌓	0.010	A	B
Total Runout	⌰	0.010	A	

One way to improve the form of the surface is to add a flatness tolerance. This tolerance compares a surface to an ideal or perfectly flat plane. A flatness tolerance does not locate the surface.





Flatness Placement



The flatness requirement is placed in a view where the controlled surface appears as an edge. The feature control frame may be on either a leader line or an extension line. Since flatness can only be applied to flat surfaces, it should never be placed next to a size dimension.

Straightness
(of an axis or center plane)

Straightness is a condition under which an element of a surface or an axis is a straight line.

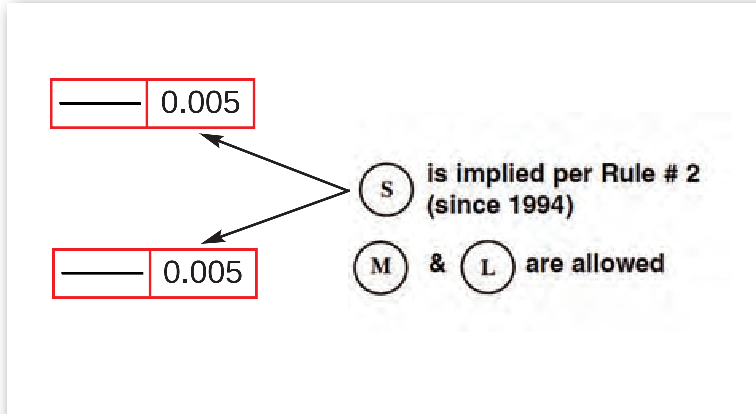
The feature control frame must be located with the size dimension.

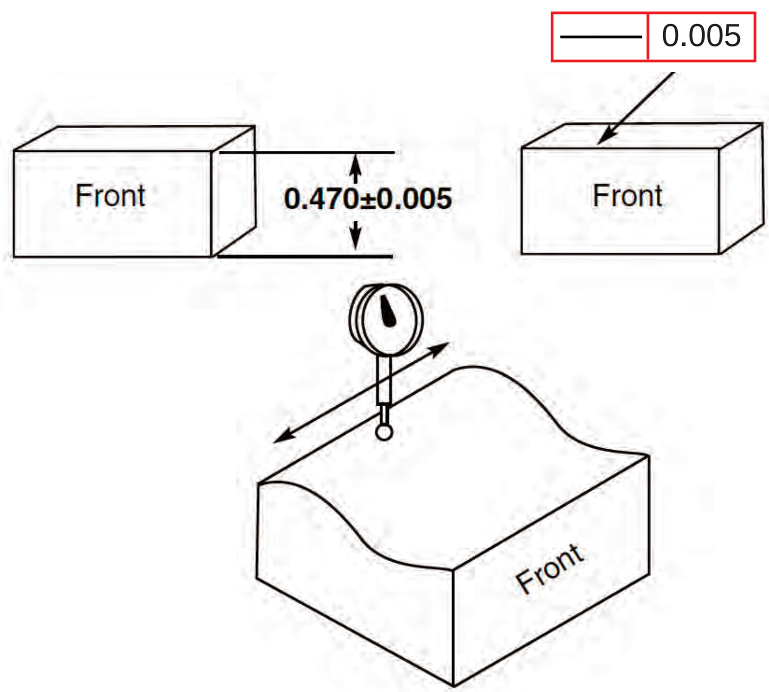
This tolerance is used as a way to override the requirement of perfect form at MMC (Rule #1).

Other tolerancing that automatically provides this control are:

Any Size Tolerances	**± 0.010**
Circular Runout	↗ | 0.006 | A
Total Runout	⌰ | 0.010 | A

The straightness tolerance can be used whenever a straight line element, axis, or center plane can be identified on a part. The tolerance zones used for straightness can be either a pair of parallel lines or a cylinder. Each line element, axis, or center plane is compared to the tolerance zone. The tolerance for line elements is shown on the drawing in a view where the elements to be controlled are shown as straight lines.

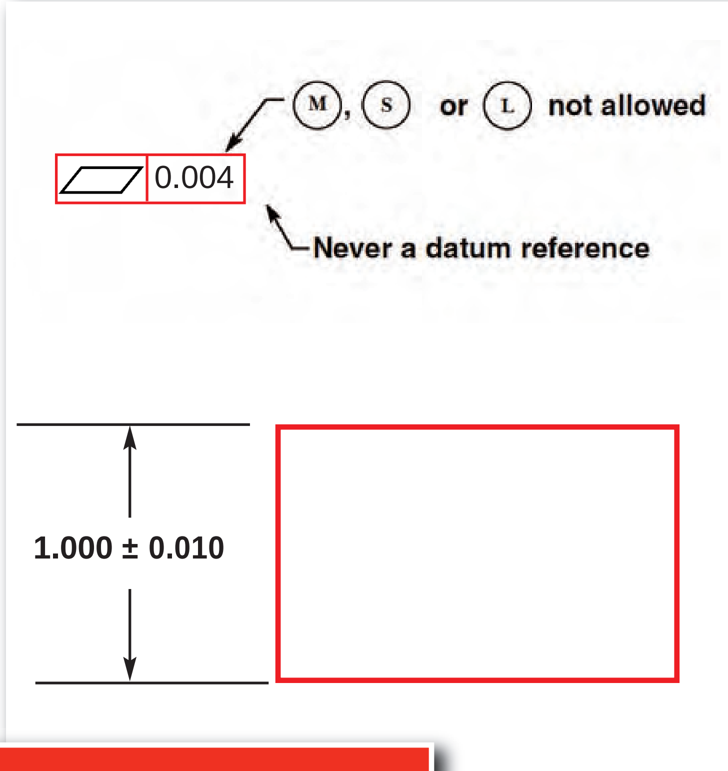




Surface Straightness
(on a flat surface, cylinder or cone)

Other tolerances that provide flatness control include:

- Any size tolerance on a feature comprised of two internal or external parallel opposed planes.



The straightness in this case would be 0.020.

- Any flat surface being controlled by:

Perpendicularity	⊥	0.008	A	
Parallelism	//	0.008	A	
Angularity	∠	0.008	A	
Profile of a Surface	⌓	0.010	A	B
Total Runout	⌰	0.010	A	
Flatness	▱	0.006		
Cylindricity	⌭	0.006		

Circularity
(roundness)

Circularity is the condition on a surface of revolution (cylinder, cone, sphere) where all points of the surface intersected by any plane (1) perpendicular to a common axis (cylinder, cone) or (2) passing through a common center (sphere) are equidistant from the center.

Other tolerances that provide circularity control include:

- Any size tolerance on a cylindrical feature or sphere.
- Any feature containing circular elements and being controlled by:

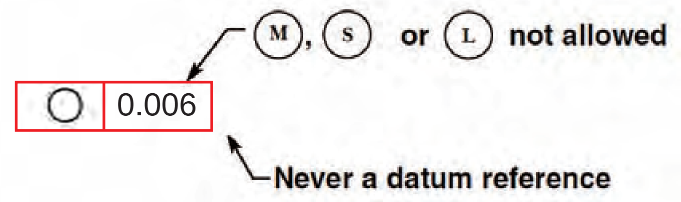
Circular Runout ↗ | 0.006 | A

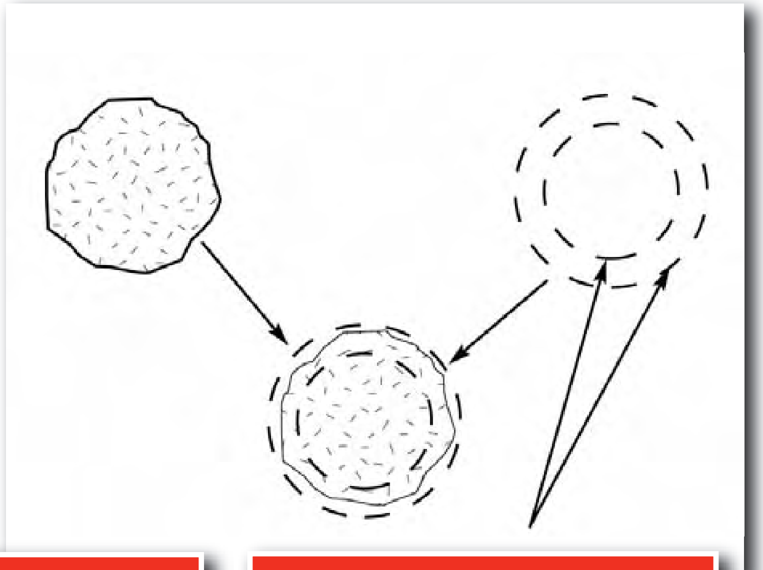
Total Runout ⌰ | 0.010 | A

Rule of thumb:
Runout tolerances are usually less expensive to verify and should be considered when circularity is desired.

The tolerance will be a leader line, which points to the feature containing the circular element(s). Circularity is similar to straightness except that the tolerance zone is perfectly circular rather than perfectly straight.

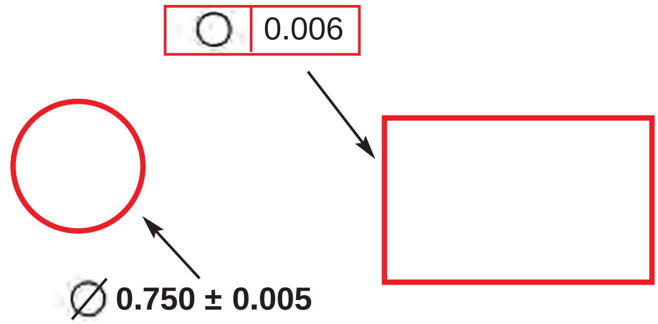
Although the circularity tolerance floats within the limits of size, it is independent of size and should not be placed next to the size dimension.





Every circular element must be within the tolerance zone.

These two diameters can be of any diameters within the size limits of the feature, provided they remain concentric and their radial difference equals the circularity tolerance.



Cylindricity

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

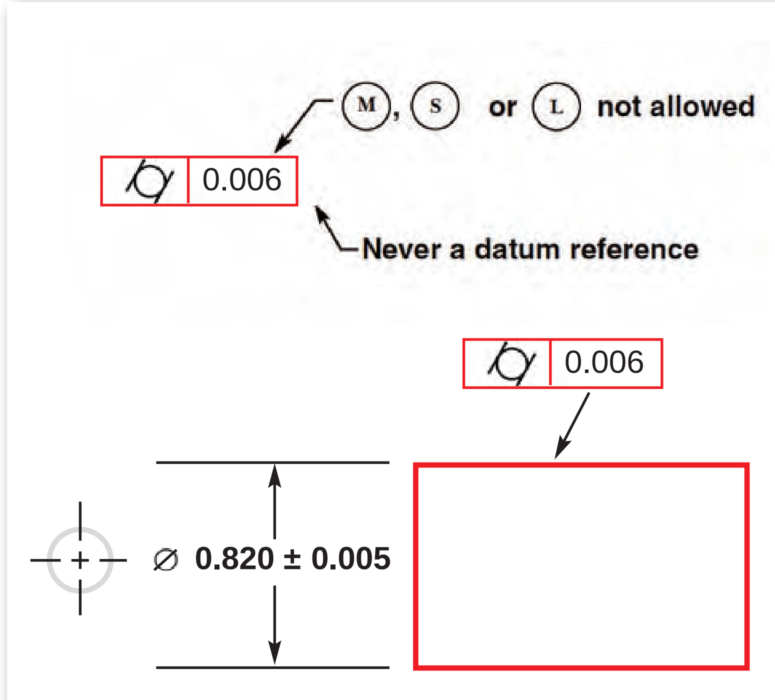
Other tolerances that provide the control of cylindricity include:

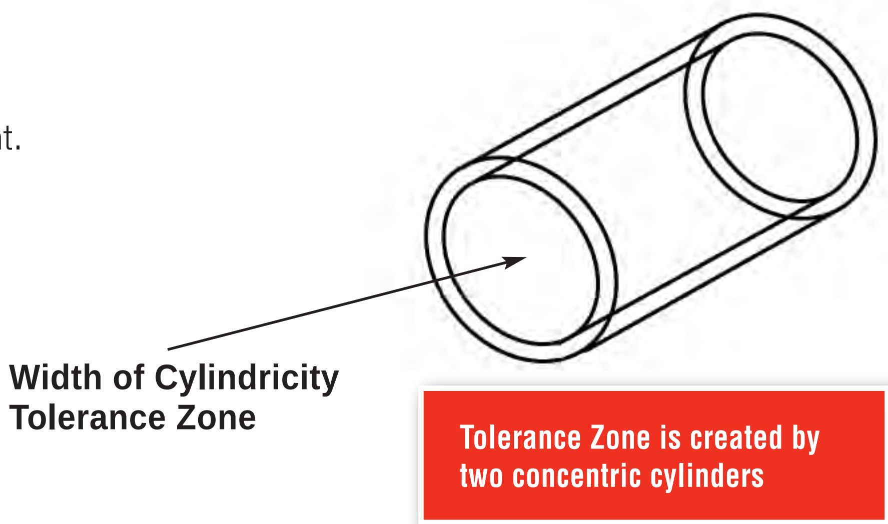
- Any size tolerance on a cylindrical feature.
- Any feature containing cylindrical features being controlled by:

Total Runout | ⌰ | 0.010 | A |

Rule of thumb:
Total runout is usually more cost effective to verify and should be considered when cylindricity is desired.

- No datum reference
- Independent of size
- May not be modified
- Does not locate or orient.

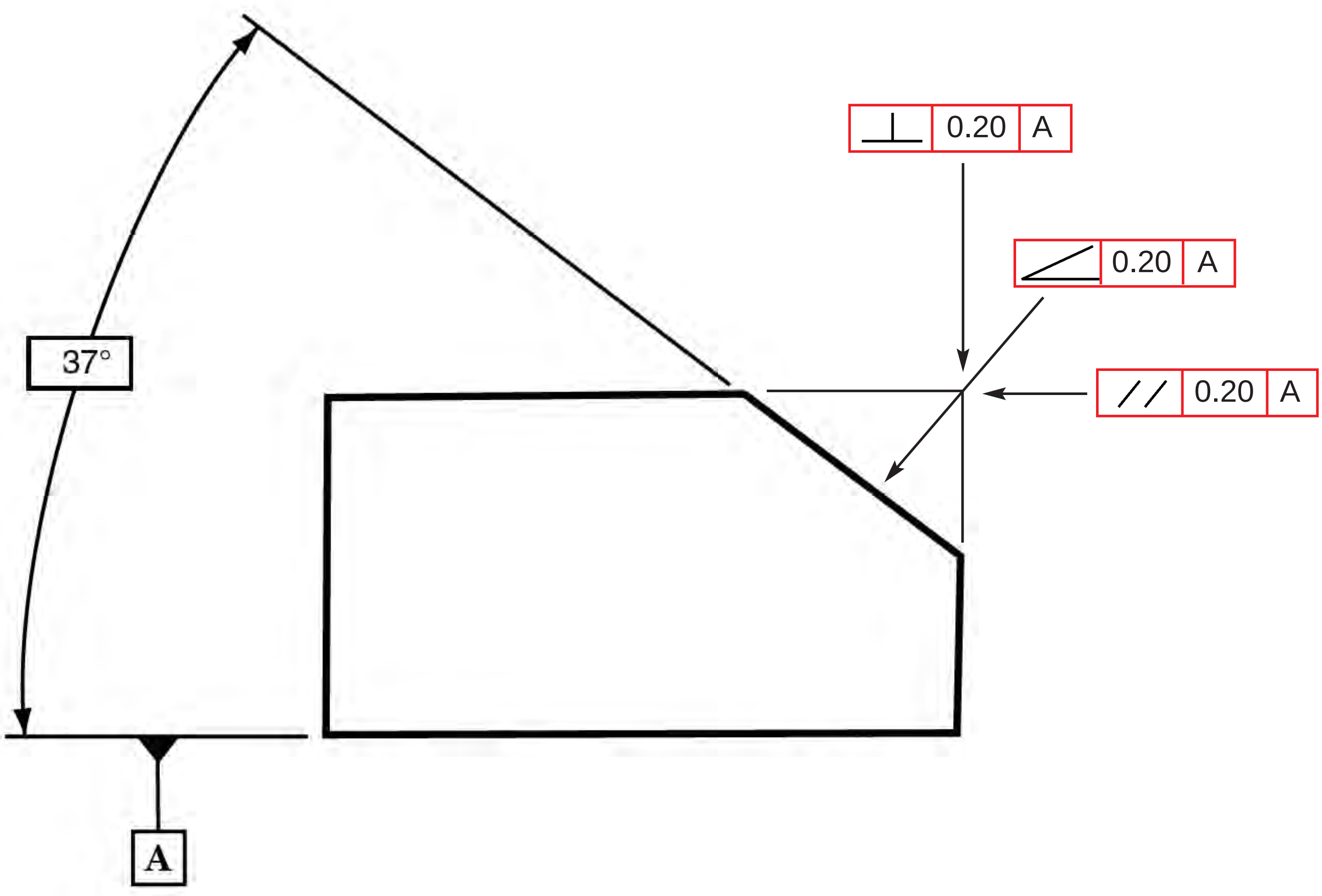




Orientation Tolerances

Orientation tolerances are applicable to related features, where one feature is selected as a datum feature and the other related to it. Orientation tolerances are perpendicularity, angularity, and parallelism.

Orientation tolerances control the orientation of a feature with respect to a datum that is established by a different part feature (the datum feature). For that reason, the tolerance will always include at least one datum reference. Orientation tolerances are considered on a "regardless of feature size" basis unless the maximum material condition modifier is added. The important thing to remember about orientation tolerances is that they do not locate features. Because of that, with the exception of perpendicularity on a secondary datum feature or a plane surface, orientation tolerances should not be the only geometric control on a feature. They should, instead, be used as a refinement of a tolerance that locates the feature.



Perpendicularity

Perpendicularity is the condition of a surface, axis, or line which is 90 degrees from a datum plane or a datum axis.

Perpendicularity is used on a secondary datum feature, relative to the primary datum.

It may be used to a tertiary datum feature not requiring location.

Other tolerances that may provide perpendicularity include:

Position | ⌖ | ⌀ 0.020 Ⓜ | A | B Ⓜ |

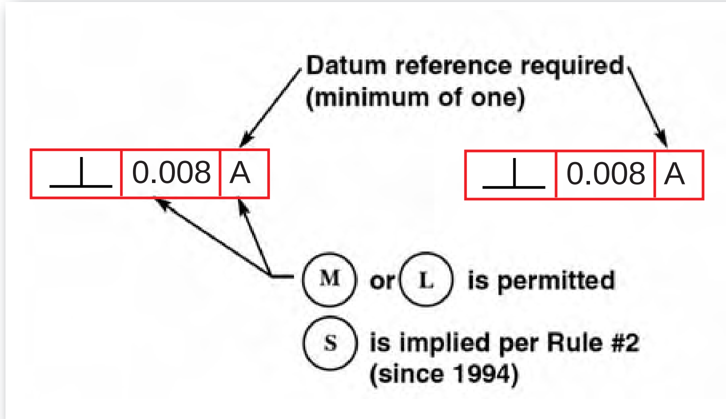
Profile of a Surface | ⌓ | 0.010 | A | B |

Total Runout | ⌰ | 0.010 | A |

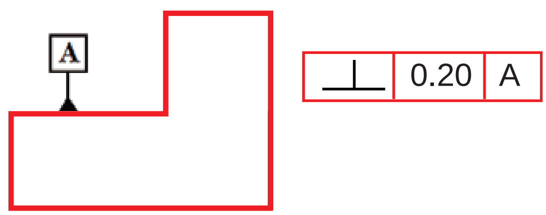
Therefore, perpendicularity should usually be used as a

| ⌖ | ⌀ 0.020 Ⓜ | A | B Ⓜ |

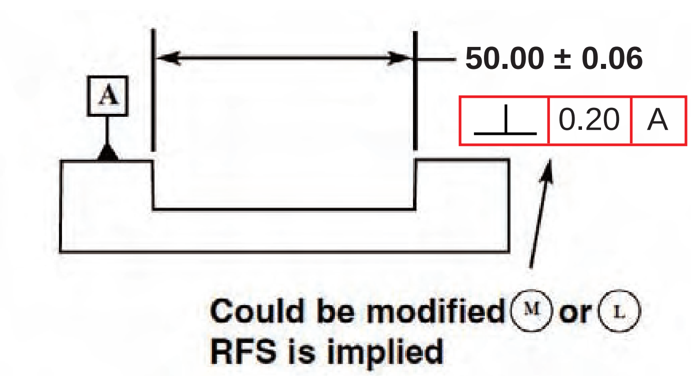
| ⊥ | ⌀ 0.008 | A |

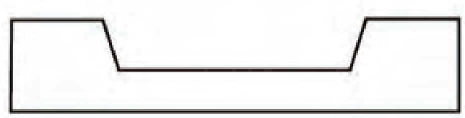


The perpendicularity tolerance is specified by being placed on an extension line. The tolerance zone is defined by a pair of parallel planes 0.2 mm apart. The tolerance zone is perfectly perpendicular to the datum plane -A-. The tolerance zone may be thought of as a flatness tolerance zone that is oriented at exactly 90 degrees to the datum.



The perpendicularity of features of size may also be controlled. The tolerance will be associated with the size dimension. When the size dimension applies to a pair of parallel planes (a slot or tab), the median or center plane is controlled by the tolerance.





Parallelism

When parallelism is applied to a flat surface, parallelism automatically provides flatness control and is usually easier to measure.

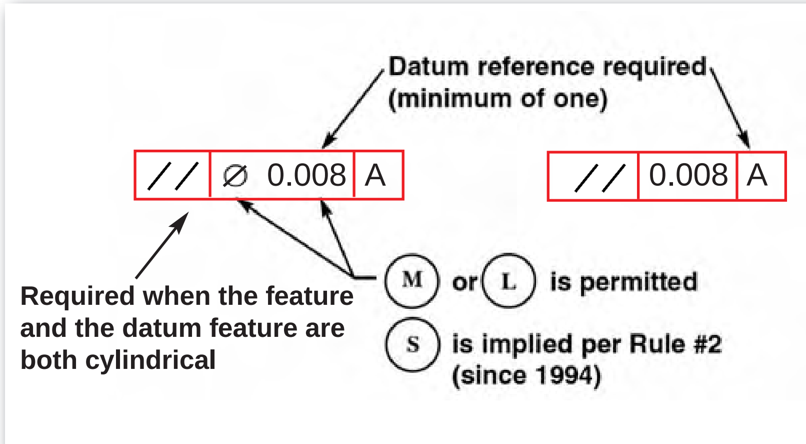
Other tolerances that may provide parallelism include:

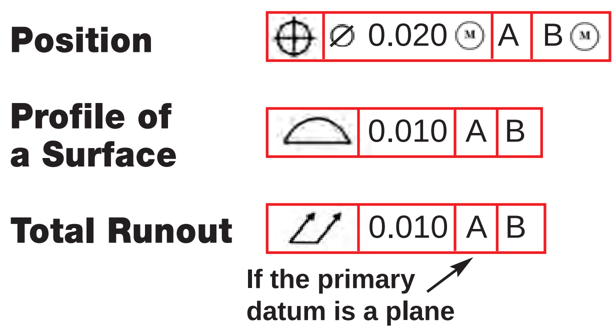
Any size tolerance on a feature composed of two internal or external parallel planes.

Features are considered parallel when the distance between them remains constant. Two lines, two surfaces, or a surface and a line may be parallel. The parallelism of features on a part is controlled by making one a datum feature and specifying a parallelism tolerance with respect to it.

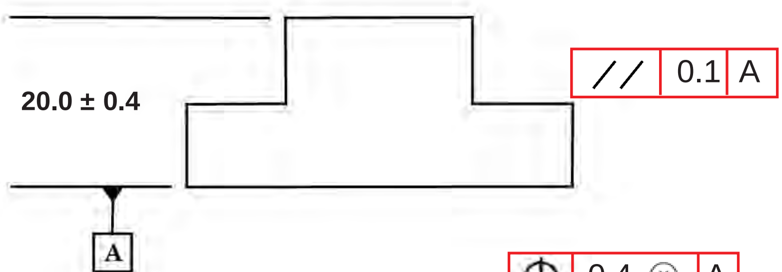
When parallelism is applied to a plane that is part of a feature of size and the other plane of that feature is the referenced datum feature, the parallelism tolerance cannot be greater than or equal to the total size tolerance or it would be meaningless since the plane's parallelism is automatically controlled by the size dimension.

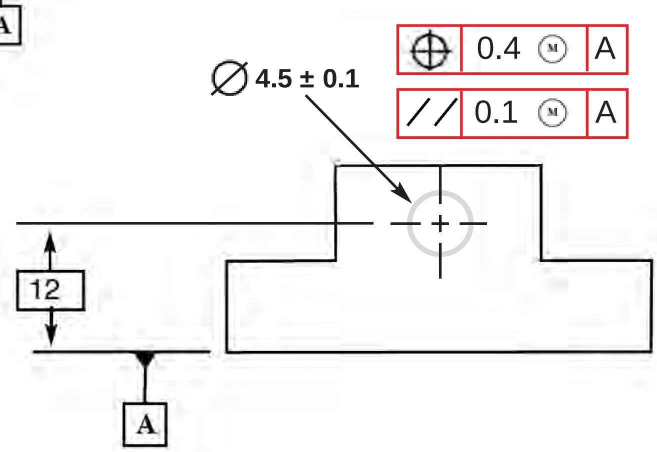
Parallelism can also be specified on an MMC basis. The MMC modifier can be on the feature tolerance, the datum feature, or both. As the feature deviates from its maximum material condition, the parallelism tolerance is increased.





Therefore, parallelism should easily be used as a refinement of Position Profile of a Surface.





Angularity

Angularity is the condition of a surface, axis, or center plane which is at a specified angle (other than 90 degrees) from a datum plane or axis.

Angularity, as a tolerance, always requires a BASIC angle.

Other tolerances that may provide angular control of features include:

- A tolerance in degrees applied to an angular dimension (not BASIC), provided there is a general note on the drawing relating toleranced dimensions to a datum reference frame.

Position | ⌖ | ⌀ 0.020 Ⓜ | A | B Ⓜ |

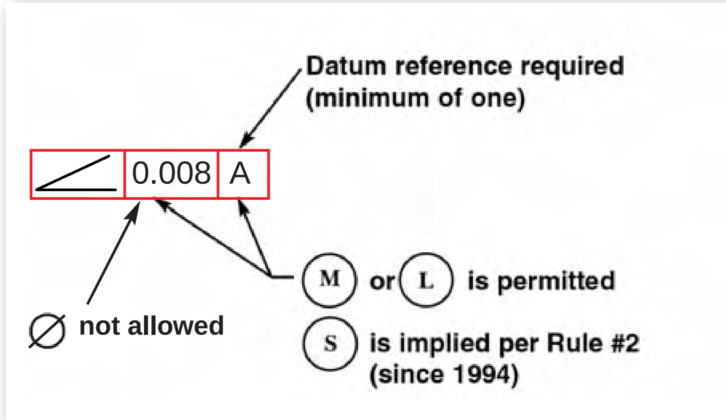
Profile of a Surface | ⌓ | 0.010 | A | B |

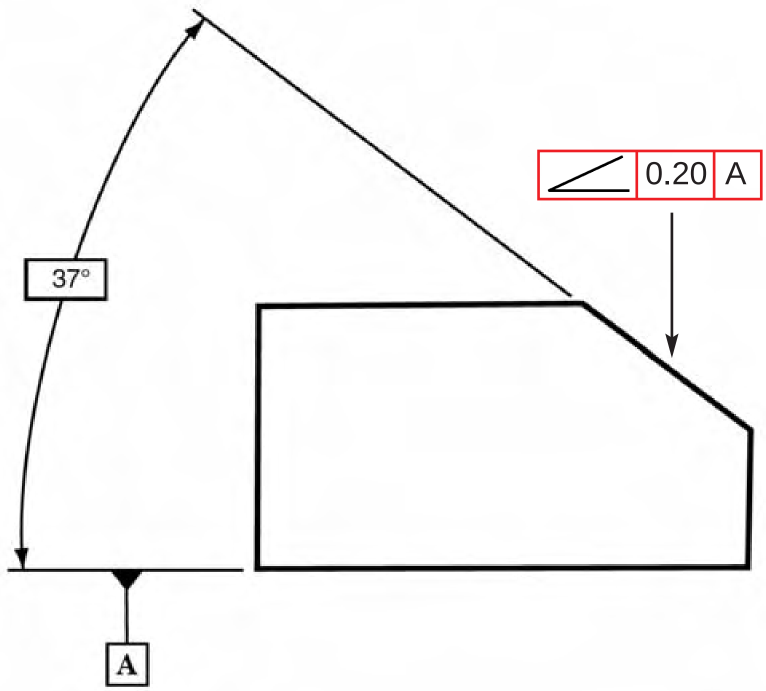
Therefore, angularity should usually be used as a refinement of one of the above:

| ⌖ | ⌀ 0.020 Ⓜ | A | B Ⓜ |

| ∠ | 0.008 | A |

Angularity is used to control the orientation of features to a datum axis or datum plane when they are at some angle other than 0 or 90 degrees. Since angularity does not locate features, it should only be considered after the feature is located. Usually a locating tolerance such as position or profile will do an adequate job of controlling the angularity and further refinement will not be necessary. A Basic Angle must always be applied to the feature from the referenced datum.



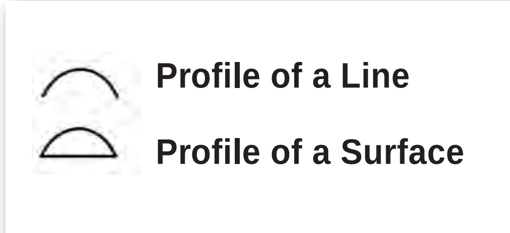


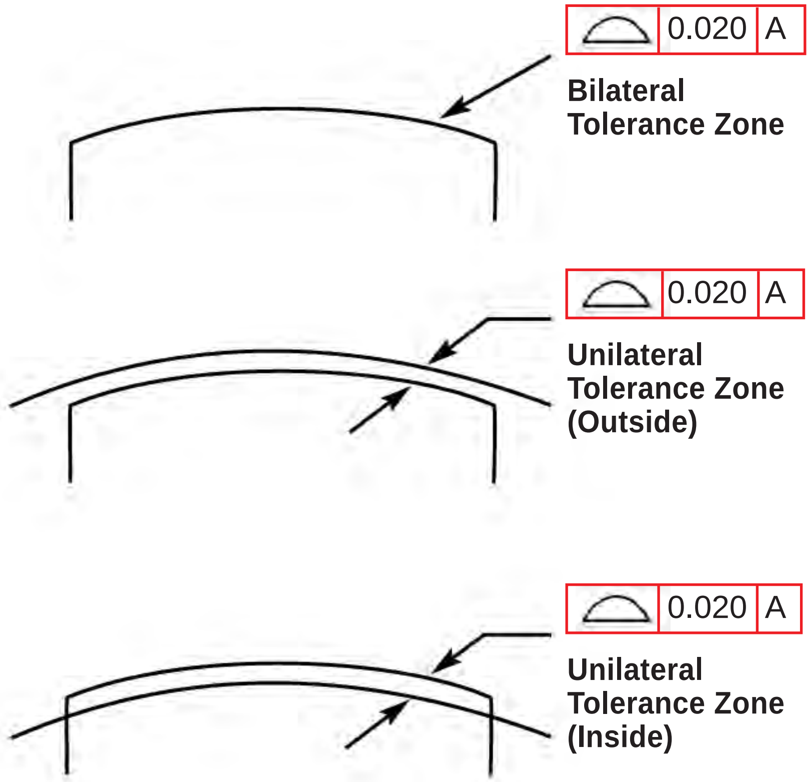
Angularity
– Must always have a datum reference
– May be modified when controlling a feature of size
– Does not locate features
– Requires a basic angle.

Profile

Profile is one of the least used--and yet most useful--geometric tolerances available. There are two types of profile tolerance: profile of a line and profile of a surface. The profile tolerances are the only geometric tolerances that may have a datum reference or may not. Without a datum reference in the feature control frame, the profile tolerance is controlling form. Profile of a line is very similar to the control seen with straightness or circularity. Profile of a surface is similar to the flatness or cylindricity tolerance. Care should be exercised in using profile without a datum. It usually makes the inspection of the part more difficult.

With a datum reference, the profile tolerance may control form, orientation, and location. Under certain conditions, profile may also control size. When a profile tolerance is used on the drawing, the tolerance is implied to be centered on the surface of the feature that has been defined by basic dimensions. If it is desired that the profile tolerance apply only in one direction, this can be illustrated on the drawing using a phantom line to indicate the side of the surface to which the tolerance should apply. This method of specifying the tolerance in only one direction is extremely useful for applications such as a punch and die in tooling or a cover on a housing where the internal and external features have an irregular shape. The basic shape of the object being controlled with profile must be dimensioned or defined using basic dimensions.





The tolerance zone is implied to be centered on the basic surface unless shown otherwise on the drawing.

Profile of a Surface

Profile of a surface is the condition permitting a uniform amount of a profile variation, either unilaterally or bilaterally, on a surface. (Profile tolerances are the only geometric tolerances where datum referencing is optional.)

Without a datum reference, profile of a surface controls the form of the surface (similar to straightness or circularity).

Form, orientation, and location may be controlled through datum referencing.

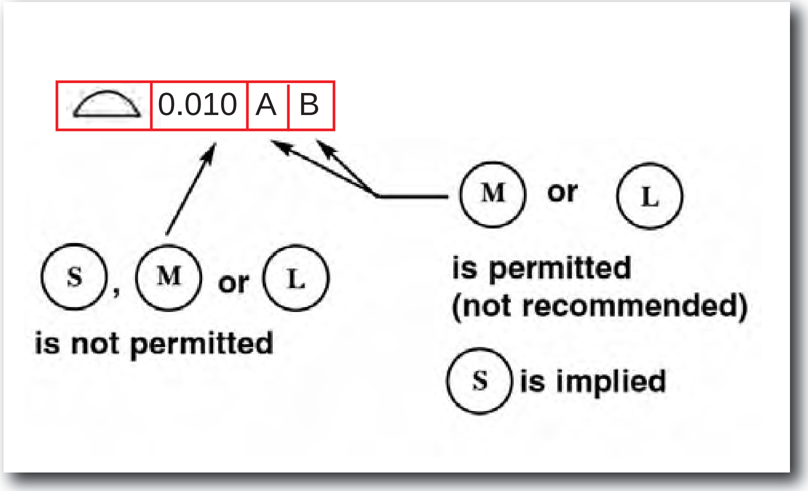
If a size dimension is made basic, profile of a surface may also control size.

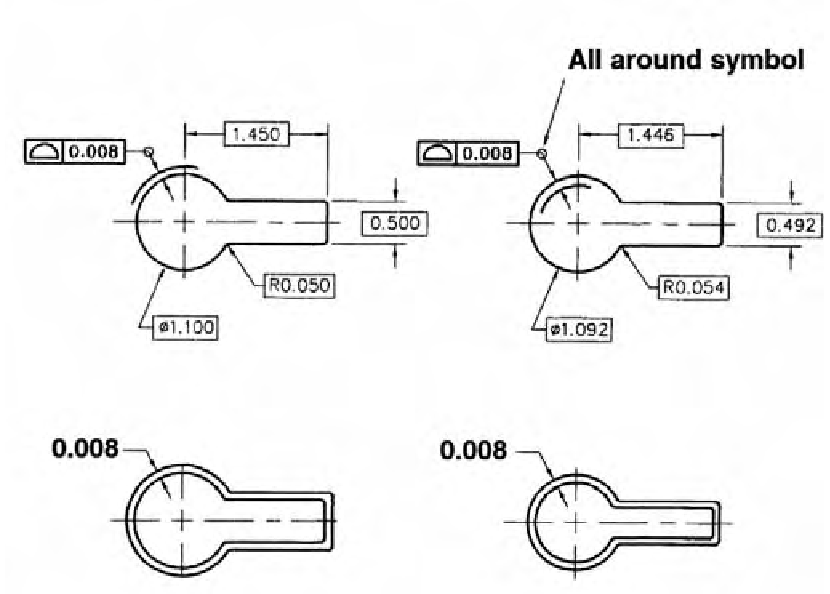
The shape of the feature must be described using basic dimensions.

The best application of profile of a surface is to locate plane and contoured surfaces.

When irregular parts must fit together, the use of unilateral profile tolerancing makes tolerance analysis easy for the designer. This approach may make manufacturing and inspection more difficult since many computer numerically controlled (CNC) machine tools and inspection machines now use the CAD file, which should usually be created at the goal or middle values.

Without a datum reference, profile of a surface controls the form of the surface (similar to straightness or circularity).





Profile of a Line

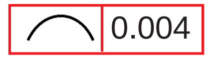
Profile of a line is the condition permitting a uniform amount of profile variation, either unilaterally or bilaterally, along a line element of a feature. (Profile tolerances are the only geometric tolerances where datum referencing is optional.)

Without a datum reference, profile of a line controls the form of lines independently within a surface (similar to straightness or circularity).

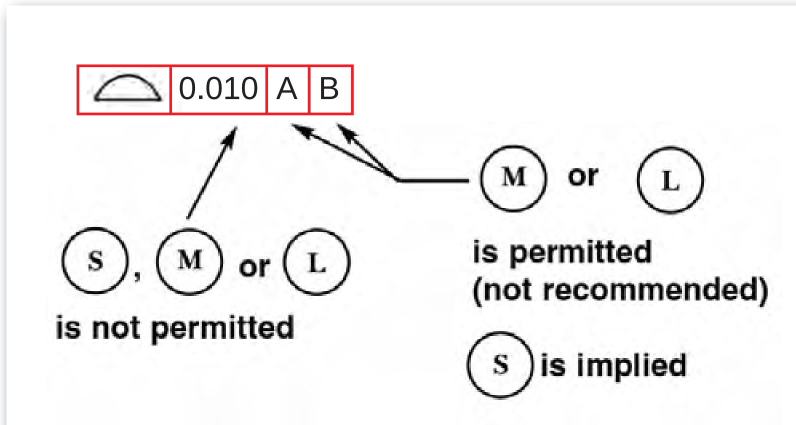
Both form and orientation are controlled through datum referencing.

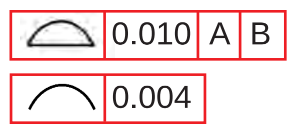
Unless dealing with thin parts, profile of a surface is a better choice for location.

The shape of the feature must be described using basic dimensions.



Without a datum reference, profile of a line controls the form of linesindependently within a surface (similar to straightness or circularity).

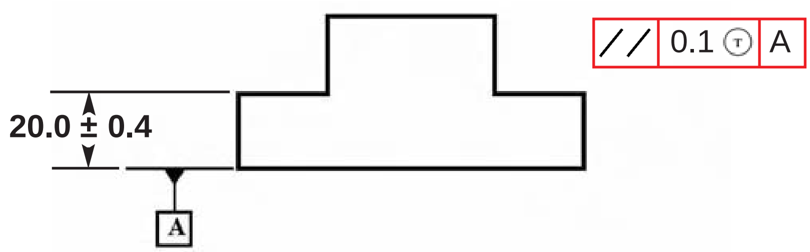


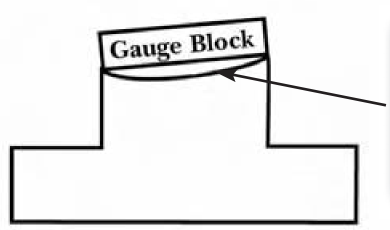


Since profile of a surface also controls the lines within the surface, profile of a line is often used to refine profile of a surface.

Tangent Plane

Tangent plane is a new concept/symbol, introduced in the 1994 Standard. Normally when a surface is inspected for Perpendicularity, Parallelism, Angularity, Profile of a Surface, or Total Runout, the flatness must also fall within the aforementioned geometric tolerance or the part would fail. Tangent Plane exempts the flatness requirement. The gauge block is intended to simulate the mating part.





Ignore the out-of-flat condition when checking parallelism.

Concentricity

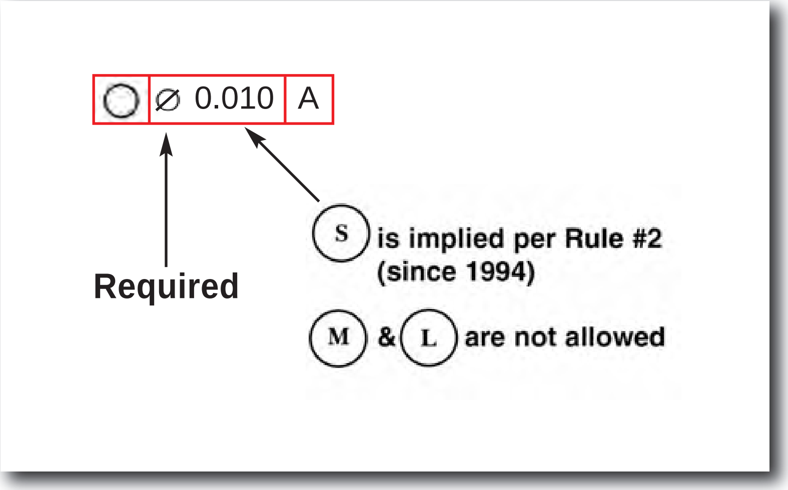
Concentricity is a condition in which two or more features (cylinders, cones, spheres, hexagons, etc.) in any combination have a common axis.

The datum(s) referenced must establish an axis.

Consider circular runout instead of concentricity:

- Runout is easier to verify
- Runout also controls the form of the feature.

Concentricity is a static attempt to control dynamic balance.



APPENDIX to Section 8
Basic Terminology for Geometric Tolerancing

actual size — An actual size is the measured size of the feature.

angularity — Angularity is the condition of a surface, axis, or center plane, which is at a specified angle (other than 90 degrees) from a datum plane or axis.

basic dimension — A dimension specified on a drawing as Basic (or abbreviated BSC) is a theoretical value used to describe the exact size, shape, or location of a feature. It is used as the basis from which permissible variations are established by tolerances on other dimensions or notes.

basic size — The basic size is that size from which limits of size are derived by the application of allowances and tolerances.

bilateral tolerancing — A bilateral tolerance is a tolerance in which variation is permitted in both directions from the specified dimension.

center plane — Center plane is the middle or median plane of a feature.

circular runout — Circular runout is the composite control of circular elements of a surface independently at any circular measuring position as the part is rotated through 360 degrees.

circularity — Circularity is the condition on a surface of revolution (cylinder, cone, sphere) where all points of the surface intersected by any plane (1) perpendicular to a common axis (cylinder, cone) or (2) passing through a common center (sphere) are equidistant from the center.

clearance fit — A clearance fit is one having limits of size so prescribed that a clearance always results when mating parts are assembled.

coaxiality — Coaxiality of features exists when two or more features have coincident axes, i.e., a feature axis and a datum feature axis.

concentricity — Concentricity is a condition in which two or more features (cylinders, cones, spheres, hexagons, etc.) in any combination have a common axis.

contour tolerancing — See profile of a line or profile of a surface.

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

datum — A datum is a theoretically exact point, axis, or plane derived from the true geometric counterpart of a specified datum feature. A datum is the origin from which the location or geometric characteristics of features of a part are established.

datum axis — The datum axis is the theoretically exact center line of the datum cylinder as established by the extremities or contacting points of the actual datum feature cylindrical surface, or the axis formed at the intersection of two datum planes.

datum feature — A datum feature is an actual feature of a part which is used to establish a datum.

datum feature symbol — The datum feature symbol contains the datum reference letter in a rectangular box.

datum line — A datum line is that which has length but no breadth or depth such as the intersection line of two planes, center line or axis of holes or cylinders, reference line for functional, tooling, or gauging purposes. A datum line is derived from the true geometric counterpart of a specified datum feature when applied in geometric tolerancing.

datum plane — A datum plane is a theoretically exact plane established by the extremities or contacting points of the datum feature (surface) with a simulated datum plane (surface plate or other checking device). A datum plane is derived from the true geometric counterpart of a specified datum feature when applied in geometric tolerancing.

datum point — A datum point is that which has position but no extent such as the apex of a pyramid or cone, center point of a sphere, or reference point on a surface for functional, tooling, or gauging purposes. A datum point is derived from a specified datum target on a part feature when applied in geometric tolerancing.

datum reference — A datum reference is a datum feature as specified on a drawing.

datum reference frame — A datum reference frame is a system of three mutually perpendicular datum planes or axes established from datum features as a basis for dimensions for design, manufacture, and verification. It provides complete orientation for the feature involved.

datum surface — A datum surface or feature (hole, slot, diameter, etc.) refers to the actual part surface or feature coincidental with, relative to, and/or used to establish a datum.

datum target — A datum target is a specified datum point, line, or area (identified on the drawing with a datum target symbol) used to establish datum points, lines, planes, or areas for special function, or manufacturing and inspection repeatability.

dimension — A dimension is a numerical value expressed in appropriate units of measure and indicated on a drawing.

feature — Feature is the general term applied to a physical portion of a part, such as a surface, hole, pin, slot, tab, etc.

feature of size — A feature of size may be one cylindrical or spherical surface, or a set of two plane parallel surfaces, each of which is associated with a dimension; it may be a feature such as hole, shaft, pin, slot, etc. which has an axis, centerline, or centerplane when related to geometric tolerances.

feature control frame — The feature control frame is a rectangular box containing the geometric characteristic symbol and the form, orientation, profile, runout, or location tolerance. If necessary, datum references and modifiers applicable to the feature of the datums are also contained in the frame.

fit — Fit is the general term used to signify the range of tightness or looseness which may result from the application of a specific combination of allowances and tolerance on the design of mating part features. Fits are of four general types: clearance, interference, transition, and line.

flatness — Flatness is the condition of a surface having all elements in one plane.

form tolerance — A form tolerance states how far an actual surface or feature is permitted to vary from the desired form implied by the drawing. Expressions of these tolerances refer to flatness, straightness, circularity, and cylindricity.

full indicator movement (FIM) (see also FIR and TIR) — Full indicator movement is the total movement observed with the dial indicator (or comparable measuring device) in contact with the part feature surface during one full revolution of the part about its datum axis. Full indicator movement (FIM) is the term used internationally. United States terms FIR, and TIR, used in the past, have the same meaning as FIM. Full indicator movement also refers to the total indicator movement observed while in traverse over a fixed noncircular shape.

full indicator reading (FIR) — Full indicator reading is the total indicator movement reading observed with the dial indicator in contact with the part feature surface during one full revolution of the part about its datum axis. Use of the international term, FIM (which, see), is recommended. Full indicator reading also refers to the full indicator reading observed while in traverse over a fixed noncircular shape.

geometric characteristics — Geometric characteristics refer to the basic elements or building blocks which form the language of geometric dimensioning and tolerancing. Generally, the term refers to all the symbols used in form, orientation, profile, runout, and location tolerancing.

implied datum — An implied datum is an unspecified datum whose influence on the application is implied by the dimensional arrangement on the drawing—e.g., the primary dimensions are tied to an edge surface; this edge is implied as a datum surface and plane.

interference fit — An interference fit is one having limits of size so prescribed that an interference always results when mating parts are assembled. 8-53 Aluminum Extrusion Manual

interrelated datum reference frame — An interrelated datum reference frame is one which has one or more common datums with another datum reference frame.

least material condition (LMC) — This term implies that condition of a part feature wherein it contains the least (minimum) amount of material, e.g., maximum hole diameter and minimum shaft diameter. It is opposite to maximum material condition (MMC).

limits of size — The limits of size are the specified maximum and minimum sizes of a feature.

limit dimensions (tolerancing) — In limit dimensioning only the maximum and minimum dimensions are specified. When used with dimension lines, the maximum value is placed above the minimum value, e.g., .300 - .295. When used with leader or note on a single line, the minimum limit is placed first, e.g., .295 - .300.

line fit — The limits of size are the specified maximum and minimum sizes of a feature.

location tolerance — A location tolerance states how far an actual feature may vary from the perfect location implied by the drawing as related to datums or other features. Expressions of these tolerances refer to the category of geometric characteristics containing position and concentricity (formerly also symmetry).

maximum material condition (MMC) — Maximum material condition is that condition where a feature of size contains the maximum amount of material within the stated limits of size, e.g., minimum hole diameter and maximum shaft diameter. It is opposite to least material condition.

maximum dimension — A maximum dimension represents the acceptable upper limit. The lower limit may be considered any value less than the maximum specified.

minimum material condition — See least material condition.

modifier (material condition symbol) — A modifier is the term sometimes used to describe the application of the “maximum material condition,” “regardless of feature size,” or “least material condition” principles. The modifiers are maximum material condition (MMC), regardless of feature size (RFS), and least material condition (LMC).

multiple datum reference frames — Multiple datum reference frames are more than one datum reference frame on one part.

nominal size — The nominal size is the stated designation which is used for the purpose of general identification, e.g., 1.400, .060, etc.

normality — See perpendicularity.

orientation tolerance — Orientation tolerances are applicable to related features, where one feature is selected as a datum feature and the other related to it. Orientation tolerances are perpendicularity, angularity, and parallelism.

parallelepiped — This refers to the shape of the tolerance zone. The term is used where total width is required and to describe geometrically a square or rectangular prism, or a solid with six faces, each of which is a parallelogram.

perpendicularity — Perpendicularity is the condition of a surface, axis, or line which is 90 degrees from a datum plane or a datum axis.

position tolerance — A position tolerance (formerly called true position tolerance) defines a zone within which the axis or center plane of a feature is permitted to vary from true (theoretically exact) position.

profile tolerance — Profile tolerance controls the outline or shape of a part as a total surface or at planes through a part.

profile of line — Profile of line is the condition permitting a uniform amount of profile variation, either unilaterally or bilaterally, along a line element of a feature.

profile of surface — Profile of a surface is the condition permitting a uniform amount of profile variation, either unilaterally or bilaterally, on a surface.

projected tolerance zone — A projected tolerance zone is a tolerance zone applied to a hole in which a pin, stud, screw, or bolt, etc. is to be inserted. It controls the perpendicularity of the hole to the extent of the projection from the hole and as it relates to the mating part clearance. The projected tolerance zone extends above the surface of the part to the functional length of the pin, screw, etc., relative to its assembly with the mating part.

regardless of feature size (RFS) — This is the condition where the tolerance of form, runout, or location must be met irrespective of where the feature lies within its size tolerance.

roundness — See circularity.

runout — Runout is the composite deviation from the desired form of a part surface of revolution during full rotation (360 degrees) of the part on a datum axis. Runout tolerance may be circular or total.

runout tolerance — Runout tolerance states how far an actual surface or feature is permitted to deviate from the desired form implied by the drawing during full rotation of the part on a datum axis. There are two types of runout: circular runout and total runout.

size tolerance — A size tolerance states how far individual features may vary from the desired size. Size tolerances are specified with either unilateral, bilateral, or limit tolerancing methods.

specified datum — A specified datum is a surface or feature identified with a datum feature symbol.

squareness — See perpendicularity.

straightness — Straightness is a condition where an element of a surface or an axis is a straight line.

symmetry — Symmetry is a condition in which a feature (or features) is (are) symmetrically disposed about the center plane of a datum feature.

tolerance — A tolerance is the total amount by which a specific dimension may vary; thus, the tolerance is the difference between limits.

transition fit — A transition fit is one having limits of size so prescribed that either a clearance or an interference may result when mating parts are assembled.

true position — True position is a term used to describe the perfect (exact) location of a point, line, or plane of a feature in relationship with a datum reference or other feature.

total indicator reading (TIR) (see also FIR and FIM) — Total indicator reading is the full indicator reading observed with the dial indicator in contact with the part feature surface during one full revolution of the part about its datum axis. Total indicator reading also refers to the total indicator reading observed while in traverse over a fixed noncircular shape. Use of the international term, FIM (which, see), is recommended.

total runout — Total runout is the simultaneous composite control of all elements of a surface at all circular and profile measuring positions as the part is rotated through 360 degrees.

unilateral tolerance — A unilateral tolerance is a tolerance in which variation is permitted only in one direction from the specified dimension, e.g., 1.400 + .000 - .005.

virtual condition — Virtual condition of a feature is the collective effect of size, form, and location error that must be considered in determining the fit or clearance between mating parts or features. It is a derived size generated from the profile variation permitted by the specified tolerances. It represents the most extreme condition of assembly at MMC.