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

Zero Positional Tolerance at MMC[1]

As I stated in the text, "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."

Some engineers don't use zero positional tolerancing at MMC because they say that the manufacturing staff will not understand it. Consequently, they put some small value such as .005 in the feature control frame with a possible .035 bonus tolerance available. (See Fig. 7-18A.) If machinists cannot read the bonus, they will produce the part within the .005 tolerance specified in the feature control frame ignoring any bonus 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 tolerancing technique offers them. Manufacturers can easily reduce costs by accepting more parts.

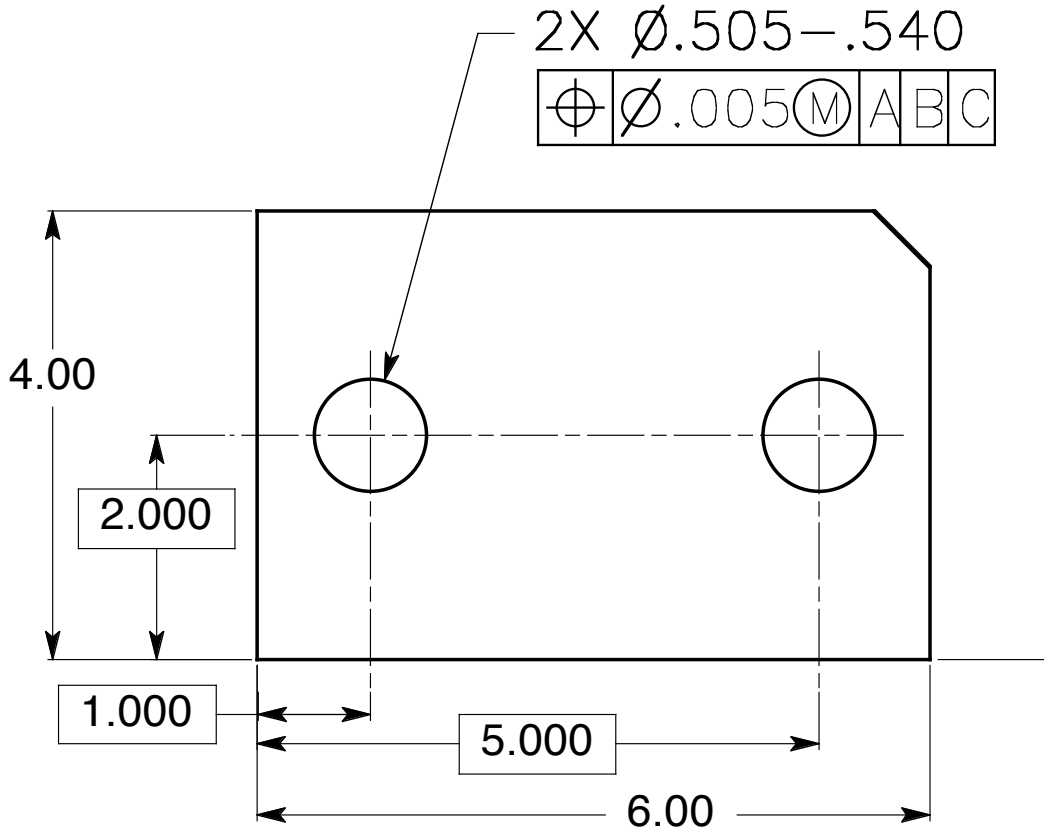

Figure 7-18A A large size tolerance and small location tolerance at MMC

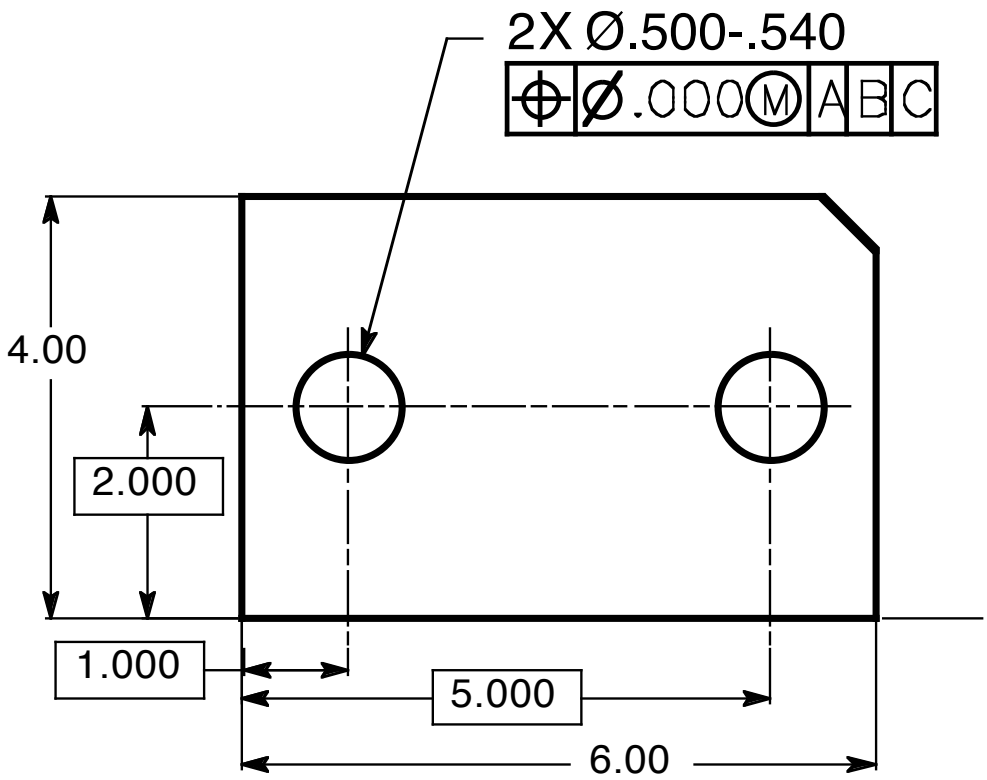

Figure 7-18B A large size tolerance and zero positional tolerance at MMC

Tolerancing often involves misunderstanding and prejudice. Consequently tolerancing is frequently based on past experience and/or a gut feeling. Engineers have told me, "Zero positional tolerancing just doesn't feel right."

Zero positional tolerancing at MMC is like a salesman who works for no salary but gets paid a commission for everything he sells. The bigger the sale, the greater the commission. In the case of zero positional tolerancing, MMC is the commission. The larger the hole, the greater the location tolerance.

Some designers erroneously believe that if they tolerance a clearance hole with zero positional tolerance at MMC such as the one in Fig. 7-18B, the machinist might produce a hole exactly .500 in diameter providing .000 location tolerance. That would be like the salesman selling his product at cost with nothing left for himself. First of all, it is very difficult for a machinist to produce a hole exactly .500 in diameter. Secondly, why would a machinist produce a hole at MMC when there is a .040 size tolerance available? Most machinists will try to produce features of size in the middle of the size tolerance range. Finally, even if the machinist does produce the hole at or near MMC, the hole must be within the location tolerance, the bonus, provided by the difference between the actual mating envelope (the hole diameter) and the maximum material condition hole size. If the hole does not fall within the location tolerance generated by the condition stated above, the hole location is out of tolerance. It is up to the machinist to produce features that satisfy the required

specifications. That is, the hole must be large enough to provide the location tolerance required.

Zero positional tolerancing is a win, win, win situation. It is easy for engineers to use, machinists like the flexibility it provides, and manufacurers save time and money by reducing scrap and rework.

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