Chapter 3 Geometric Tolerancing

Objective:

Chapter 3 presents an overview of Geometric Tolerancing and inspection using gages. The fundamentals of design tolerances and their interpretation are presented in detail. The basics of ASME Y14.5 (Form Geometric Tolerance Specification) are presented. Single and multiple data are introduced.

Outline:

3.1 Background

In Chapter 2, we learned about engineering design and traditional tolerance specification. As products have gotten increasingly sophisticated and geometrically more complex, the need to better specify regions of dimensional acceptability has become more apparent. Traditional tolerances schemes are limited to translation (linear) accuracies that get appended to a variety of features. In many cases bilaterial, unilaterial and limiting tolerance specifications are adequate for both location as well as feature size specification. Unfortunately, adding a translation zone or region to all features is not always the best alternative. In many cases in order to reduce the acceptable region, the tolerance specification must be reduced to insure that mating components fit properly. The result can be a high than necessary manufacturing cost. In this chapter, we will introduce geometric tolerances; how they are used; and how they are interpreted.

In 1973, the American National Standards Institute, Inc. (ANSI) introduced a system for specifying dimensional control called Geometric Dimensioning and Tolerancing (GD&T). The system was intended to provide a standard for specifying and interpreting engineering drawings, and was referred to as ANSI Y14.5 – 1973. In 1982, the standard was further enhanced and a new standard ANSI Y14.5 – 1982 was born. In 1994, the standard was further evolved to include formal mathematical definitions of geometric dimensioning and tolerancing, and became ASME Y14.5 – 1994.

3.2 Geometric tolerances – ASME Y14.5

Geometric tolerancing specifies the tolerance of geometric characteristics. Basic geometric characteristics as defined by the ASME Y14.5M 1994 standard include

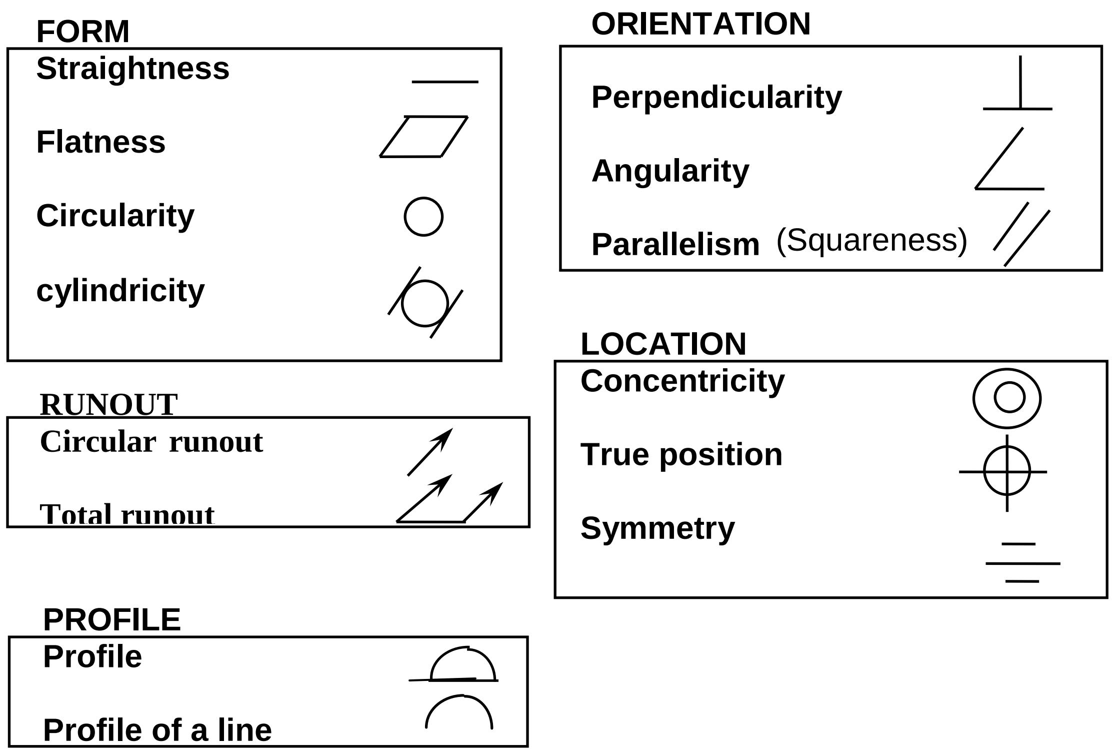
Straightness	Perpendicularity
Flatness	Angularity
Roundness	Concentricity
Cylindricity	Runout
Profile	True position
Parallelism	

Symbols that represent these features are shown in Table 3.1. To specify the geometric tolerances, reference features – planes, lines, or surfaces – can be established. Geometric tolerance of a feature (lines, surface, etc.) is specified in a feature control frame (Figure 3.1). Each frame consists of: 1) a tolerance symbol, 2) the tolerance value, 3) a modifier to the tolerance value, and 4) pertinent datum information. The diameter symbol (Table

3.1) may be placed in front of the tolerance value to denote the tolerance is applied to the hole diameter. The meaning of the modifier will be discussed later and omitted here. Following the modifier could be from zero to several datums (Figure 3-1 and Figure 3.2). For tolerances such as straightness, flatness, roundness, and cylindricity, the tolerance is internal; no external reference feature is needed. In this case, no datum is used in the feature control frame.

A datum is a plane, surface, point(s), line, axis, or other information source on an object. Datums are assumed to be exact, and from them, dimensions similar to the reference-location dimensions in the conventional drawing system can be established. Datums are used for geometric dimensioning and frequently imply fixturing location information. The correct use of datums can significantly affect the manufacturing cost of a part. Figure 3-3 illustrates the use of datums and the corresponding fixturing surfaces. The 3-2-1 principle is a way to guarantee the workpiece is perfectly located in the three-dimensional space. Normally three locators (points) are used to locate the primary datum surface, two for secondary surface, and one for tertiary surface. After the workpiece is located, clamps are used on the opposing side of the locators to immobilize the workpiece.

Table 3-1 Geometric tolerancing symbols (ASME Y14.5M-1994 GD&T (ISO 1101, geometric tolerancing; ISO 5458 positional tolerancing; ISO 5459 datums; and others))



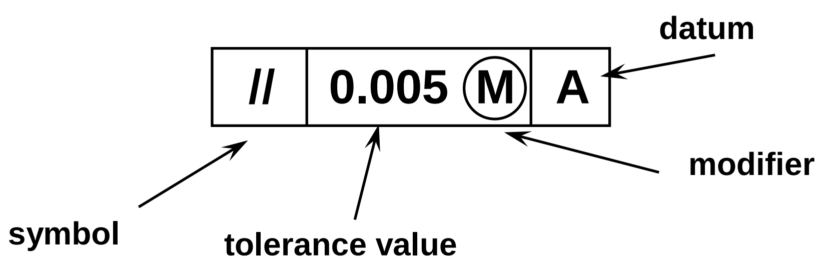


Figure 3-1 Feature control frame

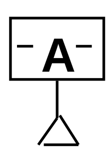


Figure 3-2 Datum symbol

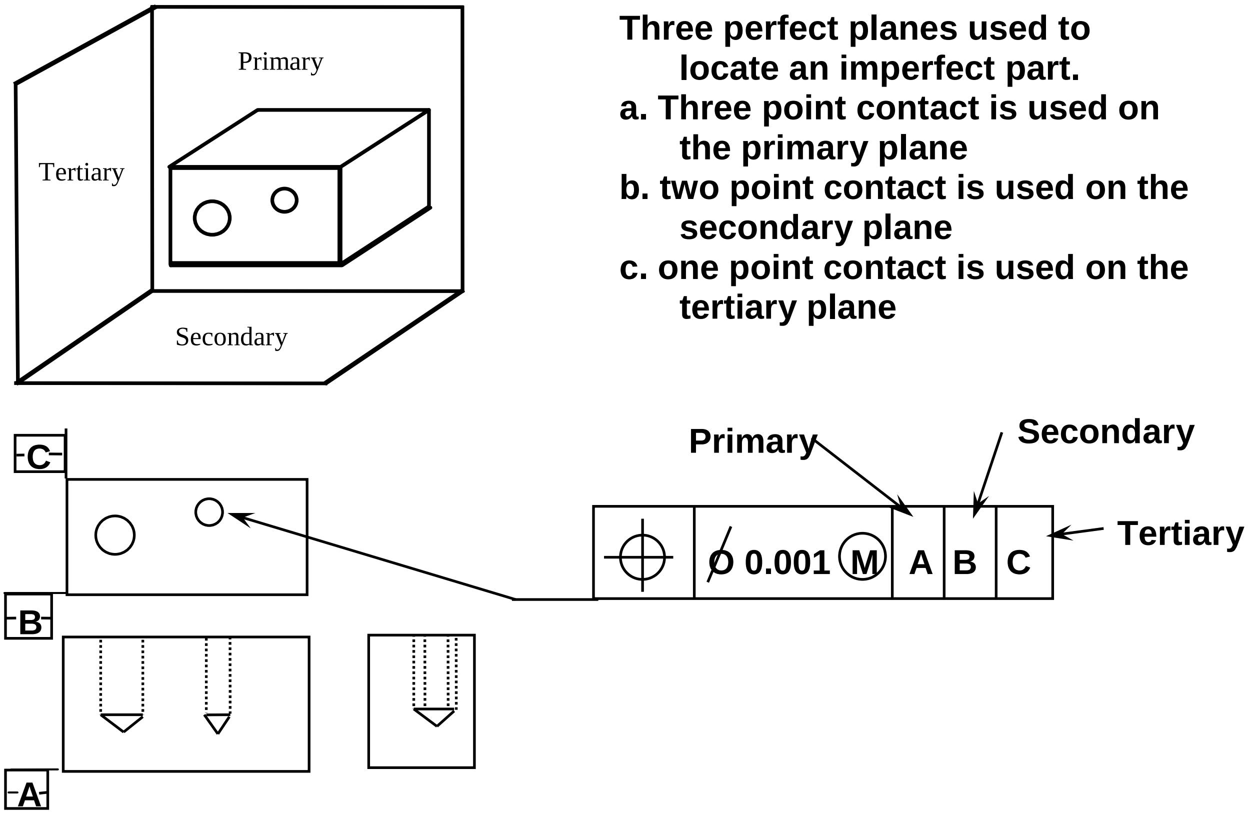


Figure 3-3 3-2-1 principle in datum specification

3.2.1 Symbol modifiers

Symbolic modifiers are used to clarify implied tolerances (Figure 3-1). There are three modifiers that are applied directly to the tolerance value: Maximum Material Condition (MMC), Regardless of Feature Size (RFS), and Least Material Condition (LMC). RFS is the default, thus if there is no modifier symbol, RFS is the default callout. MMC can be used to constrain the tolerance of the produced dimension and the maximum designed dimension. It is used to maintain clearance and fit. It can be defined as the condition of a part feature where the maximum amount of material is contained. For example, maximum shaft size and minimum hole size are illustrated with MMC as shown in Figure 3-5. LMC specifies the opposite of the maximum material condition. It is used for maintaining interference fit and in special cases to restrict the minimum material to eliminate vibration in rotating components. MMC and LMC can be applied only when both of the following conditions hold:

1. Two or more features are interrelated with respect to the location or form (e.g., two holes). At least one of the features must refer to size.
2. MMC or LMC must directly reference a size feature.

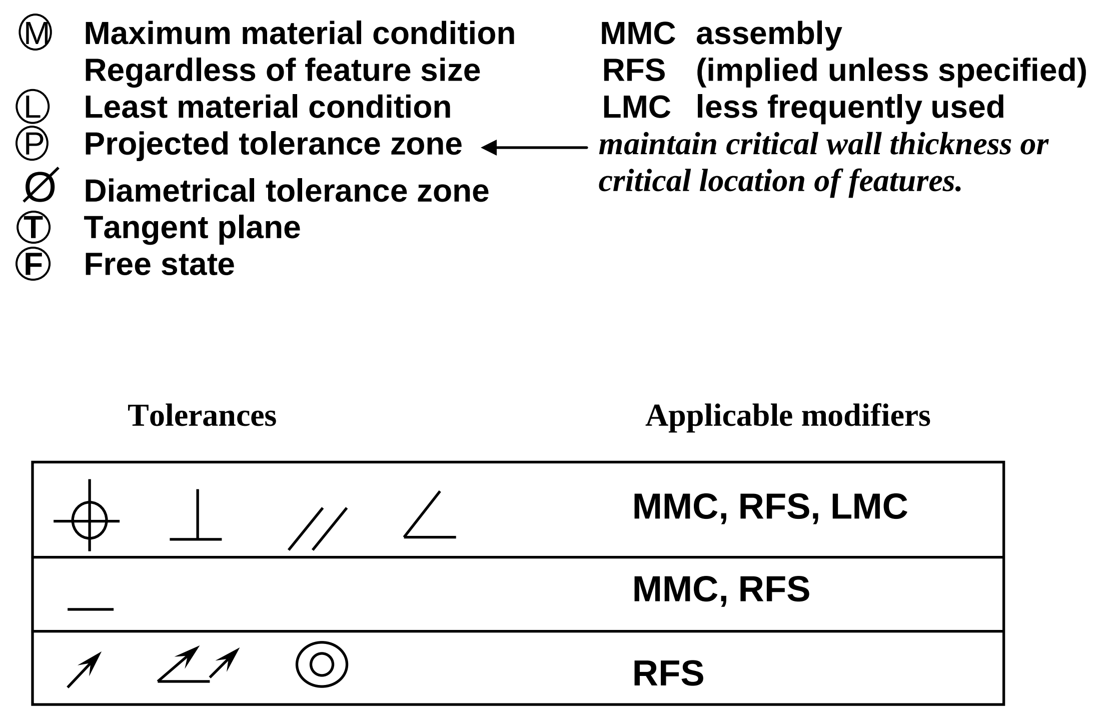


Figure 3-4 Modifiers and applicability

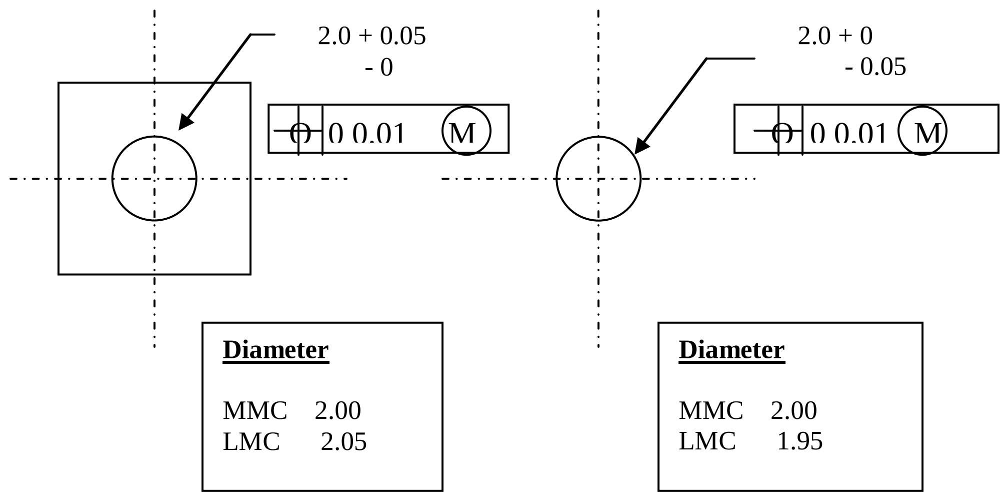


Figure 3-5 Maximum material diameter and least material diameter

When MMC or LMC is used to specify the tolerance of a hole or shaft, it implies that the tolerance specified is constrained by the maximum or least material condition as well as some other dimensional feature(s). For MMC, the tolerance may increase when the actual produced feature size is larger (for a hole) or smaller (for a shaft). Because the increase in the tolerance is compensated by the deviance of size in production, the final combined hole-size error and geometric tolerance error will still be larger than the anticipated smallest hole. Figure 3-6 illustrates the allowed tolerance under the produced hole size. The allowed tolerance is the actual acceptable tolerance limit; it varies as the size of the produced hole changes. The specified tolerance is the value.

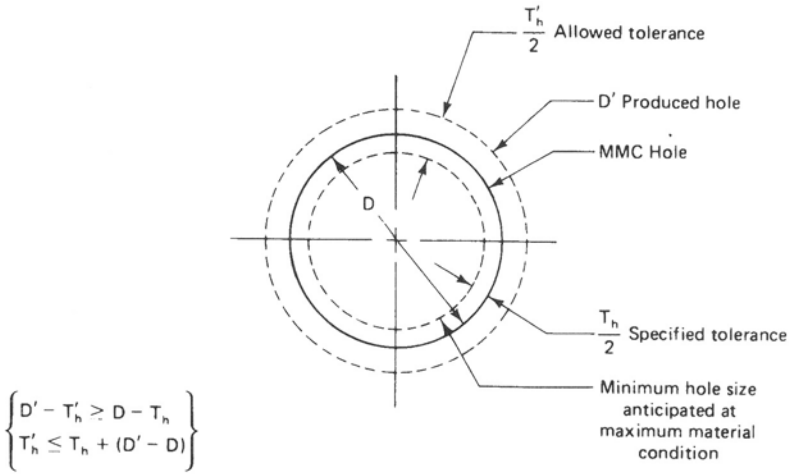


Figure 3-6 Allowed tolerance under the produced hole size

Projected tolerance zone is used in assembly of two parts with a gap in between them. The assembly of engine block with cylinder head is a good example. A gasket is inserted in between the engine block and the cylinder head. When specifying the position of bolt holes on the cylinder head, the positional error is measured at a thickness of the gasket to ensure the mating with the bolt on the engine block. Figure 3-7 illustrates the projected tolerance zone is 0.25" over the top of the hole. The dimension above the feature control frame specifies 0.375" diameter holes with UNC (Unified Coarse) 16 threads per inch screw threads. 2B stands for number 2 fit and internal thread.

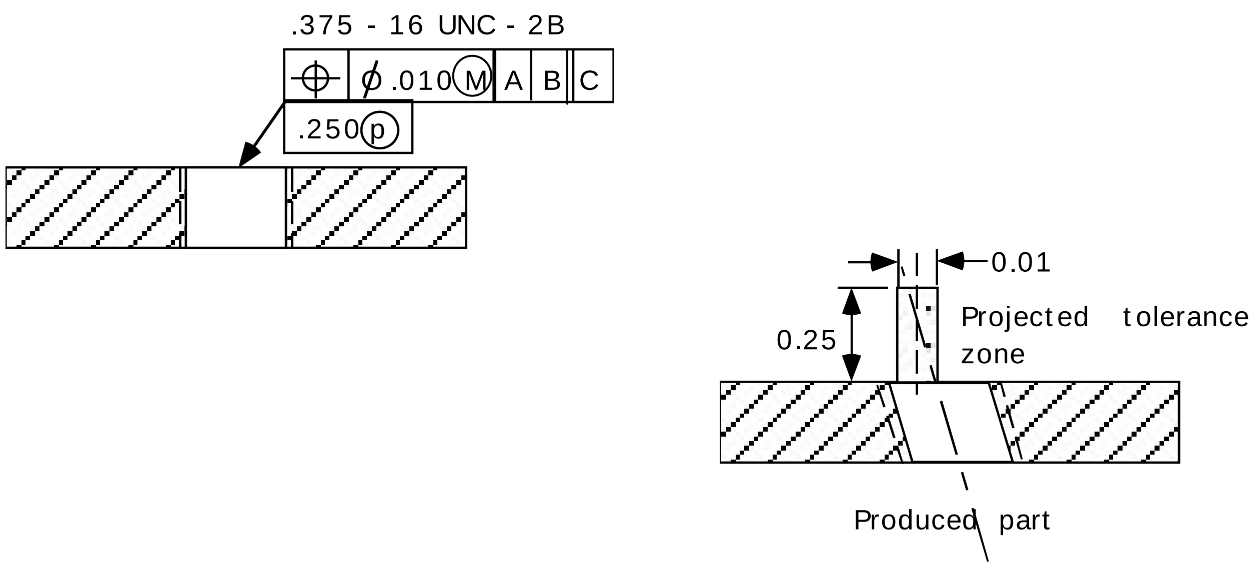


Figure 3-7 Projected tolerance zone

In the following sections, the Mathematics based definitions of geometric tolerances are introduced. The gage based definitions are presented in the Appendix. The math based definition is more precise; however, for technologists with less math education, the gage based definition is preferred.

3.2.2 Straightness (ASME Y14.5-1994)

Straightness is a condition where an element of a surface, or an axis, is a straight line. A straightness tolerance specifies a tolerance zone within which the considered element or derived median line must lie. A straightness tolerance is applied in the view where the elements to be controlled are represented by a straight line.

Straightness of a Derived Median Line

Definition: Straightness tolerance for the derived median line of a feature specifies that the derived median line must lie within some cylindrical zone whose diameter is the specified tolerance.

A straightness zone for a derived median line is a cylindrical volume consisting of all points $\vec{P}$ satisfying the condition:

$$\left|\hat{T} \times (\vec{P} - \vec{A})\right| \leq \frac{t}{2} \tag{3.1}$$

where

$\hat{T}$ = direction vector of the straightness axis

$\vec{A}$ = position vector locating the straightness axis

t = diameter of the straightness tolerance zone (tolerance value)

This feature is illustrated in Figure 3-8 for a cylindrical surface and Figure 3-9 for a planar surface.

Conformance. A feature conforms to a straightness tolerance t_0 if all points of the derived median line lie within some straightness zone as defined above with $t = t_0$. That is, there exist $\hat{T}$ and $\vec{A}$ such that with $t = t_0$, all points of the derived median line are within the straightness zone.

Actual value. The actual value of straightness for the derived median line of a feature is the smallest straightness tolerance to which the derived median line will conform.

3.2.3 Straightness of Surface Line Elements

Definition. A straightness tolerance for the line elements of a feature specifies that each line element must lie in a zone bounded by two parallel lines that are separated by the specified tolerance and that are in the cutting plane defining the line element.
A straightness zone for a surface line element is an area between parallel lines consisting of all points $\vec{P}$ satisfying the condition:

$$\left|\hat{T} \times (\vec{P} - \vec{A})\right| \le \frac{t}{2} \qquad (3.2)$$

and

$$\hat{C}_p \cdot (\vec{P} - \vec{P}_s) = 0$$
$$\hat{C}_p \cdot (\vec{A} - \vec{P}_s) = 0$$
$$\hat{C}_p \cdot \hat{T} = 0$$

where

$\hat{T}$ = direction vector of the center line of the straightness zone
$\vec{A}$ = position vector locating the center line of the straightness zone
t = size of the straightness zone (the separation between the parallel lines)
$\hat{C}_p$ = normal to the cutting plane defined as being parallel to the cross product of the desired cutting vector and the mating surface normal at $\vec{P}_s$
$\vec{P}_s$ = point on the surface, contained by the cutting plane

Figure 3-8 illustrates a straightness tolerance zone for surface line elements of a cylindrical feature. Figure 3-9 illustrates a straightness tolerance zone for surface line elements of a planar feature.

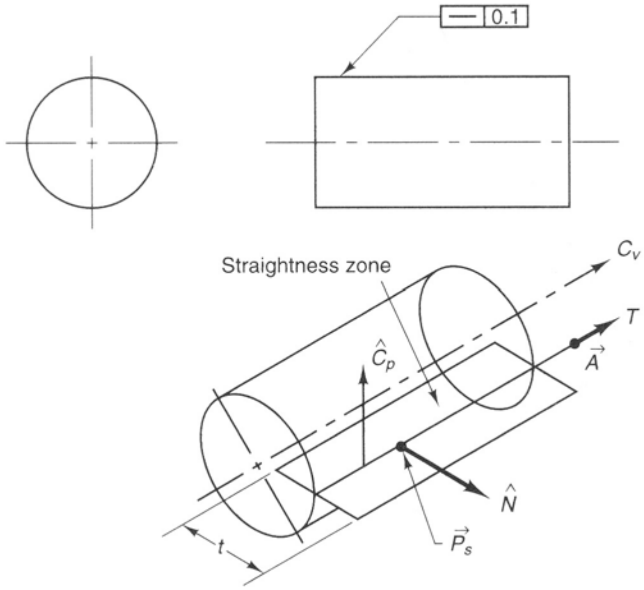


Figure 3-8 Evaluation of straightness of a cylindrical surface.

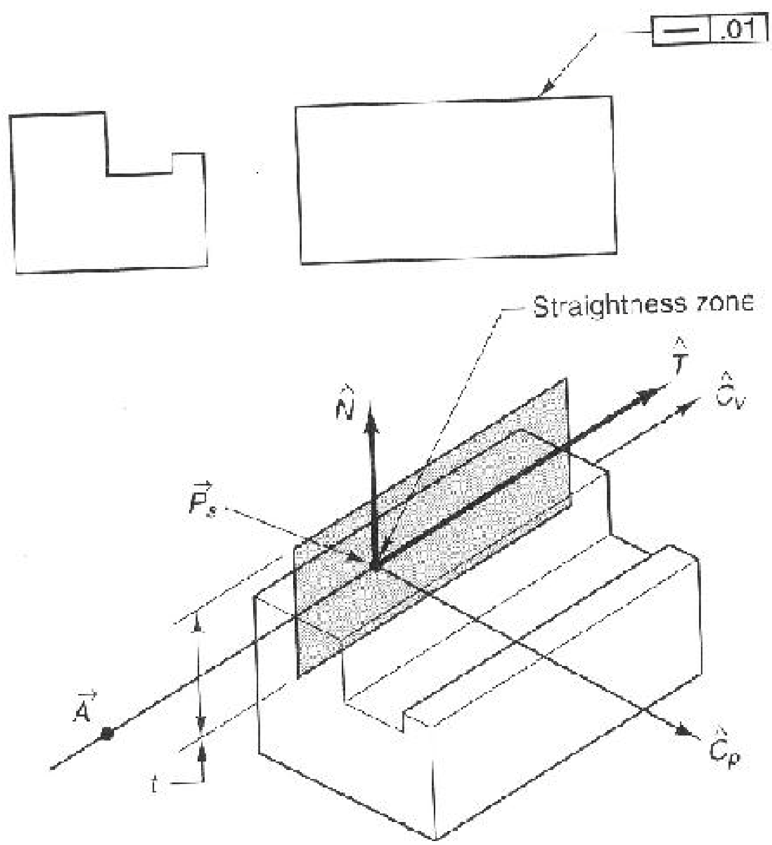


Figure 3-9 Evaluation of straightness of a planar surface

Conformance. A surface line element conforms to the straightness tolerance t_0 for a cutting plane if all points of the surface line element lie within some straightness zone as defined before with $t = t_0$. That is, there exist $\hat{T}$ and $\vec{A}$ such that with $t = t_0$, all points of the surface line element are within the straightness zone.

A surface conforms to the straightness tolerance t_0 if it conforms simultaneously for all toleranced surface line elements corresponding to some actual mating surface.

Actual value. The actual value of straightness for a surface is the smallest straightness tolerance to which the surface will conform.

3.2.4 Flatness

Flatness is the condition of a surface having all elements in one plane. A flatness tolerance specifies a tolerance zone defined by two parallel planes within which the surface must lie.

(a) *Definition*. A flatness tolerance specifies that all points of the surface must lie in some zone bounded by two parallel planes which are separated by the specified tolerance.

A flatness zone is a volume consisting of all points $\vec{P}$ satisfying the condition:

$$\left|\hat{T} \times (\vec{P} - \vec{A})\right| \leq \frac{t}{2} \tag{3.3}$$

where

$\hat{T}$ = direction vector of the parallel planes defining the flatness zone
$\vec{A}$ = position vector locating the midplane of the flatness zone
t = size of the straightness zone (the separation between the parallel lines)

(b) *Conformance.* A feature conforms to a flatness tolerance t_0 if all points of the feature lie within some flatness zone as defined before, with $t = t_0$. That is, there exist $\hat{T}$ and $\vec{A}$ such that with $t = t_0$, all points of the feature are within the flatness zone.

(c) *Actual value.* The actual value of flatness for a surface is the smallest flatness tolerance to which the surface will conform.

32.5 Circularity (roundness). 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.

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 described in (a) and (b) above.

(a) *Definition.* A circularity tolerance specifies that all points of each circular element of the surface must lie in some zone bounded by two concentric circles whose radii differ by the specified tolerance. Circular elements are obtained by taking cross-sections perpendicular to some spine. For a sphere, the spine is 0-dimensional (a point), and for a cylinder or cone, the spine is 1-dimensional (a simple, nonself-intersecting, tangent-continuous curve). The concentric circles defining the circularity zone are centered on, and in a plane perpendicular to, the spine.

A circularity zone at a given cross-section is an annular area consisting of all points $\vec{P}$ satisfying the conditions:

$$\hat{T} \cdot (\vec{P} - \vec{A}) = 0 \qquad (3.3)$$

and

$$\left| |\vec{P} - \vec{A}| - r \right| \le \frac{t}{2} \qquad (3.4)$$

where

$\hat{T}$ = for a cylinder or cone, a unit vector that is tangent to the spine at $\vec{A}$. For a sphere, $\hat{T}$ is a unit vector that points radially in all directions from $\vec{A}$

$\vec{A}$ = position vector locating a point on the spine

r = radial distance (which may vary between circular elements) from the spine to the center of the circularity zone ($r > 0$ for all circular elements)

t = the size of the circularity zone

Figure illustrates a circularity zone for a circular element of a cylindrical or conical feature.

(b) *Conformance.* A cylindrical or conical feature conforms to a circularity tolerance t_0 if there exists a 1-dimensional spine such that at each point $\vec{A}$ of the spine, the circular element perpendicular to the tangent vector $\hat{T}$ at $\vec{A}$ conforms to the circularity tolerance t_0. That is, for each circular element, there exist $\vec{A}$ and r such that with $t = t_0$, all points of the circular element are within the circularity zone.

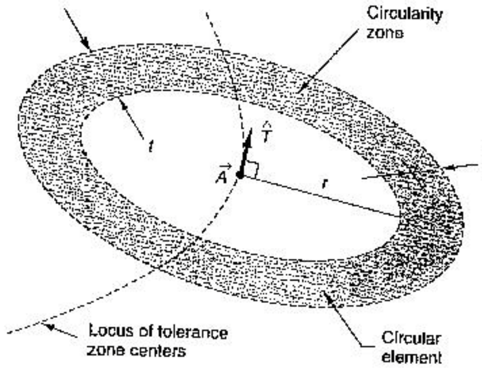


Figure 3.30 Illustration of a circularity tolerance zone for a cylindrical or conical feature.

A spherical feature conforms to a circularity tolerance t_0 if there exists a point (a 0-dimensional spine) such that each circular element in each cutting plane containing the point conforms to the circularity tolerance t_0. That is, for each circular element, there exist $\hat{T}$, r, and a common $\vec{A}$ such that with $t = t_0$, all points of the circular element are within the circularity zone.

(c) *Actual value.* The actual value of circularity for a feature is the smallest circularity tolerance to which the feature will conform.

3.2.6 **Cylindricity.** Cylindricity is a condition of a surface of revolution in which all points of the surface are equidistant from a common axis. A cylindricity tolerance specifies a tolerance zone bounded by two concentric cylinders within which the surface must lie. In the case of cylindricity, unlike that of circularity, the tolerance applies simultaneously to both circular and longitudinal elements of the surface (the entire surface). Note: The cylindricity tolerance is a composite control of form that includes circularity, straightness, and taper of a cylindrical feature.

(a) *Definition.* A cylindricity tolerance specifies that all points of the surface must lie in some zone bounded by two coaxial cylinders whose radii differ by the specified tolerance.

A cylindricity zone is a volume between two coaxial cylinders consisting of all points $\vec{P}$ satisfying the condition:

$$\left| \|\hat{T} \times (\vec{P} - \vec{A})\| - r \right| \leq \frac{t}{2} \qquad (3.5)$$

where

$\hat{T}$ = direction vector of the cylindricity axis
$\vec{A}$ = position vector locating the cylindricity axis
r = radial distance from the cylindricity axis to the center of the tolerance zone
t = size of the cylindricity zone

(b) *Conformance.* A feature conforms to a cylindricity tolerance t_0 if all points of the feature lie within some cylindricity zone as defined before with $t = t_0$. That is, there exist $\hat{T}, \vec{A}$, and r such that with $t = t_0$, all points of the feature are within the cylindricity zone.

(c) *Actual value.* The actual value of cylindricity for a surface is the smallest cylindricity tolerance to which it will conform.

3.2.7 **Profile control.** A profile is the outline of an object in a given plane (two-dimensional figure). Profiles are formed by projecting a three-dimensional figure onto a plane or taking cross-sections through the figure. The elements of a profile are straight lines, arcs, and other curved lines. With profile tolerancing, the true profile may be defined by basic radii, basic angular dimensions, basic coordinate dimensions, basic size dimensions, undimensioned drawings, or formulas.

(a) *Definition.* A profile tolerance zone is an area (profile of a line) or a volume (profile of a surface) generated by offsetting each point on the nominal surface in a direction normal to the nominal surface at that point. For unilateral profile tolerances, the surface is offset totally in one direction or the other by an amount equal to the profile tolerance. For bilateral profile tolerances, the surface is offset in both directions by a combined amount equal to the profile tolerance. The offsets in each direction may, or may not, be disposed equally.

For a given point $\vec{P}_N$ on the nominal surface there is a unit vector $\hat{N}$ normal to the nominal surface whose positive direction is arbitrary; it may point either into or out of the material. A profile tolerance t consists of the sum of two intermediate tolerances t_+ and t_-. The intermediate tolerances t_+ and t_- represent the amount of tolerance to be disposed in the positive and negative directions of the surface normal $\hat{N}$, respectively, at $\vec{P}_N$. For unilateral profile tolerances, either t_+ or t_- equals zero; t_+ and t_- are always nonnegative numbers.

The contribution of the nominal surface point $\vec{P}_N$ toward the total tolerance zone is a line segment normal to the nominal surface and bounded by points at distances t_+ and t_- from $\vec{P}_N$. The profile tolerance zone is the union of line segments obtained from each of the points on the nominal surface.

(b) *Conformance.* A surface conforms to a profile tolerance t_0 if all points $\vec{P}_S$ of the surface conform to either of the intermediate tolerances t_+ or t_- disposed about some corresponding point $\vec{P}_N$ on the nominal surface. A point $\vec{P}_S$ conforms to the intermediate tolerance t_+ if it is between $\vec{P}_N$ and $\vec{P}_N + \hat{N}t_+$. A point $\vec{P}_S$ conforms to the intermediate tolerance t_- if it is between $\vec{P}_N$ and $\vec{P}_N - \hat{N}_{t_-}$. Mathematically, this is the condition that there exists some $\vec{P}_N$ on the nominal surface and some u, $-t_- \leq u \leq t_+$, for which $\vec{P}_S = \vec{P}_N + \hat{N}_u$.

(c) *Actual value.* For both unilateral and bilateral profile tolerances, two actual values are necessarily calculated: one for surface variations in the positive direction and one for the negative direction. For each direction, the actual value of profile is the smallest intermediate tolerance to which the surface conforms. Note that no single actual value may be calculated for comparison to the tolerance value in the feature control frame, except in the case of unilateral profile tolerances.

$$|\hat{T} \cdot (\vec{P} - \vec{A})| \le \frac{t}{2} \qquad (3.6)$$

where

$\hat{T}$ = direction vector of the planar orientation zone
$\vec{A}$ = position vector locating the midplane of the planar orientation zone
t = size of the planar orientation zone (the separation of the parallel planes)

The planar orientation zone is oriented such that, if $\hat{D}_1$ is the direction vector of the primary datum, then

$$|\hat{T} \cdot \hat{D}_1| = \begin{cases} |\cos \Theta| & \text{for a primary datum axis} \\ |\sin \Theta| & \text{for a primary datum plane} \end{cases} \qquad (3.7)$$

where Θ is the basic angle between the primary datum and the direction vector of the planar orientation zone.

If a secondary datum is specified, the orientation zone is further restricted to be oriented relative to the direction vector, $\hat{D}_2$, of the secondary datum by

$$|\hat{T}' \cdot \hat{D}_2| = \begin{cases} |\cos \alpha| & \text{for a secondary datum axis} \\ |\sin \alpha| & \text{for a secondary datum plane} \end{cases} \qquad (3.8)$$

where $\hat{T}'$ is the normalized projection of $\hat{T}$ onto a plane normal to $\hat{D}_1$, and α is the basic angle between the secondary datum and $\hat{T}'$. $\hat{T}'$ is given by

$$\hat{T}' = \frac{\hat{T} - (\hat{T} \cdot \hat{D}_1)\hat{D}_1}{|\hat{T} - (\hat{T} \cdot \hat{D}_1)\hat{D}_1|} \qquad (3.9)$$

Figure 3.11 shows the relationship of the tolerance zone direction vector to the primary and secondary datums. Figure 3.12 illustrates the projection of $\hat{T}$ onto the primary datum plane to form $\hat{T}'$.

(b) *Conformance.* A surface, center plane, tangent plane, or axis S conforms to an orientation tolerance t_0 if all points of S lie within some planar orientation zone as defined before with $t = t_0$. That is, there exist $\hat{T}$ and $\vec{A}$ such that with $t = t_0$, all points of S are within the planar orientation zone. Note that if the orientation tolerance refers to both a primary datum and a secondary datum, then $\hat{T}$ is fully determined.

(c) *Actual value.* The actual value of orientation for S is the smallest orientation tolerance to which S will conform.

3.2.9 Cylindrical orientation zone.

(a) *Definition.* An orientation tolerance that is preceded by the diameter symbol specifies that the toleranced axis must lie in a zone bounded by a cylinder with a diameter equal to the specified tolerance and whose axis is basically oriented to the primary datum and, if specified, to the secondary datum as well.

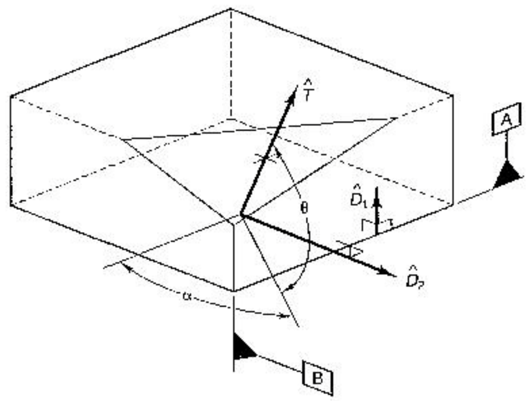
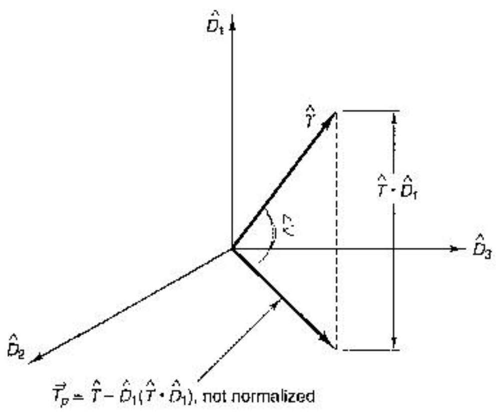


Figure 3.11 Planar orientation zone with primary and secondary datum planes specified.



$$\text{Normalized, } T' = \frac{\vec{T}_p}{|\vec{T}_p|} = \frac{\hat{T} - \hat{D}_1(\hat{T} \cdot \hat{D}_1)}{|\hat{T} - \hat{D}_1(\hat{T} \cdot \hat{D}_1)|}$$

Figure 3.12 Projection of tolerance vector onto primary datum plane.

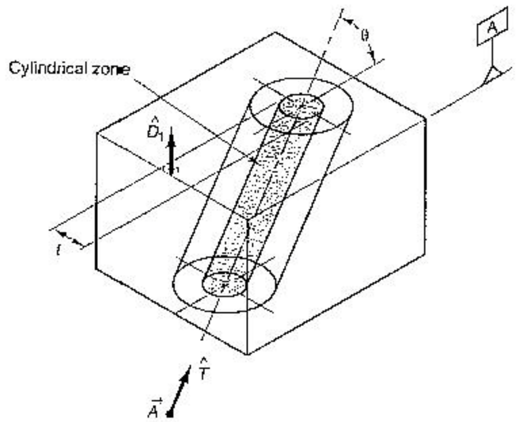


Figure 3.13 Orientation zone bounded by a cylinder with respect to a primary datum plane.

A cylindrical orientation zone is a volume consisting of all points $\vec{P}$ satisfying the condition

$$|\hat{T} \times (\vec{P} - \vec{A})| \le \frac{t}{2} \qquad (3.10)$$

where

$\hat{T}$ = direction vector of the axis of the cylindrical orientation zone
$\vec{A}$ = position vector locating the axis of the cylindrical orientation zone
t = diameter of the cylindrical orientation zone

The axis of the cylindrical orientation zone is oriented such that if $\hat{D}_1$ is the direction vector of the primary datum, then

$$|\hat{T} \cdot \hat{D}_1| = \begin{cases} |\cos \Theta| \text{ for a primary datum axis} \\ |\sin \Theta| \text{ for a primary datum plane} \end{cases} \qquad (3.11)$$

where Θ is the basic angle between the primary datum and the direction vector of the axis of the cylindrical orientation zone.

If a secondary datum is specified, the orientation zone is further restricted to be oriented relative to the direction vector, $\hat{D}_2$, of the secondary datum by

$$|\hat{T}' \cdot \hat{D}_2| = \begin{cases} |\cos \alpha| \text{ for a secondary datum axis} \\ |\sin \alpha| \text{ for a secondary datum plane} \end{cases} \qquad (3.12)$$

where $\hat{T}'$ is the normalized projection of $\hat{T}$ onto a plane normal to $\hat{D}_1$, and α is the basic angle between the secondary datum and $\hat{T}'$. $\hat{T}'$ is given by

P 7

$$\hat{T}' = \frac{\hat{T} - (\hat{T} \cdot \hat{D}_1)\hat{D}_1}{|\hat{T} - (\hat{T} \cdot \hat{D}_1)\hat{D}_1|} \tag{3.13}$$

Figure 3.13 illustrates a cylindrical orientation tolerance zone.

(b) *Conformance.* An axis S conforms to an orientation tolerance t_0 if all points of S lie within some cylindrical orientation zone as defined before with $t = t_0$. That is, there exists $\hat{T}$ and $\vec{A}$ such that with $t = t_0$, all points of S are within the orientation zone. Note that if the orientation tolerance refers to both a primary datum and a secondary datum, then $\hat{T}$ is fully determined.

(c) *Actual value.* The actual value of orientation for S is the smallest orientation tolerance to which S will conform.

3.2.10 Linear orientation zone

(a) *Definition.* An orientation tolerance that includes the notation EACH ELEMENT or EACH RADIAL ELEMENT specifies that each line element of the toleranced surface must lie in a zone bounded by two parallel lines that are (1) in the cutting plane defining the line element, (2) separated by the specified tolerance, and (3) are basically oriented to the primary datum and, if specified, to the secondary datum as well.

For a surface point $\vec{P}_S$, a linear orientation zone is an area consisting of all points $\vec{P}$ in a cutting plane of direction vector $\hat{C}_P$ that contains $\vec{P}_S$. The points $\vec{P}$ satisfy the conditions

$$\hat{C}_P \cdot (\vec{P} - \vec{P}_S) = 0 \tag{3.14}$$

and

$$|\hat{T} \times (\vec{P} - \vec{A})| \leq \frac{t}{2} \tag{3.15}$$

where

- $\hat{T}$ = direction vector of the center line of the linear orientation zone
- $\vec{A}$ = position vector locating the center line of the linear orientation zone
- $\vec{P}_S$ = point on S
- $\hat{C}_P$ = normal to the cutting plane and basically oriented to the datum reference frame
- t = size of the linear orientation zone (the separation between the parallel lines)

The cutting plane is oriented to the primary datum by the constraint

$$\hat{C}_P \cdot \hat{D}_1 = 0 \tag{3.16}$$

If a secondary datum is specified, the cutting plane is further restricted to be oriented to the direction vector of the secondary datum $\hat{D}_2$, by the constraint

$$\begin{aligned} |\hat{C}_P \cdot \hat{D}_2| &= |\cos \alpha| \text{ for a secondary datum axis} \\ |\hat{C}_P \cdot \hat{D}_2| &= |\sin \alpha| \text{ for a secondary datum plane} \end{aligned} \tag{3.17}$$

The position vector $\vec{A}$, which locates the center line of the linear orientation zone, also locates the cutting plane through the following constraint:

$$\hat{C}_P \cdot (\vec{P}_S - \vec{A}) = 0 \qquad (3.18)$$

If a primary or secondary datum axis is specified, and the toleranced feature in its nominal condition is rotationally symmetric about that datum axis, then the cutting planes are further restricted to contain the datum axis as follows:

$$\hat{C}_P \cdot (\vec{P}_S - \vec{B}) = 0 \qquad (3.19)$$

where $\vec{B}$ is a position vector that locates the datum axis. Otherwise, the cutting planes are required to be parallel to one another.

The direction vector of the center line of the linear orientation zone, $\hat{T}$, is constrained to lie in the cutting plane by

$$\hat{C}_P \cdot \hat{T} = 0 \qquad (3.20)$$

The center line of the linear orientation zone is oriented such that, if $\hat{D}_1$ is the direction vector of the primary datum, then

$$|\hat{T} \cdot \hat{D}_1| = \begin{cases} |\cos \Theta| & \text{for a primary datum axis} \\ |\sin \Theta| & \text{for a primary datum plane} \end{cases} \qquad (3.21)$$

where Θ is the basic angle between the primary datum and the direction vector of the linear orientation zone.

Figure 3.14 illustrates an orientation zone bounded by parallel lines on a cutting plane for a contoured surface.

(b) *Conformance.* A surface, center plane, or tangent plane S conforms to an orientation tolerance t_0 for a cutting plane $\hat{C}_P$ if all points of the intersection of S with $\hat{C}_P$ lie within some linear orientation zone as defined before with $t = t_0$. That is, there exist $\hat{T}$ and $\vec{A}$ such that with $t = t_0$, all points of S are within the linear orientation zone.

A surface S conforms to the orientation tolerance t_0 if it conforms simultaneously for all surface points and cutting planes $\hat{C}_P$.

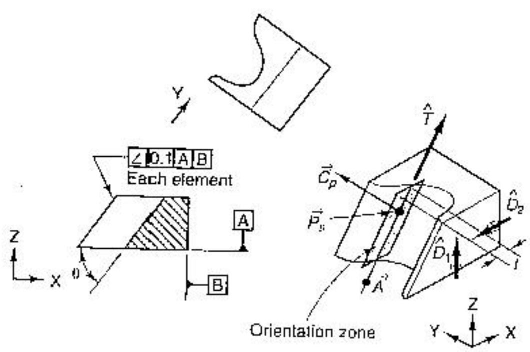


Figure 3.14 Orientation zone bounded by parallel lines.

36

Note that if the orientation tolerance refers to both a primary datum and a secondary datum, then $\boldsymbol{T}$ is fully determined.

(c) *Actual value.* The actual value of orientation for S is the smallest orientation tolerance to which S will conform.

3.2.11 **Runout tolerance.** Runout is a composite tolerance used to control the functional relationship of one or more features of a part to a datum axis. 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.

Surfaces constructed around a datum axis are those surfaces that are either parallel to the datum axis or are at some angle other than 90° to the datum axis. The mathematical definition of runout is necessarily separated into two definitions: one for surfaces constructed around the datum axis, and one for surfaces constructed at right angles to the datum axis. A feature may consist of surfaces constructed both around and at right angles to the datum axis. Separate mathematical definitions describe the controls imposed by a single runout tolerance on the distinct surfaces that comprise such a feature. Circular and total runout are handled later in this chapter.

Evaluation of runout (especially total runout) on tapered or contoured surfaces requires establishment of actual mating normals. Nominal diameters, and (as applicable) lengths, radii, and angles establish a cross-sectional *desired contour* having perfect form and orientation. The desired contour may be translated axially and/or radially, but may not be tilted or scaled with respect to the datum axis. When a tolerance band is equally disposed about this contour and then revolved around the datum axis, a volumetric tolerance zone is generated.

Circular Runout

Surfaces Constructed at Right Angles to a Datum Axis

(a) *Definition.* The tolerance zone for each circular element on a surface constructed at right angles to a datum axis is generated by revolving a line segment about the datum axis. The line segment is parallel to the datum axis and is of length t_0, where t_0 is the specified tolerance. The resulting tolerance zone is the surface of a cylinder of height t_0.

For a surface point $\vec{P}_S$, a circular runout tolerance zone is the surface of a cylinder consisting of the set of points $\vec{P}$ satisfying the conditions

$$|\hat{D}_1 \times (\vec{P} - \vec{A})| = r \qquad (3.22)$$

and

$$|\hat{D}_1 \cdot (\vec{P} - \vec{B})| \le \frac{t}{2} \qquad (3.23)$$

where

r = radial distance from $\vec{P}_S$ to the axis
$\hat{D}_1$ = direction vector of the datum axis
$\vec{A}$ = position vector locating the datum axis

$\vec{B}$ = position vector locating the center of the tolerance zone

t = size of the tolerance zone (height of the cylindrical surface)

(b) *Conformance.* The circular element through a surface point $\vec{P}_S$ conforms to the circular runout tolerance t_0 if all points of the element lie within some circular runout tolerance zone as defined before with $t = t_0$. That is, there exists $\vec{B}$ such that with $t = t_0$, all points of the surface element are within the circular runout zone.

A surface conforms to the circular runout tolerance if all circular surface elements conform.

(c) *Actual value.* The actual value of circular runout for a surface constructed at right angles to a datum axis is the smallest circular runout tolerance to which it will conform.

Surfaces Constructed Around a Datum Axis

(a) *Definition.* The tolerance zone for each circular element on a surface constructed around a datum axis is generated by revolving a line segment about the datum axis. The line segment is normal to the desired surface and is of length t_0, where t_0 is the specified tolerance. Depending on the orientation of the resulting tolerance zone will be either a flat annular area or the surface of a truncated cone.

For a surface point $\vec{P}_S$, a datum axis $[\vec{A}, \hat{D}_1]$, and a given mating surface, a circular runout tolerance zone for a surface constructed around a datum axis consists of the set of points $\vec{P}$ satisfying the conditions:

$$\frac{\hat{D}_1 \cdot (\vec{P} - \vec{B})}{|\vec{P} - \vec{B}|} = \hat{D}_1 \cdot \hat{N} \qquad (3.24)$$

and

$$\left\| \vec{P} - \vec{B} \right| - d \right| \le \frac{t}{2} \qquad (3.25)$$

$$\hat{N} \cdot (\vec{P}_S - \vec{B}) > 0 \qquad (3.26)$$

where

$\hat{D}_1$ = direction vector of the datum axis

$\vec{A}$ = position vector locating the datum axis

$\hat{N}$ = surface normal at $\vec{P}_S$ determined from the mating surface

$\vec{B}$ = point of intersection of the datum axis and the line through $\vec{P}_S$ parallel to the direction vector $\hat{N}$

d = distance from $\vec{B}$ to the center of the tolerance zone as measured parallel to $\hat{N}$ ($d \ge t/2$)

t = size of the tolerance zone as measured parallel to $\hat{N}$

Figure 3.15 illustrates a circular runout tolerance zone on a noncylindrical surface of revolution.

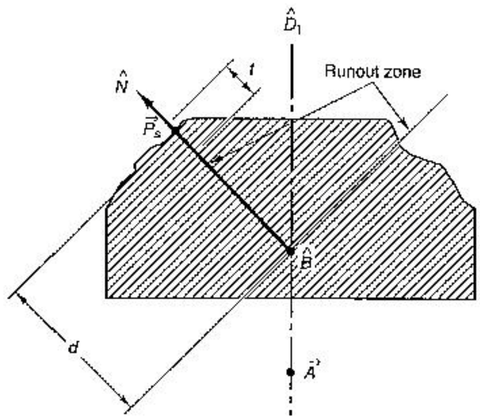


Figure 3.15 Circular runout tolerance zone.

(b) *Conformance.* The circular element through a surface point $\vec{P}_s$ conforms to the circular runout tolerance t_0 for a given mating surface if all points of the circular element lie within some circular runout tolerance zone as defined before with $t = t_0$. That is, there exists d such that with $t = t_0$, all points of the circular element are within the circular runout tolerance zone.

A surface conforms to a circular runout tolerance t_0 if all circular elements of the surface conform to the circular runout tolerance for the same mating surface.

(c) *Actual value.* The actual value of circular runout for a surface constructed around a datum axis is the smallest circular runout tolerance to which it will conform.

Total Runout

Surfaces Constructed at Right Angles to a Datum Axis

(a) *Definition.* A total runout tolerance for a surface constructed at right angles to a datum axis specifies that all points of the surface must lie in a zone bounded by two parallel planes perpendicular to the datum axis and separated by the specified tolerance.

For a surface constructed at right angles to a datum axis, a total runout tolerance zone is a volume consisting of the points $\vec{P}$ satisfying

$$|\hat{D}_1 \cdot (\vec{P} - \vec{B})| \leq \frac{t}{2} \qquad (3.27)$$

where

$\hat{D}_1$ = direction vector of the datum axis
$\vec{B}$ = position vector locating the midplane of the tolerance zone
t = size of the tolerance zone (the separation of the parallel planes)

(b) *Conformance.* A surface conforms to the total runout tolerance t_0 if all points of the surface lie within some total runout tolerance zone as defined before with $t = t_0$. That is, there exists $\vec{B}$ such that with $t = t_0$, all points of the surface are within the total runout zone.

(c) *Actual value.* The actual value of total runout for a surface constructed at right angles to a datum axis is the smallest total runout tolerance to which it will conform.

Surfaces Constructed Around a Datum Axis

(a) *Definition.* A total runout tolerance zone for a surface constructed around a datum axis is a volume of revolution generated by revolving an area about the datum axis. This area is generated by moving a line segment of length t_0, where t_0 is the specified tolerance, along the desired contour with the line segment kept normal to, and centered on, the desired contour at each point. The resulting tolerance zone is a volume between two surfaces of revolution separated by the specified tolerance.

Given a datum axis defined by the position vector $\vec{A}$ and the direction vector $\hat{D}_1$, let $\vec{B}$ be a point on the datum axis locating one end of the desired contour, and let r be the distance from the datum axis to the desired contour at point $\vec{B}$. Then, for a given $\vec{B}$ and r, let $C(\vec{B}, r)$ denote the desired contour. (Note: Points on this contour can be represented by $[d, r + f(d)]$, where d is the distance along the datum axis from $\vec{B}$.) For each possible $C(\vec{B}, r)$, a total runout tolerance zone is defined as the set of points $\vec{P}$ satisfying the condition

$$|\vec{P} - \vec{P}'| \le \frac{t}{2} \qquad (3.28)$$

where

$\vec{P}'$ = projection of $\vec{P}$ onto the surface generated by rotating $C(\vec{B}, r)$ about the datum axis

t = size of the tolerance zone, measured normal to the desired contour.

(b) *Conformance.* A surface conforms to a total runout tolerance t_0 if all points of the surface lie within some total runout tolerance zone as defined before with $t = t_0$. That is, there exist $\vec{B}$ and r such that with $t = t_0$, all points of the surface are within the total runout tolerance zone.

(c) *Actual value.* The actual value of total runout for a surface constructed around a datum axis is the smallest total runout tolerance to which it will conform.

3.2.12 **Free state variation.** Free-state variation is a term used to describe distortion of a part after removal of forces applied during manufacture. This distortion is principally due to weight and flexibility of the part and the release of internal stresses resulting from fabrication. A part of this kind, for example, a part with a very thin wall in proportion to its diameter, is referred to as a non-rigid part. In some cases, it may be required that the part meet its tolerance requirements while in the free state. In others, it may be necessary to simulate the mating part interface in order to verify individual or related feature tolerances. This is done by restraining the appropriate features. The restraining forces are those that would be exerted in the assembly or functioning of the part. However, if the dimensions and tolerances are met in the free state, it is usually not necessary to restrain the part unless the effect of subsequent restraining forces on the concerned features could cause other features of the part to exceed specified limits.

3.3 Interpreting Geometric Specifications

Hole size and position

We will use the simple part shown in Figure 3.16 to illustrate how position and feature size interact. The part is first specified using the modifier.

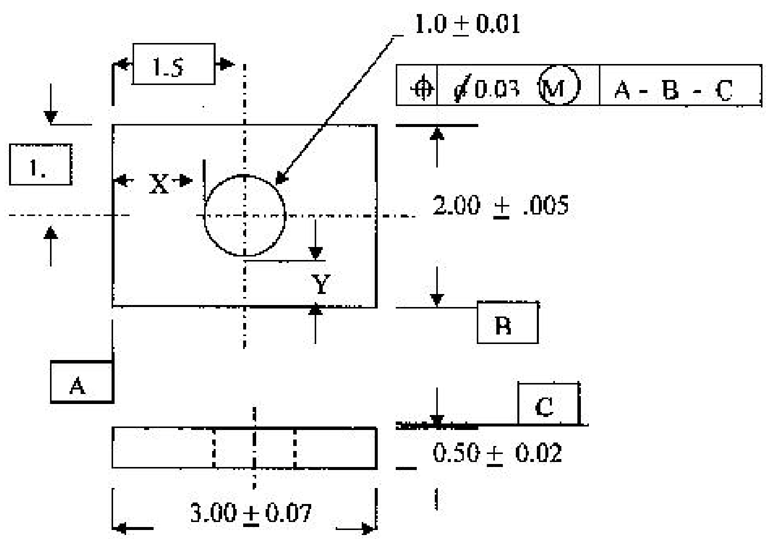


Figure 3-16 Part to illustrate true position

X_M – the largest value for x

Y_M – the largest

X'_M – the smallest value for x

Y'_M – the smallest value for y

INTERPRETATION

The size of the hole feature can be from 0.99 to 1.01. The MMC diameter for the hole is for a size .99. For the example:

Ǿ (actual)	⊕
.99	0.03
1.00	0.04
1.01	0.05

For D* (the actual measured value), the position tolerance will vary depending on the hold diameter. If D* = 0.99, then the allowable location error is contained in a circular region of diameter 0.03. If we seek to compute $X_{M,}$ then the maximum size will correspond to a location residing as far right of Datum-A- as allowable. In this case, 1.5+.03/2 =1.515. Subtracting D*/2 (0.99/2) from this yields X_M to be 1.020. Interestingly for any acceptable value for D and true position, X_M will remain the same. This is because of the way that MMCs are specified. X_M in this case will provide a maximum acceptable boundary for any acceptable diameter, D* and acceptable position tolerance zone. X'_M will provide the same acceptable boundary for the right side of the hole. $X'_M - X_M$ will yield the maximum diameter that will always assemble into the hole. The same calculations can be made for Y'_M and $Y_{M.}$ The calculations will produce the same diameter.

The tolerances for the symmetric features (holes) are now specified using Form Geometry symbols, e.g., ⊕, O, //. The hole features in the figure are specified as MMC entities. *This means that a **virtual size** for assembly is specified. This **virtual size** is specified as the MMC for all of the part features, and represents the minimum opening for all such labeled entities.* For female features such as holes, the virtual size is specified as:

$$\varepsilon_v = \varepsilon_s - \tau_s - \gamma_s \tag{3.29}$$

where: ε_v is the virtual size of the feature

ε_s is the nominal size specification of the feature
τ_s is the negative tolerance specification (for holes)
γ_s is the form geometric value (for holes)

For male features, such as a shaft, the virtual size is specified as:

$$\varepsilon_v = \varepsilon_s + \tau_s + \gamma_s \tag{3.30}$$

For the hole shown in Figure 3-10, the virtual size of the hole is:

$$\varepsilon_v = 1.000\text{-}0.01\text{-}0.03 = 0.96 \text{ inches}$$

Virtual size information is useful for a variety of reasons. It can be used to determine assembleability, interference, etc. The virtual size for the part in Figure 3-16 is shown in Figure 3-11. As can be seen in the figure, the virtual size is the maximum material condition for the entire part. It represents the silhouette for all possible combinations of acceptable parts stacked on top of each other. Therefore it provides the condition for which all mating components can be assembled. This is a key feature for design as well as for inspecting components. It also only relates to parts specified using the MMC modifier.

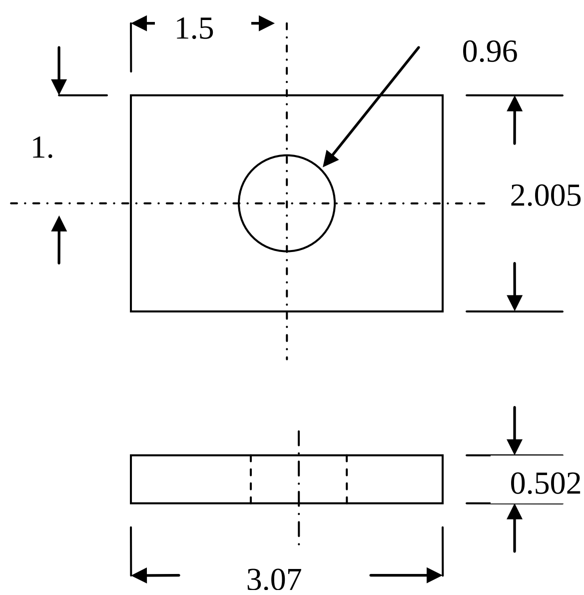


Figure 3-17. Virtual size for the part shown in Figure 3-16.

Inspection equipment can be broadly classified as *general-purpose* or *special-purpose equipment*. To measure reasonably simple parts or very low-volume items, general-purpose inspection equipment is normally used. To inspect very intricate and/or high-volume parts, special gages are normally designed in order to reduce the amount of time required for the (calibration and) inspection process.

For the part shown in Figure 3-16, a single datum set is used and the lone feature on the part is specified with a MMC modifier. Because of the interpretation above, this means that a virtual part exists and that special gages can be used to inspect the part, rather than having to use general purpose gages, such as micrometers and/or Vernier calipers. This can significantly reduce the inspection time and improve the quality of the inspection process.

In order to inspect the part, "snap gages" would be used to qualify the size requirements for the block portion of the bracket. The GO size is set to the largest acceptable size dimension and the NO GO size is set to the smallest acceptable size.

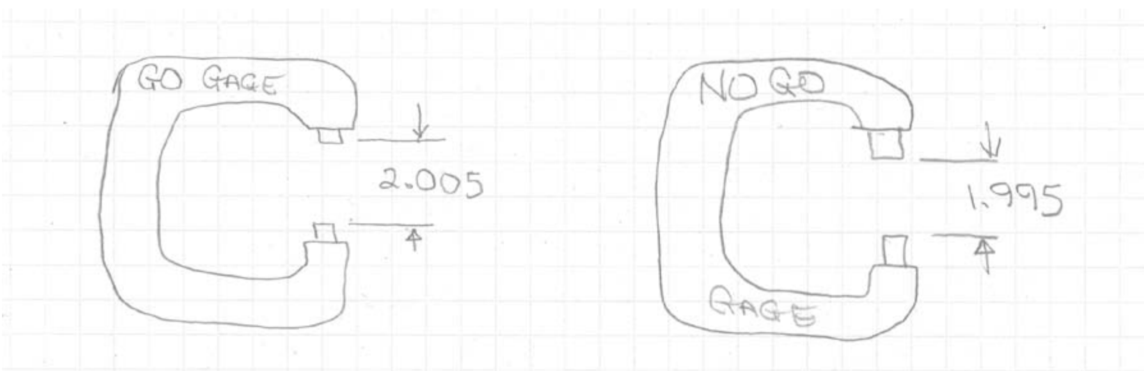


Figure 3-18 Snap gages for the part height

The snap gage to inspect the part height in Figure 3-16 is shown in Figure 3-18. Similar snap gage for the width and depth would be used.

For the hole feature, a typical GO and NO GO gage can be used. The Plug gage to inspect the hole in the sample part is shown in Figure 3-19

.

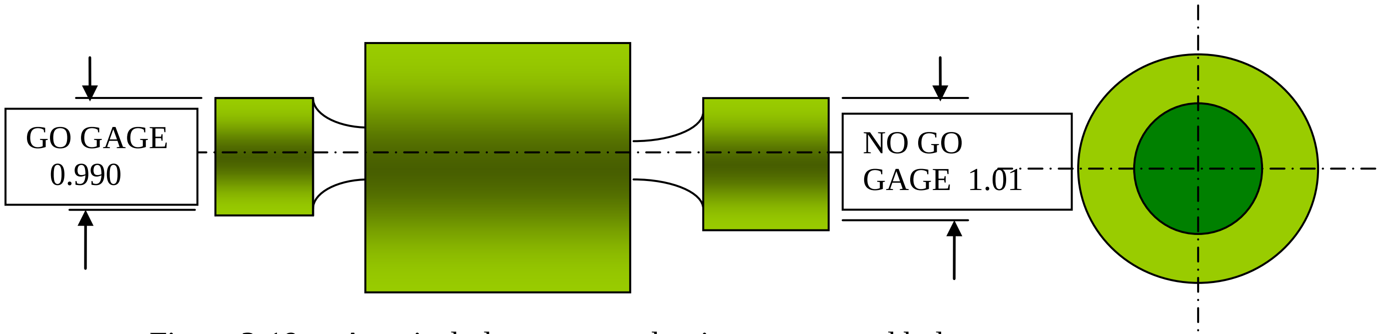


Figure 3-19 A typical plug gage used to inspect a round hole.

In order to inspect feature location for the hole, a location gage would be used. The size for the “pin” locator is determined using

$D_P = D_M - \oplus_M$ (for hole features)

$D_P = D_M + \oplus_M$ (for shaft features)

The location gage for the part would look like the gage shown in Figure 3-20. Note that the location planes correspond to datum surfaces –A- and –B-. The standard for gage accuracy is that it should be at least 10 times more precise than the part.

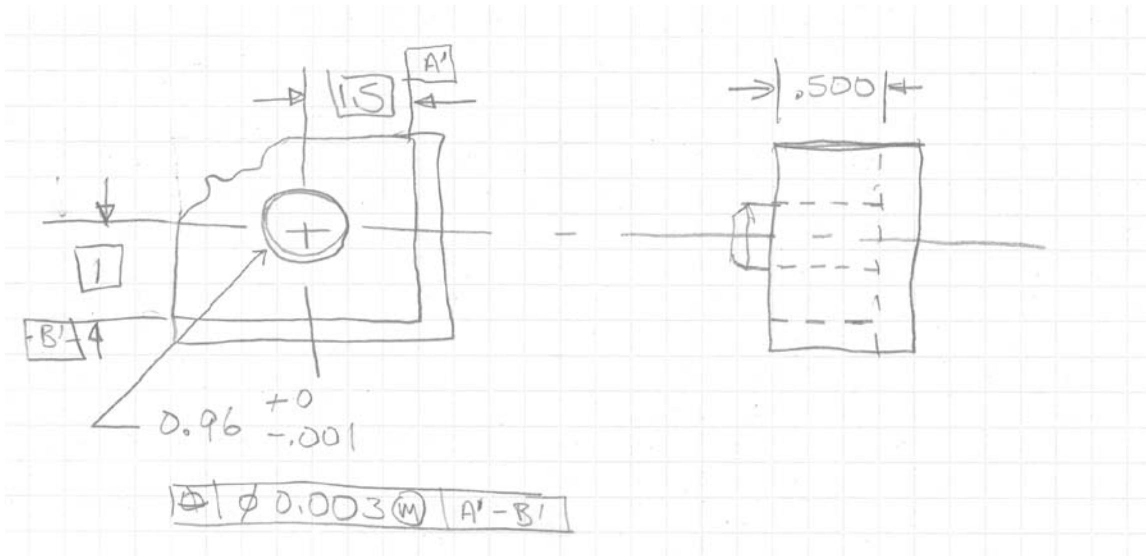


Figure 3-20 Location gage for illustrative parts

The inspection schema described will only work for parts qualified using MMC. If the hole were specified for RFS or LMC then conventional inspection equipment (micrometers and Vernier calipers) would have to be used.

3.4 Multiple Part features and datums

In the previous section, we looked at interpreting a part with a single hole feature. In this section, we will begin with a similar part with three holes and a single datum. We will then alter the specification of the part from one datum to two datums, and discuss the why and hows of interpreting the part. This has become more and more of an engineering requirement, because part datums are used to define critical relationships and dependencies that features have to other features.

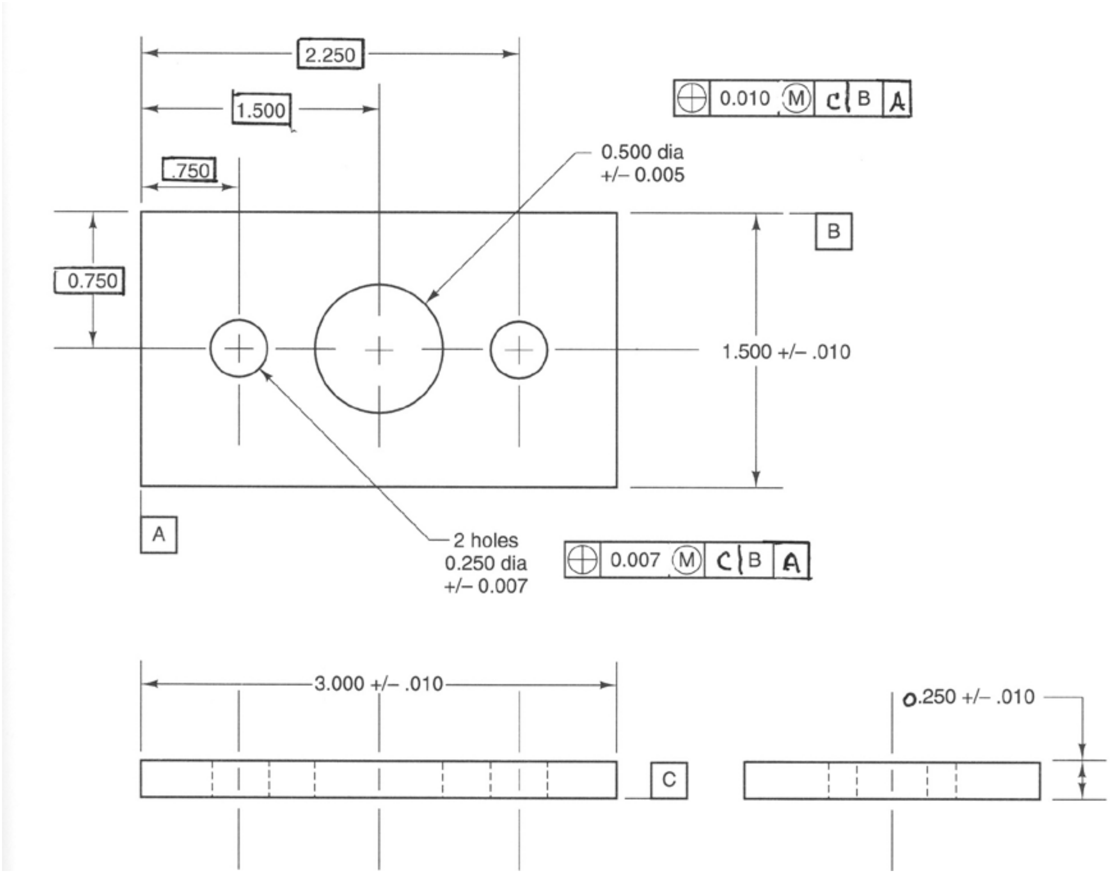


Figure 3-21. Sample part with three hole features.

We begin our discussion with the part shown in Figure 3-21. The part is a simple component with three hole features are all specified from a single part datum. The part is further specified using MMC modifiers. Each of the hole features on the part can again be treated as independent features with one part coordinate system. Each of the holes is interpreted with respect to the part coordinate system or datums A – B – C . The part height, width and depth would again be inspected with snap gages (given that a high enough volume was to be inspected). The holes would also be inspected with plug gages. This part of the inspection would insure that all of the part sizes were either within specification or that the part was defective. The last component to inspect is the location of each of the holes. Since there is only one datum and all of the features are called out with an MMC modifier, a virtual part exists and a location gage can be used to inspect the hole locations. The diameter for the pins would again be calculated by

$D_P = D_M - \oplus_M$

or D_p<large> = 0.495 – 0.010 = 0.485

and

D_p<small> = 0.247 – 0.007 = 0.240

Since all three holes are called out from the same datum set, all three hole locations can be assessed using a single location gage. The gage to inspect this part is shown in Figure 3-22.

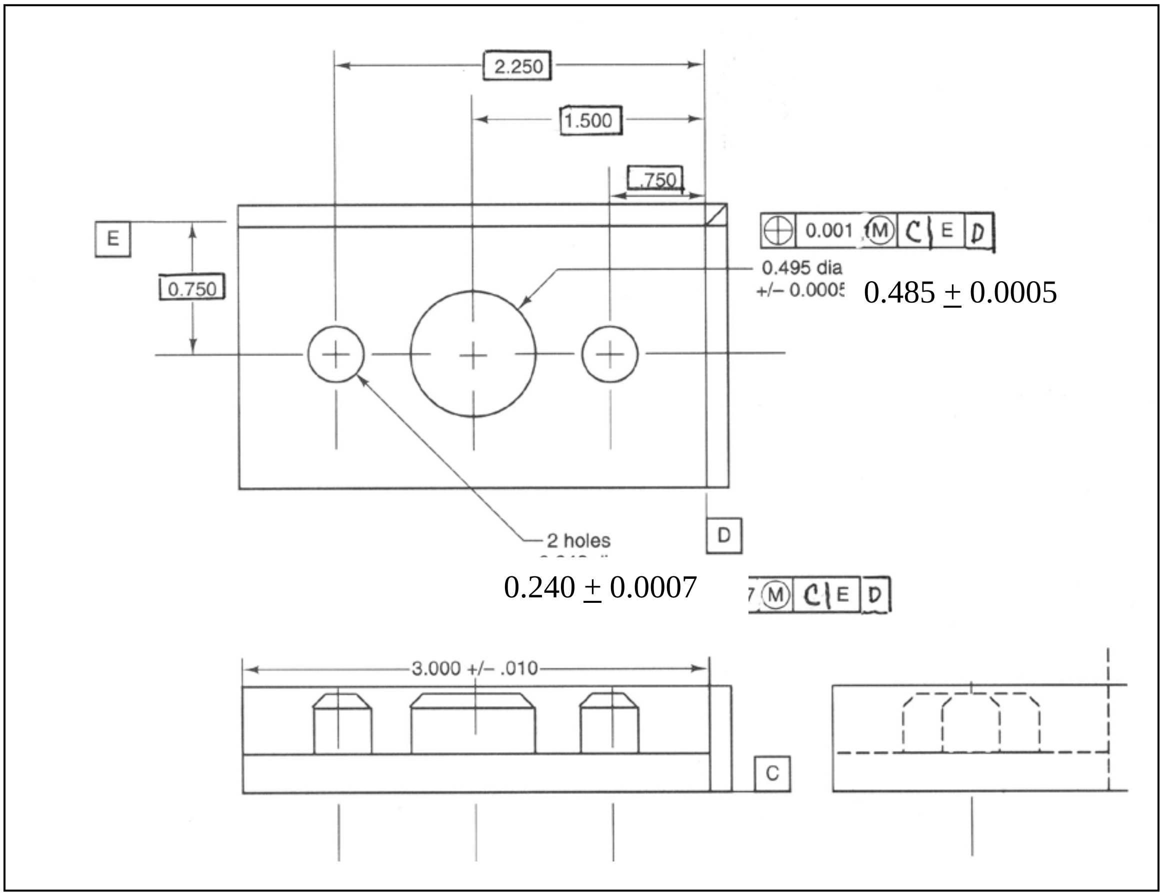


Figure 3-22 Location gage for part shown in Figure 3-21.

A small variant in the part is shown in Figure 3-23 . In the figure, the dimensions are all the same as previously. The only difference is that a second datum set is used to define the location of the two smaller holes (C – B – D), and datum D is qualified at MMC. The reason for qualifying a part like this might be that the larger hole might serve as an anchor for a part with two smaller pins fitting into the other holes.

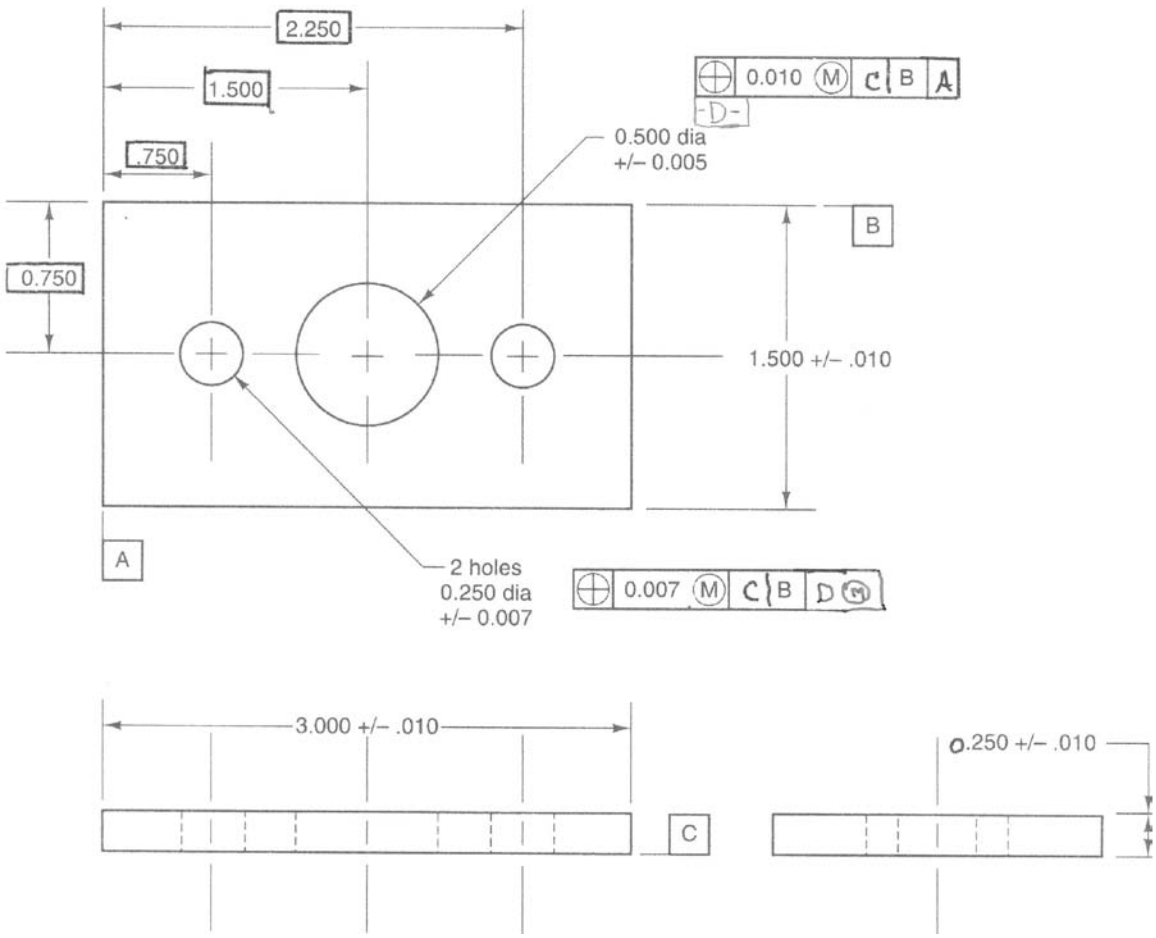


Figure 3-23 Part similar to the part shown in Figure 3-21 but with two datums.

The size features for this part would again be inspected as they were for the previous part. The location requirements however will be very different. The large hole is called out with respect to the major part coordinates (A- B- C), so it will be inspected in much the same manner as for the previous part. The location gage for the large hole is shown in Figure 3-24. Since there are two datum that are used, the location for all of the holes can not be qualified at the same time – a second location gage will be necessary. The location for the small holes is called out with respect to datum C – B – D. These datum will provide the locating surfaces for the smaller holes. Location for the smaller holes will now be qualified in the original part A axis using the large hole (datum feature D). The location gage for the small holes is shown in Figure 3-19.

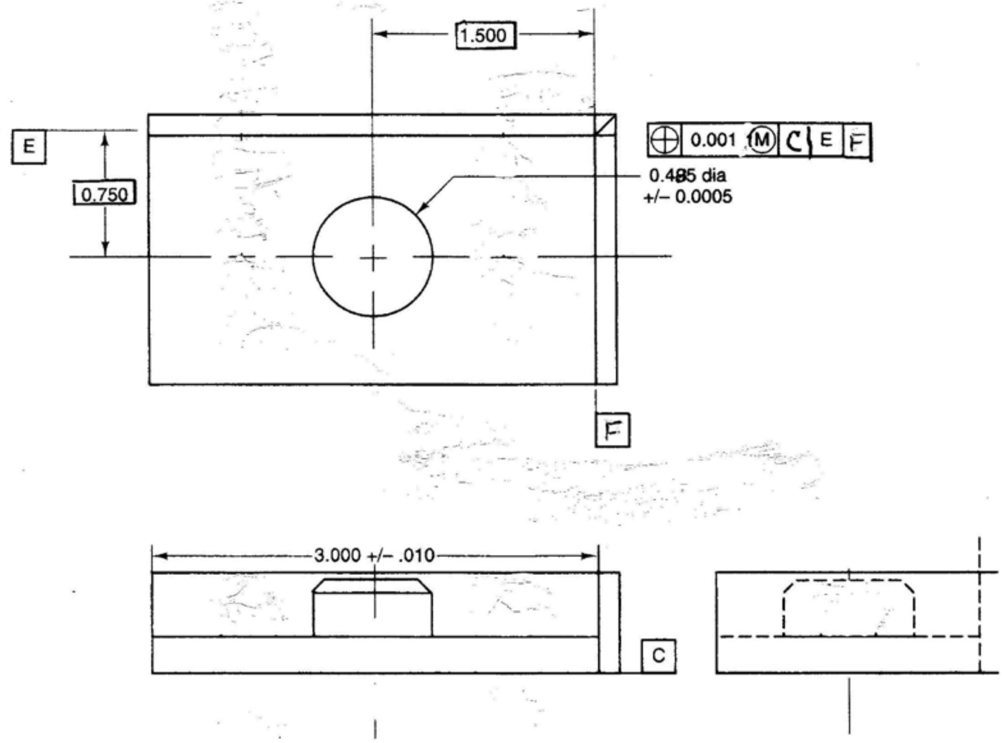


Figure 3-24 Inspection location gage for large hole.

The gage in Figure 3-24 is just like the previous gages that were developed. This is because the large hole is called out using an MMC modifier and the hole is referenced from the general part coordinate system (datum). The gage shown in Figure 3-25 is different in that rather than using a plane to locate from the part's –A- datum, the large hole is used (at MMC) to position the small holes in this direction.

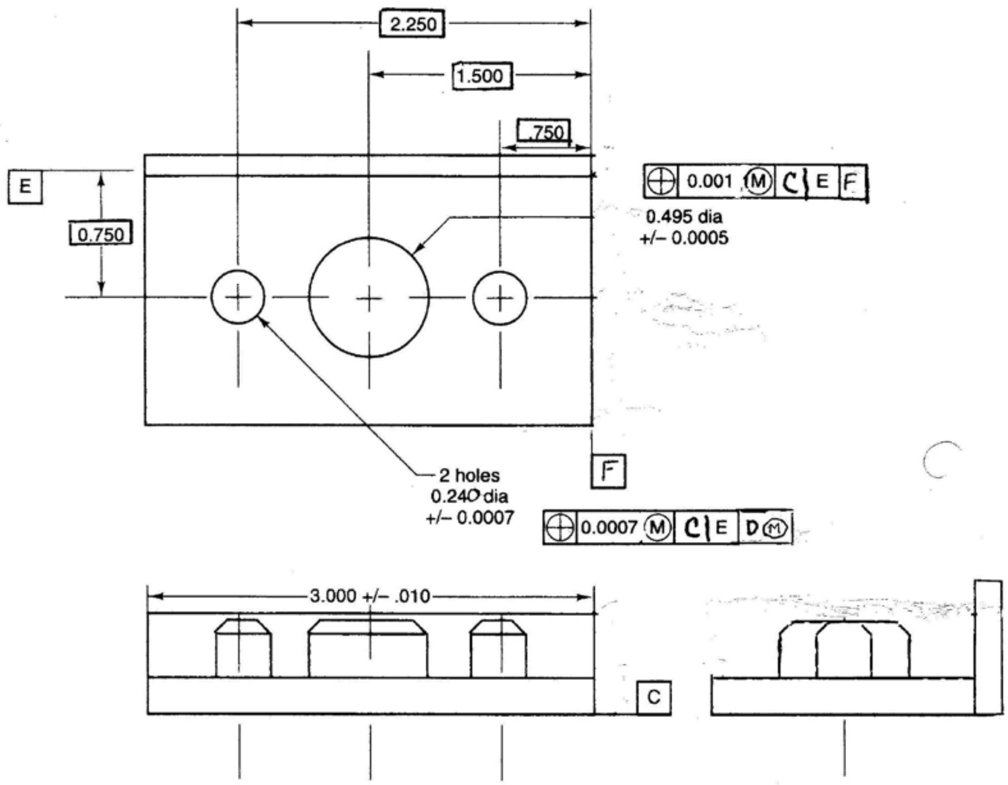


Figure 3-25. Inspection location gage for smaller holes.

3.4.1 Multiple datum parts

Parts often have inter-feature functionality, in that a component may assemble over 2 or 3 features where the performance is related to how a feature relates to another feature. In this case, multiple datum are used to characterize this functionality. The part shown in Figure 3-26 has two distinct datum, where the circular boss relates directly to the major part coordinate system. The hole on the boss requires a special location relationship with respect to the boss itself. In this case the boss becomes a datum for the part (-F), and the hole on the boss is called out with respect to the boss at MMC. In many cases this implies that the boss will first be machined and then the part will be refixtured with respect to the boss rather than the original part coordinate system. This part is another part that would require two different location gages to fully qualify: first, the gage to qualify the boss, and then a gage to qualify the hole on the boss.

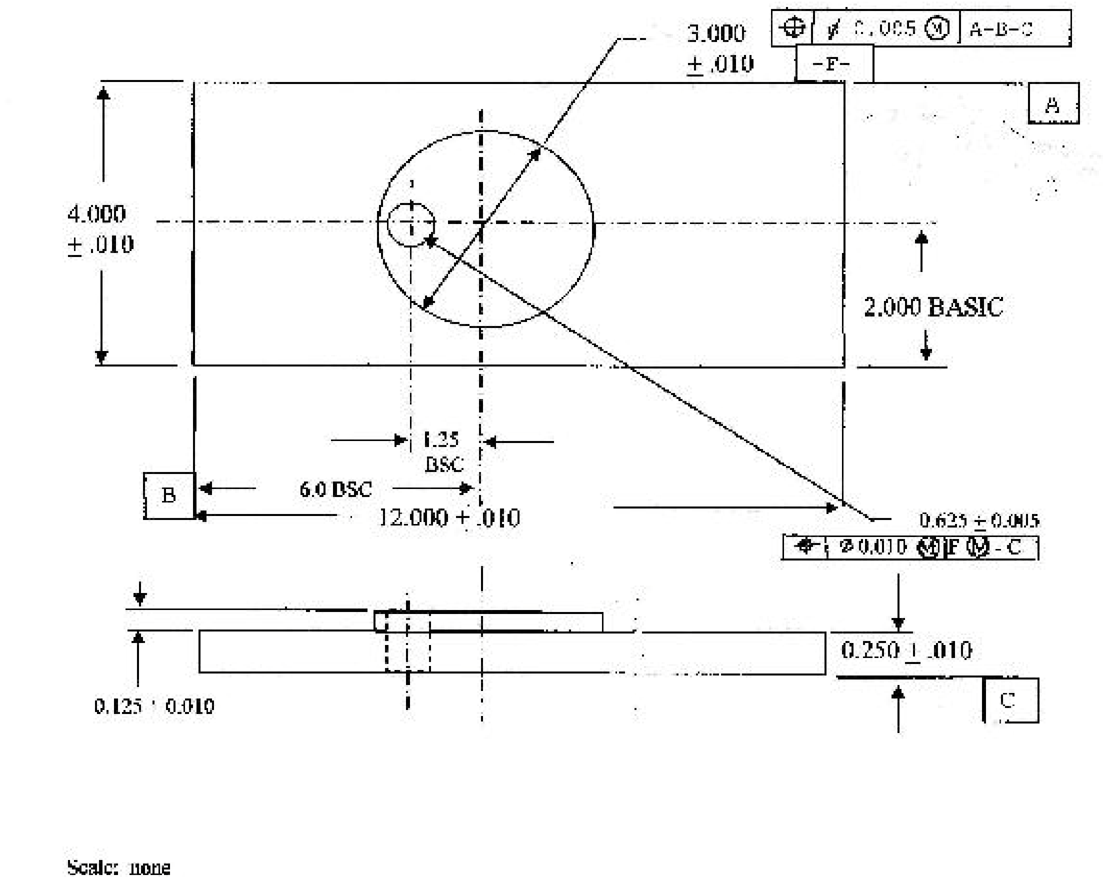


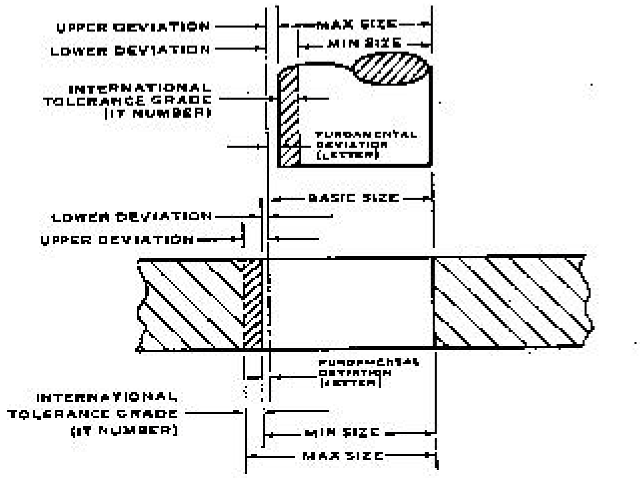
Figure 3-26 Part with 2 different feature datums.

3.4.2 Limit and Fit

Since Whitney first demonstrated the concept of interchangeable parts, fit has become a critical manufacturing issue. In fact, a tolerancing system for mating components was developed as part of both U.S. Standards and International Standards (ISO) [Gooldy, 1999] [ISO286-1]. This standard provides a table of recommended specifications for hole/shaft fit for various functionalities.

Three classes of fit *are used to specify mating interaction. These classes of fit are: (1) clearance fits, (2) transitional fits, and (3) interface fits.* Figure 3-21 illustrates this basic concept. As their names imply, a clearance fit indicates that a clearance remains between the shaft and the hole after they have been assembled, allowing the shaft to rotate or move about the major hole axis. The USAS standard uses nine subclasses of fit to describe hole/shaft fit. These subclasses range from RC1, a close sliding fit where no perceivable play can be observed, to RC9, a loose running fit where the shaft fits more loosely into the hole. The designer simply selects from the subclasses the one that best fits the needs of the part, knowing that the higher the specification subclass, the less expensive the manufacturing.

Transition fits are normally used to specify tolerance for parts that are stationary. Location clearance fits (LC1 to LC11) are used for parts that are assembled together and can be disassembled for service. The accuracy for these components is not exact. Transition location fits (LN1 to LN6) are specified when the location accuracy is of importance but a smaller clearance or interference is acceptable.



	ISO SYMBOL		DESCRIPTION	
	Hole Basis	Shaft Basis		
Clearance Fits	H11/c11	C11/h11	*Loose running* fit for wide commercial tolerances or allowances on external members.	More Clearance
Clearance Fits	H9/d9	D9/h9	*Free running* fit not for use where accuracy is essential, but good for large temperature variations, high running speeds, or heavy journal pressures.	More Clearance
Clearance Fits	H8/f7	F8/h7	*Close running* fit for running on accurate machines and for accurate location at moderate speeds and journal pressures.	More Clearance
Clearance Fits	H7/g6	G7/h6	*Sliding* fit not intended to run freely, but to move and turn freely and locate accurately.	More Clearance
Clearance Fits	H7/h6	H7/h6	*Locational clearance* fit provides snug fit for locating stationary parts; but can be freely assembled and disassembled.	More Clearance
Transition Fits	H7/k6	K7/h6	*Locational transition* fit for accurate location, a compromise between clearance and interference.	More Interference
Transition Fits	H7/n6	N7/h6	*Locational transition* fit for more accurate location where greater interference is permissible.	More Interference
Interference Fits	H7/p6	P7/h6	*Locational interference* fit for parts requiring rigidity and alignment with prime accuracy of location but without special bore pressure requirements.	More Interference
Interference Fits	H7/s6	S7/h6	*Medium drive* fit for ordinary steel parts or shrink fits on light sections, the tightest fit usable with cast iron.	More Interference
Interference Fits	H7/u6	U7/h6	*Force* fit suitable for parts which can be highly stressed or for shrink fits where the heavy pressing forces required are impractical.	More Interference

Figure 3-27: Preferred metric limits and fits. (Copied with permission from ASME Y14.5 – 1994)

Figure 3.27

(cont'd)

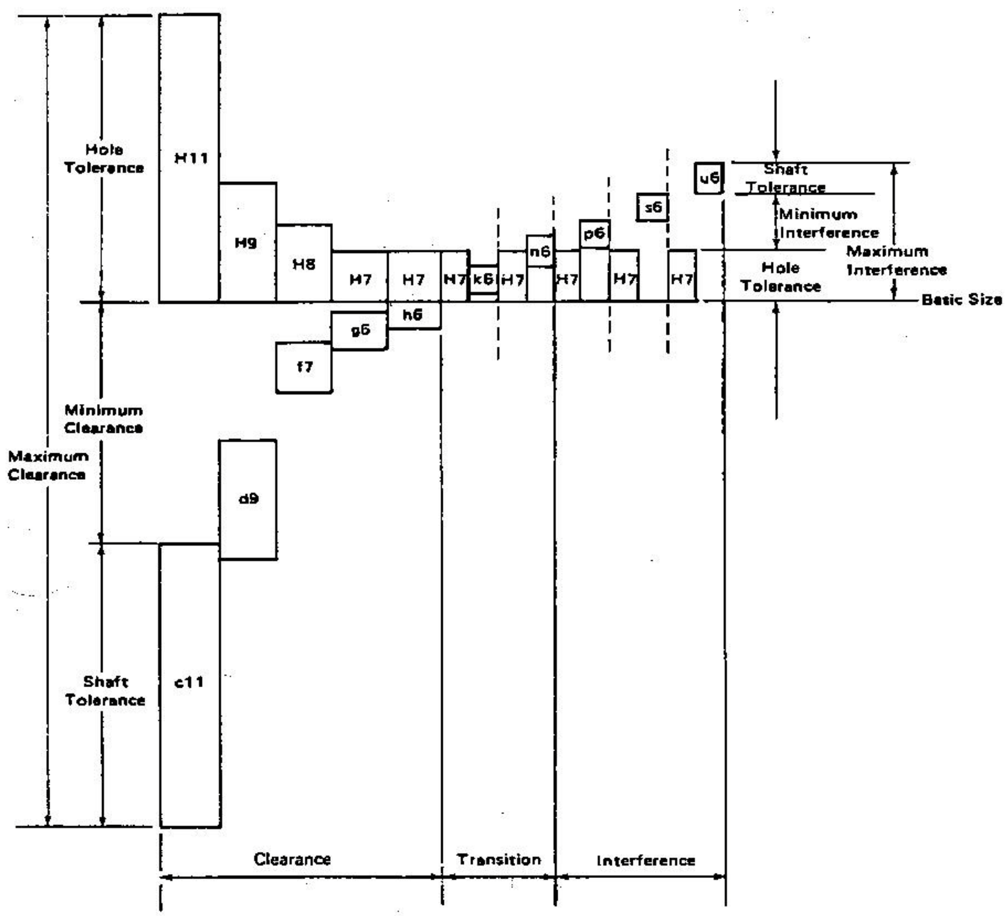


Figure 3.27 (cont'd) Hole basis fits.

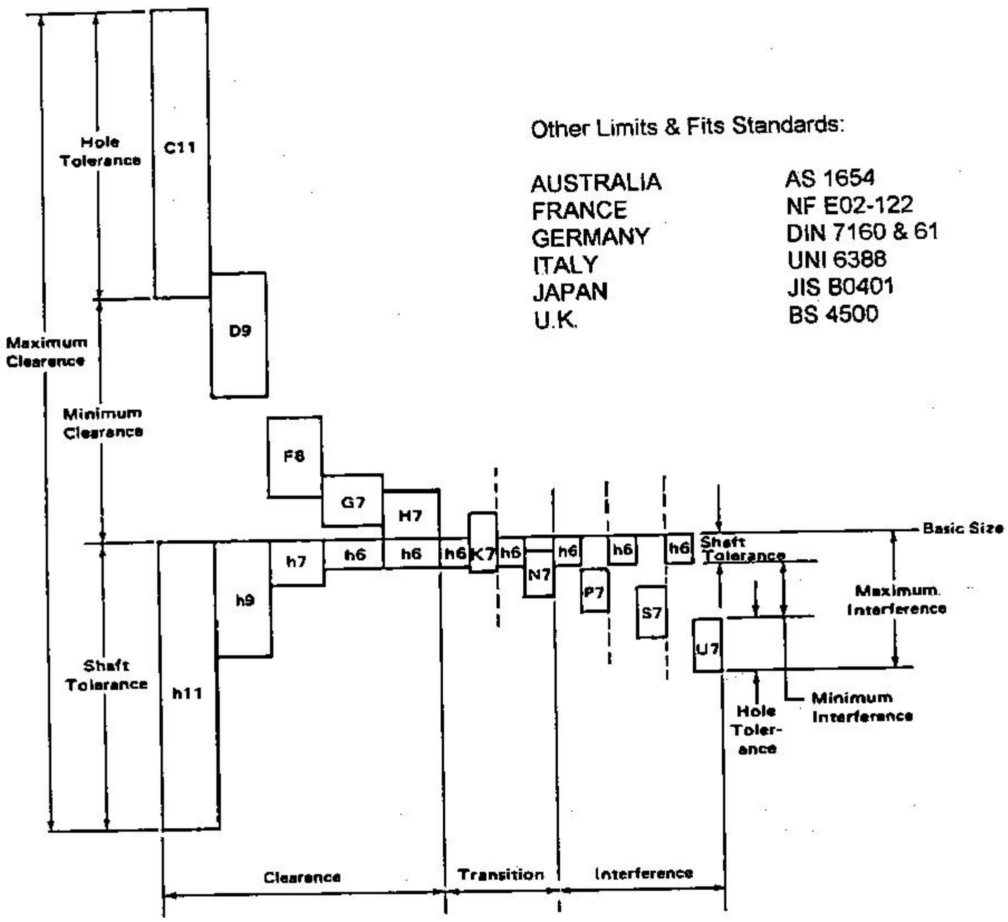


Figure 3.27 (Cont’d) Shaft basis fits.

Location interference fits (LN1 to LN3) are specified when both rigidity and accuracy are required. Location interference fit parts can be assembled and disassembled but not without special tooling (usually a shaft or wheel puller) and considerable time. Other interference parts normally require special operations for assembly. Tight drive fits (FN1) are used on parts requiring nominal assembly pressure. Force fits (FN5) are used for drive applications where the hole element is normally heated to expand the diameter prior to assembly. Tables that contain the specification for various classes of fit are available in most design handbooks.

3.5 Chapter Summary

In this chapter, we presented the basics of ASME Y14.5 along with discussions on the use and interpretation of this standard. As engineering products become more complex and as tolerances get smaller and smaller, the need to more precisely describe the acceptable part variability becomes greater and greater. The chapter provided many simple parts to illustrate the basic concepts of form and location. Although the parts used in the discussion were simple, they illustrate the same interpretation specifics that one would find in a more complicated part. Most times, part complexity is more related to the number of features on a part rather than the interactions of these features.

3.6 Review Questions

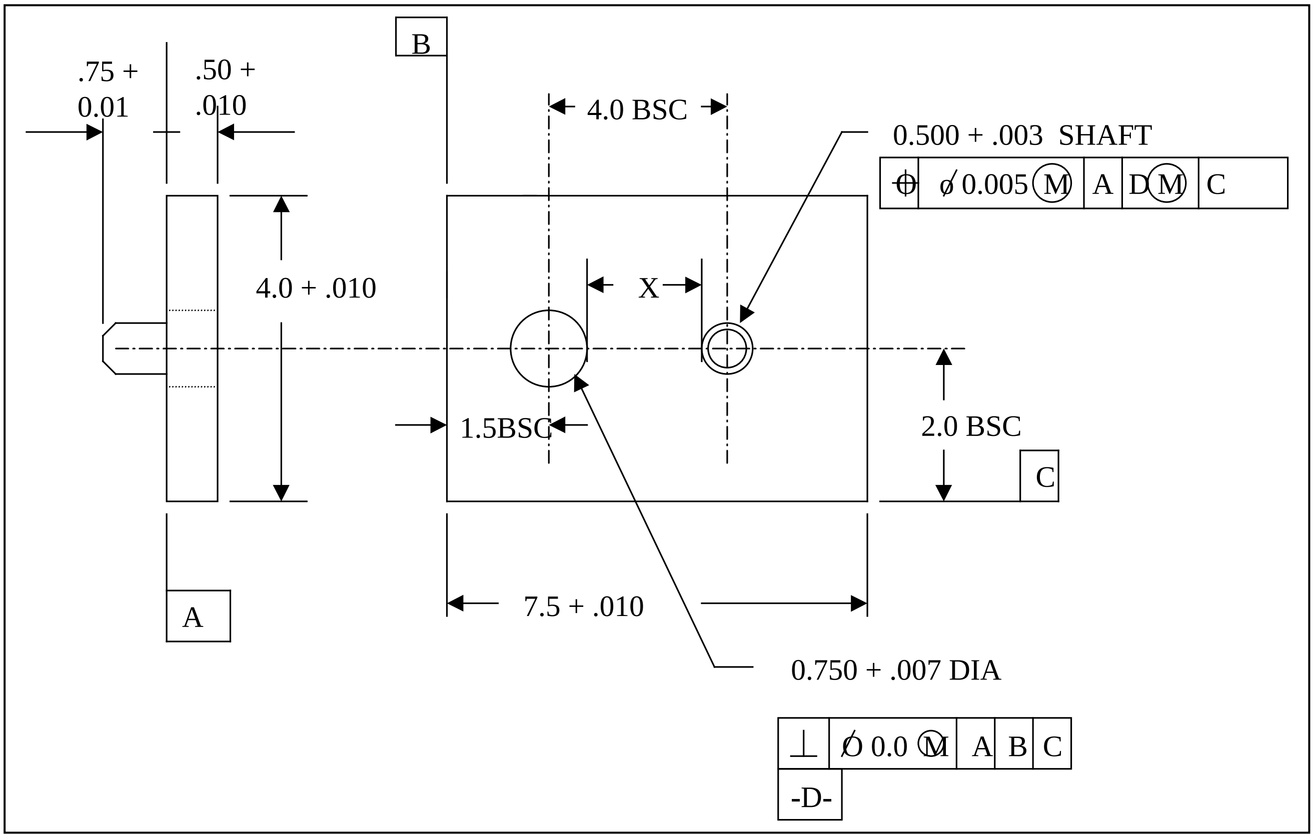
3.1 Describe why ASME Y14.5 is important for product specification.

3.2 What is virtual size? What are the specification requirements to have a virtual size?

3.3 What are the different classes of fit? Describe specific instances when these classes would be used as part of a design.

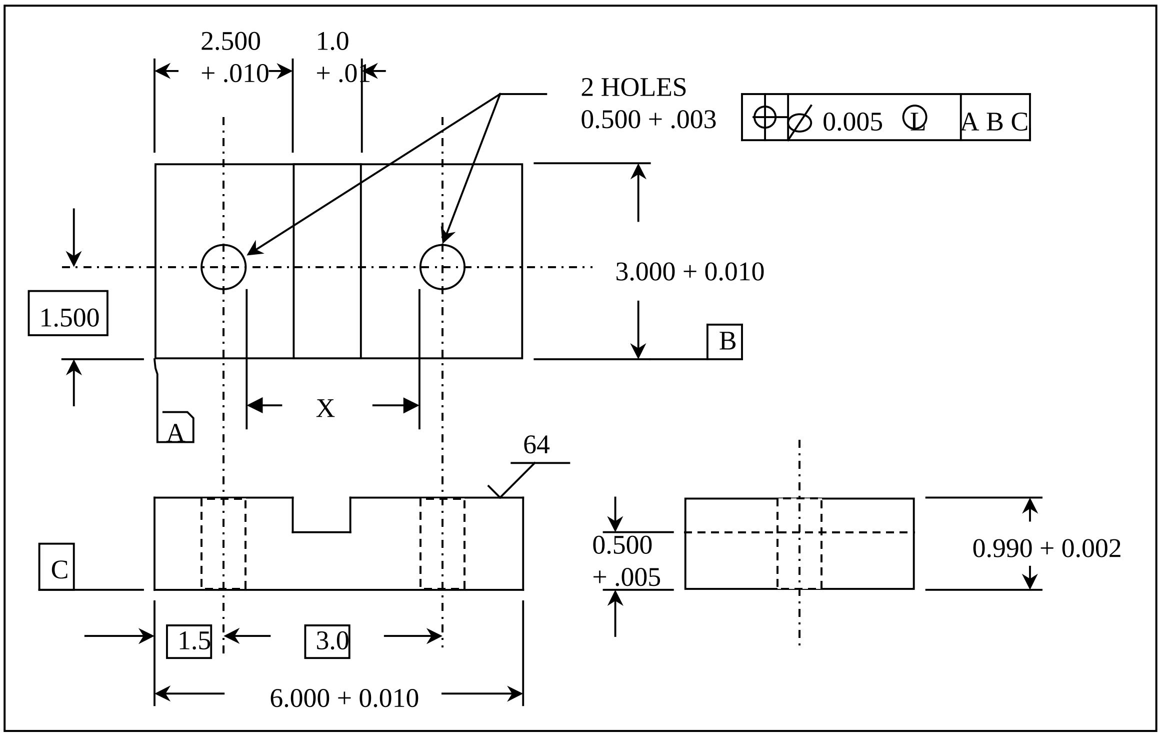
3.7 Review Problems

3-1. For the part shown below, determine the maximum and minimum value of the dimension labeled as X.

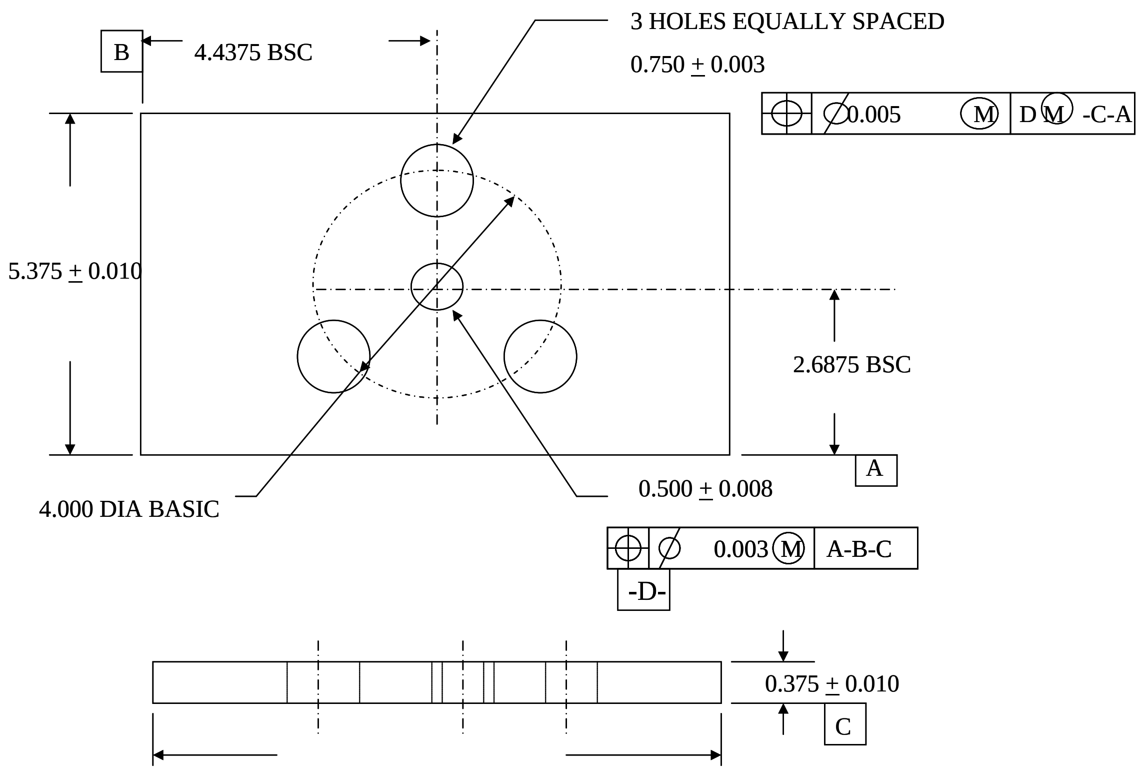


3-2 For the part shown in 3-1, draw the VIRTUAL SIZE (if there is one) as well as the gaging (if any can be used) that would be required for inspection of the part. Assume that the part is a high volume product.

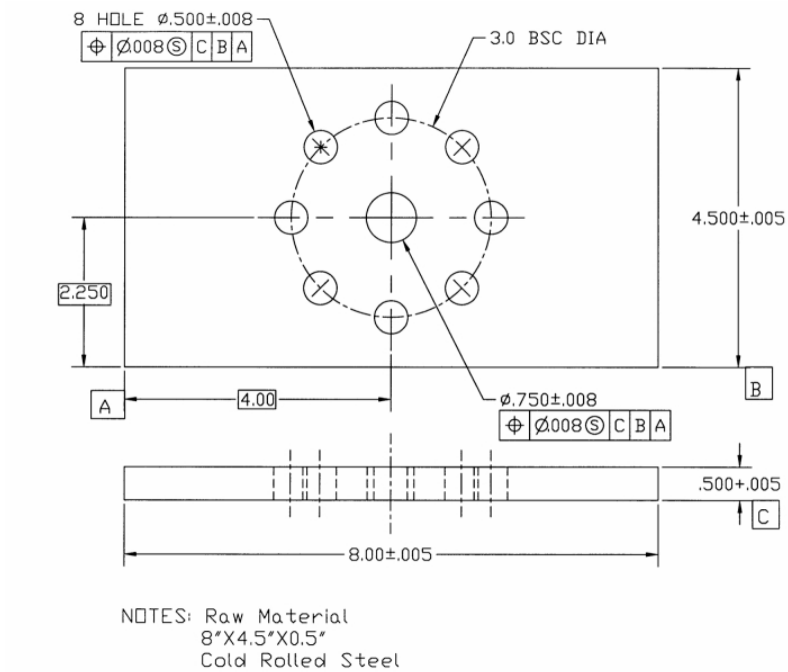
3.3 For the part shown below, describe how the part would be inspected if it were to be produced in large quantities. What is the minimum and maximum acceptable value of X?



3.4 For the part shown below, describe an efficient inspection schema if thousands of parts will be inspected.



3.5 For the part shown below, develop an efficient inspection schema for inspecting thousands of components. Does the schema change if the modifier for position is changed from RFS to MMC?



3.7 References:

ANSI, “Surface Texture,” ANSI Standard B46.1-1978.

ANSI, for ANSI and ISO document: http://global.ihs.com/.

ASME Y14.36M-1996; Surface Texture Symbols.

ASME Y14.5M-1994; Geometric Dimensioning and Tolerancing.

Earle, James H., (2000). Graphics for Engineers: with AutoCAD release 14 & 2000, Upper Saddle River, NJ : Prentice-Hall.

Foster, L.W. (1994). Geo-Metrics III: The Application of Geometric Tolerancing Techniques. Reading, MA: Addison-Wesley.

Gooldy, G. (1999). Dimensioning, Tolerancing and Gaging applied. Englewood Cliffs, NJ: Prentice Hall.

ISO 1302:1994; Technical Drawings - Method of indicating surface texture.

ISO CD 1302; Geometrical Product Specifications (GPS) - Indication of Surface texture.

ISO, for ISO standard see: http://www.iso.ch/iso/en/ISOOnline.frontpage.

Suh, N.P. (1982). “Qualitative and Quantitative Analysis of Design and Manufacturing Axioms,” CIRP Annals, 31, 333-338.

Voelcker, H.B. (1993). “A Current Perspective on Tolerancing and Metrology,” Manufacturing Review, 6,4.

Voelcker, H.B. and A.A.G. Requicha (1977). “Geometric Modeling of Mechanical Parts and Processes,” Computer, 10,12, 48-57.