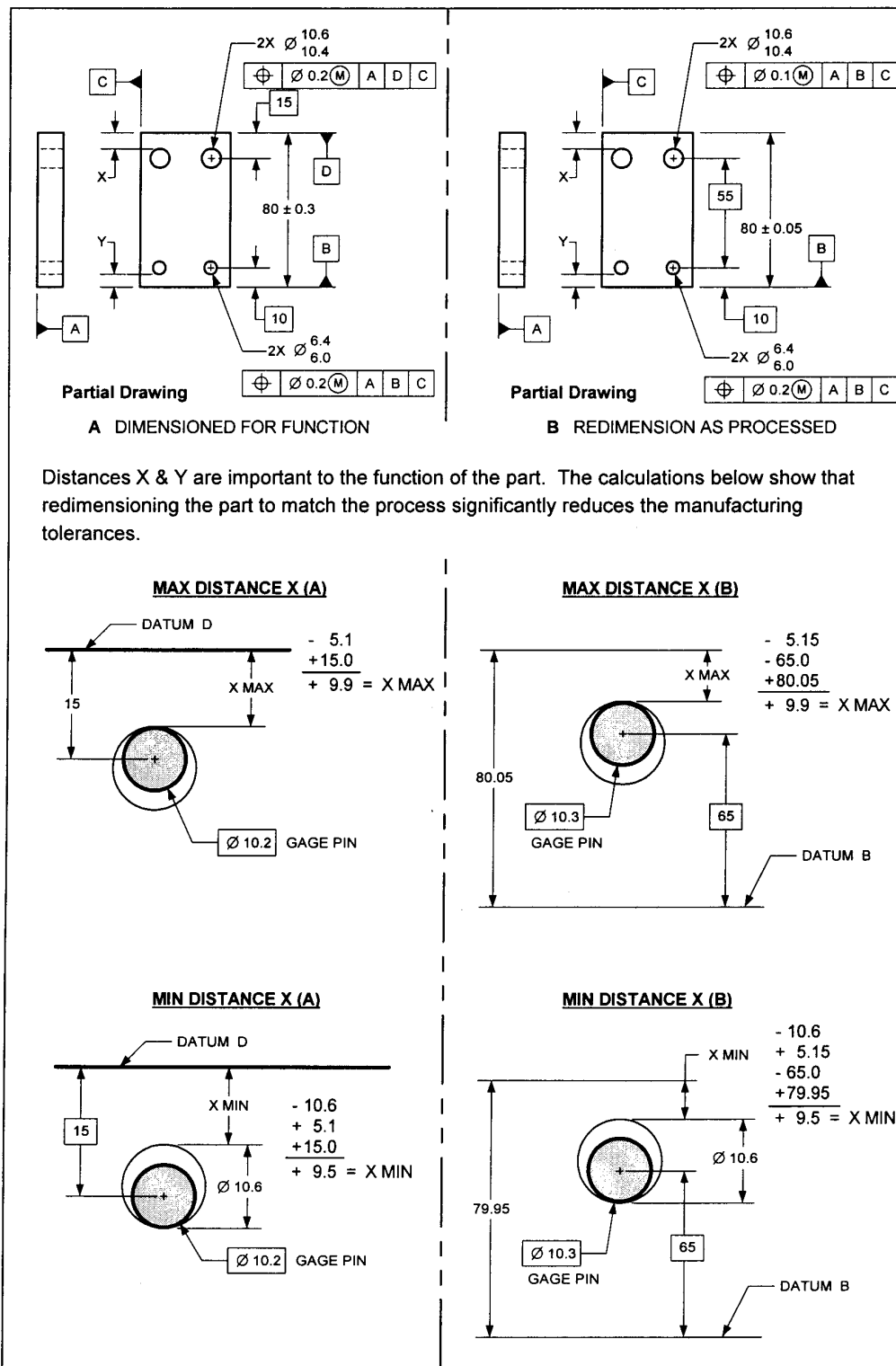


The misunderstanding of datums existing on the part often results from two conditions. One is a lack of understanding of the datum system. The other is a lack of understanding (or disbelief in) the functional dimensioning philosophy. Datum planes and axes do not exist on the part. Datums are theoretical and are simulated by the inspection equipment. Datum features exist on the part. Misunderstanding the difference between datums and datum features can lead to errors in specifying datum targets and inspection of parts.

The second misconception is that datum features should be chosen by the process engineer. This misunderstanding results from a lack of understanding, or belief in the functional dimensioning philosophy. Datum features should be part features which either mount and locate the part in its assembly or are involved in communicating a functional relationship for a dimension.

In some cases, the datum features specified on the part (based on function) are not convenient for use during processing of the part. The process engineer is then faced with the choice of using different surfaces to manufacture the part than will be used to inspect the part, or request to revise the datum feature specifications to represent the manufacturing process. Both choices have shortcomings.

If the datum features are revised to reflect the manufacturing process, it destroys the functional relationships of the dimensions. The dimensions lose their relationship to the customer requirements. On the surface, it appears the part is easier to produce. However, rearranging datum specifications to match the manufacturing process often results in less tolerance for manufacturing as well. To demonstrate this, let's look at the example in Figure 13-7.



13-7 Functional Datum vs. Redimensioned for Process

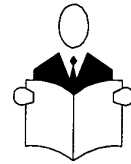
In Figure 13-7A, the part has four holes: two sets of holes at each end of the part. One set of holes is dimensioned from datum B and the other set is dimensioned from datum D. The function of the part requires that each set of holes be held in relation to the edge of the part they are nearest to. Distances X and Y are important to the function of the part. If the part is processed using the datums shown, distance X would be 9.5 - 9.9 and distance Y would be 6.5 - 7.1. This tolerance range results from the position callouts that control each set of holes to their respective datums. The resulting distances X and Y are acceptable to the function of the part. To achieve the hole relationship from each end of the part would require the part to be processed by placing it in a fixture twice. This is not a desirable condition for manufacturing.

Let's look at what happens if the part is redimensioned to optimize the process. In Figure 13-7B, the process uses different surfaces on which to locate during the manufacture of the part. Then the process and inspection will originate from one set of surfaces, while the functional requirement will originate from another set of surfaces. Also, since the functional relationship of the holes to the edges of the piece must be maintained, there is considerably less tolerance available for the manufacture of the part. In order to maintain the distance X as defined above, manufacturing tolerances would have to be reduced to allow for the tolerance stack. When all four holes are produced from datum B, then distance X is the result of two tolerances: the 80 ± 0.3 overall height and the positional tolerance for hole location. In order to maintain distance X of 9.5 - 9.9, the tolerance on the 80 dimension would be reduced to ± 0.05 (over an 80% reduction!) and the position tolerance is reduced from 0.2 to 0.1.

The drawing should specify the datums based on function. Removing datum D from the drawing and revising the position callout to reference datum B (as shown in Figure 13-7B) does not help manufacturing or engineering. It requires tighter tolerances for manufacturing and masks the functional requirements for engineering. Instead of revising the datum scheme on the part, it is better to make adjustments to the process plan. Two options exist.

One option to revise the process plan is to use two operations, one for each set of holes. The cost of the extra handling should be offset by the savings that result from the additional tolerances. If the savings do not offset the cost of the extra handling of the part, then use the following option.

Leave the datums on the drawings as shown to reflect the functional requirements of the part. Continue to process the part as planned, using different surfaces for the process than will be used for inspection. This is not a desirable condition, but it is sometimes unavoidable.



Author's Comment

Although it isn't mandatory, it is advantageous to use the functional datums for processing. Whenever a functional relationship is divided into two or more sub-dimensions to accommodate manufacturing, the sub-dimensions will have less tolerance. Also, functional parts may be scrapped.