Geometric Dimensioning and Tolerancing

Geometric Dimensioning and Tolerancing (GD&T) as defined in ASME and ISO standards offers a great many benefits for clear, complete and unambiguous product specifications. Standards, such as the one entitled ASME Y14.5M-1994 Dimensioning and Tolerancing, allows users to show design requirements and their tolerances in ways not available with plus and minus tolerances alone. For example:

1. Functional Gages can be utilized to verify parts will fit into the assembly under the worst possible mating conditions. Cylindrical tolerance zones displayed in feature control frames can be assigned to cylindrical features. This makes tolerance stack-up analysis easier and more reliable. A typical feature control frame looks like this: <υ?δ0.3μ?Α?Β?Χ>. This type of control makes it possible to measure the produced features with cylindrical gage pins and gage holes for orientation and location controls. Geometric tolerances, like the position tolerance shown in this feature control frame, are easily and functionally measured in this way (see figures 1-1 thru 1-3).

2. Fixed and Floating Fastener assembly tolerances (such as position tolerances) can be calculated using simple formulas that can be applied in seconds. Their tolerances are based on manufacturing capabilities and cost considerations and can be tested through easy to understand formulas to assure all parts accepted by inspection will assemble (see figures 2-1 thru 2-3). Also, tolerances that are projected out of threaded holes to simulate the screws make it much more likely that parts will assemble without binding (see figures 2-5 and 2-6).

3. Specified datums act as origins of measurement for orientation and location. They act as a coordinate system and make it possible for everyone to set the produced parts up in the same way. This allows collected inspection data to be repeatable, functional and representative of mating features (see figures 3-1 thru 3-5).

4. Additional tolerances based on produced feature sizes can be allowed though the use of the maximum and least material condition symbols without the loss of quality or functionality. When combined with the fact that cylindrical tolerance zones provide about 57% more manufacturing tolerances than the equivalent square plus and minus tolerance zone, design engineers who use geometric dimensioning and tolerancing will consistently design highly functional parts at minimum manufacturing costs (see figures 4-1 thru 4-4).

5. Features that are on a common axis or center plane can be toleranced to control how far off center they are allowed to shift from one another (see figures 5-1 thru 5-6). These are just a few of the benefits of Geometric Dimensioning and Tolerancing. The following sections describe these benefits in more detail and how they can be applied to typical design situations. Also included at the end of this document, Appendix A provides a general description of the tolerance zones derived by each the geometric tolerances.
**Functional Gages**

For companies who mass produce components, the application of geometric tolerancing can greatly facilitate inspection when combined with the use of functional gages. Like other types of receiver gages, functional gages offer quick and easy determination of a part’s conformance with its geometric tolerances. Gages of this type are especially effective in determining compliance with position tolerances.

For example, consider the part in figure 1-1, the bolt hole pattern and outside diameter are simultaneously positioned with respect to datum plane A and datum axis B. In addition, datum feature B has a perpendicularity requirement with respect to datum A. Given this tolerancing scheme a single functional gauge can be used to simultaneously inspect the perpendicularity control and both position controls. The dimensional characteristics of the gauge are shown in figure 1-2.

![Figure 1-1 Datums and Geometric Tolerances](image.png)
By utilizing this functional gage, inspectors can easily determine if produced parts are in compliance with their geometric controls. If the parts fit into the gage and seat with a minimum of 3 points of high point contact on datum feature A, all geometric tolerances (other than flatness) have been met. The flatness control and the part’s size limits are separately verified. See Figure 1-3 for a demonstration of the use of this functional gage.

Figure 1-2 Functional Gage

Figure 1-3 Inspection Procedure
Fixed and Floating Fastener Assemblies

The calculation of position tolerances utilizing formulas for fixed and floating fastener assembly conditions offers one of the greatest benefits of Geometric Dimensioning and Tolerancing. When used correctly, these formulas ensure assemblability with the maximum geometric tolerances arithmetically available. For examples of fixed and floating fastener assembly tolerance calculations see figures 2-1 and 2-3.

Floating Fastener Assemblies

An assembly of mating parts containing clearance holes that are fastened with bolts and nuts is considered to be a floating fastener assembly. The fasteners are free to float within the resultant boundaries of the clearance holes. See Figure 2-1.

To ensure the parts can assemble without interference, the following formula is used to derive the position tolerances applied to the holes on both parts.

\[ T = H - F \]

Where:
- \( T \) = position tolerance
- \( H \) = Minimum diameter of clearance hole (the hole’s maximum material condition)
- \( F \) = Maximum diameter of fastener (the fastener’s maximum material condition)

Given the assembly in figure 2-1, if the design requirement is to use M10 bolts and a smallest clearance hole size of 10.1, we can apply the formula below. From this, the correct positional tolerance on each part can be calculated which will guarantee these two parts will assemble with their fasteners, even at worse case conditions.

\[ T = H - F \]
\[ T = 10.1 - 10 \]
\[ T = 0.1 \text{ for all clearance holes on both parts} \]
Using a position tolerance of 0.1 to locate each 4-hole pattern, the tolerance scheme applied to each part could be as shown in figure 2-2.

Figure 2-2 Annotated Part Models for Floating Fastener Assembly

Also, if the positional tolerance is predefined it is possible to determine the smallest clearance hole size from the formula that will also guarantee the parts will assemble:

$$ H = F + T $$

For example, if the requirement was that the fastener size is to be M10 and the position tolerance 0.2, then:

$$ H = 10.0 + 0.2 $$

$$ H = 10.2 $$

Using a maximum material size (smallest hole size) of 10.2 on the two 4-hole patterns, the tolerance scheme applied to each part could be as shown in figure 2-3.

Figure 2-3 Formula Used to Specify Hole Size on 4-hole Patterns
**Fixed Fastener Assembly**

An assembly of mating parts containing clearance holes on one part and threaded holes on the other is considered to be a fixed fastener assembly. The fasteners are in a fixed position in the threaded holes. See Figure 2-3.

![Figure 2-3 Fixed Fastener Assembly](image)

To ensure the parts assemble without interference the following formula is used to derive the total position tolerance available to be divided between the part with the clearance holes and the part with the threaded holes:

\[ T = H - F \]

Where:

- **T** = Position tolerance to be divided between the parts
- **H** = Minimum diameter of clearance hole (the hole’s maximum material condition)
- **F** = Maximum diameter of fastener (the fastener’s maximum material condition)

Given the assembly in figure 2-3, if the design requirement is to use M10 bolts and a smallest clearance hole size of 10.1, we can apply the formula below.

\[ T = H - F \]
\[ T = 10.4 - 10.0 \]
\[ T = 0.4 \]

This position tolerance may be divided equally between the two parts, or a larger position tolerance given to the threaded holes on the basis that they are harder to manufacture. In this case, it was determined the threaded holes would receive 0.3 of the 0.4 position tolerance and the remaining 0.1 would be assigned to the clearance holes. See figure 2-4.
It should be considered that the parts may experience binding when the screw is threaded into the holes. This is caused by the threaded holes using part, or all, of their position tolerance as out of perpendicularity tolerance. This problem can be corrected by applying a Projected Tolerance Zone to the threaded holes. This concept is explained in the next section.
Fixed Fastener Case with Projected Tolerance Zones

A major issue to consider when dimensioning and tolerancing fixed fastener assemblies is binding problems caused when threaded holes are produced out-of-square. Position tolerances control both angle and location. If threaded holes are out of position, but perfectly perpendicular to their primary datum, no problem occurs. But, when threaded holes positioned with the formulas explained above, lean out of perpendicularity to the primary datum, it may not be possible to insert the screws into the holes to fasten the parts together or the fasteners may bind with the clearance holes.

To prevent these types of interferences, a projected tolerance zone is applied to the position tolerance of the threaded holes at a height equal to the maximum projection of the fasteners. The position tolerance zone then ceases to exist within the threaded holes. It exists solely outside of the threaded hole emulating the screw as it projects from the hole. See figure 2-5.

![Figure 2-5 Tolerance Zones with and without a Projected Tolerance Zone]

As can be seen in figure 2-6, the part with the clearance holes has a maximum thickness of 41, therefore the position tolerance zone required to contain the axis of the pitch cylinder for each threaded hole needs to be projected to a minimum height of 41. Remember, the position tolerance zone for the threaded holes lies entirely outside of the part when projected tolerance zones are used. See figure 2-7.
Figure 2-6 Fixed Fastener Assembly with Projected Tolerance Zones

Figure 2-7 Projected Tolerance Zone

Tolerance zone lies entirely outside and projected above datum A
Datum Reference Frames

Datum referencing allows designers to clearly state the required feature relationships of the product. They also provide clear guidelines for manufacturing and inspection from which to produce and inspect parts.

Consider the tolerance scheme shown in Figure 3-1(a). Do you think the two 50 dimensions are meant to control the two planes as shown in Figure 3-1(b)? Or do you think the dimensions are meant to control the location of the hole as shown in Figure 3-1(c)? There are design situations that validate the need for both cases. For example, PC boards and automobile panels rarely use the edges of the part from which to measure the holes. Similarly angle brackets rarely use the holes of the part from which to measure the edges.

Figure 3-1 Interpreting Plus and Minus Tolerances
Additionally, if the interpretation was meant to locate the hole from the edges of the part, is the location of the hole to be inspected in one or two setups? When using two setups, the hole is checked independently from each plane. See Figure 3-2(a). Note how the square tolerance zone turns into a skewed parallelogram on the actual part. However, when using one setup, there are two possibilities based on which plane is used to establish initial contact with the angle plate. In figure 3-2(b) the bottom plane is used to establish initial contact, and in figure 3-2(c) the right plane is used first. These inconsistencies are due to the fact that the origins of measurement are implied, rather than specified and given an order of precedence. Therefore, the interpretation is left up to the individual manufacturing engineer and inspector.

![Figure 3-2 Inspection Criteria](image_url)
With geometric tolerancing and properly applied datum schemes it’s clear what features are being controlled and what they are being controlled to. If, using the previous example, the requirement is to control the outer boundary of the part relative to the holes, the part could be tolerated as shown in Figure 3-3.

Figure 3-3 Profile of a Boundary
Conversely, if the holes are to be related to the outside planes, the part could be tolerated as shown in Figure 3-4.

Figure 3-4 Position of Hole Pattern

In each case, it is clear which features are being tolerated and what their origins of measurement (datum reference frame) are. For manufacturing the order in which the datums are specified shows them how to set the part up to achieve the required results. For inspection, they are given a clear, concise definition of the functional requirements to be met. Therefore, their measurement techniques will collect repeatable attribute (good vs. bad) data or variables data (how far the features have strayed from perfect and in what direction).

Fundamentally, datum features are used to establish datum reference frames, which are defined by a set of 3 mutually perpendicular planes. This constitutes a coordinate system that establishes the origins of measurement.
Consider the tolerance scheme shown in Figure 3-4. The position tolerance states that the two holes are to be positioned relative to the datum reference frame established by datum planes A, B, and C.

One way to simulate this datum reference frame would be with the fixture shown in Figure 3-5. To do this, the part would be placed into the fixture by: 1) assuring the part’s datum feature A has a minimum of 3 points of high point contact with the fixture base (simulator for datum A); 2) sliding the part along datum feature simulator A until datum feature B has a minimum of 2 points of high point contact against the side rail (simulator for datum B); and 3) sliding the part along datum feature simulators A and B until datum feature C has a minimum of 1 point of high point contact against the other side rail (simulator for datum C). The location of the two holes can then be inspected from the two side rails (B and C) while seated on A.

Figure 3-5 Fixture used to Check Part Shown in Figure 3-4
Maximum Material Condition

When functionality allows, the use of the maximum material condition (MMC) concept may be applied to the tolerated features and datum features of size. This provides the largest possible finished product tolerance (which translates into the least amount of production costs) while still guaranteeing that the manufactured parts will assemble without interference. MMC principles also are the key component that facilitates the use of functional gaging which provides the quickest and most direct method of inspecting a part. The concept of MMC is mainly used between the mating features of two parts.

To illustrate the MMC principle, consider the part and tolerances shown in Figure 5-1. The position tolerance applied to the 2-hole pattern states that a cylindrical tolerance of 0.2 is allowed when the features are produced at their MMC size. For a hole, the MMC size is the smallest allowable hole size, which in this case would be a diameter of 25.2. The MMC principle also allows for additional “bonus” positional tolerance to be given if the holes are produced larger than 25.2. The difference between the actual hole size and the MMC size is added to the original position tolerance as the “bonus” tolerance. In this example, there is a potential gain of 0.2 calculated by subtracting the holes MMC size from the holes least material condition (LMC) size, which for a hole feature is its largest allowable size. The result is a total possible positional tolerance of 0.4. This would be the maximum position tolerance for each hole in its allowed deviation from the perfect (basic) location of 100 millimeters from hole to hole.

Figure 4-1 Hole Pattern Toleranced at Maximum Material Condition (MMC)
The reasoning behind the “bonus” tolerance is that as the holes get larger there is additional clearance for the mating feature/fastener to fit through. For example, the part shown in Figure 4-1 is compared against a functional gauge shown in Figure 4-2. The gage pins are dimensioned to be a diameter of 25. When the distance between the holes is 100.4 and they are at their MMC size of 25.2, interferences occur. At that size, the maximum axis separation allowed between the holes would be 100.2. See Figure 4-2(a). If the holes are opened up (produced larger) to their LMC size of 25.4, the part is able to slide onto the gauge. With both holes produced at 25.4, the maximum axis separation between the holes could be 100.4 and the two holes would function just fine. See Figure 4-2(b).

(a) When the 2 holes are at their MMC size and the distance between their centers is 100.4 interferences occurs when the part is superimposed onto the gage. The area of interference is shown in red on the right

(b) When the 2 holes are at their LMC the larger sized holes provide additional clearance, allowing the part to fit onto the gage without interference

Figure 4-2 Functional Gauge Check with Features at MMC and LMC
**Datums Features Referenced at MMC**

A similar concept applies to datum features referenced at MMC in a feature control frame. In this case, the additional clearance provided by the datum features is commonly referred to as allowable “datum shift” or a “pattern shift”. Datum shift (or pattern shift) does not increase the size of the position tolerance for each hole in the pattern, since that would allow the holes to move closer to one another or further away from one another. Instead, it allows the holes as a group to shift an additional amount away from the datum feature axis.

To see the benefits derived by referencing datum features at MMC, refer to Figure 4-3. On this part the 2 holes are positioned to datum plane A and datum axis B (the large center hole), which is referenced at MMC.

![Figure 4-3 Datum Referenced at Maximum Material Condition](image)

If the part is produced as shown in Figure 4-4(a) with the two holes and center datum hole at their MMC size, no difference exists between the gage pins and the sizes of the 2-hole pattern and datum hole B. Since the holes are produced shifted off center of datum feature axis B by 0.2 (difference between the actual value 49.8 and nominal value 50), the part does not fit the gage (even if the line fit conditions were considered as clearance).

In Figure 4-4(b), even though the 2 hole pattern is produced at the same size as their gage pins and still shifted off center of datum feature axis B by 0.2, since datum feature B is produced at its LMC of 50.4, the allowable datum shift/pattern shift provided from datum feature B is enough to remove the interference caused even when the two holes are produced at their MMC size. The part is simply shifted to one side by the radial growth of datum feature B (0.2). The two holes would then slip down over their gage pins and datum hole B would receive its gage pin. The gage pin that represents B would not fit into the center of the B hole, but instead fits in while shifted off to one side by 0.2.
(a) With the datum feature at a diameter 50 it fits concentric with the 50 mm gage pin, which causes the two holes to interfere with the 25 mm gage pins. The area of interference is shown in red when the part is superimposed onto the gage.

(b) With the datum feature at 50.4 mm the part can be shifted to the left by 0.2 mm, which allows the 2X holes to fit over the 25 mm gauge pins.

Figure 4-4 Datum Shift or Pattern Shift
**Features on a Common Axis, Center Plane, or Centerline**

Having multiple features shown on a common axis, center plane, or centerline poses unique problems when trying to control their relationships with plus and minus tolerances. Consider the part shown in Figure 5-1 which shows three diameters on a common axis dimensioned and tolerated, in typical fashion, with only diameter tolerances. Given this dimension and tolerance scheme, what is the allowable variation between the axes of the three features? The answer is unknown. The only checks required by inspection are to ensure the 3 diameters are within their respective tolerance. Although features shown on a common axis are implied to be coaxial there must be a tolerance stating how far the axes are allowed to shift from one another. Geometric tolerancing provides several options for controlling the relationships of features on a common axis, which are selected based on the parts intended function. See Figure 5-2.

---

**Figure 5-1 Feature Designed on a Common Axis**

![3D part model and drawing view](image-url)
The tolerance scheme shown in 5-2(a) is applicable for housings or assemblies where primary requirement is assemblablility. The tolerance scheme shown in 5-2(b) is applicable for assemblies, typically rotating assemblies, where balance is a concern and there is a need to control the roundness of the features. Finally, the tolerance scheme shown 5-2(c) is applicable for assemblies where balance is a concern, but where the roundness is not a major concern or does not need refinement.
Features Designed on a Common Center Plane

Features shown on a common center plane can also be problematic and difficult for manufacturing and inspection to determine the parts intended functionality. Consider the plus and minus tolerance scheme shown on the part in figure 5-3(b). Similar to the case described above with feature shown on a common axis, there is no tolerance controlling the relationship between the center planes of the widths shown at 100 and 200. Figure 5-3(c) shows an example of how this can be alleviated using a plus and minus tolerance to control the center width to the left side of the part, however, the tolerance scheme does not provide direct control of the relationship between the two center planes, which may be critical to the design. With geometric tolerancing the relationships can be clearly stated with direct control over the variation between the two center planes. Figure 5-4 shows examples of how this can be accomplished using position and symmetry tolerances.
Figure 5-4 Geometric Tolerances for Features Designed on a Common Center Plane

Similar to the use of position to control coaxiality, the use of position to control the centering of planes, shown in 5-4(a), is applicable for parts where primary requirement is assemblability. And similar to the situations where concentricity is used, symmetry tolerances, shown in 5-4(b), is applicable for assemblies where balance is a primary concern.

Features Dimensioned on Common Centerlines

Features shown on common centerlines can be a point of confusion and ambiguity, especially when the features are related to multiple dimensions with different tolerances. For example, consider the part and the plus and minus tolerances shown in Figure 5-5. What is the allowable deviation (vertical direction) of the 2X 10 diameter holes and the center bore where shown on common centerlines? Is it +/- 1 as defined by the 40+/-1 dimension? Or is it +/- 2 as defined by the 50 +/-2 dimension? Or is it something else? Additionally, are the 45+/-1 dimensions used to control the location of the 15 diameter holes to the center bore or to the 10 diameter holes shown on the common vertical center line with the center bore? Knowing the functional intent of the design is critical for manufacturing to make the parts correctly and is critical for inspection to verify that the manufactured parts will accomplish the intended task. If the plus and minus tolerance scheme is converted to geometric tolerancing, these ambiguities are eliminated. The location and orientation tolerances for the 10 and 15 diameter holes and the center bore are clearly stated in the position feature control frames applied to each feature. See Figure 5-6.

Figure 5-5 Features on Common Centerline
Figure 5-6 Features on Common Centerline Controlled with Position Tolerances
Appendix A
Geometric Tolerances

The following descriptions provide general information on each of the geometric tolerances with reference to the ASME Y14.5M-1994 standard, with examples of the most commonly used cases. The descriptions and examples are not complete, for complete descriptions of each tolerance refer to the appropriate ASME or ISO standard.
**Flatness**

Flatness is the condition of a surface where all elements lie in a plane.

*Tolerance Zone:* The flatness tolerance specifies a zone bounded by two parallel planes separated by the specified tolerance.

*Datums:* No.

*Material Condition Modifiers:* No.

*Conformance:* A feature conforms to its flatness tolerance if all points of the feature lie within the tolerance zone.

---

**Figure A-1 Flatness**

a) Flatness applied to a Plane

b) Tolerance zone: two parallel planes separated by the specified tolerance

c) Conformance: all surface points lie within a flatness zone no greater than 0.25
**Circularity**
Circularity (excluding spheres) is the condition of a surface where all points of the surface in a plane perpendicular to an axis are equidistance to that axis.

*Tolerance Zone*: The circularity tolerance specifies a zone bounded by two concentric circles whose radial separation is the specified tolerance.

*Datums*: No.

*Material Condition Modifiers*: No.

*Conformance*: A feature conforms to its circularity tolerance if, for each cross sectional element (checks are done independently), all points of the feature lie within the tolerance zone.

![Figure A-2 Circularity](image)

- **a)** Circularity applied to a Hole and Boss
- **b)** Tolerance zone: two concentric circles whose radial separation is the specified tolerance
- **c)** Conformance: the smallest circularity zone to which the feature conforms

Figure A-2 Circularity
Cylindricity
Cylindricity is the condition of a surface where all points of the surface are equidistant to a common axis.

Tolerance Zone: The cylindricity tolerance specifies a zone bounded by two concentric cylinders whose radial separation is the specified tolerance.

Datums: No.

Material Condition Modifiers: No.

Conformance: A feature conforms to its cylindricity tolerance if all points of the feature lie within the tolerance zone.

Figure A-3 Cylindricity
Straightness - Surface
Straightness of a surface is the condition where an element of a surface is a straight line.
Tolerance Zone: The straightness of a surface tolerance specifies a zone bounded by two parallel lines separated by the specified tolerance that are oriented to the view plane in which the tolerance is stated.
Datums: No.
Material Condition Modifiers: No.
Conformance: A feature conforms to its straightness tolerance if, for each cross sectional element, all points of the feature lie within the tolerance zone.

Figure A-4 Surface Straightness
**Straightness - Axis**

Straightness of an axis is the condition where an axis is a straight line.

*Tolerance Zone:* The straightness of an axis tolerance specifies a zone bounded by a cylinder with a diameter equal to the specified tolerance.

*Datums:* No.

*Material Condition Modifiers:* Optional.

*Conformance:* A feature conforms to its straightness tolerance if all points on the derived median line lie within the tolerance zone.

---

**Figure A-5 Axis Straightness**

a) Axis Straightness applied to a Boss

b) Tolerance zone: cylindrical volume whose diameter is the specified tolerance.

c) Conformance: the smallest straightness zone to which the feature conforms
**Parallelism**
Parallelism is the condition of a surface, center plane, or axis (derived median line) being equidistant from one or more datum planes or a datum axis.

**Tolerance Zone:** A parallelism tolerance specifies a zone:
- a. bounded by two parallel lines or planes separated by the specified tolerance that are parallel to the datum plane or axis
- b. bounded by cylinder with a diameter equal to the specified tolerance whose axis is parallel to the datum plane or axis

**Datums:** Required.

**Material Condition Modifiers:** Optional on both the datum and considered features.

**Conformance:** A feature conforms to its parallelism tolerance if all elements of the surface, axis, or center plane lie within the tolerance zone.

---

**Figure A-6 Parallelism Applied to a Plane**

a) Parallelism applied to a Plane

b) Tolerance zone: two parallel planes separated by the specified tolerance that are parallel to datum plane A

c) Conformance: the smallest parallelism zone to which the feature conforms
**Perpendicularity**

Perpendicularity is the condition of a surface, center plane, or axis (derived median line) being at a right angle to a datum plane or axis.

*Tolerance Zone:* A perpendicularity tolerance specifies a zone:

- bounded by two parallel lines or planes separated by the specified tolerance that are perpendicular to the datum plane or axis
- bounded by cylinder with a diameter equal to the specified tolerance whose axis is perpendicular to the datum plane or axis

*Datums:* Required.

*Material Condition Modifiers:* Optional on both the datum and considered features.

*Conformance:* A feature conforms to its perpendicularity tolerance if all elements of the surface, axis, or center plane lie within the tolerance zone.

---

**Figure A-7 Perpendicularity Applied to a Hole**

a) Perpendicularity applied to a Hole

b) Tolerance zone: a cylindrical boundary with a diameter equal to the specified tolerance that is perpendicular to datum plane A

c) Conformance: the smallest perpendicularity zone to which the feature conforms
**Angularity**

Angularity is the condition of a surface, center plane, or axis (derived median line) being at a specified (basic) angle to a datum plane or axis.

**Tolerance Zone**: An angularity tolerance specifies a zone:

- c. bounded by two parallel lines or planes separated by the specified tolerance that is at the specified (basic) angle to the datum plane or axis
- d. bounded by cylinder with a diameter equal to the specified tolerance whose axis is at a specified (basic) angle to the datum plane or axis

**Datums**: Required.

**Material Condition Modifiers**: Optional on both the datum and considered features.

**Conformance**: A feature conforms to its angularity tolerance if all elements of the surface, axis, or center plane lie within the tolerance zone.

---

**Figure A-8 Angularity Applied to a Plane**

- a) Angularity applied to a Plane
- b) Tolerance zone: two parallel planes separated by the specified tolerance that is at the specified (basic) angle to datum plane A
- c) Conformance: the smallest angularity zone to which the feature conforms
Position
A position tolerance is a composite control that controls the location and/or orientation of a center plane, axis, or point with respect to one or more datum features.

Tolerance Zone: A position tolerance specifies a zone located and oriented with respect to the datum features:
   a. bounded by two planes separated by the specified tolerance
   b. bounded by cylinder with a diameter equal to the specified tolerance
   c. bounded by a sphere with a diameter equal to the specified tolerance

Datums: Optional.
Material Condition Modifiers: Optional on both the datum and considered features.
Conformance: A feature conforms to its position tolerance if all elements of the center plane, axis, or point lie within the tolerance zone.

![Figure A-9 Position Applied to a Hole Pattern](image)

Figure A-9 Position Applied to a Hole Pattern
Concentricity
Concentricity is the condition where the median points of all diametrically opposed
elements of a feature of revolution are congruent with a datum axis.

_Tolerance Zone:_ The concentricity tolerance specifies a zone bounded by a cylinder with
a diameter equal to the specified tolerance.

_Datums:_ Required.

_Material Condition Modifiers:_ No.

_Conformance:_ A feature conforms to its concentricity tolerance if the median points of all
diametrically opposed elements of the feature lie within the tolerance zone.

---

**a)** Concentricity applied to a boss

**b)** Tolerance zone: a cylindrical boundary with a
diameter equal to the specified tolerance that is
concentric with the datum axis

**c)** Conformance: the smallest
concentricity zone to which all derived
median points of the feature conform

---

*Figure A-10 Concentricity Applied to a Boss*
Symmetry
Symmetry is the condition where the median points of all opposed elements of a feature of are congruent with the axis or center plane of the datum feature.

Tolerance Zone: The symmetry tolerance specifies a zone bounded by two parallel lines or planes separated by the specified tolerance.

Datums: Required.

Material Condition Modifiers: No.

Conformance: A feature conforms to its symmetry tolerance if the median points of all opposed elements of the feature lie within the tolerance zone.

![Figure A-11 Symmetry Applied to a Notch](image)

- a) Symmetry applied to a notch
- b) Tolerance zone: two parallel planes separated by the specified tolerance that is congruent with datum plane A
- c) Conformance: the smallest symmetry zone to which all derived median points of the feature conform

Figure A-11 Symmetry Applied to a Notch
Circular Runout
Circular runout is a composite control of a features roundness and coaxiality with respect to a datum axis. When applied to features constructed at 90 degrees to the datum axis circular runout controls the wobble of a plane surface.

Tolerance Zone: Applied to features revolved about the datum axis the circular runout tolerance can be simulated with a zone bounded by two circles concentric with the datum axis whose radial separation is the specified tolerance.

Datums: Required.
Material Condition Modifiers: No.

Conformance: A feature conforms to its circular runout tolerance if, for each cross sectional element (checks are done independently), all points of the feature lie within the tolerance zone.

Figure A-12 Circular Runout Applied to a Boss

36
Total Runout
Total runout is a composite control of a features circularity, straightness, coaxiality, angularity, taper, and profile with respect to a datum axis. When applied to features constructed at 90 degrees to the datum axis total runout controls perpendicularity and flatness.

Tolerance Zone: Applied to features revolved about the datum axis the circular runout tolerance can be simulated with a zone bounded by two cylinders concentric with the datum axis whose radial separation is the specified tolerance.

Datums: Required.

Material Condition Modifiers: No.

Conformance: A feature conforms to its total runout tolerance if all surface elements of the feature lie within the tolerance zone.

Figure A-13 Total Runout Applied to a Boss
**Line Profile**

Line profile is a 2 dimensional control that controls the line elements of a feature along a specified direction.

*Tolerance Zone:* Line profile specifies a 2 dimensional zone composed of two profiles equally disposed (bi-lateral case) about the nominal geometry separated by the specified tolerance.

*Datums:* Optional.

*Material Condition Modifiers:* Optional on the datum features.

*Conformance:* A feature conforms to its line profile tolerance if all 2D elements the feature lie within the tolerance zone.

---

**Figure A-14 Line Profile**
**Surface Profile**
Surface profile is a 3 dimensional control that controls the length, width and/or circumference of the considered feature or features.

*Tolerance Zone*: Surface profile specifies a 3 dimensional zone composed of two uniform boundaries equally disposed (bi-lateral case) about the nominal geometry separated by the specified tolerance.

*Datums*: Optional.

*Material Condition Modifiers*: Optional on the datum features.

*Conformance*: A feature conforms to its surface profile tolerance if all elements of the feature lie within the tolerance zone.

![Figure A-15 Surface Profile](image)

- **a)** Surface profile applied all around
- **b)** Tolerance zone: two uniform boundaries equally disposed about the nominal surface separated by the specified tolerance
- **c)** Conformance: the smallest surface profile zone to which the feature conforms