MEASUREMENT OF GEOMETRIC TOLERANCES IN MANUFACTURING

JAMES D. MEADOWS

MEASUREMENT OF GEOMETRIC TOLERANCES IN MANUFACTURING

MANUFACTURING ENGINEERING AND MATERIALS PROCESSING
A Series of Reference Books and Textbooks

FOUNDING EDITOR

Geoffrey Boothroyd
University of Rhode Island
Kingston, Rhode Island

1. Computers in Manufacturing, *U. Rembold, M. Seth, and J. S. Weinstein*
2. Cold Rolling of Steel, *William L. Roberts*
3. Strengthening of Ceramics: Treatments, Tests, and Design Applications, *Harry P. Kirchner*
4. Metal Forming: The Application of Limit Analysis, *Betzalel Avitzur*
5. Improving Productivity by Classification, Coding, and Data Base Standardization: The Key to Maximizing CAD/CAM and Group Technology, *William F. Hyde*
6. Automatic Assembly, *Geoffrey Boothroyd, Corrado Poli, and Laurence E. Murch*
7. Manufacturing Engineering Processes, *Leo Alting*
8. Modern Ceramic Engineering: Properties, Processing, and Use in Design, *David W. Richerson*
9. Interface Technology for Computer-Controlled Manufacturing Processes, *Ulrich Rembold, Karl Armbruster, and Wolfgang Ülzmann*
10. Hot Rolling of Steel, *William L. Roberts*
11. Adhesives in Manufacturing, *edited by Gerald L. Schneberger*
12. Understanding the Manufacturing Process: Key to Successful CAD/ CAM Implementation, *Joseph Harrington, Jr.*
13. Industrial Materials Science and Engineering, *edited by Lawrence E. Murr*
14. Lubricants and Lubrication in Metalworking Operations, *Elliot S. Nachtman and Serope Kalpakjian*
15. Manufacturing Engineering: An Introduction to the Basic Functions, *John P. Tanner*
16. Computer-Integrated Manufacturing Technology and Systems, *Ulrich Rembold, Christian Blume, and Ruediger Dillman*
17. Connections in Electronic Assemblies, *Anthony J. Bilotta*
18. Automation for Press Feed Operations: Applications and Economics, *Edward Walker*
19. Nontraditional Manufacturing Processes, *Gary F. Benedict*
20. Programmable Controllers for Factory Automation, *David G. Johnson*
21. Printed Circuit Assembly Manufacturing, *Fred W. Kear*
22. Manufacturing High Technology Handbook, *edited by Donatas Tijunelis and Keith E. McKee*
23. Factory Information Systems: Design and Implementation for CIM Management and Control, *John Gaylord*
24. Flat Processing of Steel, *William L. Roberts*

25. Soldering for Electronic Assemblies, *Leo P. Lambert*
26. Flexible Manufacturing Systems in Practice: Applications, Design, and Simulation, *Joseph Talavage and Roger G. Hannam*
27. Flexible Manufacturing Systems: Benefits for the Low Inventory Factory, *John E. Lenz*
28. Fundamentals of Machining and Machine Tools: Second Edition, *Geoffrey Boothroyd and Winston A. Knight*
29. Computer-Automated Process Planning for World-Class Manufacturing, *James Nolen*
30. Steel-Rolling Technology: Theory and Practice, *Vladimir B. Ginzburg*
31. Computer Integrated Electronics Manufacturing and Testing, *Jack Arabian*
32. In-Process Measurement and Control, *Stephan D. Murphy*
33. Assembly Line Design: Methodology and Applications, *We-Min Chow*
34. Robot Technology and Applications, *edited by Ulrich Rembold*
35. Mechanical Deburring and Surface Finishing Technology, *Alfred F. Scheider*
36. Manufacturing Engineering: An Introduction to the Basic Functions, Second Edition, Revised and Expanded, *John P. Tanner*
37. Assembly Automation and Product Design, *Geoffrey Boothroyd*
38. Hybrid Assemblies and Multichip Modules, *Fred W. Kear*
39. High-Quality Steel Rolling: Theory and Practice, *Vladimir B. Ginzburg*
40. Manufacturing Engineering Processes: Second Edition, Revised and Expanded, *Leo Alting*
41. Metalworking Fluids, *edited by Jerry P. Byers*
42. Coordinate Measuring Machines and Systems, *edited by John A. Bosch*
43. Arc Welding Automation, *Howard B. Cary*
44. Facilities Planning and Materials Handling: Methods and Requirements, *Vijay S. Sheth*
45. Continuous Flow Manufacturing: Quality in Design and Processes, *Pierre C. Guerindon*
46. Laser Materials Processing, *edited by Leonard Migliore*
47. Re-Engineering the Manufacturing System: Applying the Theory of Constraints, *Robert E. Stein*
48. Handbook of Manufacturing Engineering, *edited by Jack M. Walker*
49. Metal Cutting Theory and Practice, *David A. Stephenson and John S. Agapiou*
50. Manufacturing Process Design and Optimization, *Robert F. Rhyder*
51. Statistical Process Control in Manufacturing Practice, *Fred W. Kear*
52. Measurement of Geometric Tolerances in Manufacturing, *James D. Meadows*

Additional Volumes in Preparation

MEASUREMENT OF GEOMETRIC TOLERANCES IN MANUFACTURING

JAMES D. MEADOWS
Institute for Engineering & Design, Inc.
Hendersonville, Tennessee

http://www.geotolmeadows.com

MARCEL
DEKKER

MARCEL DEKKER, INC. NEW YORK • BASEL

Library of Congress Cataloging-in-Publication Data

Meadows, James D.
Measurement of geometric tolerances in manufacturing / James D. Meadows.
p. cm. -- (Manufacturing engineering and materials processing ; 52)
Includes index.
ISBN: 0-8247-0163-1 (alk. paper)
1. Tolerance (Engineering). I. Title. II. Series.
TS172.M43 1998
620'.0045—dc21 98-17836
CIP

This book is printed on acid-free paper.

Headquarters
Marcel Dekker, Inc.
270 Madison Avenue, New York, NY 10016
tel: 212-696-9000; fax: 212-685-4540

Eastern Hemisphere Distribution
Marcel Dekker AG
Hutgasse 4, Postfach 812, CH-4001 Basel, Switzerland
tel: 44-61-261-8482; fax: 44-61-261-8896

World Wide Web
http://www.dekker.com

The publisher offers discounts on this book when ordered in bulk quantities. For more information, write to Special Sales/Professional Marketing at the headquarters address above.

Current printing (last digit):
10 9 8 7 6 5

Preface

Sometimes we learn best by seeing, reading, or hearing about the tribulations of others. We analyze what they have faced, how they dealt with it, and whether they were successful or failed miserably. As a consultant, I have visited hundreds of companies and listened to thousands of problems, and then logged the best and the worst of the problem-solving approaches used. This book relates many of the most interesting situations and discusses relevant ethical concerns.

This is a technical book about measurement of geometric characteristics and how they are practiced on the shop floor. But it is also a book about quality, company politics, and how otherwise logical professionals can perform in the most illogical of ways when placed under the pressures of too little time, too little input, and too much misinformation. It shows how ego, bias, and vested interests can destroy a product and compromise a person's principles.

In my travels I have seen that common sense is anything but common. Many of the stories in this book are amusing and some downright unbelievable. But take my word for it: I couldn't make this stuff up. Much of it is just too weird. Just be glad it isn't happening to you. But if it is—and it very well may be—please tell me about it. Someday, I may want to write another book.

This is the third book I've had published by Marcel Dekker, Inc., and I must admit I never thought it would come together this easily. As always, I remain indebted to Michael Gay of Nashville CAD, Inc. for his great illustrations, Kay DuVall for her help in putting this book together, Jeannie Winchell for overseeing and guiding all of my projects since the beginning, and my editors at Marcel Dekker, Inc.

I would also like to thank Mike Groszko of Chrysler Quality Institute for his assistance in putting together the SPC unit. Finally, I would like to thank Carl Lance for all of the GD&T insight he has given me over the last 20 years. He is truly one of the most knowledgeable people I have ever met.

James D. Meadows

Contents

Preface *iii*

1. **Introduction to Measurement Principles: Once Upon a Time . . . in the Land of Measurement 1**

2. **Geometric Product Definition and Measurement, Inspection, and Gaging Principles 7**
   - 2.1 Virtual Condition Illustration 11
   - 2.2 Verification of Position with Open Set-Up 14

3. **Reading the Feature Control Frame**
   - 3.1 Just Talk the Talk 19
   - 3.2 How to Read a Feature Control Frame 21
   - 3.3 How to Read a Feature Control Frame for Purposes of Measurement 23

4. **Functional Product Definition Creates Functional Inspection 33**
   - 4.1 Inspecting Parts as They Are Used 33
   - 4.2 Functionality Used to Suggest Manufacturing Procedures 34
   - 4.3 Multiple Possibilities of Dimensioning and Tolerancing for Functionality and Inspectability 36
   - 4.4 I Can't Measure That! 38
   - 4.5 Functionality vs. Producibility 41
   - 4.6 The Things I've Seen (Where Tolerances Come From) 42
   - 4.7 Hog-Tied and Cow-Chipped (Importance of Geometric Definition) 44

4.8 Correct and Complete
(Tips on Functional Product Definition) 46
4.9 Miles to Go Before We Sleep
(Importance of Geometric Definition) 50
4.10 Plastic Parts and Stretch Tolerances 52
4.11 Design, Inspection, Production, and Prototype Needs
and Capabilities Regarding Dimensioning and
Tolerancing 54
4.12 Implying a Manufacturing Sequence on Complex Part
Configurations 57

5. Principles of Measurement for Geometric Characteristics 63
5.1 Geometric Characteristic Symbols 63
5.2 Straightness—Form Category 64
5.3 Flatness ⏥ Form Category 76
5.4 Circularity ○ Form Category 79
5.5 Cylindricity ⌭ Form Category 89
5.6 Parallelism // Orientation Category 93
5.7 Perpendicularity ⊥ Orientation Category 97
5.8 Angularity ∠ Orientation Category 102
5.9 Profile of a Surface ⌓ Profile Category 106
5.10 Composite Profile vs. Two Single-Segment Profile
Controls 109
5.11 Coplanarity ⌓ (Profile for Coplanar Surfaces)
Profile Category 113
5.12 Conicity (Profile for Cones) ⌓ Profile Category 117
5.13 Profile of a Line ⌒ Profile Category 119
5.14 Circular Runout ↗ Runout Category 126
5.15 Total Runout ⌰ Runout Category 128
5.16 Concentricity ◎ Location Category 135
5.17 Symmetry ⌯ Location Category 139
5.18 Inspecting Threaded Holes 144
5.19 The Boundary Concept 150
5.20 Composite Positional Tolerancing vs. Two Single-Segment
Positional Tolerancing Controls 154
5.21 Rules Sheet 160

6. Comparison of Controls 161
6.1 Food Chain of Symbology 161
6.2 2-D vs. 3-D Controls 162
6.3 Comparison of Coaxiality Controls
(Position, Concentricity, Circular Runout, Total Runout) 162
6.4 Concentricity vs. Runout 166

7. Datums 169
7.1 When Sweet Seeds Grow Rancid Fruit (How to Select Datum Features) 169
7.2 Part Stabilization (Datums for Flexible Parts) 172
7.3 Sequential Part Definition vs. Simultaneous Requirement (The Game of Leaders and Followers) 176
7.4 Datum Features 178
7.5 Simultaneous and Separate Requirements 185
7.6 Datum Target Assignment 189
7.7 Oddly Configured and Curved Surfaces as Datum Features 196
7.8 Secondary and Tertiary Datum Features of Size 198
7.9 Free-State Variation in Sheet Metal Parts/Fixturing Sheet Metal 203

8. Material Conditions (Modifiers) Meaning and Uses 211
8.1 Maximum Material Condition 211
8.2 The Maximum Material Condition Symbol and Its Ramifications 214
8.3 Allowed vs. Actual Deviation From True Position 220
8.4 Least Material Condition (Meaning and Uses) 230
8.5 Regardless of Feature Size 231

9. Virtual and Resultant Condition Boundaries 235
9.1 Virtual Condition—A Functional Boundary 235
9.2 Surface vs. Axis Verification 239
9.3 Zero Positional Tolerancing at MMC for a Greater Yield 241

10. Gaging—Principles, Terms, and Examples 245
10.1 Gage General Principles 245
10.2 Gaging Size Limits 250
10.3 Terms 254
10.4 A Few Major Measurement Concepts 257
10.5 Gaging a Complex Sheet Metal Part 295
10.6 General Principles and Terms of Tolerance Compliance 300

11. Merging Theory with Reality 303
11.1 Merging Theory with Reality (Having the Right Measurement Tool) 303
11.2 Theory vs. Reality (Uncertainties) 304
11.3 Reactions to a Non-Perfect Datum Establishment 306

12. How to Improve Quality 309
12.1 The Zen of Quality 309
12.2 Tying Measurements to the Machine 311
12.3 How Many Points Should I Probe? 311

12.4 The Search for Common Sense (Statistics Out of Control) 311
12.5 Quality Issues from the Afterlife (Slogans vs. Programs and Planning) 315
12.6 Focus and Excel (Maximizing Your Abilities) 317
12.7 Competing in a Global Economy (Great Products at an Affordable Cost) 319

13. Inspection Planning 321
13.1 Dimensional Inspection Planning 321
13.2 Methodical Measurement 331

14. Source of Measurement Error 343
14.1 What We Know Is Often Untrue and Other Sources of Measurement Error 343
14.2 CMM Error Compensation 344
14.3 Certification vs. Calibration 347

15. Graphical Inspection Techniques and Examples 351
15.1 Graphical Techniques and Inspection Reporting 351
15.2 Comparison of Measurement Techniques and Equipment (Advantages and Disadvantages) 375

16. Teamwork Issues 381
16.1 Unity of Purpose Affects the Bottom Line 381
16.2 Troubled Times and Technical Leaps 385
16.3 Concurrent Engineering—Key to Our Future 387
16.4 Magical, Mystical Stuff (Doing Something vs. Doing Nothing) 389
16.5 Avoiding Tiger Pits . . . While Helping a Leopard Change Its Spots 391

17. Training Issues 395
17.1 I Wonder as I Wander 395
17.2 Training: An Ongoing Process 396

18. Statistical Process Control 399
18.1 Faulty Implementation—Wonderful Concepts 399
18.2 Concepts of Statistical Process Control 405
18.3 Calculating Percent of Nonconforming 414
18.4 Statistical Tolerancing 421
18.5 Glossary of Terms 423
18.6 Symbology for SPC Formulas 426

19. Letters: Common Problems and Their Solutions 429

Index 451

1

Introduction to Measurement Principles

Once Upon a Time . . . in the Land of Measurement

In the history of the world of manufacturing, the shop floor has often been tasked with doing quick, rough checks on the products undergoing production. These measurement procedures sometimes were visual checks by the manufacturer to determine flaws in the parts. When this was insufficient, instrumentation was employed, but usually of a less sophisticated nature than would perhaps later be utilized in the inspection lab. Still, some of the earliest and most effective gages were mating parts that had already been produced and were plucked from a barrel to serve as a test of functionality. The part just finished or nearing completion was tried on the potential mating part. If it assembled, it was assumed the newly formed part was functional and, therefore, acceptable.

After a time, it was realized that this type of test did not assure that the part would assemble with any of the mating parts plucked from the barrel, but perhaps only with some—or even one. This lack of interchangeability was sometimes unacceptable. We were, in these instances, creating matched sets of parts. What was often desirable, instead, was the creation of a part that would assemble with any of the mating parts produced so that, for example, a person could take a broken part in his automobile or tractor into the parts store and buy an off-the-shelf replacement.

At some point, it was decided a gage that represented the worst condition of mating parts would be needed to gage the capability of newly produced parts to fit into the assembly and be compatible with any part already in place. These gages are commonly called receiver-type or functional gages. Some of the initial gages were meant to inspect only size requirements. They are still used to this day. They are called GO gages. GO gages check maximum material conditions and, in doing so, contribute to checks of geometric form. They are designed to "go" over the largest shafts or into the smallest holes allowed by size requirements. A NO-GO gage is often used or simulated to inspect least material conditions. Their job would be not to "go" over the smallest cross section of the shaft or into the largest cross section of a hole. If they "go," it is assumed size at least material condition has been violated. The same as if the GO gage refuses to "go" over the entire shaft or into the entire hole, it is assumed size at maximum material condition has been violated.

But, as we all know, there is often more to a feature reacting functionally than just its size. Functional gages check geometric characteristic requirements, such as perpendicularity, position of features and patterns of features, such as holes, slots, shafts, tabs, and other features of size. They check to determine whether the boundary generated by the collective effect of worst case size and applicable geometric tolerances has been violated. These boundaries are often constant boundaries that exist between mating features, confining the shaft within and keeping the surface of the hole outside of it. It is assumed that if the hole never moves inside of this boundary, it never occupies space meant for the shaft. And it is also assumed if the shaft never moves outside of the boundary, it never occupies space reserved for the hole. Never crossing this line of the worst case mating boundary by either assures us these features will assemble without interference.

The problem with these gages has always been that because they employ no probes or indicators, just hard planar rails, shafts, holes, and other physical configurations meant to be the physical embodiment of a worst case scenario, they give little or no information except whether or not the part will fit into the assembly. They don't say how good or bad a feature is, or in what direction it might be straying. This information is often considered vital to improving a manufacturing procedure by charting or graphing deviations from geometric perfection. These pieces of variables data are regularly employed to statistically control a manufacturing process. So, other mechanisms of inspection are sometimes needed to collect this information, not to replace, but to augment, the attribute (good/bad) information given by functional gages.

These collectors of variables data, such as dial indicators and electronic probes, tell the story of a feature's deterioration from geometric perfection. From this, it is possible to determine that within a large run of parts critical features are getting worse as the run continues and, if allowed to resume in that manner, will eventually exceed the boundaries of functionality and no longer assemble or operate.

With the proliferation of computers, we have connected many of our variables data collectors directly to them or fed the information collected into computerized programs written to analyze the data. But, with these came another source of error. The algorithms that are written aren't always correct. This allows data to be improperly analyzed, sometimes causing functional parts to be rejected and nonfunctional parts to be accepted.

Controversy also ensues as to the use of touch probes. Optimal probe sizes, the number of points to take to represent a surface, and the distribution of these points on the part are areas of controversy. Then, scrutiny of what the software is doing with the collected points becomes acute. Many times, the software is supposed to duplicate what the parts rest on or clear in an assembly but, instead, computes best fit surfaces. This means that high points that are critical to determine assembly capability are treated as no more important than low surface points. It means that origins of measurement, often called datums, are formed, not from the high points of surfaces but, rather, as best fit planes or axes from all points collected. Since high points protrude farther than these best fit simulations, parts are often accepted that have no chance of assembling, hanging up instead on the high points.

Fixturing would help. Placing parts on fixtures as simple as surface plates and angle plates would allow inspectors to probe them instead of datum surfaces. And, since they rest against high points on the parts, software problems would be lessened. But many are told that because a computer is involved, all possible problems will be solved inside the mysterious world of software algorithms, without fixturing, and to just trust the computer—which many do, and they suffer for their trusting nature.

In the late 1980s, a group of quality engineers from Westinghouse Corporation sent out a cry of alarm to the U.S. military and later to others who would listen. They chronicled their finds of unreliable and nonrepeatable measurements taken with computer-aided measurement machines, pointing out errors in a variety of areas. It was discovered that software was written incorrectly by programmers well schooled in writing software but either unaware of or unconcerned about technical problems found in their software. It was also discovered that measurement machine manufacturers were giving instruction to users of their mechanisms that exacerbated the problems. Too few points taken to establish representative datum planes and axes were suggested and fixturing was often discouraged as expensive, time-consuming, and unnecessary. Too few points and/or an unwise distribution of points taken to represent surfaces under test drove the problem to the brink of, in many cases, disaster.

When the military took the problem very seriously, even offering to shut down every such machine doing work for them, another cry, equally desperate, was heard. This came from those with a vested interest in the status quo. Greed and pride prompted many to attack the findings, downplay the problems, and denigrate the alarmed. Many of those crying the loudest were powerful and tried to destroy all efforts to cure the problems, even to the point of suggesting we change the definitions of datum establishment and product inspection to simply say what the flawed software and measurement machines were doing was correct.

To a large part, both sides of this battle were successful. Many wanted to believe all things were right with the world of measurement, and those with a prideful or monetary interest in perpetuating this falsehood were ready to tell them what they wanted to hear. On the other side, ANSI committees were formed to resolve the problems. Committees on correct mathematics, measurement methods, sources of common errors, functional gaging, and fixturing were formed and began to try to make things better.

In the end, it was discovered that those who wanted to listen did and those who found it easier to belittle the efforts or divert them from their intended path set about doing exactly that. Perhaps it was because some of the problems seemed unsolvable, or because some of those involved in collecting points could not accept a merging of strengths between gaging disciplines (for example, some fixturing with some probing), that this story has no happy ending. Maybe it was just that once started down the path of ill-conceived procedures some were unwilling to consider another tactic.

As far back as 1988, Russ Shelton and Klaus Ulbrich of EMD performed tests whose results were shocking to many and ignored by others. The response to their findings seemed to be that no action was taken to correct the problems raised. Touch-trigger probes for data collection are used by about 90% of the world's CMMs. These probes are slow and plodding in their efforts to collect enough points to represent a surface. They lower, hit, raise, move, lower, and hit again. Analog probes or drag probes (as they are sometimes referred to), which are faster at collecting a large sampling of surface points, are expensive and most CMM users can't afford them. Therefore, the touch probe users continue on in the mistaken belief that they are getting a representative sample of a surface by taking relatively few points, which is all they have time to take given the technology with which they are stuck. Shelton and Ulbrich tested six sample surfaces to construct a plane. Each was about one square inch. All were lapped aluminum. They represented many different deformities such as gouges, concave and convex surfaces, etc. Each was scanned in a grid that ended up with about 2500 evenly spaced data points. I quote from them: "Depending on the specific sample, it took between 80 and 357 points (average 191) per square inch to achieve a 90 percent certainty of form, and between 48 and 227 points (average 102) per square inch to achieve a 75 percent certainty of form." Compare this to the number of points commonly probed by the average inspector to validate or represent a surface and the horror of our current situation should begin to sink in.

Although there is more information available now on these problems than ever before and more solutions are available, it is also true that the spreaders of repackaged, badly flawed software procedures and expedient, poorly thought out recommendations on general set-ups and tool and equipment usage still abound, even flourish, to this day. Wonderful equipment and software, capable of evaluating data correctly, are often ignored in favor of a flashier name and a better sales pitch. Buyers are commonly uneducated as to what measurement mechanisms and their accompanying software should be doing, so they fall prey to a smooth line and a knife-bearing pat on the back by a slick salesperson who says "Buddy" and "Pal" a lot.

We are still in a tub of trouble out here. Our systems are full of holes and we're looking left when we should be bailing with an unrestrained fervor. But it is hard to create zealots for change when the general consensus is that we're sliding along in greased grooves. Those who experience the world of politics and cover-up with an acute feeling of nausea are seen as relatives of Chicken Little shrieking that "the sky is still falling!"

Many give up, battered and bruised on the outside, with a pure heart still beating—but only faintly audible. Perhaps it is true that they shoot the messengers of bad tidings and that it is easier to entertain the troops with a sweet song and a fast shuffle than to end the war. One thing is certain in the land of the blind—the one-eyed man is not king; he is shunned because he is different.

2

Geometric Product Definition and Measurement, Inspection, and Gaging Principles

Let us start with a comparison of the plus and minus system for location and what has often been called "true positioning."

In Figure 2-1, this simple part uses plus and minus tolerancing to locate the

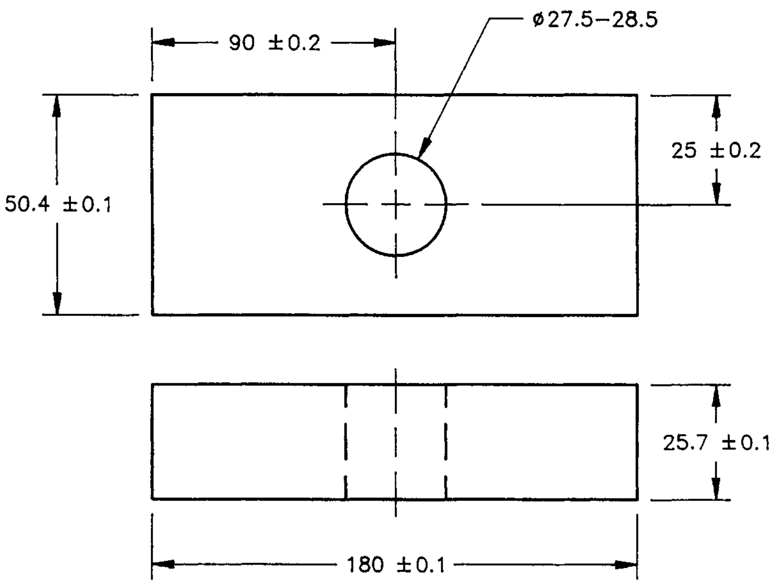


FIGURE 2-1

hole. In reality, we cannot be certain whether the hole is to be measured from the edges of the part, or the edges of the part are to be measured from the hole. This ambiguity could cause difficulties in determining set-ups for both manufacturing and inspection. It also makes it difficult to discern the exact configuration and location of the tolerance zone. If the origin of measurement were to be the axis of the hole, then tolerance zones could be assumed to contain the edges of the part and perhaps even be perceived to control the location and profile/form of the two edges.

For this example, however, even though that interpretation could be easily argued as valid, let us assume the hole is to be measured from existing edges of the part. If so, the drawing says we would like the hole to be 25 mm from one edge and 90 mm from another. Knowing we are imperfect beings and need a tolerance on that, we are allowed a plus and minus tolerance in each direction of 0.2. This generates a 0.4 square tolerance zone in which the actual axis of the hole must lie. The tolerance zone is centered around the mean dimensional location of the 25 and 90 mm specifications.

We can see the tolerance zone is square, but the hole (in two-dimensional terms) is round. The tolerance zone is not reflective of the shape of the hole it protects. Curious. However, we know this is often done because of part function, so let us examine the mating part. It is a rectangular plate with a shaft diameter mounted on it (see Fig. 2-2).

The mating situation is simply that we would like the shaft to fit into the hole with the seating perpendicular surfaces firmly against one another and the edges of the parts (from which the 25 and 90 mm dimensions emanate) to line up flush. That would not dictate a square tolerance zone. Sometimes a tolerance zone is created as nonreflective of hole shape to protect wall thickness where a too thin wall is a possibility from the hole surface to the outside of the part. But with only a 27.5 to 28.5 diameter hole and a 0.4 square location tolerance zone,

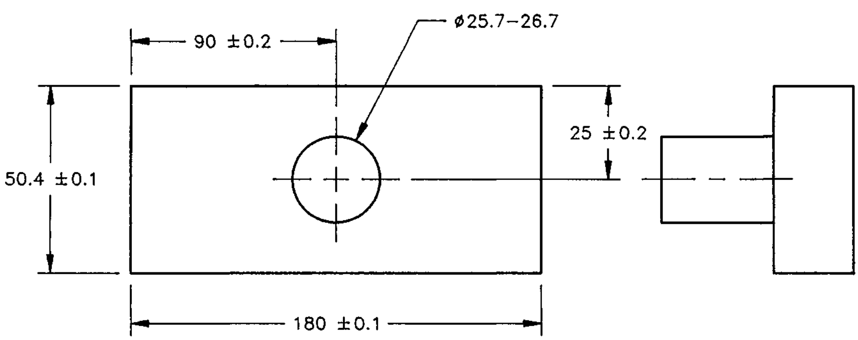


FIGURE 2-2 Mating part.

one can easily see the shortest distance of 25 millimeters of an inch is not capable of creating so thin a wall that the hole weakens the part to a point of danger.

So, there is no logic in having a round hole with a square tolerance zone considering the difficulties that zone places on us in tolerance analysis, worst case boundary calculations, and functional gage design.

Let us examine some of the decisions that would have to be made if the hole were drilled. If a machinist were asked to drill the hole, one would have to choose what surface to lay the part down on first to come in contact with the machine (drill press) table. The drawing allows us to choose one of *at least* two. The primary surface strikes a minimum of three high points of contact and eliminates part movement in two rotational and one linear direction.

Since the part can still move, the machinist slides a rail or angle plate against another surface chosen from at least two surfaces and slides into two point minimum high points of contact to eliminate two more degrees of freedom (one rotational and one linear). Whatever surface is left is used as a tertiary feature requiring a one point minimum high point of contact to eliminate the last degree of linear movement. The machinist then clamps the part into that orientation, measures from those rails and/or angle plates 25 and 90 mm and drills the hole. The problem occurs when the part must be checked. There is no repeatability factor here.

When the parts inspector gets this part and must set it up to inspect it, he or she must make the same choices as did the machinist. Suppose he or she chooses differently. How many times and in how many different part orientations must one set the part to determine that the part does not meet the drawing requirements? If the inspector sets it up the first time and it checks good, the part will most likely be accepted as within tolerance. Still, since the drawing does not indicate how this part seats in the assembly, there is no assurance that it has been inspected in a functional manner. It may check within tolerance but not assemble since it may not have been inspected in the same manner that it fits into the assembly.

But, if the first set-up allows the part to check bad, the inspector might have chosen different primary, secondary, and tertiary surfaces than the part uses in the assembly. The part might check within specifications with a different set-up (part orientation). So, the inspector resets it and resets it, knowing each different set-up might be the one that allows the part to be bought off as good. Each set up takes time, and time and money are being wasted.

If the designer had used some simple criteria to choose the set-up surfaces, the machinist and inspector would have had no choices to make. The drawing could have reflected the designer's knowledge of datum feature selection criteria (for example, part function and representation of the mating situation).

The same part using geometric dimensioning and tolerancing might appear as in Figure 2-3. With this drawing, the primary datum feature is A (3 point high point minimum contact), the secondary B (2 point high point minimum contact

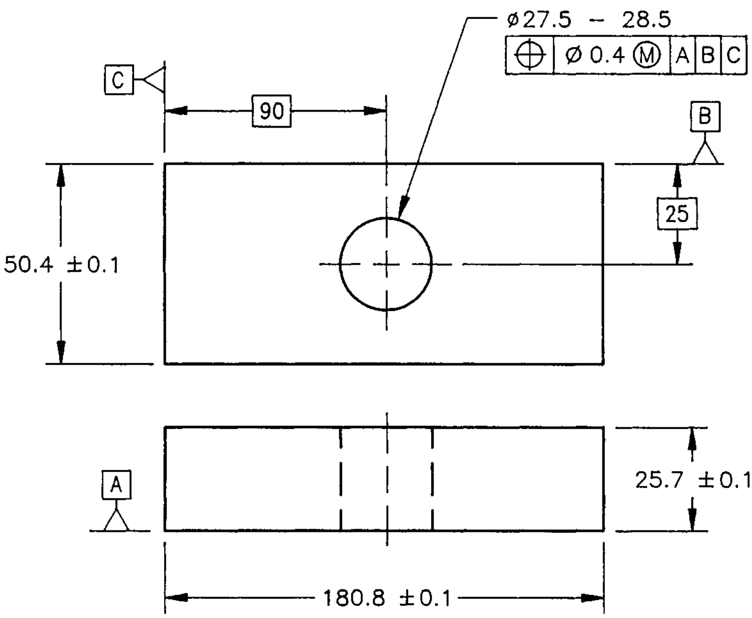


FIGURE 2-3

while maintaining A), the tertiary C (1 point high point minimum contact while maintaining A and B).

We know this is not because that is how they appear in the alphabet, but rather because this is how they appear in the information block (called a feature control frame) read from left to right. It is located directly below the size limits of the hole so as to localize much of the information about the hole.

It tells us we want to position the hole so the position symbol is the first piece of information given. The tolerance zone is described as a diameter so as to reflect the hole shape. The 0.4 square has been replaced by a 0.4 diameter tolerance zone in which the actual axis of the hole must lie. This is a design change surely—but one that will work because the mating shaft will be dimensioned and toleranced in a way to allow it to mate with this new requirement.

Although initially a smaller tolerance, the M in a circle will allow more as the hole grows from 27.5 toward the 28.5 size limit. The datum features assure more repeatability and, together with the potentially larger tolerance, make the part more cost effective to produce and inspect. A Ø28.5 hole is allowed Ø1.4 position tolerance.

Datum feature A locks in perpendicularity of the tolerance zone to it and, therefore, protects perpendicularity of the hole. This is a multipurpose control because, besides datum A controlling perpendicularity, B and C locate the tolerance zone axis (25 exactly from B and 90 exactly from C) and, consequently, specify the hole axis's perfect location. The dimensions from those surfaces are boxed to indicate an exact specification.

Now we know that if we were able to make the hole perfect, it would be perfectly perpendicular to the plane formed by the three highest points of surface A and 25 mm from the plane formed by the two highest points of B and 90 mm from the plane formed by one point of high point contact from C. Each plane is dependent upon and mutually perpendicular to the other two, forming what is commonly called a datum reference frame. This datum reference frame eliminates the 6 degrees of part freedom (previously described), orients the part, and positions the hole for repeatability.

Since the hole can't be made exactly where the datums and basic dimensions indicate, the tolerance zone of Ø0.4 is used to allow hole axis deviation from perfection, or more than Ø0.4 as the hole grows. The reason the circled M was used was a recognition of reality in this situation. The reality is simply that if the hole moves and/or leans Ø0.4 at a diameter of 27.5, the size shaft that can fit into that hole (in a manner described in the mating part drawing and mating situational requirements) is 27.5 minus 0.4 or a diameter of 27.1.

So 27.1 is the boundary we are protecting, and it isn't just important some of the time on some of the parts, but rather all of the time and on all of the parts. A combination of the smallest hole size and the allowed out-of-perfect perpendicularity and location on this part reduces its effective mating size. That's why we must design the parts so the hole never infringes on a boundary of 27.1 that belongs to the mating shaft, and the mating shaft never infringes on a boundary of 27.1 belonging to the hole. This complied with, material never interferes.

So, if the hole was produced at a diameter of 28, we could allow the positional tolerance on the hole location for that produced part to grow (by virtue of the circled M) to 0.9, because 28 minus 0.9 still protects a 27.1 boundary. At 28.5, we could allow 1.4, because 28.5 minus 1.4 is still 27.1. The part is allowed more hole positional tolerance based on its actual produced size. It is all functional and very cost-effective, because a hole allowed to move or lean away from its perfect location has the effect of reducing the area available for a shaft to insert into at that location.

2.1 VIRTUAL CONDITION ILLUSTRATION

This virtual condition boundary is generated by the hole's size and geometric imperfection. As the hole moves away from its perfect position or leans out of perfect perpendicularity, the area available for a mating pin to go into the hole at that location and angle is diminished, creating a virtual condition boundary. This boundary is smaller than the smallest hole by the amount of the allowed positional tolerance. Therefore, the pin must be smaller than or equal to the virtual condition of the hole, if it is to be inserted without interference under all circum-

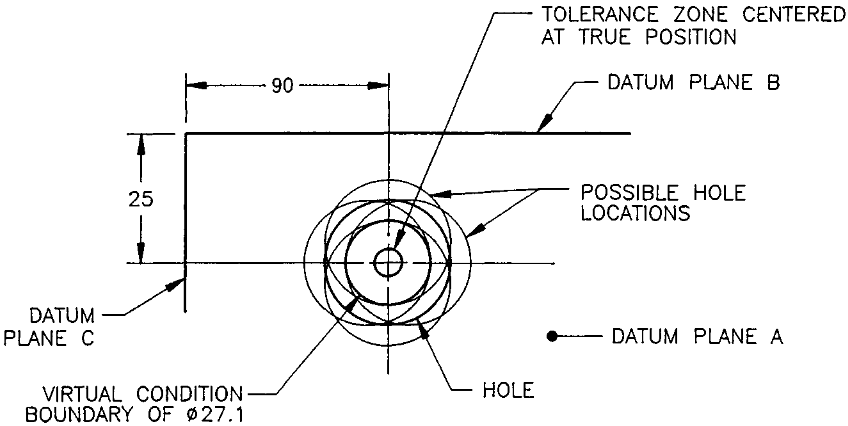


FIGURE 2-4

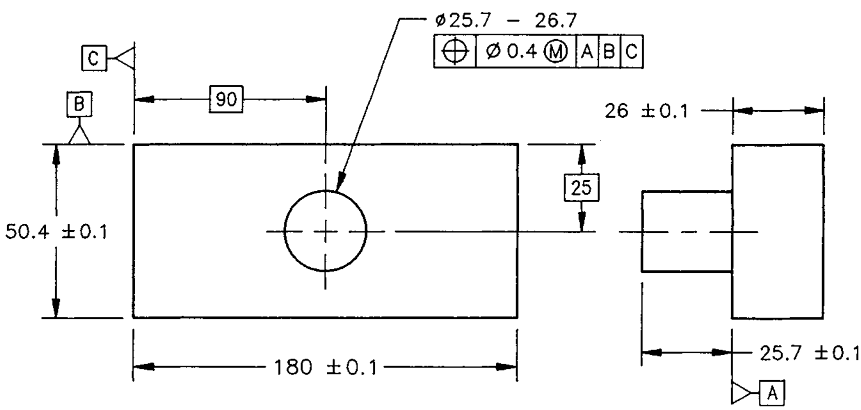


FIGURE 2-5

stances. This example (Fig. 2-4) shows the collective effect of the actual size and the geometric tolerance applicable at that size for a hole controlled for position at maximum material condition (MMC). This is the worst mating boundary.

By the same token, a mating shaft made at 26.7 diameter would only have available to it a positional tolerance of Ø0.4 before it approached the worst mating boundary of Ø27.1, but at Ø25.7 it would be allowed Ø1.4 (provided it is made within its size limits) because 25.7 plus 1.4 is 27.1. Shafts, and for that matter all external features of size that move or lean away from their perfect locations, have the effect of increasing the area needed for the shaft to insert into at those locations.

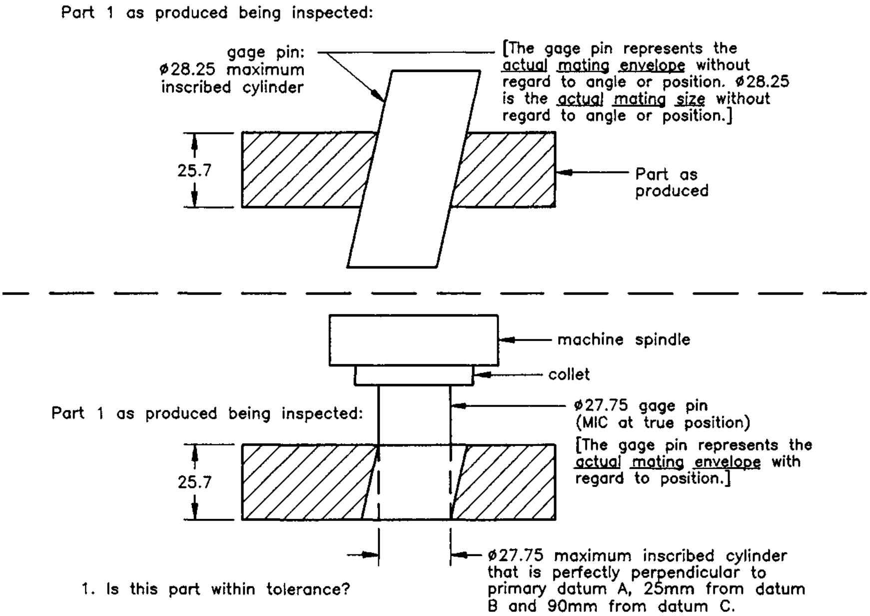


FIGURE 2-6

One possible mating part design appears in Figure 2-5. This is just one example. This text shows many techniques to assist you in the usage of the ASME Y14.5 standard (geometric dimensioning and tolerancing).

Figure 2-6 is a possible inspection procedure performed on a part produced for the specification shown in Figure 2-3.

With the information collected as shown above we can deduce in two different ways that the produced part meets its positional tolerance.

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

OR

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

2.2 VERIFICATION OF POSITION WITH OPEN SET-UP

This section will explore one often-used method of verifying the location of positionally toleranced features on parts, such as shown in Figure 2-7.

Before the part is inspected for the shaft's compliance with its positional control, size requirements should be verified. Least material condition (LMC—smallest shaft size) should be checked at every two opposing points, such as a micrometer-type measurement would accomplish. Maximum material condition (MMC—largest shaft size) should be checked for violations of a perfect cylindrical envelope, such as a GO gage would accomplish. Then, if functional gaging is not available or variable data are required, many measurement techniques may be employed to inspect the position of the shaft. A computerized coordinate measuring machine (CMM) is a tool commonly used for this purpose and will be discussed later in this book.

A surface plate type set-up (often referred to as an open set-up) may also be used. Because of the MMC symbol next to the geometric tolerance, the feature mating size—smallest cylindrical hole that will fit over the shaft without regard to orientation or location—must be determined. This can usually be accomplished quite simply through the use of fine or adjustable gages. This minimum circumscribed cylinder size will be used as the factor in determining "bonus" positional tolerance to be added to the original 0.4 diameter positional tolerance. Let us assume the feature mating size is Ø12.2.

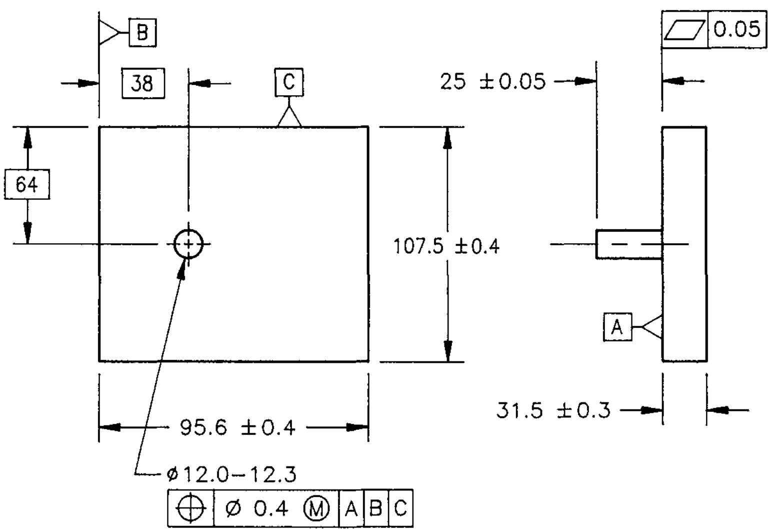


FIGURE 2-7

The part can then be positioned into the datum reference frame. For example, it would be nice if datum feature A could be pushed against an angle plate of sufficient size to allow feature A to establish the minimum of 3 high point contact to simulate datum plane A. But, of course, if the datum feature is on the same side of the part as the shaft being inspected, that would not be an easy thing to do. So, if that is the case, the primary datum feature may be established by clamping the opposing side of the part to the angle plate and, through the use of shim stock or leveling screws and a dial indicator, the datum feature can be "indicated in." In other words, if the datum feature is inaccessible to mount on, the shim stock (by placing varying thicknesses of it between the part surface and the angle plate at appropriate locations) can be used to make the datum feature parallel to the surface of the angle plate. This will have the effect of negating (within a range) the out-of-parallelism between the surface that is accessible (and on which you will mount the part to the angle plate) and the datum feature.

While maintaining this contact and orientation, the angle plate could be placed on a surface plate or machine table and datum feature B brought into a minimum 2 high point contact with this surface to simulate datum plane B. Another plate could then be brought into contact with 1 high point of datum feature C to position the part so that datum plane C is not only established, but established 90° to an edge of the first angle plate and 90° to the surface plate (or machine table) on which the part was mounted. This allows us to rotate the angle plate 90° and have datum planes B or C simulated by the surface plate (or machine table) on which the angle plate (which holds our part) is resting.

Let us assume we have done this and our part is oriented in its simulated datum reference frame. A height gage with an affixed dial indicator can now be used to determine the location of the shaft from simulated datums B and C.

Step 1: Rotate the angle plate so that datum plane B is simulated by the surface plate.

Step 2: Zero out the dial indicator on the surface plate and record the vernier reading from the height gage.

Step 3: Raise the height gage indicator to the top of the shaft and run the indicator over the shaft as close to the part as possible. Keep adjusting the gage height until the same zero is recorded on the top of the shaft by the indicator. (We are looking for the deviation from the 38 basic dimension but must later—in Step 5—take into consideration the pin diameter.) Take the reading of the height of the top of the pin from the vernier scale on the height gage.

Step 4: Subtract the reading taken in Step 2 from the reading taken in Step 3.

Step 5: Let's assume the difference in those readings equals 44.4. Sub-

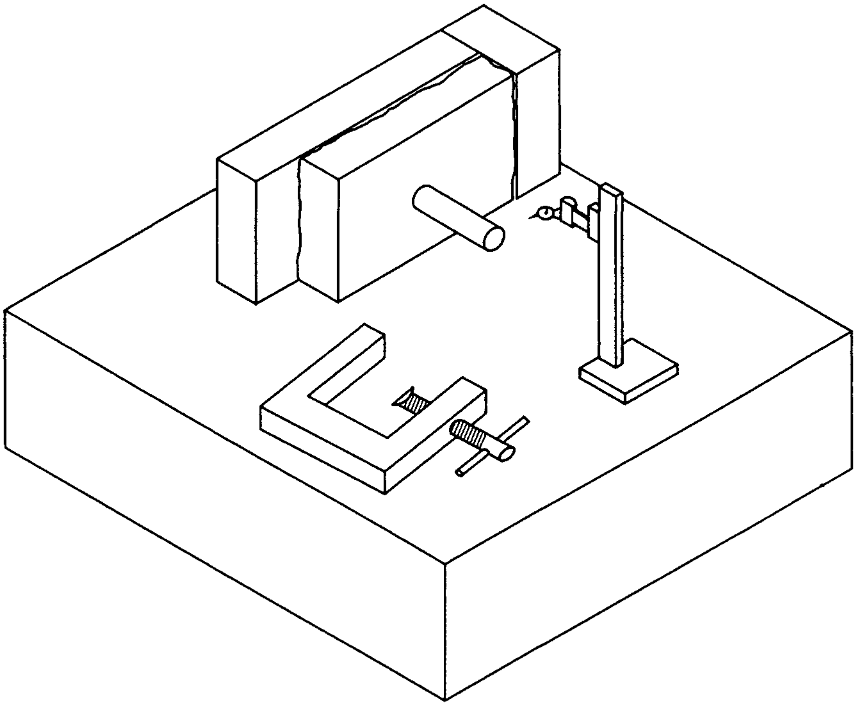


FIGURE 2-8

tract from this number one-half of the diameter of the shaft (minimum circumscribed cylinder).

12.2 divided by 2 = 6.1 Thus, 44.4 minus 6.1 = 38.3

We know we wanted to be at 38 but ended up at 38.3, a deviation of 0.3. Record this 0.3 deviation.

Step 6: Rotate the angle plate 90° so that datum C is now simulated by the surface plate. Repeat Steps 2–5 to determine the distance of the maximum deviation of the shaft axis from datum C. Let's assume the worst distance of the axis from C is 64.4—a maximum deviation from 64 of 0.4.

Step 7: We now know the bonus tolerance—the difference between MMC (12.3) and its actual feature mating size (12.2)—is 0.1. Since we started with 0.4, this gives us a total of a 0.5 diameter positional tolerance zone allowed. We also know the actual deviation from true position is 0.3 from datum B and 0.4 from datum C.

Step 8:

$$Z = 2\sqrt{x^2 + y^2}$$

Diameter of axial out-of-position | Deviation from true position from B | Deviation from true position from C

$$Z = 2\sqrt{0.3^2 + 0.4^2}$$

$$Z = 2\sqrt{0.09 + 0.16}$$

$$Z = 2\sqrt{0.25}$$

$$Z = 2\,(0.5)$$

$$Z = 1$$

FIGURE 2-9

Since we are allowed to be out of position a diameter of 0.5 and are actually out a little more than a diameter of 1, the feature is rejected as produced. If it had checked good, this process would also be performed at the other end of the shaft (farthest from the primary datum) and, if time permits, at various places in between.

When the indicator is brought into actual contact with the feature surface, rather than in contact with a tool representing mating size, surface variations must be taken into consideration. This direct contact type of check is sometimes made more valid if differential readings (180° from one another) are taken down the shaft and compared to determine the actual derived axial location. If any surface variations on the shafts or holes being checked are complex, this direct contact type of inspection technique can be time-consuming and difficult.

A similar procedure (such as described) will work for holes as well as shafts, and with slight modification is also accurate for other features of size such as elongated holes, slots, tabs, bosses, etc. Coordinate measuring machines (CMM) make this procedure easier and less time-consuming while using the same or similar basic steps. A functional gage may be used to collect attribute (accept or reject) data and is even faster than a CMM, but this type of receiver gage will not collect variables data, as will the CMM and the surface plate set-up.

However, "soft functional gages," which use data collected by (for example) a CMM, can give variables data concerning the direction and amount a feature has deviated from geometric perfection. These soft gages, which are computer generated, will be discussed more later in this text, as will the CMM and other measurement machines.

[**Inspection Note for Holes:** The maximum inscribed cylinder minus the maximum inscribed cylinder with perfect perpendicularity equals the out-of-perpendicularity of a produced hole axis. Also, the maximum inscribed cylinder minus the maximum inscribed cylinder with perfect position equals the out-of-position of a produced hole axis.]

3

Reading the Feature Control Frame

(For a Functional Understanding and to Guide Product Measurement)

3.1 JUST TALK THE TALK

Many people read only the symbols and not their implications. Without reading in the implications, it takes longer to understand what the control is trying to convey. I have some guidelines that I use when creating or interpreting a drawing, and generally they hold up if others involved in the process use similar techniques.

1. I read the diameter symbol as "of the axis" or "the axis may be out of"; therefore, a control that starts out $\boxed{\oplus \mid \varnothing}$ is read "position of the axis" or if you start with the diameter symbol and read backward, it would read: "The axis may be out of position."

2. The next thing one must understand is that the feature control frame is tolerancing. The dimensioning is somewhere else, usually also on the field of the drawing, but a completely separate component of the drawing. The dimensioning states the desired relationships, but the tolerancing states the allowed deviation from those relationships. More specifically, the feature control frame is there to state the allowed deviation from what would be considered geometrically perfect. The next item—the tolerance—does this. For example: $\boxed{\oplus \mid \varnothing.030}$ says: "The axis may be out of position a diameter of .030."

3. Now it is time to state or imply at what size the feature or features may be out of geometric perfection (in this case, out of perfect position). $\boxed{\oplus \mid \varnothing.030Ⓜ}$ could be read "The axis may be out of position a diameter of .030 if the controlled feature or features are produced at Maximum Material Condition." Of course, this implies that if the features of size being controlled depart from maximum material condition (MMC), staying within size limits, they

would be allowed to be out of their perfect position (move away from their true position) more.

Use of the Ⓜ is an indicator that the feature being controlled mates with something. It states that an alteration in size away from the MMC indicates an additional tolerance is called for, an excellent signal that the feature mates. That is when that statement would be true. If an Ⓛ, or least material condition (LMC) symbol, had been used, that would have been an excellent indication of a need to preserve material as in situations where the designer is concerned primarily with wall thickness problems that might occur. It is also used on casting drawings when it is a concern that there be enough material on the casting to be able to clean up in subsequent machining operations that may be called for on other drawings.

In both of these situations, the Ⓢ (which may be implied by omission), or regardless of feature size (RFS) symbol, may have worked, but would, by virtue of the fact that it allows *no* additional "bonus tolerance" to be drawn from size limits, have made the product potentially cost more. In fact, the circled M and L don't normally improve the function of a product. They are used in place of the circled S almost exclusively to reduce product cost. And as long as functionality is preserved, the use of the "bonus tolerance" symbols is a great idea.

There are times when the use of anything other than the RFS concept would endanger the ability of the product to function. For example, if balance is a concern, the use of the circled S symbol (especially after any datum features of size) is a good place from which to begin your final decision. This decision about whether or not to use the circled S is often based on how fast the features spin, how many are being made (because in a large run of parts it is often desirable to use functional gaging and the use of the RFS symbology makes that *very difficult*), and the material being used, among other factors.

For me, the use of |⌖|∅.001 Ⓢ|B Ⓢ| is an indication that this feature either does not mate with anything or is a spinning part or both. The circled S after the .001 says that this feature probably does not mate with anything, unless what it mates with expands into or onto it. The circled S after the datum feature (B) indicates that (a) the part spins and I am concerned with balance, or that (b) I do not want inspection to buy parts that can only be assembled off center (again maybe a balance item), or that (c) datum feature B does not mate with anything, or that (4) maybe B does mate with something, but that mating item is a completely separate component that would have no effect on the location of the feature being toleranced by this control.

4. Next comes the datum feature(s). First, it is important to understand the nature of the position symbol. If there is more than one feature in the pattern being controlled (a pattern of two or more holes, for example), then the position symbol itself begins by controlling the relationship between those holes. |⌖|∅.030 Ⓢ| applied to such a pattern already has the ability to control the posi-

tion between the holes to within that tolerance per hole. In other words, a basic dimension of 1.500 between two holes within a pattern would have a range of deviation similar to 1.500 plus or minus .030, to put the hole axes anywhere from 1.470 to 1.530 from one another.

If looked at in this context, datums are an extra added attraction, to lend orientation or additional location from part edge planes or other datum axes or centerplanes generated, for example, by holes or slots. So we say, *in addition to being positioned to one another*, to what else are the holes positioned? |⌖|⌀.030Ⓜ|A|B|C| could be read: "The hole axes may be out of position to one another and datums A, B, and C a diameter of .030 if they are made at MMC." This also implies that any hole not made at MMC but within size limits may be out of perfect position more than .030, and that the holes probably mate with shafts. If the hole axes are meant to be (drawn) perpendicular to datum plane A, then the same .030 tolerance zones that were controlling only the hole-to-hole tolerance are now oriented perfectly perpendicular to datum plane A. If the hole axes are within those dimensions given from each to the hole pattern, then the .030 tolerance zones are further restricted as stationary with their centers at precisely that location. If the hole axes stay within their respective tolerance zones, then they are not only in the allowed positional tolerance from one another (and the allowed perpendicularity tolerance to datum plane A) but now finally within the allowed positional tolerance from datum planes B and C.

This really is a universal language of symbols that, if learned properly, can be used to communicate with other interested parties, in other departments, other companies, and even other countries. It also will open up to you a whole array of other augmenting documents about product measurement, mathematics, software writing, manufacturing, and quality techniques that have either been written around this Dimensioning and Tolerancing standard or use it as an integral component of recommended procedures therein.

3.2 HOW TO READ A FEATURE CONTROL FRAME

Symbology contained in a feature control frame can convey information vital to the functionality and measurement of a part. To understand this symbology, one must not only be able to read the individual name of a symbol but also understand the implications and ramifications of the symbols when strung together in sentence or paragraph form. This section begins to explain some of the vital nuances in reading part functionality from a feature control frame.

Just as we have replaced many of the words on signs with symbols, we are replacing many of the notes on our engineering drawings with symbols. These symbols can, indeed, be read as one would read a sentence to describe how a part functions or is to be measured or made. For example:

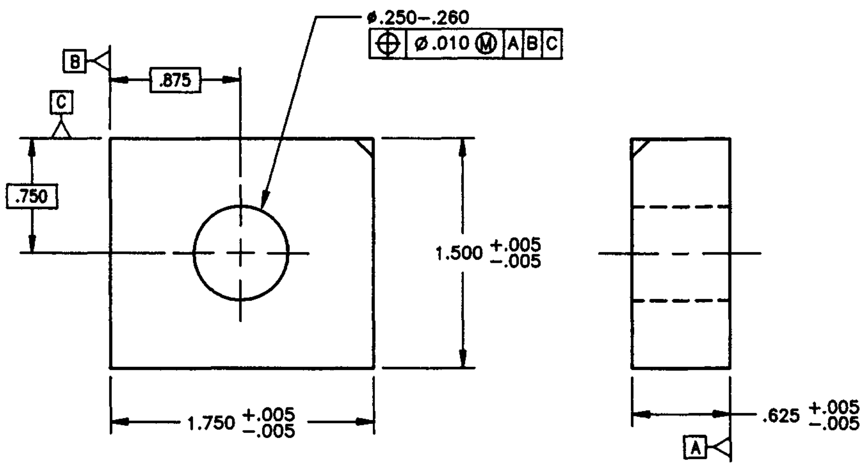


FIGURE 3-1

This feature control frame can be read, starting with the diameter sign:

Ø = The axis may be out of

⊕ = position

Then, reading the diameter sign again:

Ø = a diameter of

.010 = .010

Ⓜ = if produced at maximum material condition (a diameter of .250)

A = to A for perpendicularity

B = and B for location (holding the .875 dimension)

C = and C for location (holding the .750 dimension)

So, as a complete sentence, it may read, "The axis of this hole may be out of position a diameter of .010 if the hole is produced at a diameter of .250 holding perpendicularity to datum A, and location to datums B and C." It may also be read, "Position of the axis of this hole must be held to A for perpendicularity and to B and C for location to within a diameter of .010 if the hole is produced at .250." It also implies some important information. It implies that this hole may be further out of its perfect position if the hole grows. For example, .011 would be allowed for a hole made at a diameter of .251 and .012 for a .252 hole.

This is so because the hole uses the MMC symbol. The MMC symbol allows the geometric tolerance of location to grow as the hole grows and thereby creates a reduction in part cost by the acceptance of more of the parts that have been produced. This increase in geometric positional tolerance is only allowed because the functional requirements of the part are not endangered by this tolerance zone growth. This is based on the information that using the MMC symbol implies. It says that a larger hole has to be less perfectly located than a smaller hole. This is most often true when a hole mates. A shaft will more easily fit into a larger hole; therefore, the location of that larger hole can afford to be less perfect.

In fact, we may choose to read the control in an entirely different manner because of the circled M in use. For example, we might read it as: "Position of a diameter that mates while the part is seated on A and the hole is located from B and C." And, if it does indeed mate, we have to assume we are trying to preserve a mating boundary. That mating boundary is the collective effect of the actual size of the hole and the positional imperfection allowed to it.

For example, when a hole moves or leans out of its perfect location in space, its available area for mating at that location and angle closes up by the amount that it has moved or leaned. Since we cannot allow it to infringe on the boundary in which the mating shaft will reside or interference may occur, we protect that mating boundary (virtual condition-MMC concept).

So, a hole produced at Ø.250 may move or lean only Ø.010, while a hole produced at Ø.255 may move or lean out of its perfect location Ø.015 because we have determined the mating boundary to be protected to be Ø.240 and:

Ø.250 – .010 = Ø.240 Ø.251 – .011 = Ø.240 Ø.252 – .012 = Ø.240
Ø.255 – .015 = Ø.240 Ø.258 – .018 = Ø.240 Ø.260 – .020 = Ø.240

Therefore, we have determined that if the holes produced at these sizes are out of their perfect location by no more than an amount that would allow them to mate with a Ø.240 or smaller shaft, they will function and should be accepted by inspection as being within their functional limits of hole location.

This is only a bare-bones sketch of all the information that can be packed into one of these controls.

3.3 HOW TO READ A FEATURE CONTROL FRAME FOR PURPOSES OF MEASUREMENT

Figure 3-2 is a good one to discuss the way to measure a part for functionality. The feature control frames listed each serve a purpose to preserve this functionality. Each can be read as an integral part of how the part works. If one control is not met, we can determine its negative effect on how the part interacts with other parts or the next part it interacts with. And, we can be reasonably certain that it does interact with other parts in an assembly by reading the progression of controls given to this figure.

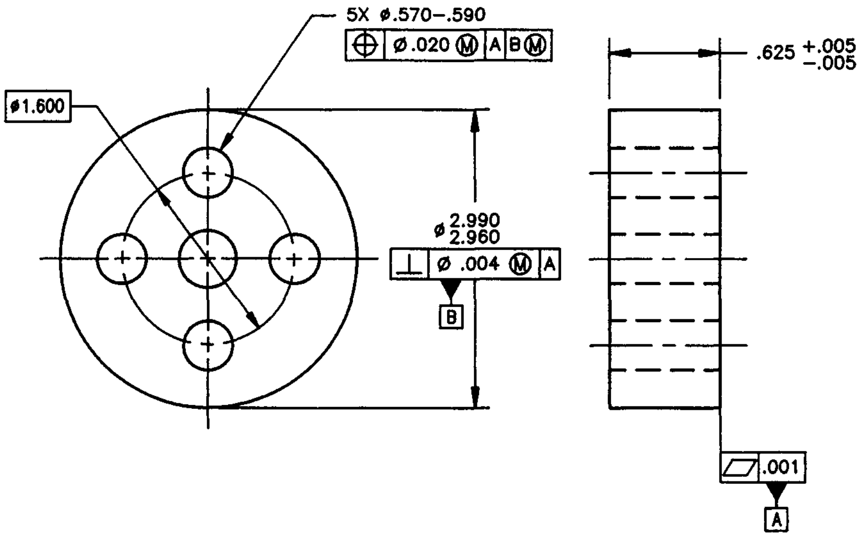


FIGURE 3-2

For example, a flatness control has been assigned to the primary datum feature. It can be speculated that this feature is a seating surface and that the flatness control is assigned to stop this part from rocking too much in the assembly. It may also be speculated that the surface it seats on has been assigned a flatness control of a similar value for the same reason.

If this is true, we know the two parts may rock on one another a maximum of the sum of their flatness tolerances. This sum would optimally provide no rock at all; but, as one can see, a flatness tolerance of zero would not be attainable, so a tolerance of .001 has been assigned to this part. This leads us to surmise that the sum of flatness allowable between the two seating surfaces may be about .002. When tolerancing a part, a balance must be struck between the perfect geometry desired and the costs associated with any tolerances assigned. When a tolerancing engineer assigns tolerances, it should be with the overall product cost in mind and how much each control will add to that cost. Too much tolerance and the part may not function. Too little tolerance and the part perhaps cannot be produced to comply or, if it can, the cost to do so may be so prohibitive that no customer is willing to pay for it. So, to be practical, we settle for what will function within acceptable parameters.

The datum plane for this primary datum A is struck ideally from the surfaces' three highest points of contact. In this case, when inspecting the flatness control, the measurement is to confirm that the datum feature surface low points and the datum plane struck from its three highest points of contact do not depart from one another by more than .001. If they do not, it will tell us that this surface resting on that plane is stabilized to within .001, doesn't rock more than that, and is in compliance with the flatness tolerance.

The second geometric control in the sequence of controls that define this part is the perpendicularity control. This control is an axial control with a cylindrical tolerance zone that is to enclose the axis of datum feature B. Datum axis B will then be struck from the datum feature, but the axis will be struck as perfectly perpendicular to datum plane A. Then this axis will act as the origin of measurement for the five-hole pattern.

We could read this perpendicularity control as "the axis may be out of perpendicularity a diameter of .004 if the feature is produced at maximum material condition to datum plane A." The implication is that if the feature is produced within size limits but at a size less than 2.990 (MMC), the tolerance zone would grow in size.

The tolerance stated in the control is a minimum. It isn't a maximum or even an average geometric tolerance for a produced feature. The feature, if produced at least material condition, would receive a tolerance of a diameter of .034 for perpendicularity of feature B to datum plane A. This is based on the functional reason for such a tolerance. This reason is easy to discover if the control is read in a different way. The second way is based on the concept that the MMC symbol is used after a geometric tolerance, most commonly if that feature mates. That being the case, the control could be read that we are holding "perpendicularity on a diameter that mates while the part seats on datum feature A."

If we were to read that into the control, we could extend the logic even more. If datum feature B mates while the part seats on A, then what does it mate with? Datum feature B is a shaft. A shaft mates with a hole. The hole that accommodates B and has a bottom that interfaces with A is a cavity of sufficient size to accept B in its worst case allowed relationship to A. That case might be considered to be B produced at its largest size and at its most allowed out-of-perpendicularity to A at that size. That size is Ø2.990, and that perpendicularity tolerance is a Ø.004. The worst case condition is the sum of those numbers, or Ø2.994.

However, the same worst case condition would occur even if datum feature B was produced at its smallest allowable size of Ø2.960, but experienced its full allowed perpendicularity tolerance to A at that size. For example:

At Ø2.960 the feature is allowed to be out of perpendicularity to A Ø.034. The sum of these numbers still equals Ø2.994.

So, we should now come to the conclusion that it is the job of the inspector at this stage of the definition to assure all concerned that datum feature B will indeed, not only comply with the tolerance, but that its tolerance, if complied with, will allow the feature to fit into a cavity as described (with a minimum area available for insertion of Ø2.994) while seated on datum feature A with three points of high point contact minimum. This is a virtual condition boundary that represents the worst case assembly condition for datum feature B.

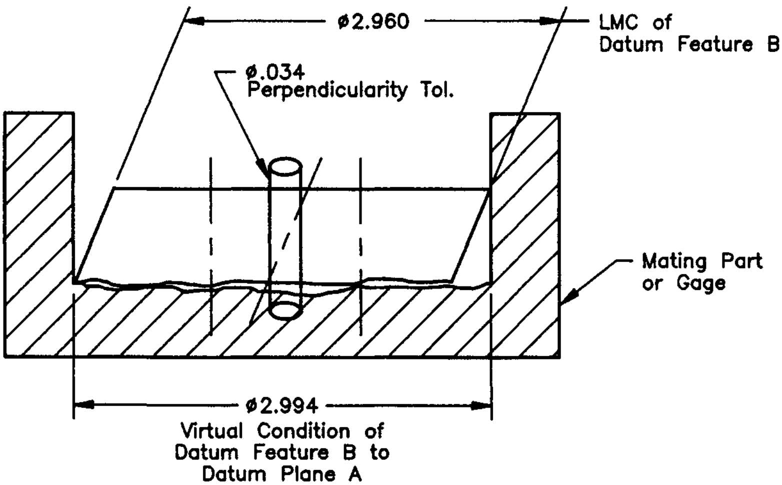


FIGURE 3-3

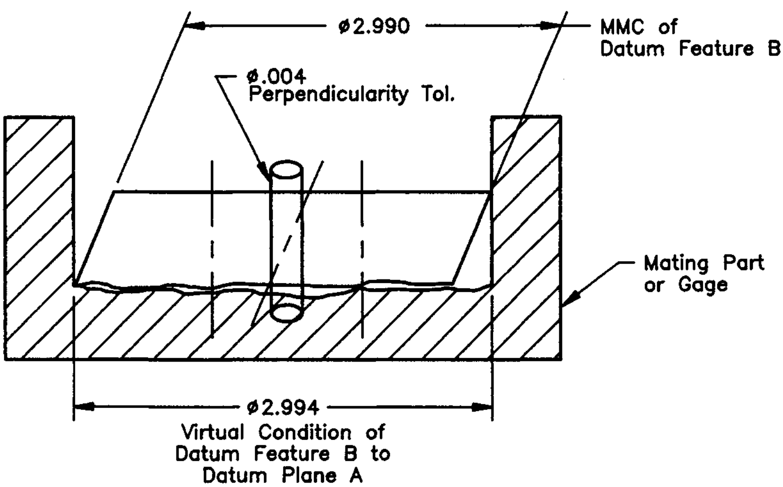


FIGURE 3-4

Sometimes, in this day and age, we forget it is the ultimate goal of the inspector to give assurances that a part or feature on the part being inspected not only meets tolerances, but that in doing so proves that the part or features will perform desired functions. By that same logic, we must look for multiple messages about functional requirements from the final geometric control on this part, the control of position.

The essence of any position control, when used on patterns of features, is to hold the desired angle and distance between the features within the pattern. The use of the position symbol relates the features within the pattern to each other. When datums are used, the relationship of the features to these datums must also be held. Sometimes, when the relationship between the features is more important than their relationship to the datums, composite position tolerancing is used (two levels of position controls sharing one position symbol). Or if position of the features to each other and to one or two of the datums is more important that the positional relationship to all of the datums, multiple single segment position controls are used (each level using its own position symbol). But, as is the case here, where not only must the features mate, but the datum features of size must mate (and with the same interfacing part in the assembly), one level of position is used to relate the features to each other and to the datums to within the same tolerance. All relationships stated within the control are of equal importance. The reason for that here is related by reading the control.

This positional control can be read as that we must hold the "position of diameters that mate while the part is seated on A and the holes are centered to B while B mates." Again, the maximum material condition symbols have been read as an implication that these features and datum feature B all mate with the same interfacing part in the assembly. If this was not true, then the MMC symbology should not have been used in the way it has been.

We have already gone over what datum feature B mates with; a cavity with a minimum diameter of 2.994. But now we know this cavity probably contains five pins (or five holes that will contain five pins eventually) that mate with the five holes being positional on the part we are discussing. Four of these pins are dimensioned on a bolt circle that matches the bolt circle on our part of Ø1.600, and 90 degrees to each other, while the other pin is centered in the cavity.

If the holes on our part have a minimum diameter (MMC) of Ø.570 and may move out of position a Ø.020 at the MMC size, then the area available for a pin to insert into right at the perfect (true) position is the difference between those two numbers, or Ø.550. If the holes were produced at a Ø.590, they would be allowed Ø.040 of movement, again creating a worst case boundary for mating conditions that is the difference between those two numbers of Ø.550.

That being so, we must assume that the pins in the cavity have been dimensioned and toleranced to have a worst mating condition boundary of their own that is Ø.550 or smaller (if a clearance is desired). They will mate with the holes on our part while the plate that the pins are mounted on (the bottom of the cavity) seats on datum feature A on our part. They will do so while datum feature B mates with the cavity.

The positional control could also have been read that we are "positioning five holes to each other for angle and distance, and to A for perpendicularity, and to B for location, to within a minimum diameter tolerance zone per hole of .020, if the holes are produced at MMC (a diameter of .570) or a maximum of Ø.040 per hole if the holes are produced at their largest size of Ø.590 (LMC)."

To make this explanation a little more clear, let's change the illustration to one with four holes instead of five. Although the meaning will be the same in most respects to the last illustration, to keep them separate, let's switch the primary datum to the letter L and the secondary to datum D.

Since the numbers are all the same (just four holes instead of five) as in Figure 3-4, the boundary explanation is the same for both.

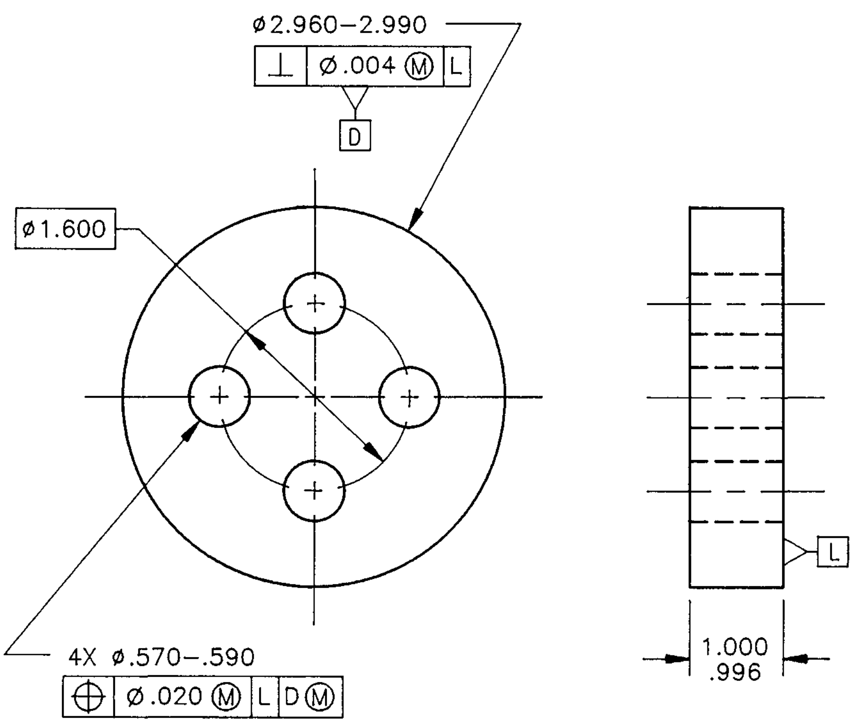
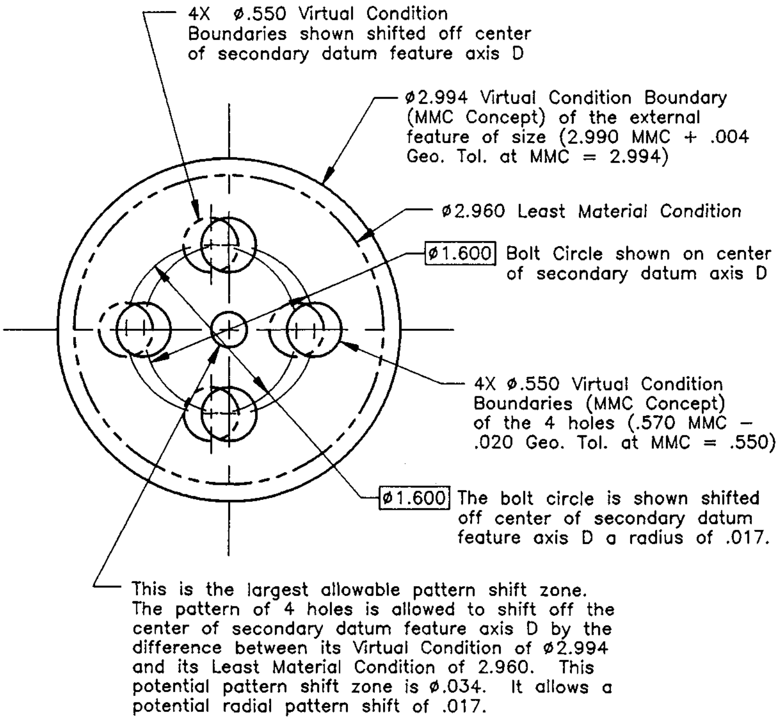


FIGURE 3-5



Since it is the datum feature that acts as the origin of measurement for the 4-hole pattern, the holes are seen as if they, as a group, have shifted in their relationship to the axis of secondary datum feature D (the actual diameter as produced) as shown above. In theory, it is viewed as the datum feature axis being allowed a departure from the datum axis, and in assembly this entire part may have to be shifted off the center of the imaginary datum axis to assemble.

FIGURE 3-6

Virtual condition boundaries (MMC concept) are generated by the collective effect of the maximum material condition of a feature of size and the geometric tolerance applicable at that size. The surface of the feature, such as the holes shown on the parts discussed, must not violate the virtual condition boundary (Ø.550) to comply with the individual positional tolerance given to it. The pattern of virtual condition boundaries theoretically begins perfectly centered to the secondary datum feature axis D (.800 radially from center), but because of the maximum material condition symbol used after datum feature D in the positional control for the hole pattern, this pattern may shift as datum feature D departs from its virtual condition boundary of Ø2.994.

This pattern shift is only allowed while maintaining the relationship between the four virtual condition boundaries of 90 degree angles to one another (and the primary datum) and their Ø1.600 bolt circle distance.

Since datum feature D may be made as small as Ø2.960 and could, in theory, be perfectly perpendicular to datum plane L at that size, the maximum pattern shift that may be experienced by the four virtual condition boundaries as a group is a Ø.034 or .017 radially. If the full amount of pattern shift is allowed, the pattern of four holes may be shifted off the center of secondary datum feature axis D radially an amount equal to one-half of the pattern shift zone (which is a Ø.034) or .017. This has the effect of allowing the four virtual condition boundaries as a group to deviate from their .800 basic dimension distance from the secondary datum feature axis of D by the amount of radial shift.

If this part mates with a part consisting of four pins in a cylindrical cavity, this pattern shift allowed by the maximum material condition symbol used after datum feature D (inside of the four hole's positional feature control frame) allows some parts to pass inspection that can only be assembled off center of the mating cavity. Possible negative effects of this are imbalance for spinning parts and nonuniformity of fit (unequal airspace between datum feature D and its counterpart cylindrical cavity). This pattern shift of the four holes as a group is in addition to the individual positional tolerance of movement around the Ø.550 virtual condition boundaries afforded to each of the holes. Size limits of each hole's MMC and LMC are verified separately from the conditions discussed here.

These geometric tolerances and the virtual condition boundaries they generate tell the inspector what he or she needs to know about the functional requirements of the parts described here. And whether the part has four or five holes and whether the datums are labeled as A and B or as L and D, we can generate a vision of worst case boundaries that can be represented in gages. These gages may be generated in software and a hard gage may be produced. Although the mating part may not be sized at exactly Ø.550 for the pins and Ø2.994 for the cavity, if our part as produced was capable of mating with such a gage, it may also be capable of fitting the worst mating part (if it was also dimensioned properly) without interference. This is so because what we have described here is a worst case scenario type receiver gage or known as a functional gage. This functional gage would be capable of checking both the perpendicularity control shown and the positional control simultaneously. It would not check feature size limits.

This gage may or may not be needed for purposes of inspection. Other inspection tools and equipment may be used to gather variables data. However, the process we just went through (visualizing the worst conditions allowed) is important and allows us to understand exactly what the drawing asks for and why.

The untoleranced gage for one of the parts discussed (the one with five holes) may appear as shown in Figure 3-7.

With only some of the dimensions and tolerances, the gage may appear as shown in Figure 3-8.

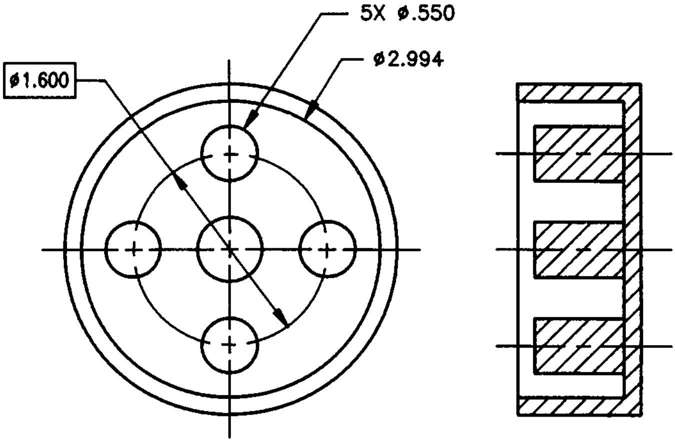


FIGURE 3-7

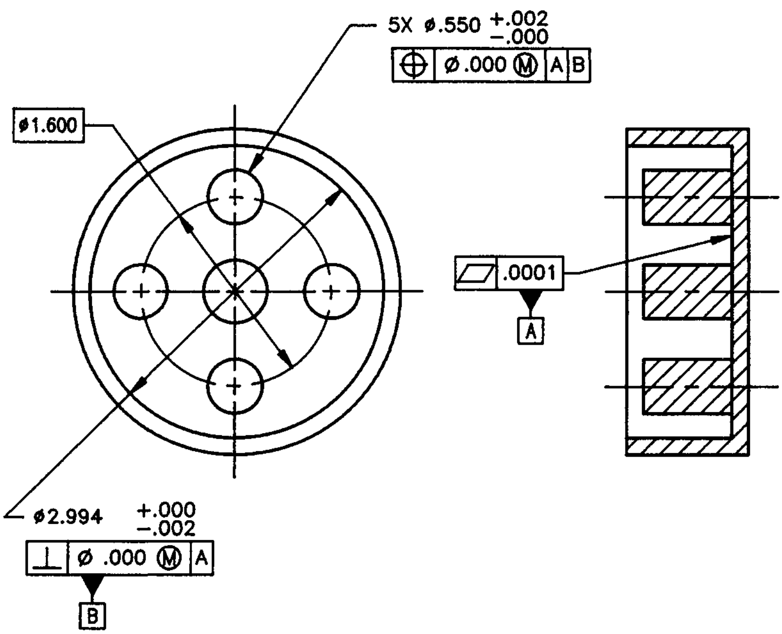


FIGURE 3-8

4

Functional Product Definition Creates Functional Inspection

4.1 INSPECTING PARTS AS THEY ARE USED

I visited a company a few years ago that made large, soft rubber parts. As soon as I entered, one of the quality engineers was waiting for me with product in hand. He shook it at me. It looked like a large piece of gelatin with holes. "Our inspectors are supposed to inspect the location of these holes," he said as the piece quivered and rippled in the air. "What do you think?" he asked, smiling. "Believe we can get it to conform to specs?" His point was well taken and graphically displayed.

I asked the designer how the part was used, and while he explained, I took notes. When he finished, I had the beginning of a good drawing note. It began, "These features are to be inspected in a restrained state," and went on to explain briefly—but specifically—how the part was restrained during use. The note was put on the drawings and a success story was born.

Recently, a client company complained that while their very long, thin beams inspected well within specified limits for parallelism, when in use they sagged so far in the middle that anything placed on them slid to the lowest point. I asked to look at some, and sure enough they looked pretty bad. The engineer scratched and shook his head as we stared up at the structure. "I don't get it. The thing inspected great according to the inspectors," he said.

"I notice it is suspended on each end with no support in the middle," I responded. He shook his head.

"How is it inspected?" I asked.

He shrugged his shoulders in a "you got me" fashion. I suggested we find out, so we visited the inspection department. As soon as we entered the room, we saw one of the beams sitting flat on a long surface plate, entirely supported along its length. An inspector was running an indicator over the surface of the beam's opposing side. He looked up at us and grinned. "Checks out great," he said. I got a look at the drawing and realized the designer had used an entire surface of the beam as a datum feature instead of using each end of that surface as datum target areas.

Thin-walled materials such as sheet metal, as well as some plastics and rubber parts, are often prone to wide variations in form when released from the restraints used to machine or form them. Holding these features to specific tolerances of geometric control requires close examination of when and how the part dictates the need for these controls. For example, if a sheet metal part is to be used in a restrained state (bolted, riveted, or otherwise clamped against several surfaces of the part), then perhaps the final shape or position (form, orientation, runout, profile, or location) of features should be judged to be in or out of control while in the restrained state.

A note added to the drawing, such as "This feature shall be inspected while restrained (bolted, clamped, etc.) to the specified datum(s) with the following torque," will allow the controlled features to be judged based on a simulation of the actual conditions the part will experience while in use. As such, any geometric tolerances assigned the features take on a realistic value, and the datum features are chosen and utilized based on proper criteria:

1. Function
2. Representation of the mating situation
3. Accessibility
4. Repeatability

4.2 FUNCTIONALITY CAN BE USED TO SUGGEST MANUFACTURING PROCEDURES

To Begin

Form control tolerances (like flatness and cylindricity), whether given as a direct geometric tolerance or deferred to the feature size limits under Rule #1, should be easily discernible.

Likewise, the tolerance on interrelationships (like angles and distance) between all features that comprise a part should be either stated directly or calculable.

Be Complete

One should look at the individual features (surfaces) first. Look at each one individually and determine whether or not its form is sufficiently toleranced.