Chapter 5

Major Concepts of Geometric Dimensioning and Tolerancing

- Major Concepts of Geometric Dimensioning and Tolerancing
  - -Converting from ± to Geometric Tolerancing
  - -Position
  - -Gaging
  - -Flatness
  - -Selecting Datum Features
  - -Perpendicularity
  - -Mating Part Tolerancing
  - -Reading the Feature Control Frames as a Language
  - -Calculating Inner and Outer Boundaries
    - Virtual Condition
    - Resultant Condition
  - -MMC vs. RFS vs. LMC
    - What they mean
    - When to use them
    - Boundaries they create
- Bonus Tolerancing Formulas … for Position, Perpendicularity, Angularity and Parallelism Modified at MMC
- Allowed vs. Actual Deviation from True Position
- A Difference between Bonus Tolerance (growth) and Datum feature shift (movement) of Tolerance Zones

Chapter Objectives

Readers will learn:

1. How to convert from plus and minus tolerancing to geometric tolerancing.
2. When Position Tolerancing applies, its tolerance zones and the boundaries it creates.
3. How to select, define and tolerance datum features.
4. How size controls form.
5. How to measure MMC and LMC.
6. How to apply geometric tolerances of flatness, perpendicularity and position in sequence.
7. The tolerance zone configuration for flatness and varieties of perpendicularity.
8. How to calculate and distribute mating part tolerances.
9. Practical Absolute Functional Gage design, dimensioning and tolerancing.
10. How to calculate bonus tolerance for position, perpendicularity, angularity and parallelism when modified at MMC.
11. The difference between using MMC symbology after geometric tolerances (tolerance zone growth) and MMB symbology after datum features (datum feature shift).
12. How to calculate when a feature complies with its position tolerance and when it does not (variables data collection and analysis using CMM type data).
13. How to use charts for inches or millimeters to determine position tolerance compliance.
14. The meaning of Actual Mating Envelope and Actual Mating Size (both oriented and/or located vs. unoriented, unlocated) for tolerance compliance.
15. How to use gages to help understand geometric controls and use of maximum material condition and maximum material boundary symbols.

Major Concepts of Geometric Dimensioning and Tolerancing

There are many situations where all four geometric contributors of size, shape, angle and location must be controlled. The only geometric characteristic symbols capable of tolerancing all four are in the profile category, most commonly, profile of a surface (which will be discussed in-depth later). But, if we give a regular feature of size like a simple cylindrical hole a size tolerance, the size limits will control size and form. We can then go about controlling angles and location.

On the following mating parts (Part #1 and Part #2), location has been accomplished with plus and minus toleranced dimensions.

FIGURE 5-1

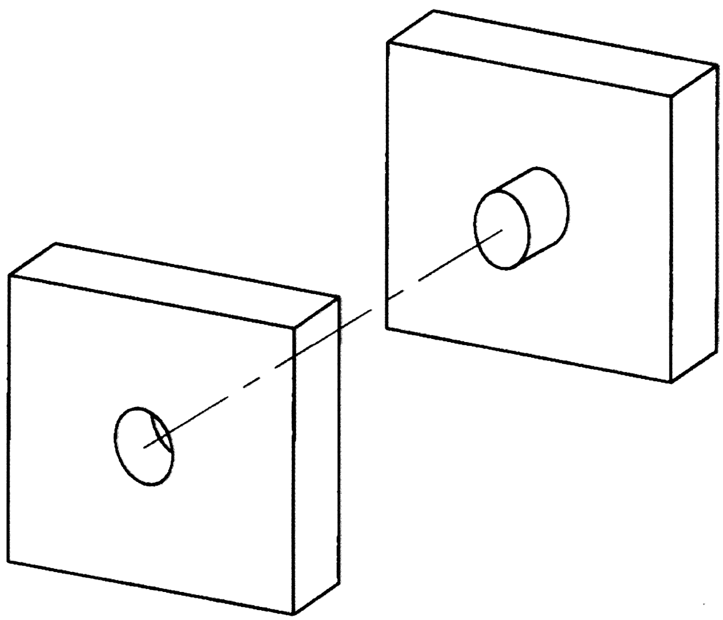
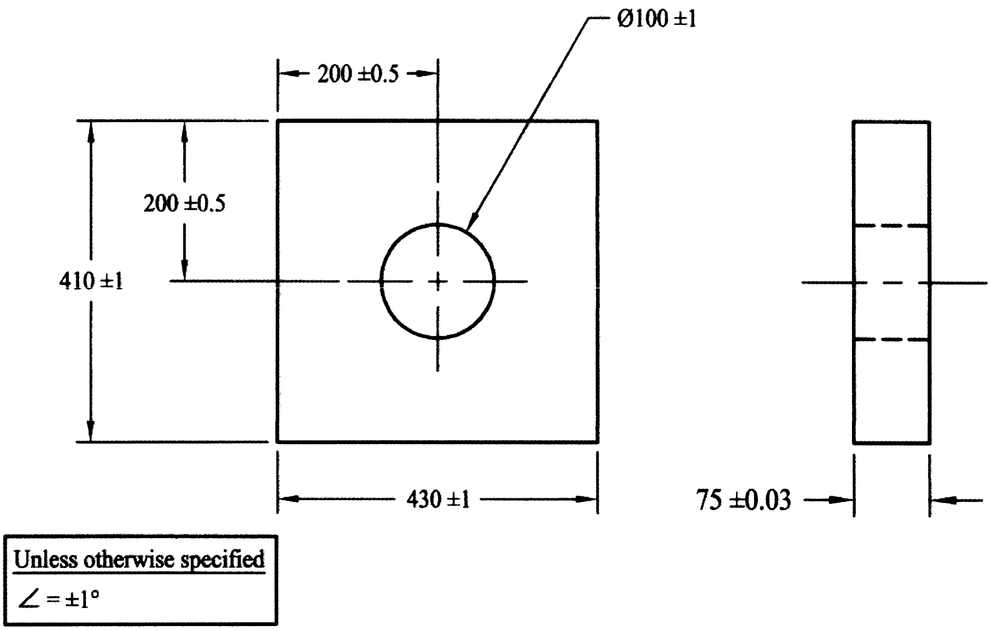


FIGURE 5-2 a & b [Part #1 and Part #2]

Part #1



Part #2

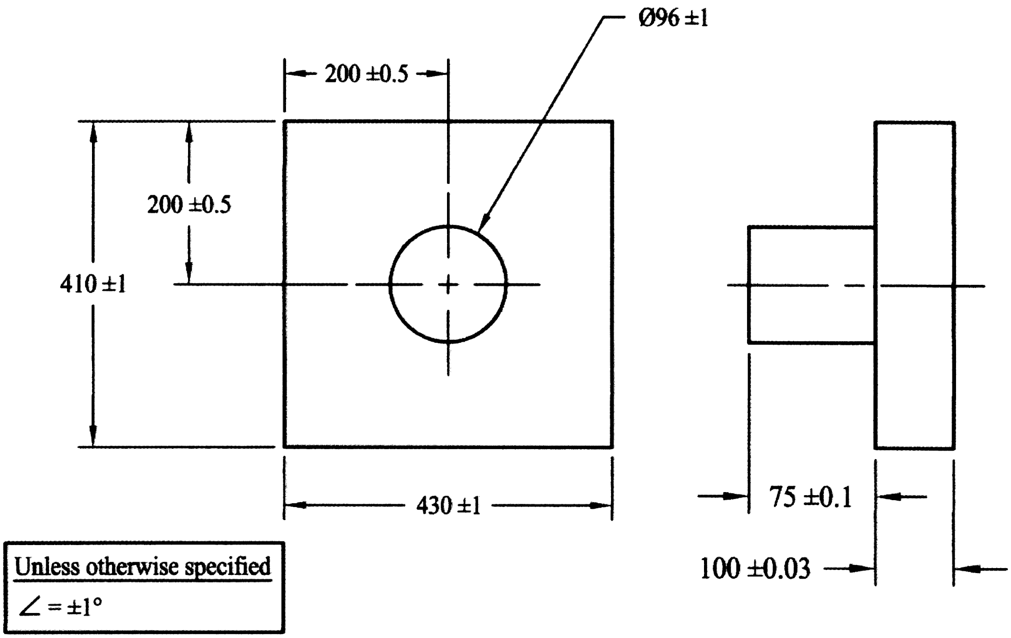
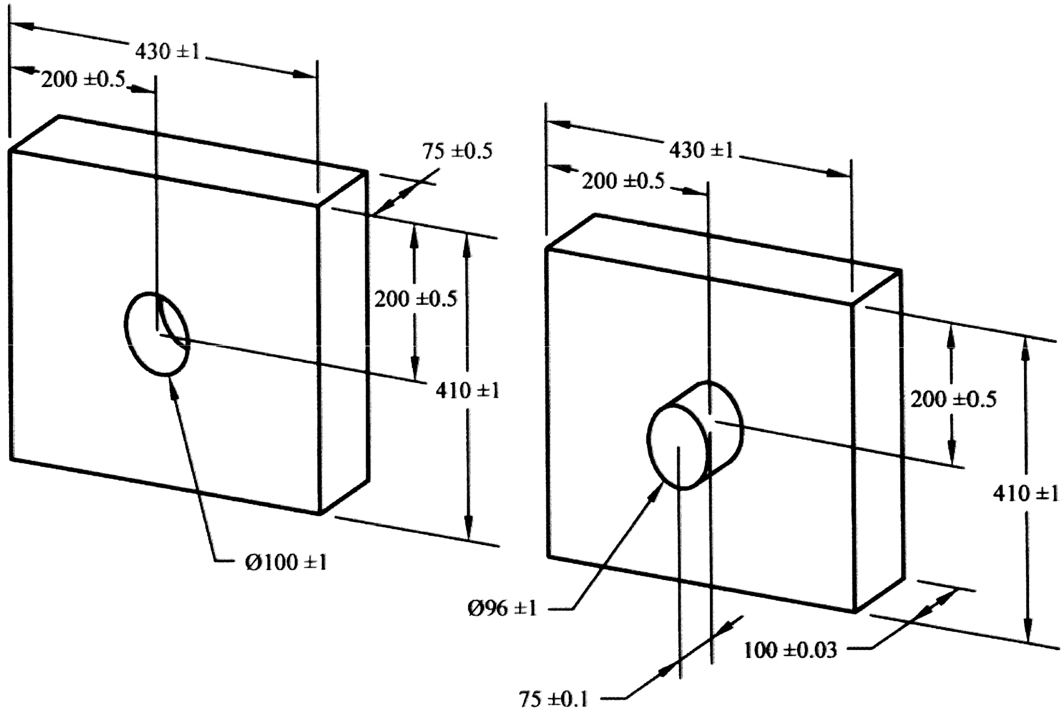


FIGURE 5-3



The angles on the part depicted here as 90° angles have been toleranced by a general tolerance note in the "unless otherwise specified" block.

Unfortunately, the type of tolerancing to control the tolerances of the relationship for location between the hole on Part #1 (as well as the shaft on Part #2) and the edges of the part is ambiguous and, therefore, insufficient.

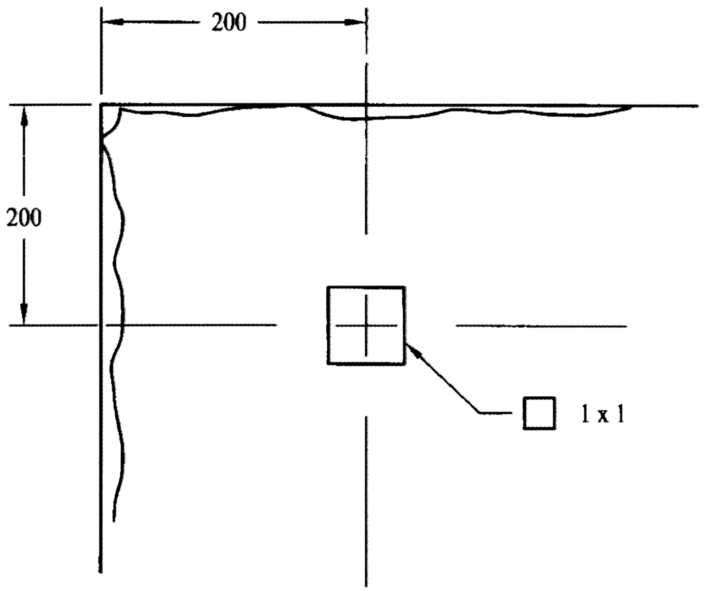
For example, depending on which features are interpreted as the origin of measurement, the tolerance zones applied to the part could be quite different. If the edges of Part #1 are seen to be the implied datum features (origins of measurement), the tolerance zone for the hole's location will appear as a square zone (2D) or parallelepipedic zone (3D) that may be 1mm by 1mm.

This zone would be seen as measured from planes formed by the part surface high points. Its center would be 200 millimeters from one of these planes and 200 millimeters from another and perpendicular to another to complete our 3 dimensional, 3 plane coordinate system.

FIGURE 5-4

One Possible Interpretation of Part #1

Interpreted similar to a Position Control:



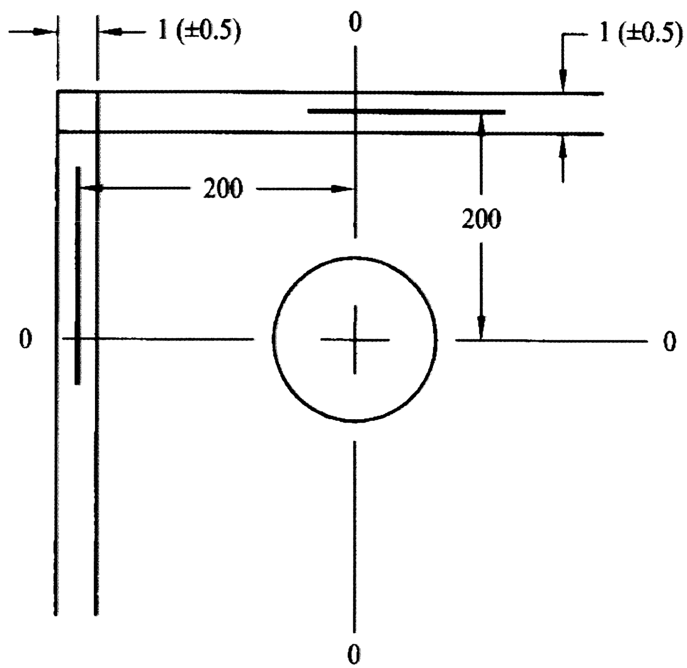
Unfortunately, we would only be guessing at what would be the implied primary datum feature, the secondary datum feature and the tertiary (third) datum feature in this datum reference frame (coordinate system). This is important since the person setting up and measuring the part would be required to know. They could give the primary datum feature a proper seat of a minimum contact of 3 high points of contact on the primary datum plane (simulated somehow in the measurement procedure).

The secondary datum feature would get a minimum of 2 points of high point contact and the tertiary would get a minimum of 1 point of high point contact. Since no datum features are specified on the part drawing, no set up of the part would be repeatable and, therefore, no measurement data would be repeatable.

Add to that the fact that another inspector might simply interpret the drawing less as a position control (which this first interpretation assumes) and more as a profile of a surface-type control. Instead of the hole measured from the edges of the part, it might be interpreted as the edges of the part are measured from the hole (which would then be assumed to be the implied datum feature). The edges of the part would be given the tolerance of plus or minus 0.5 millimeters centered on the 200 millimeter dimensions to the edges.

FIGURE 5-5

Part #1 interpreted similar to a Profile Control:



This interpretation would not only locate the surfaces from the hole axis, but would (if interpreted as an implied profile of a surface control) hold a form (flatness) and angle (perpendicularity) control on the surfaces. All this is, of course, idle speculation, since the drawing has no specific true interpretation. It is poorly toleranced, ambiguous, and badly in need of improvement.

Correcting the Geometric Definition and Its Tolerancing Scheme

Step 1

This improvement starts off with the assignment of datum features. The primary datum feature on each part should be:

1) the seating surfaces,
2) the surfaces that need the most physical contact in the assembly, and
3) the surfaces that dictate the angle at which these two parts will assemble.

The answer to this selection can be seen in an assembly view of the side view of both parts as follows:

FIGURE 5-6

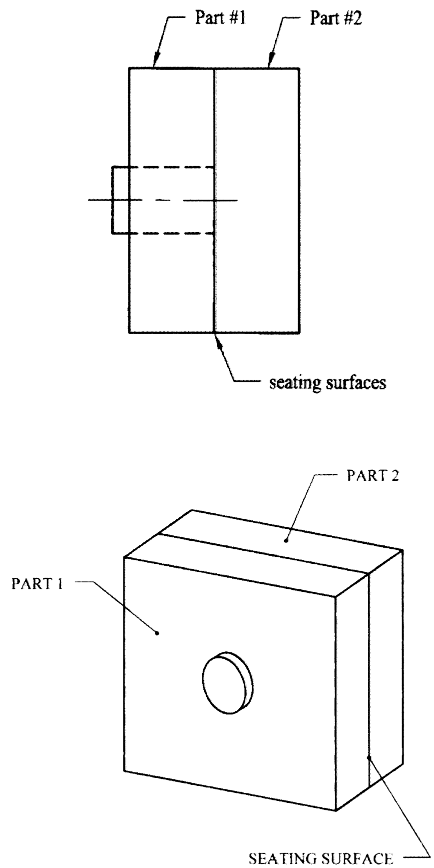
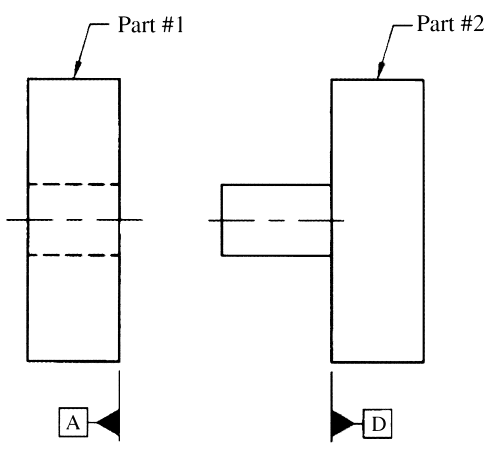


FIGURE 5-7

Therefore, the primary datum features are assigned to both parts.



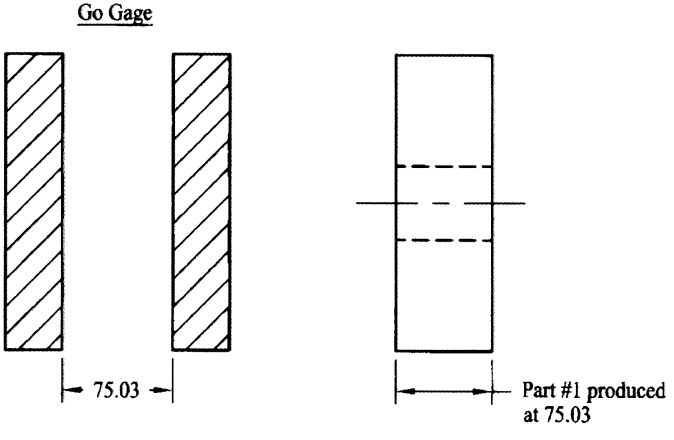
The only letters not used as datum features are I, O and Q. They look too much like numbers. Also, the order of the letters as they are used in the alphabet is unimportant. The only thing of importance is how these datum features will be used (referenced) in the feature control frames (geometric tolerances) on the parts.

We must dwell on these primary datum features and ask the question, "What characteristic of geometry must each have to seat in the assembly without rocking too much?" Since this will be the first control assigned to each part, it cannot be a relationship control. This feature establishes the primary datum feature on each part. It is first; therefore, there is nothing to relate it to. So, it can't be perpendicular or parallel to anything because at this stage of the definition, there is nothing to relate it to. It is first. All we can do is give it a form tolerance.

Since the entire surface on each part seats on the entire surface of the other part, this control must apply to the entire surface. It must be a surface control (3D), not a line element control (2D) like straightness. Since the surface has a planar shape, it must be controlled by flatness. Rule #1 says the size tolerance given to the width of the part in the side view already controls the flatness, straightness and parallelism of both sides.

On Part #1, the size tolerance is 75±0.03. On Part #2, the size tolerance is 100±0.03. Rule #1 says they must each have perfect form if they are produced at MMC (75.03 for Part #1 and 100.03 for Part #2). Only as they depart from the MMC, may they depart from perfect form. For example, on Part #1 the size is verified with a simulation of a GO gage at MMC and at cross sections at LMC. A GO gage could be two parallel rails at 75.03 apart as shown in FIGURE 5-8.

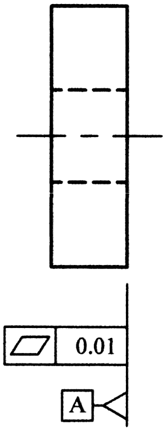
FIGURE 5-8 [GO Gage]



If the part is produced at 75.03 at all cross-sectional measurements, the only way it would (even in theory) fit the GO gage is if it has perfect flatness, straightness and parallelism. But, if the part was produced smaller than 75.03 (MMC), it could be less than perfect in its form by its departure from MMC. Since the LMC of 74.97 is the smallest the part may be and still comply with the size tolerance, the most it can depart from the MMC of 75.03 is 0.06. So, even without a flatness feature control frame, the flatness is controlled by Rule #1 to within 0.06. Any additional flatness control would have to refine (be smaller than) the 0.06 tolerance. Otherwise, it would be meaningless, since the size tolerance would control flatness better. In this case, due to cost, manufacturing capability and functional requirements, the flatness (tolerance) has been assigned as 0.01. For example:

FIGURE 5-9

Step 1

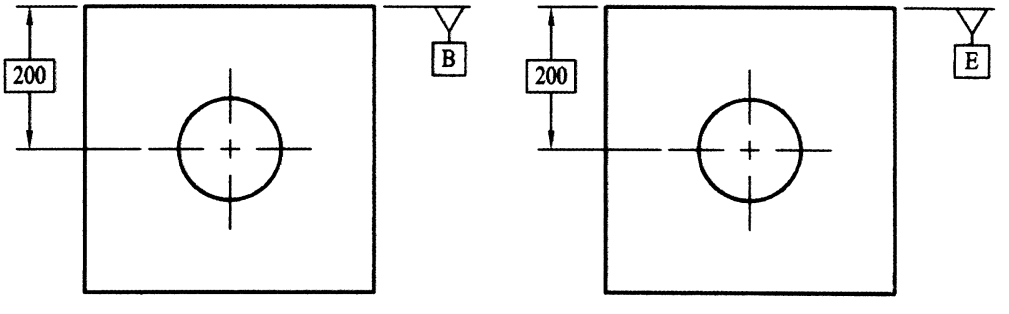


Since both parts, when seated on one another, will dictate how much they might rock when assembled, and the parts are the same size, a flatness tolerance of 0.01 is also assigned to primary datum feature D on Part #2. What we do on one part is commonly done on the mating part. Step 2 makes us choose between the hole and the part edges as the origin of measurement. For this example, we will choose the edges.

Step 2

The secondary datum features on each part will be one of the two sides of the part from which the 200mm dimensions originate. The longer side would, having more surface area, lend more to part stability. The top edge in the front view of both parts is 429-431 millimeters long, about 20 millimeters longer than the left edge (the other candidate). So, the top edge will become the secondary datum feature on both Part #1 and Part #2. It is important that we choose the same edge on both parts, if possible, since we will be aligning these edges during assembly.

FIGURE 5-10

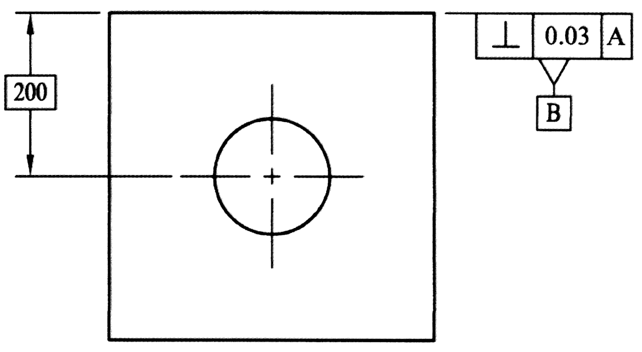


Each of these surfaces labeled datum feature B and datum feature E will need a dimension that is basic (a boxed 200mm dimension) originating at the datum plane (constructed by a minimum of 2 high points of contact on the datum feature surface) and leading us to the hole axis. This basic 200mm dimension will be a target for manufacturing to shoot for. The tolerance, instead of being plus or minus 0.5 on the 200mm dimension, will instead be calculated in Step 4 as a position tolerance that will allow the axis of the hole on Part #1 and the shaft on Part #2 to deviate from the perfect location represented by the 200mm dimensions from the (datum planes formed by the) part's edges.

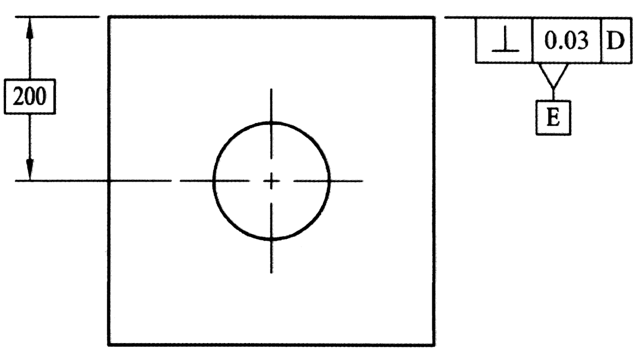
These edges, which are secondary datum features (and in Step 3, tertiary datum features), must be related with a tolerance back to the primary datum. The relationship depicted between the secondary surface and the primary plane on each part is one of 90° or perpendicularity. We could simply allow the general tolerance note of ±1° to tolerance this angle, but in this case, we will apply a more uniform tolerance zone of perpendicularity. It will form 2 parallel planes 0.03 apart. Both planes will be 90° to the primary datum plane (formed by a minimum of 3 high points of contact from the primary datum feature). If the surface is in the tolerance zone, it will be perpendicular to within 0.03mm and it will be flat to within 0.03mm.

FIGURE 5-11 [Step 2]

Part #1

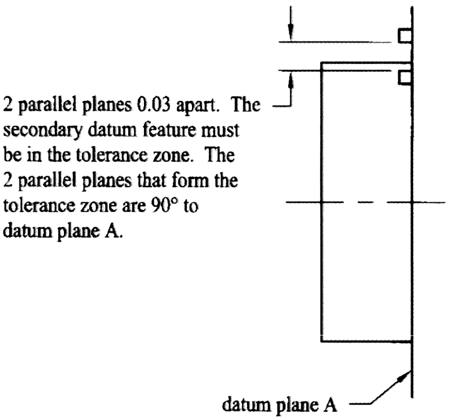


Part #2



On Part #1, the tolerance zone would appear as follows in the side view:

FIGURE 5-12



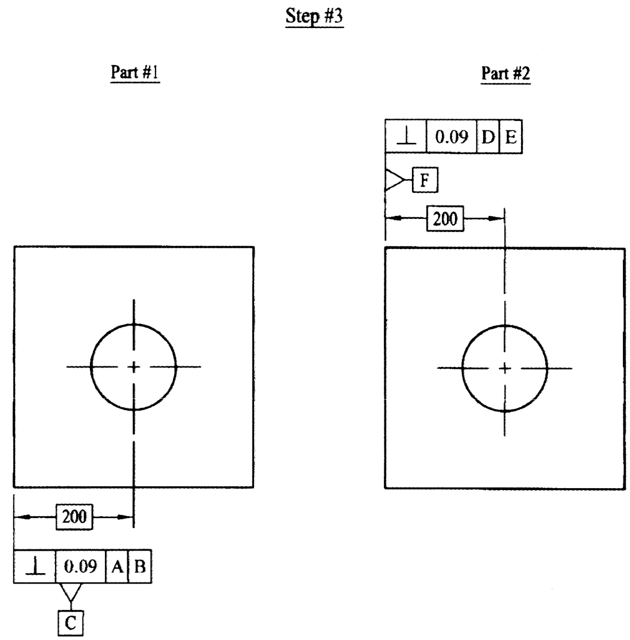
Step 3

Step #3 is to assign the remaining surface from which the other 200 millimeter dimension originates as the tertiary datum feature and to relate it to within a tolerance back to the primary and secondary datum planes. The tolerance of perpendicularity of 0.03 in Step #2 was 3 times the tolerance of flatness on the primary datum feature in Step #1 of 0.01. It is common to increase the tolerances on features to be greater than tolerances on features from which they are measured. So, if datum feature A has a tolerance of 0.01, then for measurement repeatability, we would like the tolerance on datum feature B to be greater.

In this case, 3 times greater is 0.03. Sometimes, for reasons of cost, manufacturing capability or simply by being overridden by a formula for calculating mating part tolerances, this is not feasible. But, when no such condition exists, a good rule of thumb is that the tolerances given to the features from which we measure (such as the primary datum feature which constructs the primary datum plane) should be tighter than that on features being measured from them (such as the secondary datum feature B in this case).

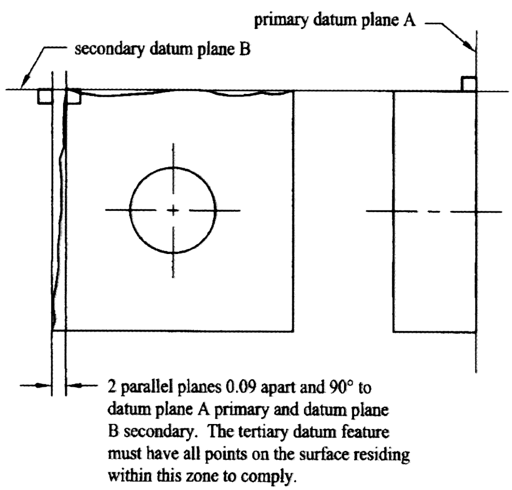
This same rule of thumb would apply to the tertiary datum features on both Part #1 and Part #2. We prefer their tolerances to be greater than those given to the secondary datum features which form the planes to which the tertiary datum features will be related. Since the tertiary datum features are depicted perpendicular (90º) to both the primary and secondary datum features on both Part #1 and Part #2, the tertiary datum features will be given a perpendicularity tolerance 3 times greater than 0.03. They appear as follows:

FIGURE 5-13



The tolerance zones for Step #3 would be two parallel planes 0.09 apart, between which must reside all points on the tertiary datum feature surface. This zone is perpendicular to both the primary and secondary datum planes. On Part #1, the zone would be as follows:

FIGURE 5-14 [Step 3, Part 1 tolerance zone]



As it was done on Step #2, Step #3 also makes certain that the 200 mm dimension is made a basic dimension by placing a box around it and transferring the tolerance (once it is recalculated) into the feature control frame (geometric tolerance) on Step 4.

Step 4

Now that all 3 datum features have been assigned and toleranced, it is time to relate the hole to the 3 mutually perpendicular planes that form the datum reference frame (coordinate system) on Part #1. Likewise, the shaft on Part #2 will be geometrically toleranced to the 3 plane reference system established on Part #2.

Since the relationship to these datum reference planes is one of perpendicularity to the primary planes and distance/location from the secondary and tertiary planes, we choose a geometric tolerance capable of controlling angle and location. The most appropriate control is a position tolerance. A position tolerance must first be calculated. Since the shaft on Part #2 is "fixed" in place, the assembly of Part #1 and Part #2 is known as a "Fixed Fastener" assembly.

The formula for a fixed fastener condition is:

MMC Hole
\- MMC Shaft
Geometric tolerance to be divided between shaft and hole

99 = MMC HOLE
\- 97 = MMC SHAFT
2 = Geometric Tolerance to be divided between Part #1 and Part #2

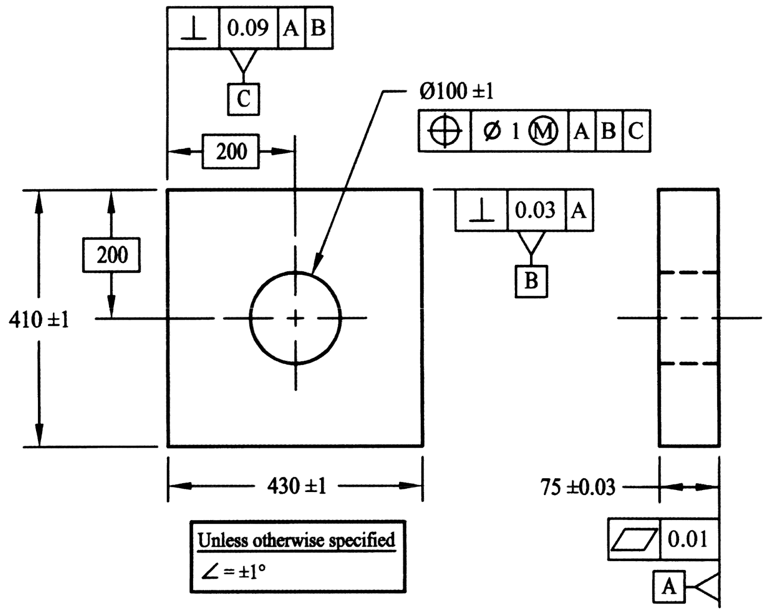
Each part will receive 1 mm of position tolerance if the tolerance is equally divided. If one of the parts was determined to need more position tolerance than the other because of manufacturing difficulty, the 2 mm of geometric tolerance may be unequally divided. In this case, each part is determined to be equally hard to manufacture and will, therefore, be assigned 1 mm of position tolerance. If Part #1 had two holes and Part #2 had two shafts, each pattern of holes or shafts would still receive 1 mm of position tolerance. So, this method holds as true for a pattern of 100 holes and 100 shafts as it does in this situation of 1 hole and 1 shaft.

Since these are mating features, the maximum material condition symbol will be used in the position control to allow a greater position tolerance as the hole is produced larger (and, therefore, mates easier) than its MMC of 99. Likewise, the maximum material condition symbol will be used in the position control to allow a greater position tolerance as the shaft is produced smaller (and, therefore, mates easier) than its MMC of 97.

Step #4 completes the drawing changes and appears with these improvements in the following figure:

FIGURE 5-15 [Part #1 and Part #2]

Part #1



Part #2

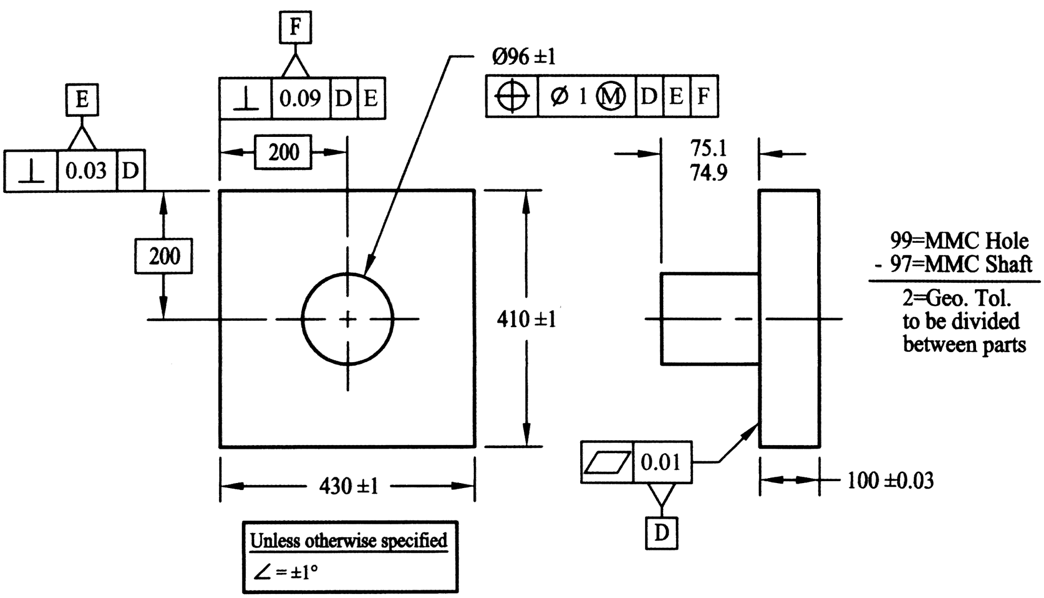
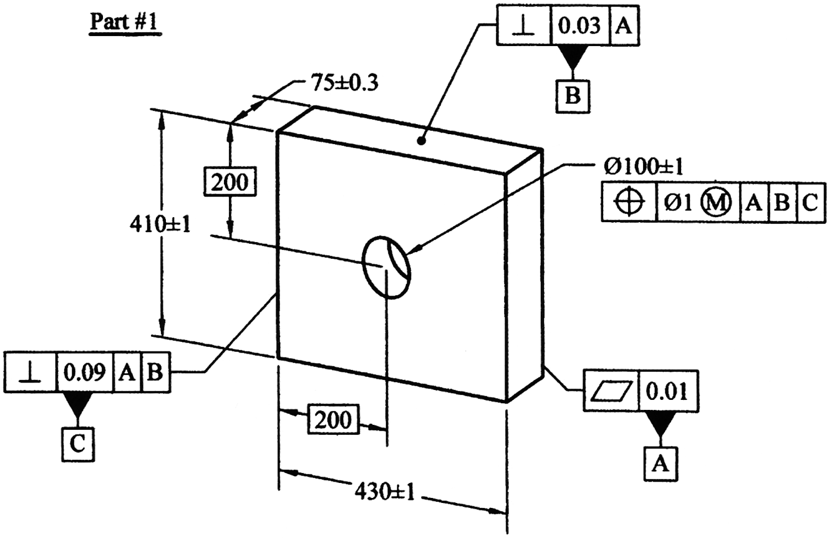
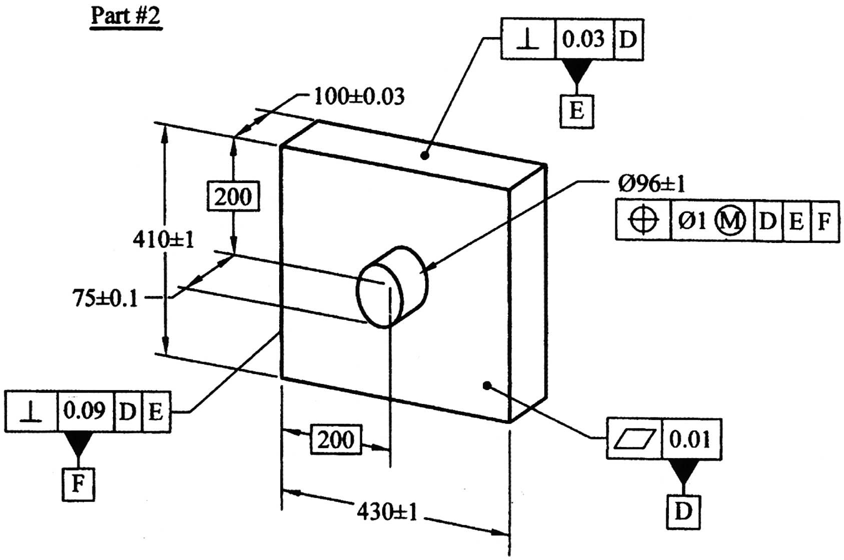


FIGURE 5-16


Part #1
⊥ 0.03 A
B
75±0.3
200
410±1
Ø100±1
⊕ Ø1 Ⓜ A B C
⊥ 0.09 A B
C
200
430±1
⏥ 0.01
A


Part #2
⊥ 0.03 D
E
100±0.03
200
410±1
75±0.1
Ø96±1
⊕ Ø1 Ⓜ D E F
⊥ 0.09 D E
F
200
430±1
⏥ 0.01
D

These feature control frames can all be read as sentences with statements and implications. For example, the position on the hole in Part #1 can be read as follows:

Reading the Feature Control Frame - Part #1

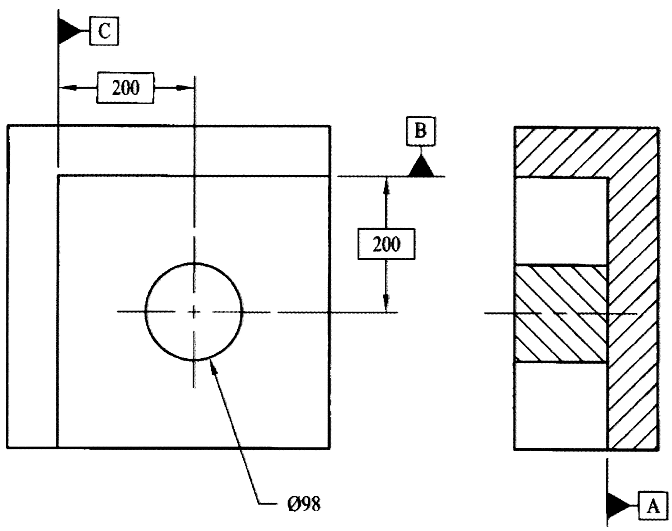
∅99-101 hole

⌖	∅1Ⓜ	A	B	C

⌖ = position
∅ = of the axis of a diameter
1 = must be held to within 1 millimeter
Ⓜ = if produced at maximum material condition (∅99) (implies a mating feature)
A = of perfect perpendicularity to datum plane A
B = and perfect location (200 millimeters) from datum plane B
C = and perfect location (200 millimeters) from datum plane C

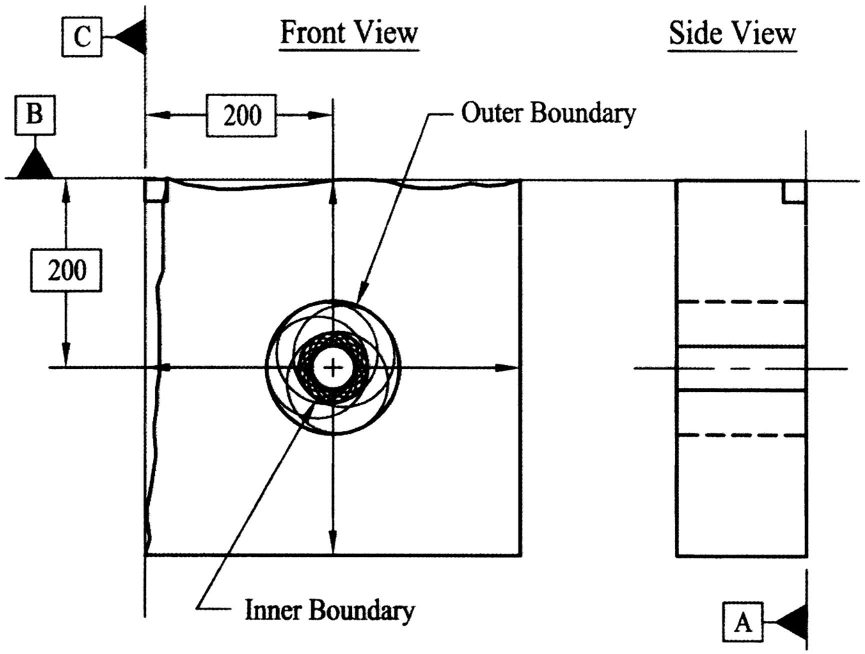
This feature control frame states the hole may move out of position a certain amount if produced at a certain size. In this, if produced at the MMC of 99, the axis may be out of position 1 mm. It also implies that more movement is allowed in direct proportion to the hole's growth from the MMC of 99. This creates an inner boundary that is a constant (constant boundaries such as this are known as virtual conditions) Ø98 and can, therefore, be gaged using a Ø98mm gage pin staged perpendicular to a gaging element representing/simulating datum feature A and located 200mm from gaging elements representing B and C. A simple gage is shown as follows:

FIGURE 5-17 [Functional Gage]
This gage is shown dimensioned, but not toleranced.



In addition to a gageable position inner boundary, a worst-case outer boundary is created that is known as a resultant condition. See the following figure for an explanation and calculation of the inner (virtual condition) boundary and the outer (resultant condition) boundary:

FIGURE 5-18 [Boundaries of Part #1]



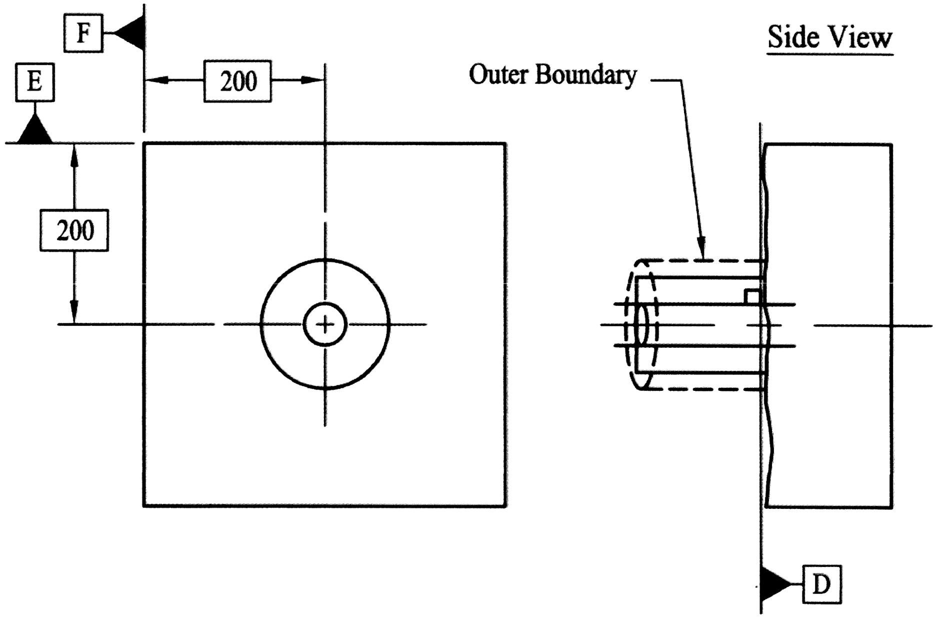
(See Part #2 side view for comparison)

Size		Geometric Tolerance	Boundary	
Ø99	-	1	= Ø98	Inner Boundary
				- worst mating condition
Ø100	-	2	= Ø98	- constant
				- virtual condition
Ø101	-	3	= Ø98	- functional gage pin size
Ø101	+	3	= Ø104	= Outer Boundary

The Outer Boundary, in this case, is known as the Resultant Condition.

Likewise, Part #2 has inner and outer boundaries. See the following figure for an explanation and calculation of these boundaries.

FIGURE 5-19 [Boundaries of Part #2]



see Part #1 for front view

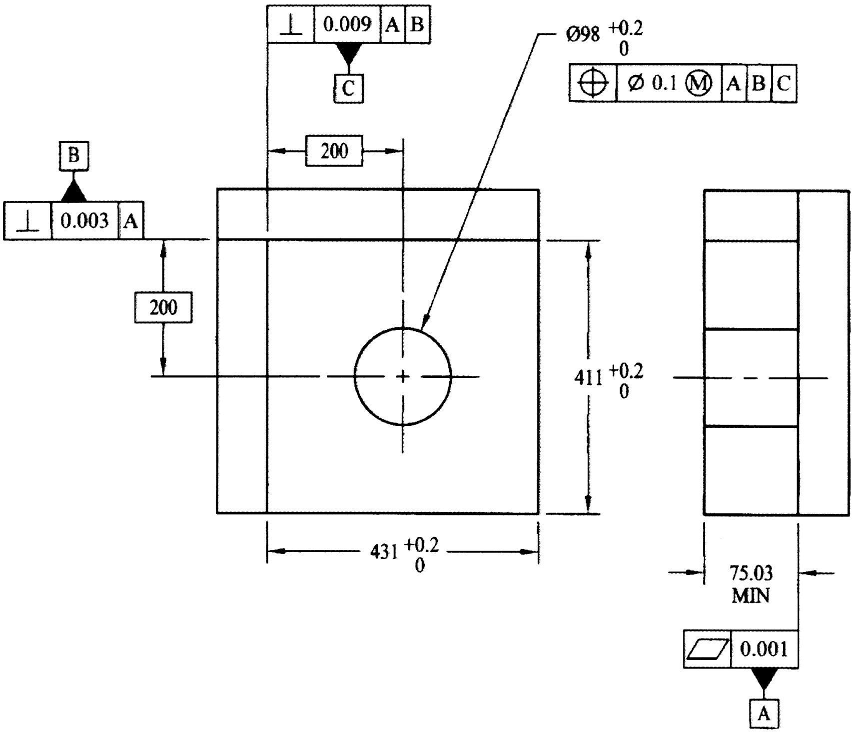
Size	Geometric Tolerance		Boundary	
Ø97	+	1	= Ø98	Outer Boundary - constant - worst mating condition - virtual condition - functional gage hole size
Ø96	+	2	= Ø98	
Ø95	+	3	= Ø98	
Ø95	-	3	= Ø92	=Inner Boundary

The Inner Boundary, in this case, is known as the Resultant Condition.

A gage may be produced to inspect for attribute data (good versus bad only) on the position of this shaft. This will not be illustrated in this unit. Rather a more complete gage for position tolerance of the hole on Part #1 is depicted. The practical absolute tolerancing method for gages per *ASME Y14.43-2003 Dimensioning and Tolerancing Principles for Gages and Fixtures* has been used in this gage design. The tolerances recommended for gages by Y14.43 are between

5% and 10% of the tolerances on the part being gaged. Notice that each tolerance used on the gage in its 4 geometric controls are 10% of the tolerances used on Part #1 for the 4 geometric tolerances on that part. Following the Practical Absolute Methodology for gages, the tolerance on the gage pin is all plus and no minus tolerance for size starting at the virtual condition of the hole being gaged. This is to prevent non-compliant parts from passing the gaging procedure. The goal is to accept no bad parts.

FIGURE 5-20 [Practical Absolute Functional Gage for Part #1]



Bonus Tolerancing Formulas
...for Position, Perpendicularity, Angularity and Parallelism Modified at MMC

HOLE

(Step 1)

Actual Size [max. inscribed cylinder]
-MMC
Bonus Geometric Tolerance

(Step 2)

Bonus Tolerance
+Original Geometric Tolerance
Total Geometric Tolerance

SHAFT

(Step 1)

MMC
-Actual Size [min. circumscribed cylinder]
Bonus Geometric Tolerance

(Step 2)

Bonus Tolerance
+Original Geometric Tolerance
Total Geometric Tolerance

The MMC symbol is allowed where it has been determined that function and fit between mating features of size will not be endangered by the addition of a proportional extra deviation from perfect form, orientation or position as the features depart from MMC (staying within their size limits). The examples given in this section deal with a positional control, but, as stated, the formulae for calculating total geometric tolerance are valid for any appropriate application of the MMC symbol to the regular feature of size being controlled (although with straightness and flatness controls at MMC, actual size is actual local size).

A different situation arises when the MMB symbol is applied to datum features of size. For example:

⌖	⌀0.25	B Ⓜ

or

⊥	0.1	B Ⓜ

You can see in these two controls that we are not worried about the extra tolerance to be gained as the features being controlled depart from MMC because they are controlled closely by the RFS concept. The features get no extra expansion in the size of their geometric tolerance zones as the features are made larger or smaller (within size limits). However, each has a datum feature of size called out at MMB that it is being controlled to. This means that as the datum feature of size departs from its own MMB, an additional shift in the tolerance zone of the feature or pattern of features being controlled to this datum is allowed.

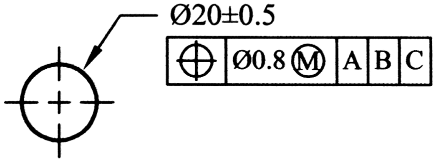
Sometimes, the amount of this additional shift is easy to figure. When the part and feature geometry is very simple (perhaps, for example, one feature controlled to one datum regular feature of size--like a plain surface to a shaft, or a clearance hole to a clearance hole, or even a coaxial situation of position with one shaft diameter controlled to another shaft diameter), the additional shift of the feature's tolerance zone is easy to calculate because it is usually a shift directly proportional to the datum feature's departure from its MMB size. But, in some situations, this allowable shift is more difficult to visualize. The more complicated part geometries (the more features and datum features of size involved) make it somewhat more difficult for a floor inspector to determine the effect this allowable shift has on part acceptance without the use of receiver (functional) gaging.

The use of receiver gages makes the calculation of bonus tolerance and allowed shift of tolerances unnecessary for part acceptance or rejection. The gage automatically does these calculations by either accepting or rejecting the part. However, it must be noted that unless the functional receiver type gages are soft gages (computer generated in software), the physical gage gives only attribute data (good vs. bad) and does not give variables data (how good or bad and why). Also, Coordinate Measuring Machines are getting better at determining datum feature shift (a.k.a. pattern shift) because of software improvements.

[**Note:** Paper gaging has been used by some inspection departments as a useful, inexpensive tool to augment their inspection procedures. When used correctly, it can, in many instances, be used in place of receiver gages. Paper gaging is used in conjunction with open set-ups (variables data collectors, such as probes and indicators). Paper gaging is simply a term used to describe that collected measurement data has been graphed out. This gives a visual display of how much and in what direction part features have deviated from perfect.]

The following charts show examples of how to calculate bonus tolerances.

FIGURE 5-21 [Examples of Calculating Bonus Tolerances]



IF A HOLE

	Actual	Less MMC	Bonus Equals	Original Pos. Tol.	Total
A.	20.5	19.5	1	0.8	1.8
B.	20.3	19.5	0.8	0.8	1.6
C.	20.1	19.5	0.6	0.8	1.4
D.	20	19.5	0.5	0.8	1.3
E	19.8	19.5	0.3	0.8	1.1
F.	19.6	19.5	0.1	0.8	0.9
G.	19.5	19.5	0	0.8	0.8

IF A SHAFT

	MMC	Less Actual	Bonus Equals	Original Pos. Tol.	Total
H.	20.5	19.5	1	0.8	1.8
I.	20.5	19.6	0.9	0.8	1.7
J.	20.5	19.8	0.7	0.8	1.5
K.	20.5	20	0.5	0.8	1.3
L.	20.5	20.1	0.4	0.8	1.2
M.	20.5	20.3	0.2	0.8	1
N.	20.5	20.5	0	0.8	0.8

For a ∅**25-26mm shaft**, the completed table below shows the allowable total tolerance using the following formula: **Bonus + Original Tolerance = Total Tolerance.**

FIGURE 5-22

Actual Feature Size	26	25.8	25.6	25.4	25.1
⌖ | Ø0.5 Ⓜ | A | B | C	A 0.5	B 0.7	C 0.9	D 1.1	E 1.4
// | Ø0.4 Ⓜ | A	F 0.4	G 0.6	H 0.8	I 1	J 1.3
⌖ | Ø0.5 | A | B | C	K 0.5	L 0.5	M 0.5	N 0.5	O 0.5
⊥ | Ø0 Ⓜ | A	P 0	Q 0.2	R 0.4	S 0.6	T 0.9
⌖ | Ø0.2 Ⓜ | L | D | C	U 0.2	V 0.4	W 0.6	X 0.8	Y 1.1
— | Ø0.3 Ⓜ	Z 0.3	AA 0.5	BB 0.7	CC 0.9	DD 1.2
⌭ | 0.05	EE 0.05	FF 0.05	GG 0.05	HH 0.05	II 0.05
⌖ | Ø0.08 Ⓜ | J | C | E	JJ 0.08	KK 0.28	LL 0.48	MM 0.68	NN 0.98

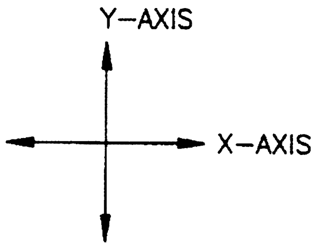
Allowed vs. Actual Deviation from True Position

The allowed deviation from true position is the size of the controlled feature's geometric tolerance zone. In the case of those features with cylindrical tolerance zones, the allowed deviation is given on the basis of a diametrical tolerance zone constructed about the true position axis. Allowed deviation takes into consideration the original tolerance given in the feature control frame plus any additional (bonus) tolerance drawn from the feature size.

This bonus tolerance is allowed for those holes or shafts using the MMC symbol. As they depart from MMC toward LMC, extra tolerance for the allowed deviation from true position is gained. In other words, the tolerance grows. The actual axis of the hole or shaft as produced must lie within this cylindrical tolerance zone which is the allowed deviation from true position (diameter basis) that is acceptable.

The actual deviation from true position may be determined by an open set-up. Coordinate Measuring Machines, optical and video comparators, height gages, gage pins, indicators and other inspection equipment may be used to locate the actual hole or shaft axis. Once this is found, other calculations are required. First, measuring in a straight line from the location datums, it must be determined how far the feature axis has been produced from its true position. This deviation must be found first measuring along the X-axis, then along the Y-axis from the datum planes.

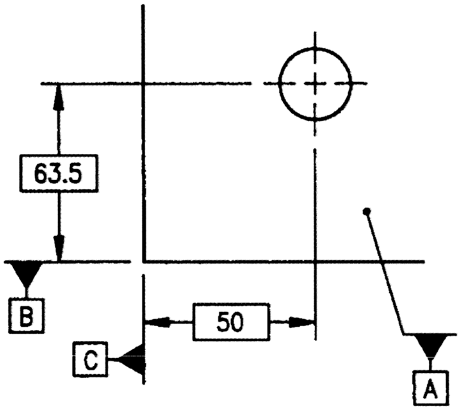
FIGURE 5-23



For example, if the feature's true position is drawn as in FIGURE 5-24, we know exactly where true position is.

After the part is produced, we must determine how far the actual hole axis is from datum C. Let's say we've checked and found the maximum deviation point of the axis is 50.12 from datum C. This is a deviation of 0.12 from true position along the X-axis. Checking along the Y-axis from datum B, we find a maximum deviation point of 63.4 or 0.1 from true position. If we use those figures (0.12 and 0.1) in the hole's actual deviation from true position radially, we get the following:

FIGURE 5-24



$$\text{Radial Deviation from} \oplus = \sqrt{x^2 + y^2}$$
$$= \sqrt{0.12^2 + 0.1^2}$$
$$= \sqrt{0.0144 + 0.01}$$
$$= \sqrt{0.0244}$$
$$= 0.1562049$$

But since we are really interested in how large a diameter would have to be drawn around true position to encompass the actual hole axis, we must multiply this answer by two. The following formula is more appropriate for our needs.

$$\text{Diametrical Deviation from} \oplus = 2\sqrt{x^2 + y^2}$$
$$= 2\sqrt{0.12^2 + 0.1^2}$$
$$= 0.3124098$$

Note: The formula for calculating the axis deviation from true position of a **spherical diameter** is:

$$2\sqrt{x^2 + y^2 + z^2}$$

FIGURE 5-25 [Inch Conversion Chart: Coordinate Measurement to Calculate the Diameter of the Actual Deviation from True Position (Z)]

INCH CONVERSION CHART

Coordinate Measurement to Calculate the Diameter of the Actual Deviation from True Position (Z)

.020	.0400	.0402	.0404	.0408	.0412	.0418	.0424	.0431	.0439	.0447	.0456	.0466	.0477	.0488	.0500
.019	.0380	.0382	.0385	.0388	.0393	.0398	.0405	.0412	.0420	.0429	.0439	.0449	.0460	.0472	.0484
.018	.0360	.0362	.0365	.0369	.0374	.0379	.0386	.0394	.0403	.0412	.0422	.0433	.0444	.0456	.0469
.017	.0340	.0342	.0345	.0349	.0354	.0360	.0368	.0376	.0385	.0394	.0405	.0416	.0428	.0440	.0453
.016	.0321	.0322	.0325	.0330	.0335	.0342	.0349	.0358	.0367	.0377	.0388	.0400	.0412	.0425	.0439
.015	.0301	.0303	.0306	.0310	.0316	.0323	.0331	.0340	.0350	.0360	.0372	.0384	.0397	.0410	.0424
.014	.0281	.0283	.0286	.0291	.0297	.0305	.0313	.0322	.0333	.0344	.0356	.0369	.0382	.0396	.0410
.013	.0261	.0263	.0267	.0272	.0278	.0286	.0295	.0305	.0316	.0328	.0340	.0354	.0368	.0382	.0397
.012	.0241	.0243	.0247	.0253	.0260	.0268	.0278	.0288	.0300	.0312	.0325	.0339	.0354	.0369	.0384
.011	.0221	.0224	.0228	.0234	.0242	.0250	.0261	.0272	.0284	.0297	.0311	.0325	.0340	.0356	.0372
.010	.0201	.0204	.0209	.0215	.0224	.0233	.0244	.0256	.0269	.0283	.0297	.0312	.0328	.0344	.0360
.009	.0181	.0184	.0190	.0197	.0206	.0216	.0228	.0241	.0254	.0269	.0284	.0300	.0316	.0333	.0350
.008	.0161	.0165	.0171	.0179	.0189	.0200	.0213	.0226	.0241	.0256	.0272	.0288	.0305	.0322	.0340
.007	.0141	.0146	.0152	.0161	.0172	.0184	.0198	.0213	.0228	.0244	.0261	.0278	.0295	.0313	.0331
.006	.0122	.0126	.0134	.0144	.0156	.0170	.0184	.0200	.0216	.0233	.0250	.0268	.0286	.0305	.0323
.005	.0102	.0108	.0117	.0128	.0141	.0156	.0172	.0189	.0206	.0224	.0242	.0260	.0278	.0297	.0316
.004	.0082	.0089	.0100	.0113	.0128	.0144	.0161	.0179	.0197	.0215	.0234	.0253	.0272	.0291	.0310
.003	.0063	.0072	.0085	.0100	.0117	.0134	.0152	.0171	.0190	.0209	.0228	.0247	.0267	.0286	.0306
.002	.0045	.0056	.0072	.0089	.0108	.0126	.0146	.0165	.0184	.0204	.0224	.0243	.0263	.0283	.0303
.001	.0028	.0045	.0063	.0082	.0102	.0122	.0141	.0161	.0181	.0201	.0221	.0241	.0261	.0281	.0301
	.001	.002	.003	.004	.005	.006	.007	.008	.009	.010	.011	.012	.013	.014	.015

Deviation in X or Y Axis

Equation: $Z = 2\sqrt{X^2 + Y^2}$

z=the diameter of actual deviation from true position of the feature axis.

x=radial deviation from true position along the axis.

y=radial deviation from true position along y axis.

FIGURE 5-26

METRIC CONVERSION CHART

Coordinate Measurement to Calculate the Diameter of the Actual Deviation from True Position (Z)

Ø POSITION ZONE (Z)

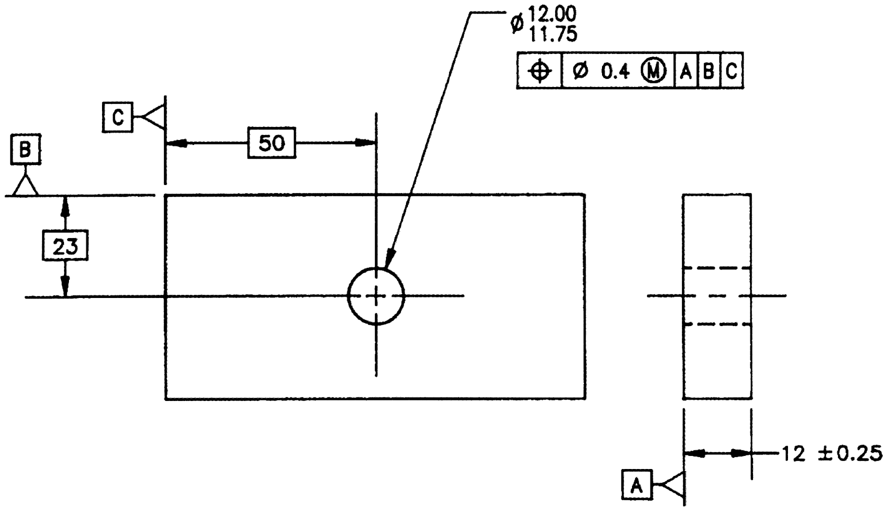
Y																									
0.50	1.001	1.003	1.007	1.013	1.020	1.028	1.038	1.050	1.063	1.077	1.093	1.109	1.127	1.146	1.166	1.187	1.209	1.232	1.256	1.281	1.306	1.332	1.359	1.386	1.414
0.48	0.961	0.963	0.967	0.973	0.981	0.990	1.000	1.012	1.025	1.040	1.056	1.073	1.092	1.111	1.132	1.154	1.176	1.200	1.224	1.250	1.276	1.302	1.330	1.358	1.386
0.46	0.921	0.923	0.928	0.934	0.941	0.951	0.962	0.974	0.988	1.003	1.020	1.038	1.057	1.077	1.098	1.121	1.144	1.168	1.193	1.219	1.246	1.273	1.301	1.330	1.359
0.44	0.881	0.884	0.888	0.894	0.902	0.912	0.923	0.936	0.951	0.967	0.984	1.002	1.022	1.043	1.065	1.088	1.112	1.137	1.163	1.189	1.217	1.245	1.273	1.302	1.332
0.42	0.841	0.844	0.849	0.855	0.863	0.874	0.885	0.899	0.914	0.930	0.948	0.967	0.988	1.010	1.032	1.056	1.081	1.106	1.133	1.160	1.188	1.217	1.246	1.276	1.306
0.40	0.801	0.804	0.809	0.816	0.825	0.835	0.848	0.862	0.877	0.894	0.913	0.933	0.954	0.977	1.000	1.024	1.050	1.076	1.103	1.131	1.160	1.189	1.219	1.250	1.281
0.38	0.761	0.764	0.769	0.777	0.786	0.797	0.810	0.825	0.841	0.859	0.878	0.899	0.921	0.944	0.968	0.994	1.020	1.047	1.075	1.103	1.133	1.163	1.193	1.224	1.256
0.36	0.721	0.724	0.730	0.738	0.747	0.759	0.773	0.788	0.805	0.824	0.844	0.865	0.888	0.912	0.937	0.963	0.990	1.018	1.047	1.076	1.106	1.137	1.168	1.200	1.232
0.34	0.681	0.685	0.691	0.699	0.709	0.721	0.735	0.752	0.769	0.789	0.810	0.832	0.856	0.881	0.907	0.934	0.962	0.990	1.020	1.050	1.081	1.112	1.144	1.176	1.209
0.32	0.641	0.645	0.651	0.660	0.671	0.684	0.699	0.716	0.734	0.755	0.777	0.800	0.825	0.850	0.877	0.905	0.934	0.963	0.994	1.024	1.056	1.088	1.121	1.154	1.187
0.30	0.601	0.605	0.612	0.621	0.632	0.646	0.662	0.680	.0700	0.721	0.744	0.768	0.794	0.821	0.849	0.877	0.907	0.937	0.968	1.000	1.032	1.065	1.098	1.132	1.166
0.28	0.561	0.566	0.573	0.582	0.595	0.609	0.626	0.645	0.666	0.688	0.712	0.738	0.764	0.792	0.821	0.850	0.881	0.912	0.944	0.977	1.010	1.043	1.077	1.111	1.146
0.26	0.522	0.526	0.534	0.544	0.557	0.573	0.591	0.611	0.632	0.656	0.681	0.708	0.735	0.764	0.794	0.825	0.856	0.888	0.921	0.954	0.988	1.022	1.057	1.092	1.127
0.24	0.482	0.487	0.495	0.506	0.520	0.537	0.556	0.577	0.600	0.625	0.651	0.679	0.708	0.738	0.768	0.800	0.832	0.865	0.899	0.933	0.967	1.002	1.038	1.073	1.109
0.22	0.442	0.447	0.456	0.468	0.483	0.501	0.622	0.544	0.569	0.595	0.622	0.651	0.681	0.712	0.744	0.777	0.810	0.844	0.878	0.913	0.948	0.984	1.020	1.056	1.093
0.20	0.402	0.408	0.418	0.431	0.447	0.466	0.488	0.512	0.538	0.566	0.595	0.625	0.656	0.688	0.721	0.755	0.789	0.824	0.859	0.894	0.930	0.967	1.003	1.040	1.077
0.18	0.362	0.369	0.379	0.394	0.412	0.433	0.456	0.482	0.509	0.538	0.569	0.600	0.632	0.666	0.700	0.734	0.769	0.805	0.841	0.877	0.914	0.951	0.988	1.025	1.063
0.16	0.322	0.330	0.342	0.358	0.377	0.400	0.425	0.453	0.482	0.512	0.544	0.577	0.611	0.645	0.680	0.716	0.752	0.788	0.825	0.862	0.899	0.936	0.974	1.012	1.050
0.14	0.283	0.291	0.305	0.322	0.344	0.369	0.396	0.425	0.456	0.488	0.522	0.556	0.591	0.626	0.662	0.699	0.735	0.773	0.810	0.848	0.885	0.923	0.962	1.000	1.038
0.12	0.243	0.253	0.268	0.288	0.312	0.339	0.369	0.400	0.433	0.466	0.501	0.537	0.573	0.609	0.646	0.684	0.721	0.759	0.797	0.835	0.874	0.912	0.951	0.990	1.028
0.10	0.204	0.215	0.233	0.256	0.283	0.312	0.344	0.377	0.412	0.447	0.483	0.520	0.557	0.595	0.632	0.671	0.709	0.747	0.786	0.825	0.863	0.902	0.941	0.981	1.020
0.08	0.165	0.179	0.200	0.226	0.256	0.288	0.322	0.358	0.394	0.431	0.468	0.506	0.544	0.582	0.621	0.660	0.699	0.738	0.777	0.816	0.855	0.894	0.934	0.973	1.013
0.06	0.126	0.144	0.170	0.200	0.233	0.268	0.305	0.342	0.379	0.418	0.456	0.495	0.534	0.573	0.612	0.651	0.691	0.730	0.769	0.809	0.849	0.888	0.928	0.967	1.007
0.04	0.089	0.113	0.144	0.179	0.215	0.253	0.291	0.330	0.369	0.408	0.447	0.487	0.526	0.566	0.605	0.645	0.685	0.724	0.764	0.804	0.844	0.884	0.923	0.963	1.003
0.02	0.057	0.089	0.126	0.165	0.204	0.243	0.283	0.322	0.362	0.402	0.442	0.482	0.522	0.561	0.601	0.641	0.681	0.721	0.761	0.801	0.841	0.881	0.921	0.961	1.001
X	0.02	0.04	0.06	0.08	0.10	0.12	0.14	0.16	0.18	0.20	0.22	0.24	0.26	0.28	0.30	0.32	0.34	0.36	0.38	0.40	0.42	0.44	0.46	0.48	0.50

FORMULA: $Z = 2\sqrt{X^2 + Y^2}$

Once computed, this actual diametrical deviation from true position must be compared against the allowed deviation. If the allowed deviation (or actual tolerance zone size) is larger than the actual diametrical deviation, the feature position is acceptable. If not, the feature must be reworked or rejected. Reworking is sometimes possible. For example, if a hole is not already made at LMC, it may be opened up (drilled larger). If the hole is modified with a MMC symbol, this procedure will enlarge the allowed deviation from true position (tolerance zone size).

The control given on the figure below can, if examined closely, explain not only how to calculate allowed additional positional tolerance but also why it is allowed. In the inspection of a part, we are often trying to simply discern whether or not the produced part or parts will function. If so, we accept them. If not, rejection or rework of the part is recommended.

FIGURE 5-27



In order to make a good judgment as to the capability of a part to function, one would like to know how the part is to be used. Sometimes, the inspector has only the design drawing from which to work. It is unfortunate when one is not given an intimate knowledge of features to be inspected. But, the reality is that this is often the case. The inspector must read the drawing as though it tells a story about the needed characteristics for conformance in order to function. If the drawing is well done, the datums and feature control frames can, indeed, give the inspector the information necessary to make good judgments about the part and also to make valid recommendations to manufacturing regarding adjustments to part processing to improve these desired characteristics.

If we read the aforementioned drawing and focus on the symbols/components of the positional control, it can tell the story of what is expected of the controlled hole. It says in part, "Position of a diameter that mates while the part is seated on surface A and the hole is located from B and C." Of course, it could be read in the more traditional manner of, "The axis may be out of position a diameter of 0.4 if the hole is produced at maximum material condition holding a relationship of perpendicularity to primary datum plane A, distance from secondary datum plane B and distance from tertiary datum plane C."

But to decode the function of the hole, the first reading and its subsequent logical implications are probably more helpful. If we derive from the positional control that we are positioning a hole that mates while A seats with a three point high point contact and we locate/measure the hole from planes B and C, we must then ask a series of questions. "If the round hole mates, what does it mate with?" Without a quantum leap in logic, we may speculate that a round hole mates with a round shaft. We might then ask, "What is the largest shaft that, in theory, could enter this hole at the desired angle and location from the datums if the hole was made within its limits of size and also out of perpendicularity and location to the listed datums the maximum amount allowed at that size?"

The answer is to be found by simply subtracting the allowed geometric tolerance from the produced hole size in each instance. If done, one finds that a constant boundary of virtual condition (MMC concept) has been protected on each hole produced. This boundary is perfectly cylindrical, perfectly oriented to datum plane A, and perfectly located from datum planes B and C. The job of the hole is to stay outside of this boundary. The mating pin/shaft is designed, dimensioned and toleranced (for size and position) to reside within this boundary.

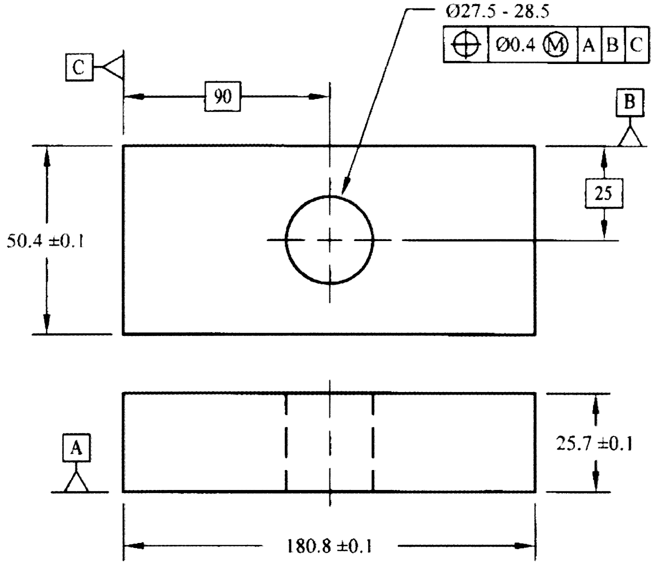
If this is discovered to be the case in parts that have been produced, the inspector accepts the parts with a fair amount of confidence that he has proven that the inspected features will work/assemble. In order to determine the boundary size on the parts under discussion, one may go to the columns labeled actual hole size (maximum inscribed cylinder) and column A, which is the allowed diameter of the deviation from true position, and subtract these two numbers. For example:

	Actual Hole Size		**A**		
(1)	∅11.75	-	0.40	=	∅11.35
(2)	∅11.80	-	0.45	=	∅11.35
(3)	∅11.90	-	0.55	=	∅11.35
(4)	∅12.00	-	0.65	=	∅11.35

The mating boundary remained constant even though the size of the produced hole and geometric positional tolerance allowed changes. The inspector is saying, "If the hole does not violate this boundary, it will mate with the worst case mating shaft if it also does not violate the boundary." If the hole resides outside of the boundary and the shaft resides inside of the boundary (which has its center/axis at true position), no interference of material will occur. Therefore, the parts will assemble and should be passed on by the inspector to assembly.

The following illustration shows a part as specified, then being measured as produced. It is first judged as a good part using the method of verifying that the virtual condition boundary has not been violated. Then, it is measured and judged to be within its positional tolerance zone. These methods are the two most common in verifying positional tolerance.

FIGURE 5-28 [Tolerance Zone vs. Boundary Verification]

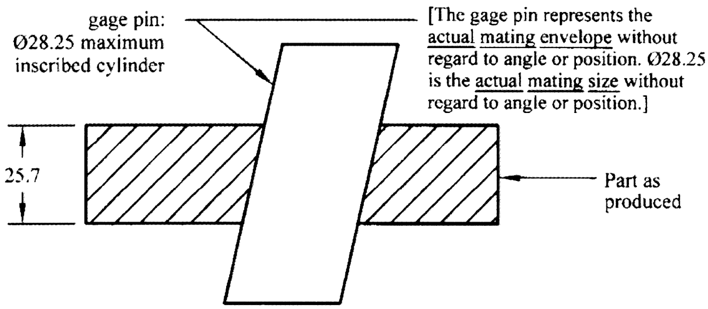


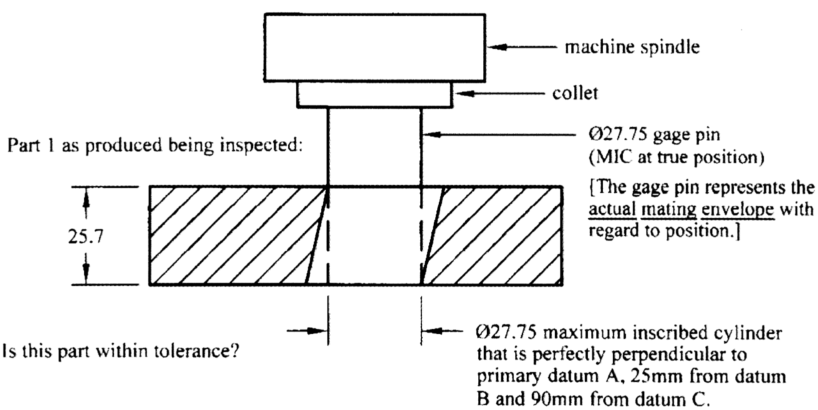
27.5 MMC
- 0.4
① 27.1 Virt. Cond. to protect
② 27.75 protected
③ Good Part

AND

① 28.25 MIC
-27.50 MMC
0.75 Bonus Geo. Tol.
+ 0.40 Orig. Geo. Tol.
Ø 1.15 Total Geo. Tol.
② 28.25 MIC
-27.75 MIC at ⌖
Ø 0.50 out of ⌖
③ Good Part

Part 1 as produced being inspected:





1. Is this part within tolerance?

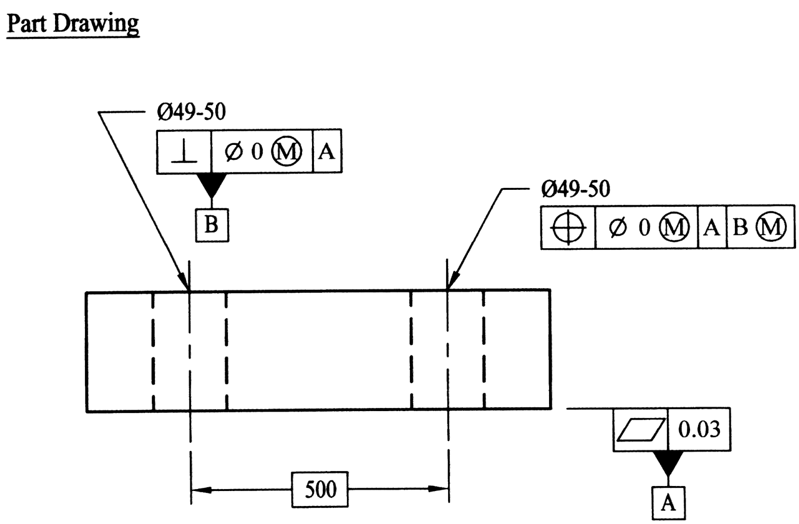
A Difference between Bonus Tolerance (Growth) and Datum Feature Shift (Movement) of Tolerance Zones

One of the most often asked questions in measurement and also in tolerancing is, "Can we take tolerance from the datum feature referenced at MMB and give it as tolerance zone growth to the features being measured from that datum?" The simple answer to that question is, "No!"

Granted, there are some isolated cases where this strategy might work out, but many more cases where it will not. Certainly, for a pattern of holes referenced to a datum regular feature of size (such as one hole), as the datum regular feature of size departs from its virtual condition (Maximum Material Boundary concept), that pattern of holes may shift as a group an additional amount. This apparent shift of the pattern of holes is actually a movement of the datum feature axis away from its imaginary datum axis. But it will appear as though the entire pattern of holes has shifted/moved. This concept is thoroughly explained in other sections of this book.

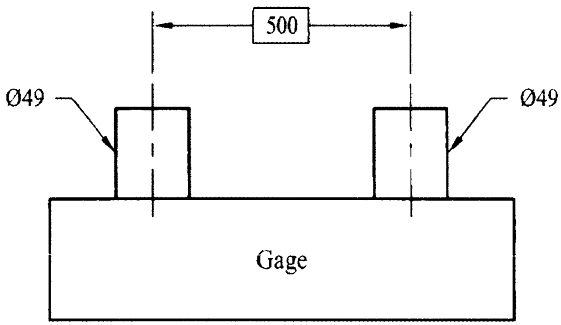
In this section, let's explore a situation that is very simple: one hole positioned to two datums. The planar primary datum will serve the purpose of perpendicularity control, while the secondary datum feature will be a hole which generates an axis that will be used to hold a 500 millimeter distance. So, datum A will be for perpendicularity and B will be for location in the following illustration.

FIGURE 5-29 [Part Drawing]



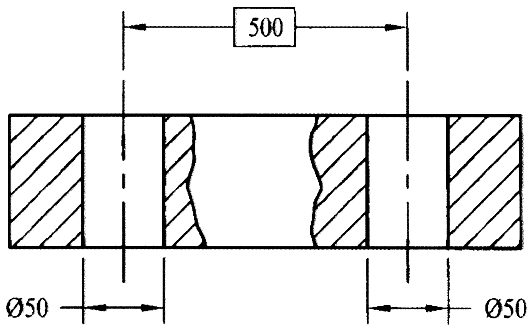
Gages can help us understand this concept very well. Think of gages as displaying the physical embodiment of the theory.

FIGURE 5-30 [Gage]



Let's start with looking at a part produced with perfect perpendicularity and perfect location and holes at ∅50.

FIGURE 5-31 [Part as Produced with perfect perpendicularity and perfect location]

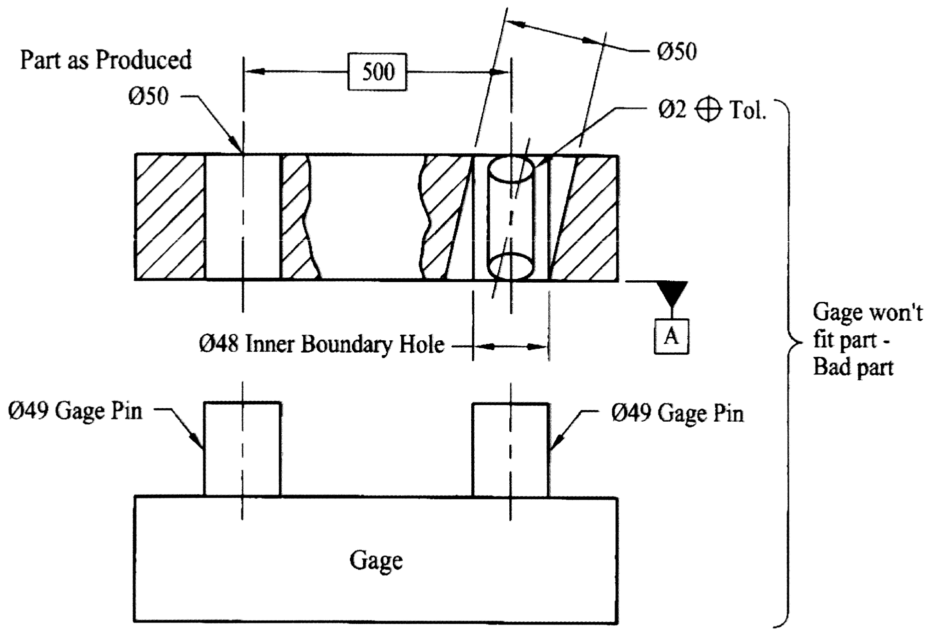


That type of produced part would seem to fit the gage no matter what we did with the clearance between the gage pins (∅49) and the part's produced holes (∅50).

50 LMC		50 LMC
-49 Virtual Cond.		-49 Virtual Cond.
∅1 Allowed ⌖ Tol.	+	∅1 Allowed ⌖ Tol. = ∅2 Total Tolerance Allowed between Holes

But, in fact, if we simply took this ∅2mm of tolerance and gave it to the hole being positioned, the part would not fit the gage if the hole used that tolerance to allow it to be out-of-perpendicularity.

FIGURE 5-32 [Part and Gage]



The functional gage will not fit into the hole being positioned and seat with the required three high point minimum contact on datum feature A. This proves the part does not comply with the position requirement.

The ∅2mm of tolerance available must be assigned where it was derived—one millimeter to datum feature B and one millimeter to the hole being positioned. Robbing tolerance from datum feature B and giving it to the hole being positioned doesn't work in this very simple situation, where there is only one hole being positioned.

This problem becomes much worse when it is a pattern of holes being positioned. Increasing the position tolerance of each hole by the growth of the datum hole would allow every hole in the pattern a greater movement away from every other hole. If this was allowed, there is no way the part would fit the gage or the assembly.

In the simple example depicted below, the following illustrations show correct distributions of the tolerances that would allow parts to pass the gage.

FIGURE 5-33 [Tolerance Zones]