Sandia Corporation

REPRINT

HOW POSITIONAL TOLERANCING CLARIFIES
DESIGN INTENT AND REDUCES PRODUCT COST

by

E S Roth

JANUARY 1960
DISCLAIMER

This report was prepared as an account of work sponsored by an agency of the United States Government. Neither the United States Government nor any agency thereof, nor any of their employees, makes any warranty, express or implied, or assumes any legal liability or responsibility for the accuracy, completeness, or usefulness of any information, apparatus, product, or process disclosed, or represents that its use would not infringe privately owned rights. Reference herein to any specific commercial product, process, or service by trade name, trademark, manufacturer, or otherwise does not necessarily constitute or imply its endorsement, recommendation, or favoring by the United States Government or any agency thereof. The views and opinions of authors expressed herein do not necessarily state or reflect those of the United States Government or any agency thereof.
DISCLAIMER

Portions of this document may be illegible in electronic image products. Images are produced from the best available original document.
LEGAL NOTICE

This report was prepared as an account of Government sponsored work. Neither the United States, nor the Commission, nor any person acting on behalf of the Commission:

A. Makes any warranty or representation, expressed or implied, with respect to the accuracy, completeness, or usefulness of the information contained in this report, or that the use of any information, apparatus, method, or process disclosed in this report may not infringe privately owned rights; or

B. Assumes any liabilities with respect to the use of, or for damages resulting from the use of any information, apparatus, method, or process disclosed in this report.

As used in the above, "person acting on behalf of the Commission" includes any employee or contractor of the Commission, or employee of such contractor, to the extent that such employee or contractor of the Commission, or employee of such contractor prepare, disseminates, or provides access to, any information pursuant to his employment or contract with the Commission, or his employment with such contractor.
HOW POSITIONAL TOLERANCING CLARIFIES DESIGN INTENT
AND REDUCES PRODUCT COST

by

E. S. Roth

January 1960
<table>
<thead>
<tr>
<th>TABLE OF CONTENTS</th>
<th>Page</th>
</tr>
</thead>
<tbody>
<tr>
<td>Introduction</td>
<td>4</td>
</tr>
<tr>
<td>The Ambiguous Bilateral System</td>
<td>5</td>
</tr>
<tr>
<td>The Positional Tolerance</td>
<td>7</td>
</tr>
<tr>
<td>The Maximum Material Condition Concept (The Bonus Tolerance)</td>
<td>8</td>
</tr>
<tr>
<td>Dictating Inspection Criteria</td>
<td>9</td>
</tr>
<tr>
<td>Paper Gaging Parts by Open Setup Inspection</td>
<td>10</td>
</tr>
</tbody>
</table>
Introduction

Companies that design, manufacture, and inspect their products in one location do not require a complete dimensioning system. Their designers are in immediate personal contact with tool and inspection departments and may constantly supplement product drawings with verbal instructions. Such personal guidance insures tool and gage compatibility, interchangeable parts and a final product that meets design requirements.

Companies with separated design, manufacturing, and inspection functions must rely almost entirely on the product drawing, as the designer may never be available for consultation. Incomplete product definition drawings, in this case, will cause disparity between drawings, production tooling, and inspection. These companies need legally binding drawings that accurately present complete engineering information. The bilateral or coordinate dimensioning system fails the decentralized company in this respect as it is incomplete, condones arbitrary inspection methods, and even implies square tolerance zones that are impossible to gage precisely. Furthermore, gages designed to inspect bilaterally dimensioned parts will consistently allow varying tolerances between part features, and as a result, will often accept parts that would be rejected by open setup inspection. Decentralized companies are currently forced to supplement their drawings with excessive specifications, and with consultations and correspondence, to stabilize design information.

Great Britain, Canada, and the United States recognized the inadequacy of the bilateral system during World War II, and all three countries had published standards defining a more complete positional tolerancing technology by 1954. Both the British and Canadian standards are national in scope and are practically identical. By 1957, three partially different standards (MIL-Std-8A, ASA, and SAE) were being published in this country. An International Drawing Standards Organization, the American-British-Canadian (ABC) Committee, has also officially adopted the positional tolerancing technology.

This paper will initially discuss the ambiguous bilateral system and then explain positional tolerancing, the bonus tolerance available through the maximum material concept, and how positional tolerancing dictates inspection criteria. It will close with a new concept called "paper gaging." Please bear in mind that this paper presents only a basic and partial explanation of a dimensioning technology that is currently increasing in scope and use.
This paper will investigate bilateral dimensioning by asking specific questions about a seemingly uncomplicated design.

Figure 1 illustrates a bilaterally dimensioned plate, including a rectangular pattern of four clearance holes which the designer wants to assemble to another identical plate. (The clearance-hole sizes have been intentionally omitted.) The designer wants to fasten both parts with four 0.500-inch diameter bolts and insists upon 100 per cent interchangeability. (The following answers to Questions 1 through 4 represent a cross section of those received by the author from engineers, tool and gage designers, draftsmen, and production men, during class instruction presented to approximately 1500 students.)

**Question 1:** What is the datum in Figure 1?

**Answers:**
- a. The lower lefthand corner of the part
- b. The intersection of any two edges
- c. The bottom surface
- d. Any corner of the part
- e. The high points on the bottom surface
- f. The high points on two surfaces
- g. The center of the part
- h. The lower lefthand hole
- i. Any two holes.

The starting point (datum) seems to be a personal determination.

Note that the answers to the following question eliminate the holes as a datum.

**Question 2:** What tool or gage design would you use to hold and locate the part shown in Figure 1 while either drilling or gaging the four-hole pattern?

**Answers:**
- a. Figure 2
- b. Figure 3
- c. Figure 4
- d. Figure 5.

Note that none of these tool or gage designs select the same datum, and that the actual starting point may vary drastically as the part varies in form. The datum should be a specific starting point for dimensions and measurements. If the part were fabricated with a tool similar to Figure 4, and then inspected with a gage similar to Figure 5, the datum might conceivably vary by 0.080.
Question 3: What minimum size clearance holes will insure 100 per cent interchangeability?

Question 4: What size gage pins (four fixed gage pins located on nominal, neglecting gagemakers' tolerances) would you use to inspect the clearance holes for location? Both answer series are placed side by side since the gage pin sizes were usually based on the clearance-hole size.

**Answers:**

<table>
<thead>
<tr>
<th>Minimum Diameter Clearance Hole Required</th>
<th>Nominal Gage Pin Diameter Required</th>
</tr>
</thead>
<tbody>
<tr>
<td>a. 0.510</td>
<td>0.500</td>
</tr>
<tr>
<td>b. 0.520</td>
<td>0.500</td>
</tr>
<tr>
<td>c. 0.528</td>
<td>0.508</td>
</tr>
<tr>
<td>d. 0.530</td>
<td>0.510</td>
</tr>
<tr>
<td>e. 0.540</td>
<td>0.520</td>
</tr>
<tr>
<td>f. 0.550</td>
<td>0.530</td>
</tr>
<tr>
<td>g. 0.556</td>
<td>0.536</td>
</tr>
</tbody>
</table>

Note that the majority of the answers indicate that gage designers subtract the total tolerance (0.020) from the minimum diameter clearance hole to determine nominal gage pin size. Also note that designers often specify clearance holes larger than actually needed, hoping to increase part acceptance rates. But the above trend indicates that, as the clearance holes increase in size, so do the gage pins, since the gage designer attempts to check the hole location to the bilateral tolerance. The designer who merely opens up clearance-hole size, without also increasing the tolerance between the holes, defeats his own purpose. Please note that none of the gages check the square tolerance zones implied by the bilateral system.

Another obvious fact about the gage answers to question 4 is that they are more restrictive than open setup inspection. An inspector would allow each hole center to fall within a 0.020 square tolerance zone. Standard gage designs reject all hole centers that fall outside of a circular tolerance zone inscribed within the 0.020 square tolerance zone. Here is proof that we do have fundamental disparity between gaging and open setup inspection with bilateral dimensioning. (See Figure 6.)

Our seemingly uncomplicated design is inherently ambiguous. We could pose additional questions, but they would only serve to further modify Figure 1. The bilateral system is obviously incomplete, ambiguous, and impossible to gage accurately. It does not accurately transmit engineering design intent. Verbal instructions are certainly required to resolve the specific problems we have uncovered. The bilateral system merely outlines the part requirements and actually allows inspection to accept or reject to arbitrary criteria. The disturbing result: as inspection criteria change, so does the design.
The remainder of this paper will examine the new positional tolerancing technology. Let us now accurately determine the minimum size clearance holes required to insure 100 per cent interchangeability.

Figure 7 illustrates the tolerance zone diagram for Figure 1: four 0.020 square tolerance zones centered about four nominal locations. Figure 7 represents both mating parts when superimposed. (Since hole patterns on both parts are dimensioned and tolerated the same, the tolerance zones on both parts will coincide even though the part edges may not line up because of variations in form.)

Figure 8 represents coincident tolerance zones for two corresponding hole locations. The most critical condition of assembly will be when the centers of both clearance holes fall diagonally opposite each other at the extreme corners of their respective coincident square tolerance zones. Figure 9 depicts both hole centers falling diagonally opposite each other. Both holes shown are the same size and are, of course, larger than the bolt that will pass through them. If a 0.500 diameter bolt can just be assembled through these two holes, it will be centered on nominal.

Figure 9 graphically proves that we must add the diagonal of the square tolerance zone (0.028) to the bolt diameter (0.500) and must specify a 0.528 diameter minimum clearance hole to insure 100 per cent interchangeability. We have established the reason for nominally located gage pins: the bolt is nominally located at the most critical condition of assembly.

Since product drawings usually allow the manufacturer a tolerance on clearance-hole size, let us arbitrarily set a maximum size limit of 0.538 diameter for our four clearance holes. Oversize holes do not affect interchangeability. Our clearance-hole size callout will be:

\[
\frac{0.528}{0.538} \text{ diameter, 4 holes.}
\]

The Positional Tolerance

Two facts are clear at this point:

1. Gages 0.020 diameter smaller than minimum clearance holes are too restrictive.

2. You cannot gage the larger-in-area square tolerance.

Figure 10 proves that both the 0.020 diameter tolerance and the 0.020 square tolerances are too restrictive. We actually have a 0.028 diameter tolerance zone, and always have had; but this zone is never legally specified on bilaterally dimensioned drawings. Figure 10 proves that clearance-hole centers may fall outside the square tolerance zone and still not affect interchangeability. As long as the two clearance holes are no farther apart than 0.028 diameter or 0.014 radius from nominal, the bolt will assemble. The
positional tolerance formula for determining minimum clearance-hole size (providing both parts contain at least three holes and are tolerated the same) is:

Add the positional tolerance specified to the bolt diameter to determine the minimum clearance holes in both parts.

The Maximum Material Condition Concept
(The Bonus Tolerance)

Figure 11 represents the same critical condition of assembly illustrated previously by Figure 9. This is a graphic representation of maximum material condition or the most critical condition of assembly: the internal features (holes) are at minimum specified size, and the external feature (bolt) at maximum specified size. Both are critical. How often are actual part features at Maximum Material Condition (MMC)? Odds are that the holes will actually be drilled close to nominal (0.534) by most manufacturers.

Figure 12 is exactly the same illustration as Figure 11 with one mating part removed, and with the 0.500 diameter bolt replaced with a 0.500 diameter gage pin fixed on nominal. We usually gage only one part at a time. The actual difference between the gage pin (0.500) and the clearance hole at MMC (0.528) is the actual tolerance allowed (0.028 diameter).

In Figure 13 the actual hole size has been increased to 0.538 diameter (we arbitrarily specified this as the upper limit of its size). The difference between the gage pin and the hole is 0.038.

Figure 14 proves that a 0.538 diameter clearance hole may actually vary within a 0.038 diameter tolerance zone, and the gage will still accept the hole. The positional tolerance technology legally allows this bonus tolerance if the hole is not at MMC by specifying "at MMC" after the positional tolerance on the drawing. This increase in locational tolerance does not affect interchangeability. (See Figure 15.)

The callout in Figure 15 states: The positional tolerance must be 0.028 diameter only when a particular hole is at MMC (0.528 in this case) and legally allows the tolerance to increase in direct proportion to clearance-hole size increase as follows:

<table>
<thead>
<tr>
<th>Clearance-Hole Size</th>
<th>Positional Tolerance Allowed</th>
</tr>
</thead>
<tbody>
<tr>
<td>0.528</td>
<td>0.028</td>
</tr>
<tr>
<td>0.529</td>
<td>0.029</td>
</tr>
<tr>
<td>0.530</td>
<td>0.030</td>
</tr>
<tr>
<td>0.531</td>
<td>0.031</td>
</tr>
<tr>
<td>etc...</td>
<td>etc...</td>
</tr>
</tbody>
</table>

The actual tolerance available is always the difference between clearance-hole and gage-pin size. Also note that the inflexible bilateral tolerances have been removed from the nominal dimensions on the drawing and replaced by a variable but controlled positional tolerance.
Two important items are specified in Figure 15. The datum for measurement (Point P) has been specifically located by three tool and gage pickup points, and the datum for the direction of measurement from datum P (datum-orient) has been identified. The datum-orient is used to align the part in the gage and during open setup inspection. Both these datums may have to be physically marked on symmetrical parts, and are always specified in the PT/MMC technology to insure only one inspection method.

Figure 15 is a complete statement of design intent -- clear enough to insist upon a single interpretation as illustrated by the gage in Figure 16. We have legalized varying tolerance zones that gage designers have recognized for many decades -- but which bilateral drawings have never authorized. Gage pin size is directly determined on positionally tolerated drawings, as the gage designer simply subtracts the positional tolerance (0.028 diameter) from the clearance-hole size at MMC (0.528 diameter) to determine his gage pin size (0.500). No guesswork here -- the gage pin size is actually specified on the drawing.

Figure 16 illustrates the gage design dictated by Figure 15. This single gage interpretation insists on a stabilized controlled product. (Also note that this gage would actually accept the holes if they were smaller than 0.528, providing their locational tolerance was tighter than 0.028 diameter.)

Figures 17, 18, and 19 compare the bilateral and positional dimensioning systems. The tolerance increase allowed (from 57 per cent to 260 per cent) should justify the use of positional tolerances for any company currently dimensioning parts with bilateral tolerances. The cost reduction possibilities are obvious when you think of the number of good parts being scrapped each day because of incorrect and ambiguous dimensioning.

Dictating Inspection Criteria

Figure 20 illustrates, in simplified form, a common design requirement. The designer wants both parts to be concentric at assembly. (They obviously may not always be the same size.) The positional tolerance technology provides the "dimensioning tool" and states that the datum must be the center of diameter X regardless of feature size (RFS). This callout dictates a gage that must pick up diameter X at its precise center regardless of its actual size.

Such a gage has been shown in simplified form in Figure 21. The actual location of the centering screws (gage pickup points) may have to be physically marked on the piece part since great accuracy is required.

The tooling shown in Figure 21 does not have to center on the part, but it would be advantageous in this case as tools and gages should be highly compatible on precision parts. Note that the product drawing has, by dictating the inspection method, accurately specified engineering requirements.

As the callout, Datum: X (RFS), does require the use of expensive centering fixtures on both tooling and gaging, it should not be specified unless
proved to be absolutely necessary by a design layout. Fortunately, most parts can be designed to eliminate such costly tooling and gaging. (See Figure 22.)

The mating parts in Figure 22 can be assembled in a slightly misaligned position without affecting their function. The callout to datum will change to Datum: X (at MMC). This callout specifies that the tooling and gaging may pick up the part datum without the use of expensive centering fixtures. The callout in Figure 22 specifies that the datum must be the center of diameter X only when it is at its most critical size (at MMC).

This means that the tool and gage can be made to the MMC size of the part datum (5.010 diameter). See Figure 23. Both the tool and gage may shift slightly if the datum (5.000 diameter) is not at MMC (5.010 diameter). This MMC callout actually specifies a "shake gage," a common and inexpensive gage design. Note that when the part datum is at its most critical size for interchangeability (5.010 diameter), the gage will center on the part and insist that all clearance holes are located from the center of diameter X.

This paper has concentrated so far on tooling and gaging the positional tolerancing technology. How will the open setup inspector fare with the PT/MMC technology? The inspector will know both the hole size and its coordinate location from datum. If the actual hole size is larger than its MMC size he can legally increase the tolerance. By making one additional computation, the open setup inspector can determine the radial displacement of an actual hole center from nominal. If this radial distance is half the positional tolerance allowed for a particular hole, the hole center is within tolerance.

Paper Gaging Parts by Open Setup Inspection

Suppose the product drawing callout is Datum: X (at MMC) which specifies a shake gage. Open setup inspectors should not shift their setups in order to accept parts. Since the inspector will have a very accurate picture of the actual part on his inspection report (including actual hole sizes, datum size, and the coordinate location of these actual holes from datum), he can make a layout of the actual part at a greatly enlarged scale. This layout is only required if some of the holes fall outside of their prescribed tolerance zones in the same direction. This layout will accurately depict the inspected part and will even show hole squareness.

Next (Figure 24), he will make a layout of the actual gage (dictated by the product drawing) to the same enlarged scale as the part. The nominal dimensions on both the part layout and the gage layout can be shortened if the part is quite large.

Figure 25 shows that by superimposing the paper gage over the part layout he can immediately determine part acceptability.

This paper gage (see Figure 26), depending on its scale, may be even more accurate than an actual gage would be. This technique would be used only if the number of parts fabricated did not warrant an actual gage.
Suppose the product drawing callout is Datum: X (at MMC), which specifies a shake gage, but the part is too large to lay out at a greatly enlarged scale. In this case, the open setup inspector will lay out only the actual hole center locations from nominal. (Figure 27 shows the datum diameter in phantom lines for reference only.)

Figure 28 is a layout of the gage tolerance zones for the particular part shown in Figure 27, including the very important "shake" tolerance zone. (The nominal dimensions on both layouts can be shortened if the part is quite large.)

Since the actual part datum falls within the shake tolerance zone when the clearance-hole centers fall in the gage tolerance zones (Figure 29), the part is acceptable. These paper gage techniques will enable open setup inspectors, for the first time, to inspect piece parts to the same criteria as an actual gage.

There are other ways of solving the problems presented and answered in this paper (decentralized companies use additional specifications, liaison, or correspondence), but, if companies with far-flung operations want to use their product drawings as the control specification, the positional tolerancing technology is their solution. The PT/MMC technology uniformly expresses and interprets tolerances. It is a complete dimensioning tool in the hands of the designer and enables him, for the first time, to fully express and control his design intent on the product drawing.
\begin{figure}
\centering
\includegraphics[width=\textwidth]{figure6.png}
\caption{FIGURE 6}
\end{figure}
4-.02 SQUARE TOLERANCE ZONES CENTERED ON NOMINAL (DESIRED) LOCATIONS

FIGURE 7
FIGURE 8
.028 DIA. TOLERANCE ZONE WILL GUARANTEE 100% ASSEMBLY

.038 DIA. CLEARANCE HOLES WITH THEIR CENTERS OUTSIDE THE SPECIFIED .02 SQUARE TOLERANCE

FIGURE 10
FIGURE 11
FIGURE 13

- .500 DIA. FIXED GAGE PIN
- .538 DIA. HOLE
- .005 EXTRA TOLERANCE
- .528 DIA. HOLE
NOTE

1. POINT P IS DEFINED BY 3 TOOL OR GAGE POINTS OPPOSITE HOLES 1 AND 4.

DATA FOR FLAT 2.25 x 2.25 IN.

D 6.00 ± .04

4.00 ± .04

3.000

1.000

1.000

2

3

4

1

P

.528
.538

DIA., 4 HOLES

POSN TOL .028 DIA. (AT MMC)

4 PLACES

DATUM: P (SEE NOTE 1)

NOTES

FIGURE 15
.02 SQUARE TOLERANCE (OPEN SET-UP INSPECTION)  
27% LARGER THAN BILATERAL GAGING TOLERANCE (.02 DIA.)

.02 DIA. TOLERANCE BILATERAL (GAGE)

FIGURE 17
.028 DIA. POSITIONAL TOLERANCE AT MMC (HOLE AT .528 DIA.) FOR BOTH GAGE AND OPEN SET-UP INSPECTION 100% LARGER THAN BILATERAL GAGING TOLERANCE (.03 DIA.)

TOLERANCE ZONE 57% LARGER THAN BILATERAL

POSITIONAL AT MMC

FIGURE 18
.038 DIA. POSITIONAL TOLERANCE WHEN HOLE IS .538 DIA. FOR BOTH GAGE AND OPEN SET-UP INSPECTION 260% LARGER THAN BILATERAL GAGING TOLERANCE ZONE

TOLERANCE ZONE 183% LARGER THAN BILATERAL.

POSITIONAL NOT AT MMC

FIGURE 19
4.375 DIA. BOLTS

.395 DIA. 4 HOLES
.405 POSN TOL .020 DIA.
(AT MMC), 4 PLACES
DATUM: X (RFS)

5.000 ± .005 DIA.

FIGURE 20
FIGURE 21

TOOL PART

FOUR .375 DIA. GAGE PINS CENTERING SCREWS ON TOOL CENTERING SCREWS ON GAGE

PART

GAGE

FIGURE 21
4-.375 DIA. BOLTS

FIGURE 22
FIGURE 24

LAYOUT OF ACTUAL PART

NOMINAL LOCATION

POSITIONAL TOLERANCE

ACTUAL HOLE CENTER
OUTSIDE THE SPECIFIED TOLERANCE ZONE
4 PLACES

ACTUAL HOLE
4 PLACES

ACTUAL PART DIA. NOT AT MMC
LAYOUT OF GAGE
(PAPER GAGE)

FIGURE 25
PAPER GAGE ACCEPTING PAPER
PART

FIGURE 26
MMC DIAMETER

ACTUAL PART NOT AT MMC

EXACT CENTER OF THE O.D.
(DATUM FOR OPEN SET UP)

ACTUAL HOLE CENTER OUTSIDE
THE POSITIONAL TOLERANCE (4 PLACES)

4 POSITIONAL TOLERANCE ZONES

LAYOUT OF PART HOLE CENTER LOCATIONS

FIGURE 27
GAGE "SHAKE" TOLERANCE (THE MEASURED DIFFERENCE IN SIZE BETWEEN THE PART DATUM AND THE GAGE DIA.)

LAYOUT OF GAGE TOLERANCE ZONES (PAPER GAGE)

FIGURE 28
FIGURE 29

PAPER GAGE
ACCEPTING PART