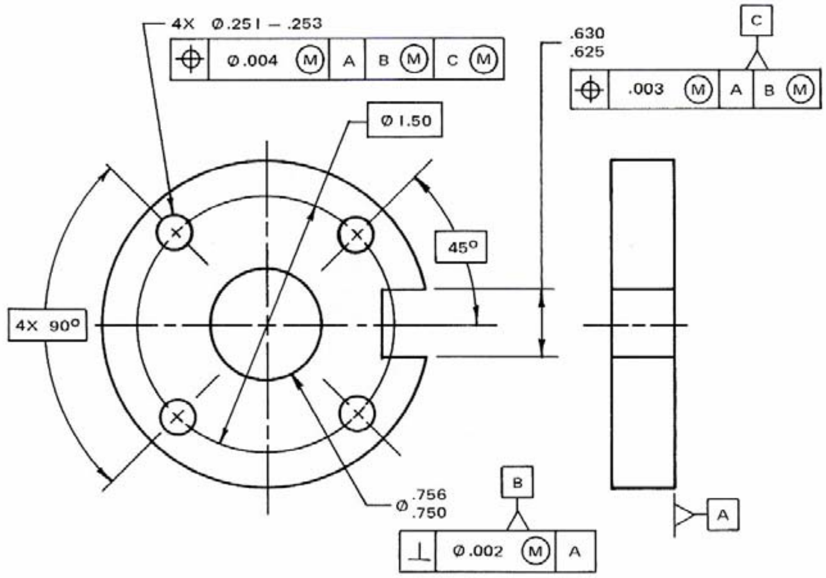
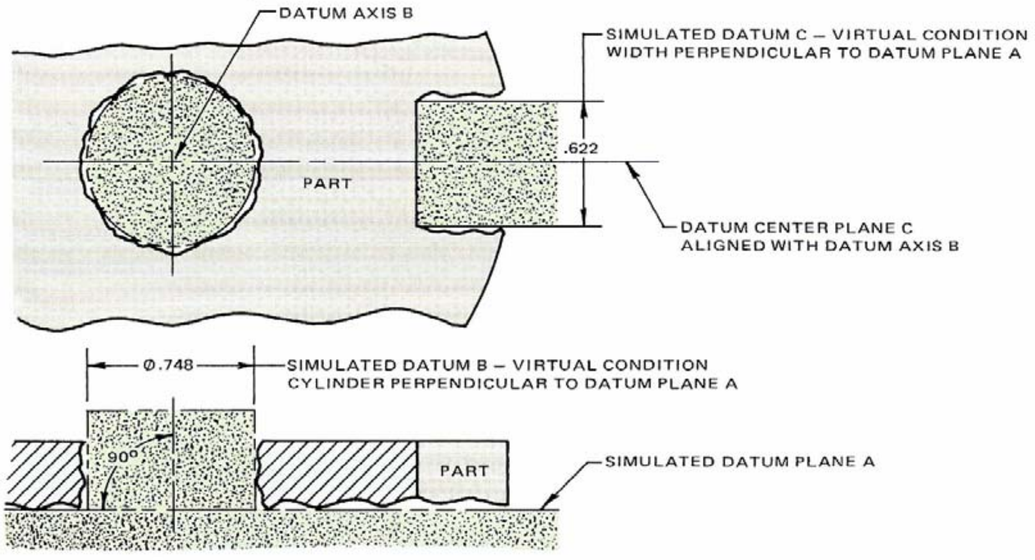
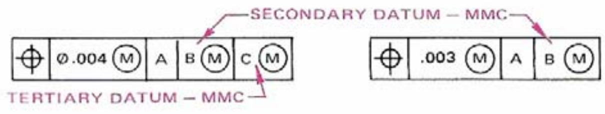


(A) DRAWING CALLOUT

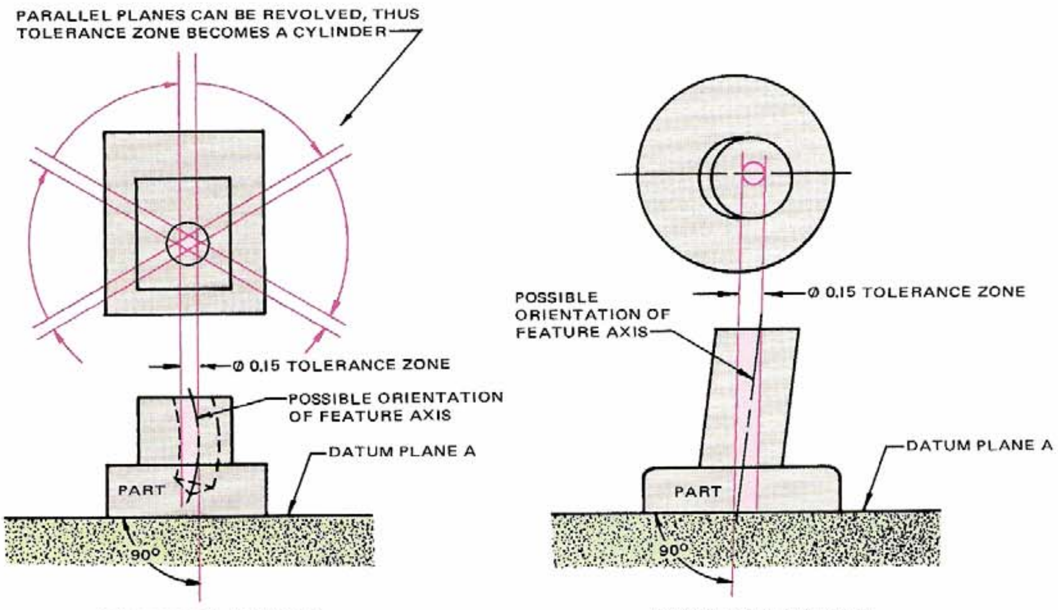


(B) DATUM FEATURE SIMULATORS



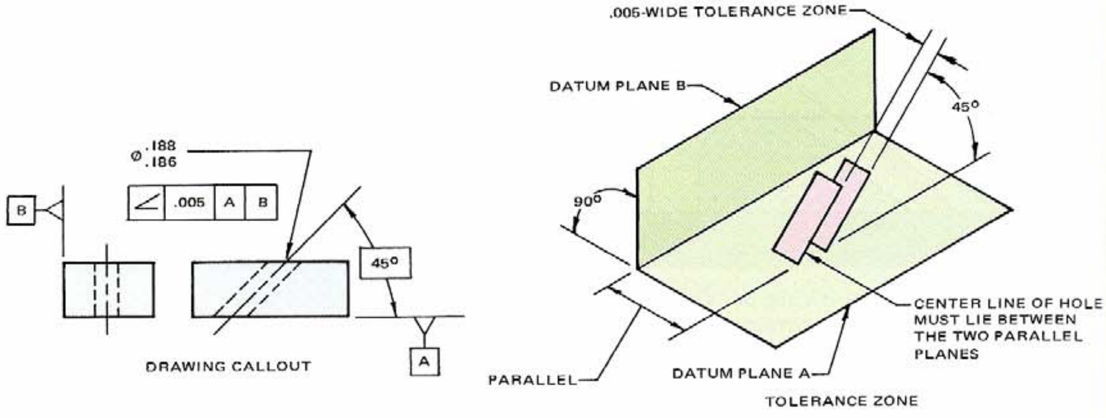
(C) SECONDARY AND TERTIARY DATUM CALLOUTS IN FEATURE CONTROL FRAME

Fig. 16-7-9 **Secondary and tertiary datum features—MMC.**

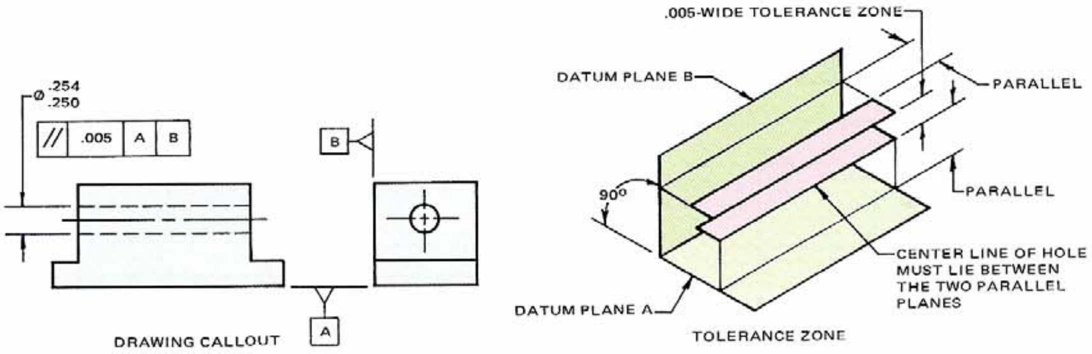


(A) INTERNAL FEATURE (B) EXTERNAL FEATURE

Fig. 16-8-5 Tolerance zones for perpendicularity shown in Fig. 16-8-2 (p. 572).

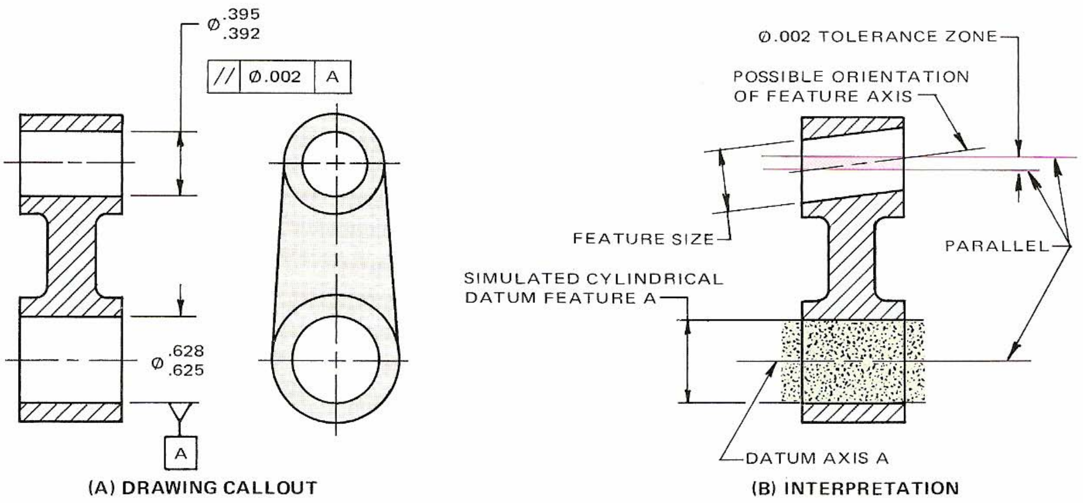


(A) ANGULARITY TOLERANCE



(B) PARALLELISM TOLERANCE

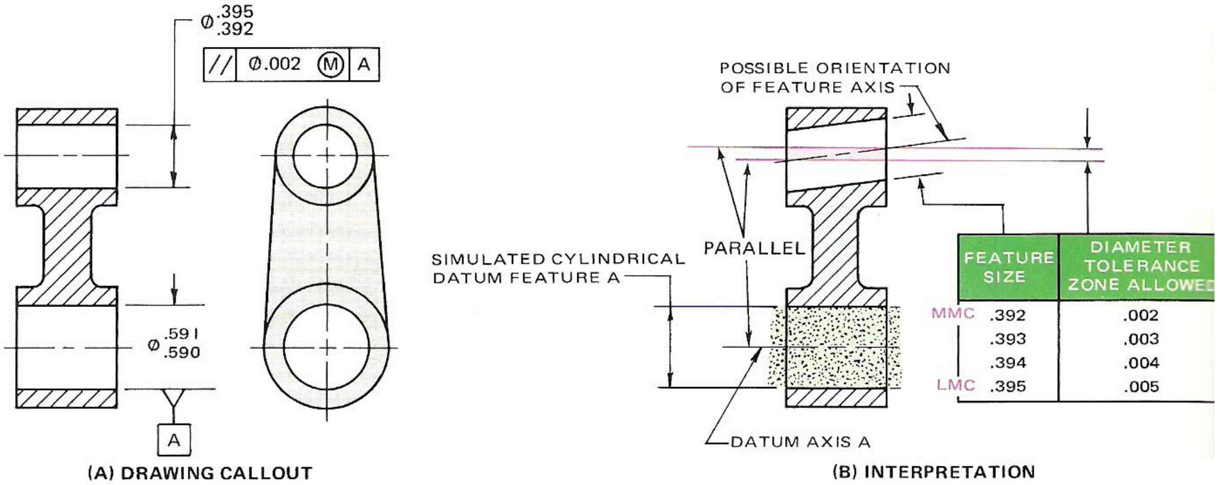
Fig. 16-8-6 Orientation tolerances referenced to two datums.



(A) DRAWING CALLOUT

(B) INTERPRETATION

Fig. 16-8-11 Specifying parallelism for an axis (both feature and datum feature RFS).



	FEATURE SIZE	DIAMETER TOLERANCE ZONE ALLOWE
MMC	.392	.002
	.393	.003
	.394	.004
LMC	.395	.005

(A) DRAWING CALLOUT

(B) INTERPRETATION

Fig. 16-8-12 Specifying parallelism for an axis (feature at MMC and datum feature RFS).

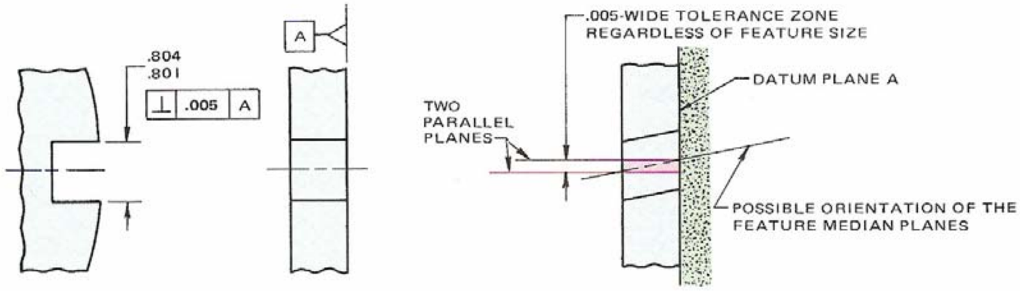
Perpendicularity for a Median Plane Regardless of feature size, the median plane of the feature shown in Fig. 16-8-13 must lie between two parallel planes, .005 in. apart, that are perpendicular to datum plane *A*. In addition, the feature center plane must be within the specified tolerance of location.

Perpendicularity for an Axis (Both Feature and Datum RFS) Regardless of feature size, the feature axis shown in Fig. 16-8-14 must lie between two parallel planes, .005 in. apart, that are perpendicular to datum axis *A*. In addition, the feature axis must be within the specified tolerance of location.

Perpendicularity for an Axis (Tolerance at MMC) When the feature shown in Fig. 16-8-15 is at the MMC (Ø 2.000), its axis must be perpendicular within .002 in. to datum plane *A*. When the feature departs from MMC, an increase in the perpendicularity tolerance, equal to the amount of such depart is allowed. Moreover, the feature axis must be within the spe ified tolerance of location.

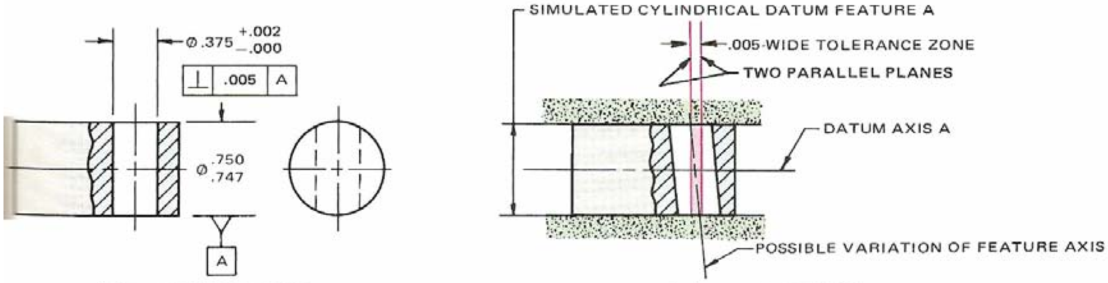
Perpendicularity for an Axis (Zero Tolerance MMC) When the feature shown in Fig. 16-8-16 (p. 578 at the MMC (Ø50.00), its axis must be perpendicular to tum plane *A*. When the feature departs from MMC, an crease in the perpendicularity tolerance is allowed equa the amount of such departure. Also, the feature axis mus within any specified tolerance of location.

Perpendicularity with a Maximum Tolerance Spe fied When the feature shown in Fig. 16-8-17 (p. 578) is MMC (Ø50.00), its axis must be perpendicular to datum pl *A*. When the feature departs from MMC, an increase in



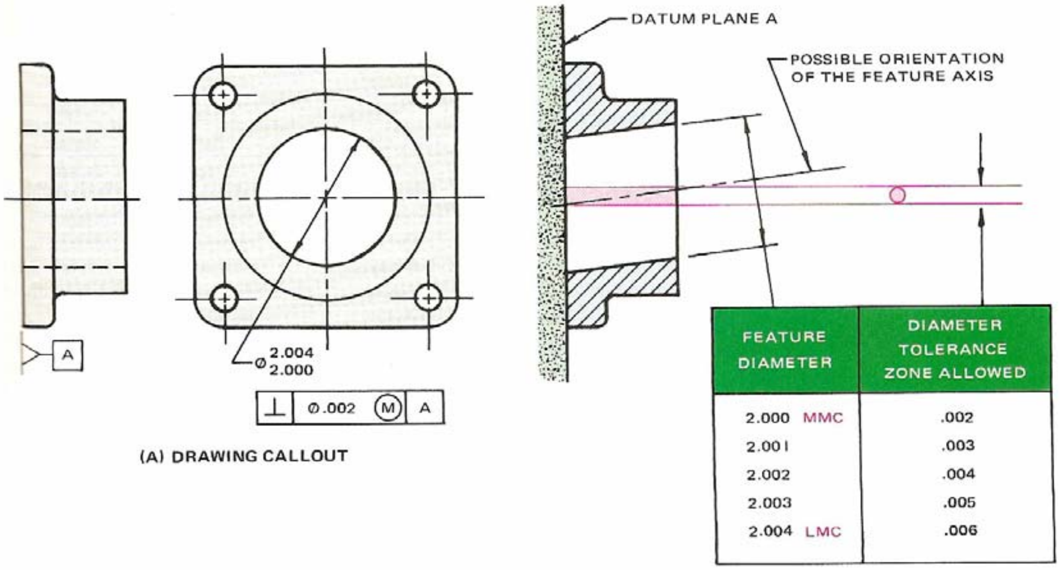
(A) DRAWING CALLOUT (B) INTERPRETATION

Fig. 16-8-13 Specifying perpendicularity for a median plane (feature RFS).



(A) DRAWING CALLOUT (B) TOLERANCE ZONE

Fig. 16-8-14 Specifying perpendicularity for an axis (both feature and datum feature RFS).



FEATURE DIAMETER	DIAMETER TOLERANCE ZONE ALLOWED
2.000 MMC	.002
2.001	.003
2.002	.004
2.003	.005
2.004 LMC	.006

(A) DRAWING CALLOUT

(B) TOLERANCE ZONE

Fig. 16-8-15 Specifying perpendicularity for an axis (tolerance at MMC).

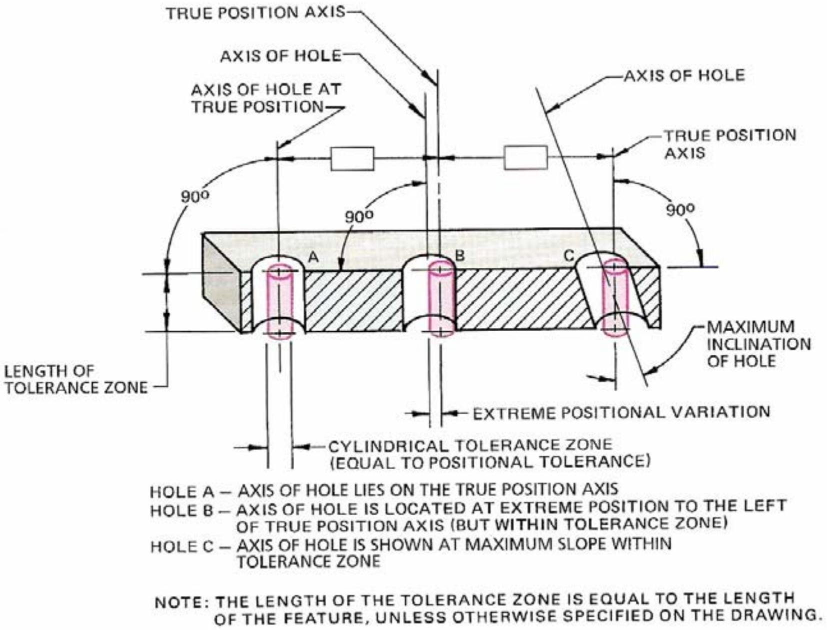
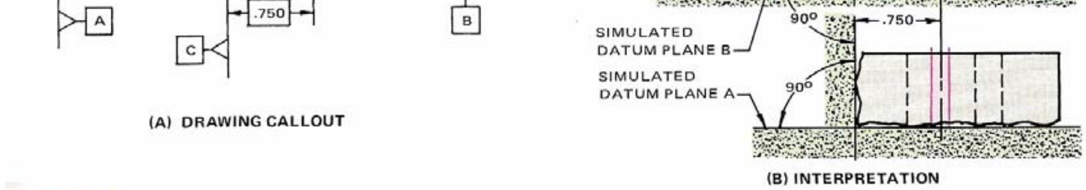


Fig. 16-9-13 Hole axes in relationship to positional tolerance zones.

positional tolerance applies. When the actual size of the feature is larger than MMC, additional positional tolerance results (Fig. 16-9-15, p. 586).

The problems of tolerancing for the position of holes are simplified when positional tolerancing is applied on an MMC basis. Positional tolerancing simplifies measuring procedures by using functional "go" gages. It also permits an increase in positional variations as the size departs from the maximum material size without jeopardizing free assembly of mating features.

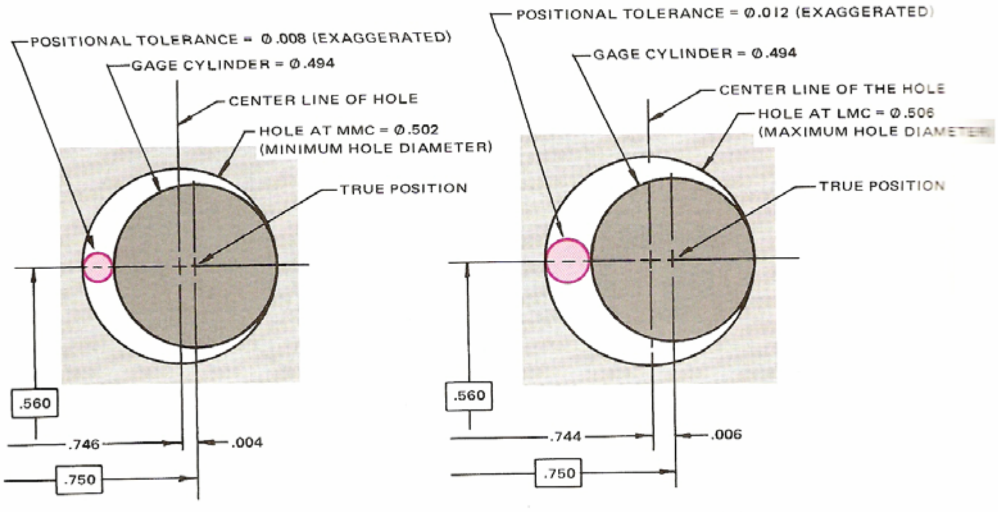
A positional tolerance on an MMC basis is specified on a drawing, on either the front or the side view, as shown in Fig. 16-9-14. The MMC symbol Ⓜ is added in the feature control frame immediately after the tolerance.



(A) DRAWING CALLOUT

(B) INTERPRETATION

Fig. 16-9-14 Positional tolerancing—MMC.



(A) HOLES AT MMC (B) HOLES AT LMC

Fig. 16-9-15 **Positional variations for tolerancing for Fig. 16-9-14, on the previous page.**

This is illustrated in Fig. 16-9-15, where the gage cylinder is shown at true position and the minimum and maximum diameter holes are drawn to show the extreme permissible variations in position in one direction.

Therefore, if a hole is at its maximum material condition (minimum diameter), the position of its axis must lie within a circular tolerance zone having a diameter equal to the specified tolerance. If the hole is at its maximum diameter (least material condition), the diameter of the tolerance zone for the axis is increased by the amount of the feature tolerance. The greatest deviation of the axis in one direction from true position is therefore:

$$\frac{H + P}{2} = \frac{.004 + .008}{2} = .006$$

where H = hole diameter tolerance
P = positional tolerance

It must be emphasized that positional tolerancing, even on an MMC basis, is not a cure-all for positional tolerancing problems; each method of tolerancing has its own area of usefulness. In each application a method must be selected that best suits that particular case.

Positional tolerancing on an MMC basis is preferred when production quantities warrant the provision of functional "go" gages, because gaging is then limited to one simple operation, even when a group of holes is involved. This method also facilitates manufacture by permitting larger variations in position when the diameter departs from the maximum material condition. It cannot be used when it is essential that variations in location of the axis be observed regardless of feature size.

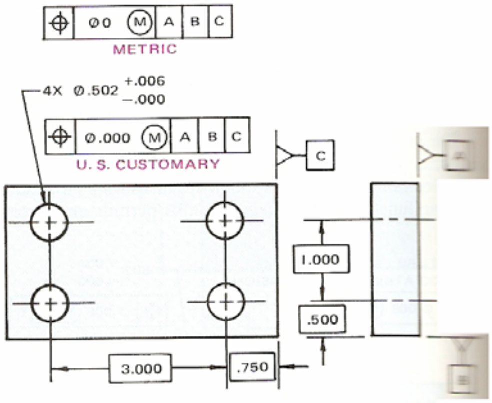


Fig. 16-9-16 **Positional tolerancing—zero MMC.**

Positional Tolerancing at Zero MMC The application of MMC permits the tolerance to exceed the value specified, provided features are within size limits and parts are acceptable. This is accomplished by adjusting the minimum size limit of a hole to the absolute minimum required for the insertion of an applicable fastener located precisely at true position, and specifying a zero tolerance at MMC (Fig. 16-9-16). In this case, the positional tolerance allowed is totally dependent on the actual size of the considered feature.

Positional Tolerancing—RFS In certain cases, the design or function of a part may require the positional tolerance or datum reference, or both, to be maintained regardless of al feature sizes. When applied to the positional tolerance rcular features, RFS requires the axis of each feature to cated within the specified positional tolerance regardless e size of the feature. This requirement imposes a closer rol of the features involved and introduces complexities rification.

sitional Tolerancing—LMC Where positional tolerancing MC is specified, the stated positional tolerance applies the feature contains the least amount of material per- ted by its toleranced size dimension. Perfect form at MMC required. Where the feature departs from its LMC size, crease in positional tolerance is allowed, equal to the nt of such departure (Fig. 16-9-17). Specifying LMC is ted to positional tolerancing applications in which MMC not provide the desired control and RFS is too restrictive.

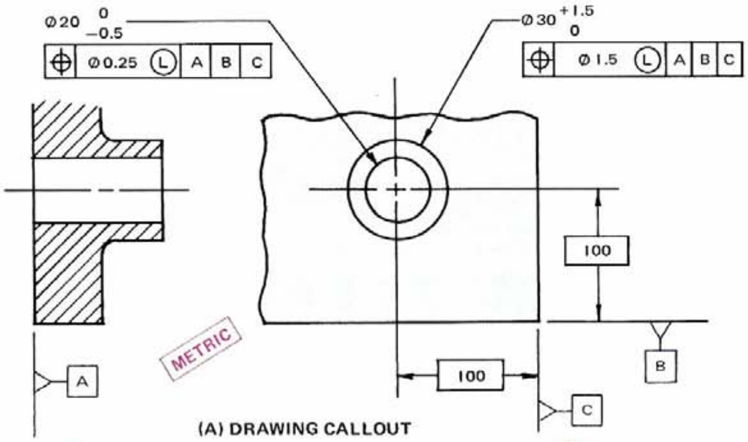
Advantages of Positional Tolerancing

It is practical to replace coordinate tolerances with a positional tolerance having a value equal to the diagonal of the coordinate tolerance zone. This provides 57 percent more tolerance area and would probably result in the rejection of fewer parts for positional errors.

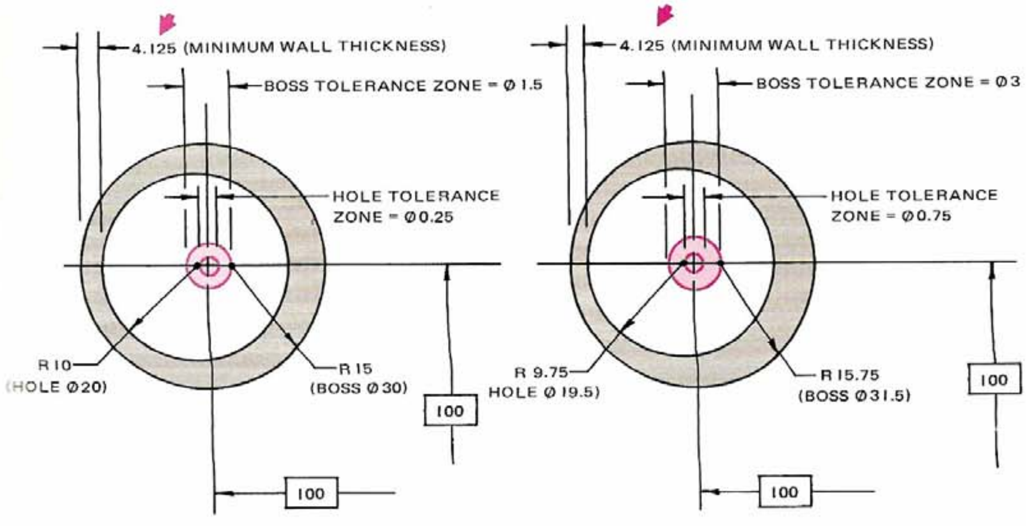
A simple method for checking positional tolerance errors is to evaluate them on a chart, as shown in Fig. 16-9-18 (p. 588). For example, the four parts listed in Fig. 16-9-19 (p. 589) were rejected when the coordinate tolerances were applied to them.

If the parts had been toleranced using the positional tolerance RFS method shown in Fig. 16-9-9 (p. 583) and given a tolerance of Ø.028 in. (equal to the diagonal of the coordinate tolerance zone), three of the parts—*A*, *B*, and *D*—would not have been rejected.

If the parts shown in Fig. 16-9-19 had been toleranced using the positional tolerance MMC method shown in



(A) DRAWING CALLOUT



(B) TOLERANCING ZONES WHEN HOLE AT LMC

(C) TOLERANCING ZONES WHEN HOLE AT MMC

16-9-17 **LMC applied to a boss and a hole.**