TECHNICAL TRAINING CONSULTANTS
856 SALT LAKE COURT SAN JOSE, CA 95133 (408) 251-5329

Zero Positional Tolerance at Maximum Material Condition[1]

Which has more tolerance, the drawing in Fig. 7-17A—a typical plus or minus tolerance for clearance holes—or the drawing in Fig. 7-17B—a zero positional tolerance at MMC? It is often assumed that a zero in the feature control frame means that there is no tolerance. This misconception occurs because the meaning of the MMC modifier in the feature control frame is not clearly understood.

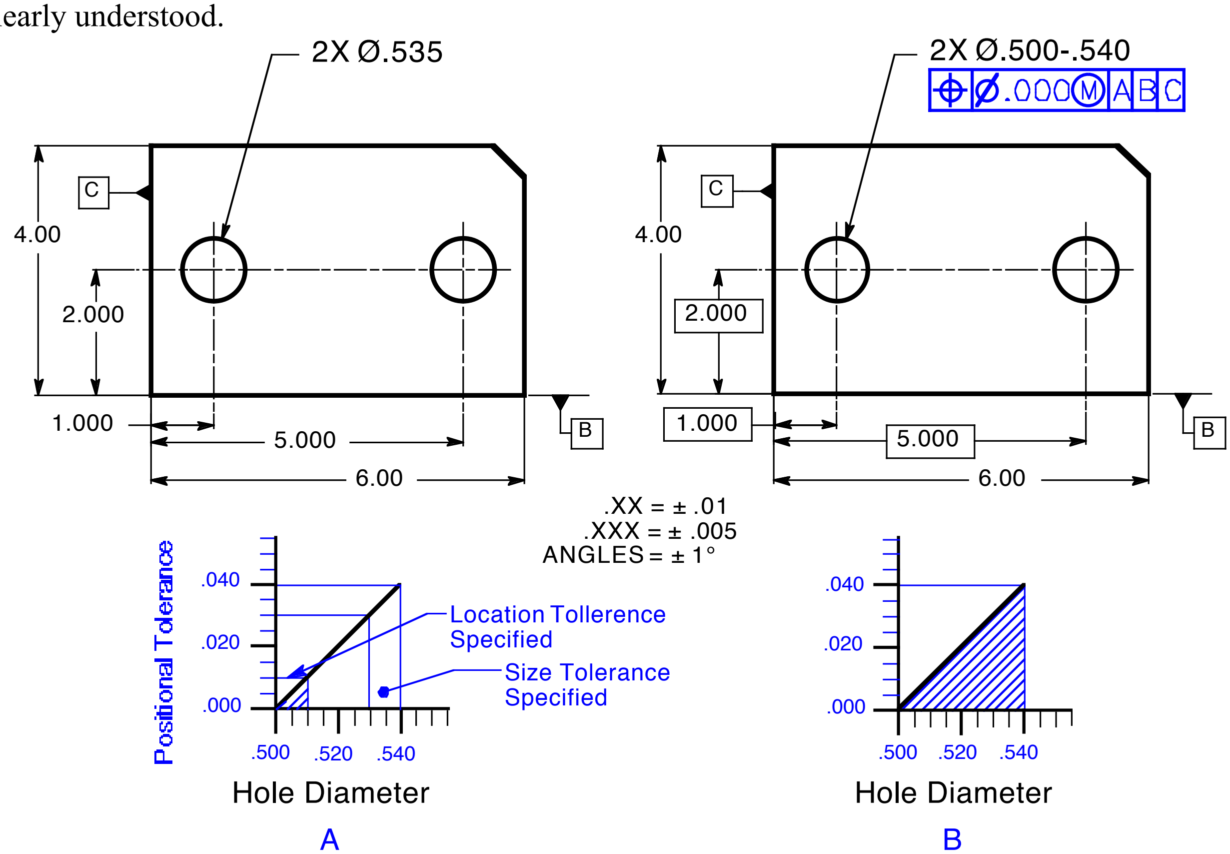

Figure 7-17 Plus or minus location tolerance compared with zero positional tolerance.

Zero tolerance is never used without an MMC or LMC modifier. Zero at RFS would, in fact, be a zero tolerance no matter what size the feature is manufactured. Where zero positional tolerance at MMC is specified, the tolerance that applies to the feature being controlled is the bonus tolerance. In many cases, the bonus is larger than the tolerance that otherwise might be specified in the feature control frame. An analysis of the part in Fig. 7-17B indicates that the holes can be produced anywhere between .500 and .540 in diameter. If the holes are actually produced at a diameter of .535, the total location tolerance available is a cylindrical tolerance zone of .035 in diameter.

[1]Cogorno, Gene R., ***Geometric Dimensioning and Tolerancing for Mechanical Design, Second Edition***, McGraw-Hill, New York, 2011, p. 121

The actual hole size, .535, minus the MMC, .500, equals a bonus tolerance of .035. GD&T reflects the exact tolerance available. For the drawing in Fig. 7-17A, the size of the holes must be between a diameter of .530 and .540. If the holes are produced at .535 in diameter, the total location tolerance available is actually a cylindrical tolerance zone of .035, just as it is in Fig.7-17B. However, because the title block tolerance is specified at ± .005, the inspector can accept the part only if the axes of the holes fall within their respective tolerance zones, which are .010 square. In this case, a tolerance of at least .025 is wasted. Tolerance is money, and no one wants to waste money!

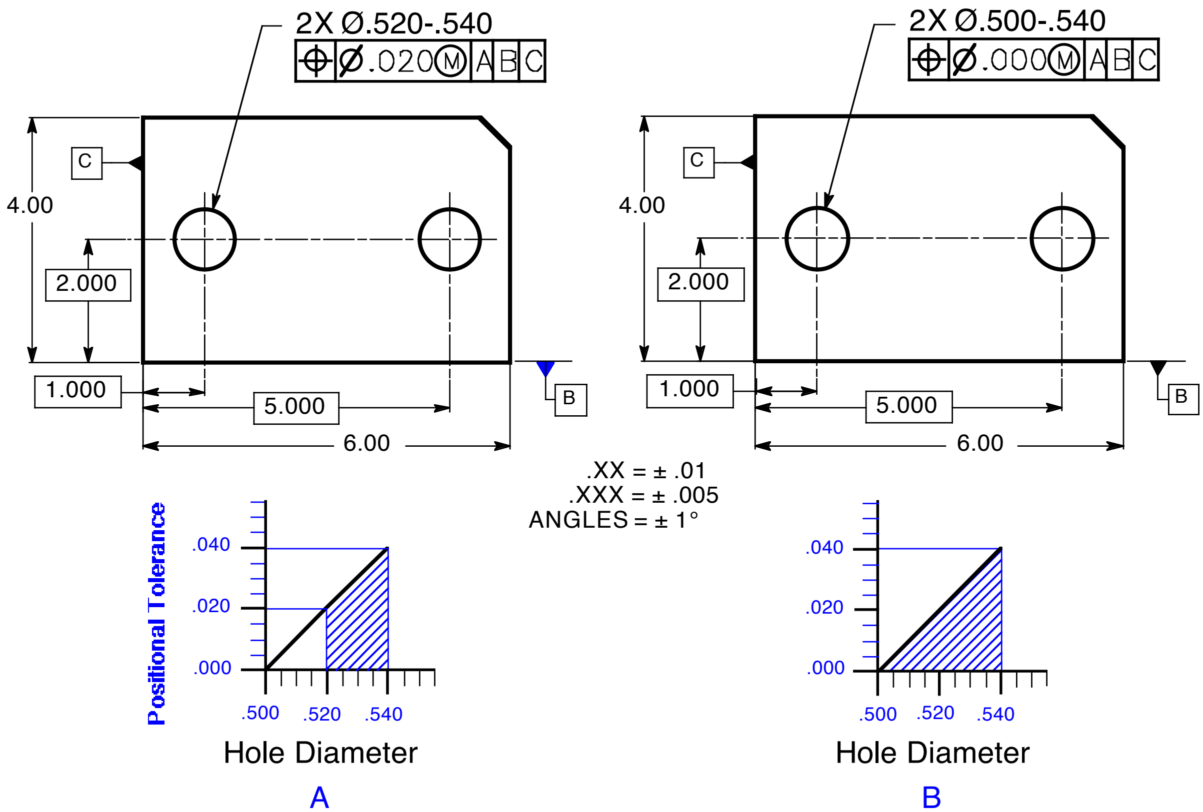

Figure 7-18 A specified position tolerance compared with zero positional tolerance.

The two parts in Fig. 7-18 are identical; they are just toleranced differently. If a part is made with the holes produced at .535 in diameter, what is the total location tolerance and the virtual condition for these holes? See Table 7-3 for the calculations. For a given hole size, the total tolerance and the virtual condition are the same whether a numerical tolerance or a zero tolerance at MMC is specified, but the range of the hole size is increased when zero positional tolerance is used.

Some engineers don't use zero positional tolerancing at MMC because they claim that the manufacturing staff will not understand it. Consequently, they put some small number such as .005 in the feature control frame with a possible .015 or .020 bonus tolerance available. If machinists cannot read the bonus, they will produce the part within the tolerance of .005 specified in the feature control frame and charge the client company for the tighter tolerance. If zero positional tolerance is used, suppliers will either know what it means, ask what it means, or not bid on the part. Actually, machinists who understand how to calculate bonus tolerance really like the flexibility this technique gives them. Inspection can easily accept more parts, reducing manufacturing costs.

Total Positional Tolerance

	Drawing **A**	Drawing **B**
Actual Hole Size	.535	.535
Maximum Material Condition	– .520	– .500
Bonus	.015	.035
Geometric Tolerance	+ .020	+ .000
Total Positional Tolerance	**.035**	**.035**

Virtual Condition

	Drawing **A**	Drawing **B**
Maximum Material Condition	.520	.500
Geometric Tolerance	– .020	– .000
Virtual Condition	**.500**	**.500**

Table 7-3 Both the total positional tolerance and the virtual condition are the same whether controlled with a numerical tolerance or zero positional tolerance at MMC

Suppose that a part is to be inspected with the drawing in Fig. 7-18*A*. The part has been plated a little too heavily, and the actual size of both holes is .518 in diameter. The inspector has to reject the part because the holes are too small. Suppose that both holes were located within a cylindrical tolerance zone of .010. Would the part assemble? The answer to this question can be determined by inspecting the part to the equivalent zero positional tolerance – the drawing in Fig. 7-18*B*. The hole size of .518 in diameter is acceptable because it falls between .500 and .540 in diameter. The location tolerance equals the bonus, which is the hole size, .518, minus MMC, .500, or a cylindrical tolerance zone .018 in diameter. The part will fit and function because only a location tolerance of .010 in diameter is required. Is it acceptable to scrap perfectly good parts? If this is a continuing problem for a particular part, submit an engineering change order converting the tolerance to a zero positional tolerance.

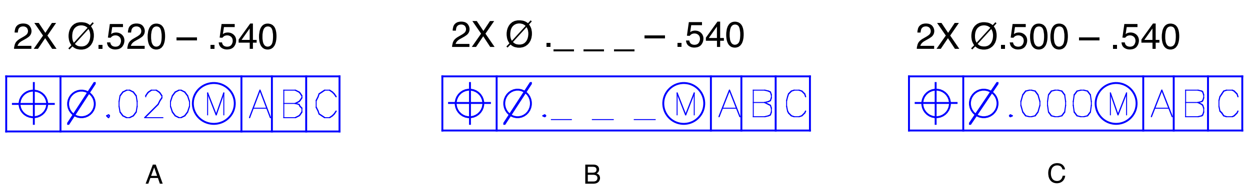

Figure 7-19 Converting a positional tolerance of a hole to a zero positional tolerance

Converting the .020 positional tolerance for the holes in Fig. 7-19*A* to a zero positional tolerance, Fig. 7-19*C*, is fairly simple. The only numbers to be changed are the MMC and the geometric tolerance, shown as blanks in Fig. 7-19*B*. The tolerance in the feature control frame is always converted to zero at MMC. A circle M symbol must follow the tolerance. Then convert the MMC of the feature to its virtual condition. In this case, the .520 MMC minus .020 geometric tolerance equals the virtual condition of .500 in diameter.

Zero tolerance is not used where the tolerance applies at RFS or where no bonus tolerance is available, as in a tolerance specified for threads or press-fit pins.