New Jersey Institute of Technology
Digital Commons @ NJIT

Theses | Theses and Dissertations

Fall 1992

Design for productivity using GD&T

Srihari G. Acharya
New Jersey Institute of Technology

Follow this and additional works at: https://digitalcommons.njit.edu/theses

Part of the Manufacturing Commons

Recommended Citation
Acharya, Srihari G., "Design for productivity using GD&T" (1992). *Theses*. 1275.
https://digitalcommons.njit.edu/theses/1275

This Thesis is brought to you for free and open access by the Theses and Dissertations at Digital Commons @ NJIT. It has been accepted for inclusion in Theses by an authorized administrator of Digital Commons @ NJIT. For more information, please contact digitalcommons@njit.edu.

Copyright Warning & Restrictions

The copyright law of the United States (Title 17, United States Code) governs the making of photocopies or other reproductions of copyrighted material.

Under certain conditions specified in the law, libraries and archives are authorized to furnish a photocopy or other reproduction. One of these specified conditions is that the photocopy or reproduction is not to be "used for any purpose other than private study, scholarship, or research." If a, user makes a request for, or later uses, a photocopy or reproduction for purposes in excess of "fair use" that user may be liable for copyright infringement,

This institution reserves the right to refuse to accept a copying order if, in its judgment, fulfillment of the order would involve violation of copyright law.

Please Note: The author retains the copyright while the New Jersey Institute of Technology reserves the right to distribute this thesis or dissertation

Printing note: If you do not wish to print this page, then select "Pages from: first page # to: last page #" on the print dialog screen

NJIT
New Jersey's Science &
Technology University

The Van Houten library has removed some of the personal information and all signatures from the approval page and biographical sketches of theses and dissertations in order to protect the identity of NJIT graduates and faculty.

DESIGN FOR PRODUCTIVITY USING GD&T

by
Srihari G. Acharya

A Thesis
Submitted to the Faculty of
New Jersey Institute of Technology
in Partial Fulfillment of the Requirements for the Degree of
Master of Science
Department of Manufacturing Engineering
October, 1992

Approval Page

Design for Productivity Using GD&T

by
Srihari G. Acharya

Dr. Steve Kotefski, Thesis Adviser
Assistant Professor, Department of Manufacturing Engineering Technology

Dr. Rajpal S. Sodhi,
Director, Manufacturing Engineering Programs and
Associate Professor, Department of Mechanical Engineering,
New Jersey Institute of Technology

Dr. Nouri Levy,
Associate Professor, Department of Mechanical Engineering

BIOGRAPHICAL SKETCH

Author: Srihari G. Acharya

Degree: Master of Science in Manufacturing Engineering

Date: October, 1992

Undergraduate and Graduate Education:

- Master of Science in Manufacturing Engineering, New Jersey Institute of Technology, Newark, NJ, 1992
- Bachelor of Science in Automobile Engineering, P.E.S. College of Engineering, Karnataka, India, 1988

Major: Manufacturing Engineering

Positions Held:

- Graduate Assistant, Physical Education Department, New Jersey Institute of Technology, Newark, NJ. (January 1991 to August 1992)
- Lecturer, P.E.S. Polytechnic, Bangalore, India. (December 1989 to December 1990)

This thesis is dedicated to my Mom and Dad.

ACKNOWLEDGEMENT

The author wishes to thank his thesis adviser, Dr. Steve Kotefski for patiently reviewing the progress of the thesis at every stage and helping him to plan it efficiently. This thesis would not have been successful but for his invaluable guidance and sincere concern.

Special thanks to professors Dr. Nouri Levy and Dr. Raj Sodhi for serving as members of the committee.

Sincere thanks to Duane Felzak of Physical Education for his understanding and support towards my academic accomplishments.

Thanks also are due to the librarians at NJIT for their help in the survey of research papers on the subject of the thesis.

Lastly, a thank you to friends Nagasimha, Manjunath and Prashanth for their unique cooperation.

TABLE OF CONTENTS

LIST OF FIGURES

Figure **Page**

CHAPTER ONE
INTRODUCTION

1.1 Introduction

In recent years, new systems and new methods have evolved to improve productivity, manufacturing quality and cost in the manufacturing environment. The advent of computerization, made things faster and easier. Still the systems have many shortcomings. The production department is still no where achieving a high level of productivity. This is attributed to many things like labor, planning, designing, production, and inspection. Engineers and scientists are focussing more and more towards developing or employing new systems or methods, ignoring the fact that many systems that exist have some basic shortcomings. One of the defects lie in the traditional design language itself. To counter this ANSI Y 14.5M-1982 has come into being. This is a design language which is clear and precise and improves productivity.
By definition geometric dimensioning and tolerancing is a technique which standardizes engineering drawing practices, with respect to the function of dimensions and tolerances.GD&T is totally different from coordinate dimensioning or conventional dimensioning. The 150 year old coordinate dimensioning lacks GD&T's precise symbology, clear rules, and quality oriented design philosophy.

GD&T has gained acceptance in an manufacturing environment because it is the link that acknowledges machining capabilities and desired parts configuration via the utilization of graphical symbols for form, fit, and function requirements. The GD&T system allows one to maximize tolerance conditions of parts, while still maintaining inter-changeability characteristics.

The technique that GD&T uses above normal drafting practices is the datum reference, basic dimensions, and various geometric control characteristics, including perpendicularity, flatness, parallelism, and such as displayed in the table (1).These requirements are generally not specified in standard print specifications, but these additional specifications will further assure product compliance. This is one of the many reasons for the wide acceptance of GD&T concepts.

The authorative document governing the use of geometric dimensioning and tolerancing in the united states is ANSI 14.5M-1982, "Dimensioning and Tolerancing." This standard evolved out of a consolidation of standards, ANSI Y14.5-1973, USASI Y14.5-1966, ASA Y14.5-1957, SAE Automotive Aerospace Drawing Standards and MIL-STD-8C, October 1963. This consolidation was accomplished over many years by committee action representing military, industrial, and educational interests. The work of the committee has had and continues to have three prime objectives:

1) to provide a single standard for practices in the united states,
2) to update existing practices in keeping with technological advances and extend the principles into new areas of application,
3) to establish a single basis and "voice" for the united states in the interest of international trade, in keeping with the united states' desire to be more active, gain greater influence, and pursue a more extensive exchange of ideas with other nations in the area of international standards development.

The historic evolution of geometric dimensioning and tolerancing in the united states is an interesting story. It suffices to say that the early introduction functional gaging, giving rise to the possibility of new techniques, along with the growing need for more specifically and economically stated engineering design requirements, has caused it's growth. Advancing product sophistication and complexity, rapid industrial expansion, diversification have all created an environment in which more exacting engineering drawing communication is not desirable but mandatory for competitive and effective operation.

Updated and expanded practices have been initiated in the present Y14.5 standard. Further expansion will no doubt occur as growth in this area continues. In the process of extending into new areas, this expansion is confronted by the challenge of ensuring progress without upsetting stability. Rapid advances in this subject, although desirable, must be tempered by the ability to make transition with no loss of continuity or understanding.

1.2 ANSI Organization

The American National Standards Institute (ANSI) is the group whose charter is tasked with the development and monitoring of various standards. In particular, we are interested with the geometric dimensioning and tolerancing system.

Development of GDT standards was initiated in the 1940's, by a Stanley Parker of Britain, He had worked on problems that Britain was faced with complications in fabricated material compatibility and inter-changeability. So the fundamentals of GDT was established and concerns of run-out, perpendicularity, concentricity, parallelism and such were addressed.

In 1957, a meeting between Britain, Canada, and the USA was held in Toronto, Canada. This meeting was to coordinate a mutual system that would establish a standard for product fabrication via documentation control. At this meeting, it was realized that the USA had no formal technique of controlling meeting, it was realized that the USA had no formal technique of controlling geometric features that are considered vital to form, fit, and function of products.

It is important to note that other systems exist such as:

1) ISO - International Standards Organization, Which influences European and Orient Symbologies.
2) ABCA - American, British, Canadian and Australian standards, which are used by the respective nations in dealing with each other.

1.3 Problem Description

In a manufacturing set-up, there are many constraints to produce a part or product specified by the design department. As discussed in section 1.1 Labor, Production planning, Designing, production and Inspection all go hand in hand towards improving productivity. We need to have sufficient, skilled and understanding laborers in a good manufacturing set-up; without which any industry will not be able to sustain the quality of competition these days. Proper planning is essential in any kind of set-up, it could be short term planning or long term planning or a mixture of the both. A plan indicates as to what our goal will be and gives an insight of the steps that have to be followed in order to achieve this goal. If planning is good, it indicates that we are on the right track. Designing, Production and Inspection is the core of any system. The objective of the plan is to design and produce a product of quality.

Assuming now that the plan is good and labor is the best that is available, the burden lies on the production department to manufacture a product to design specifications and quality. Many a times the design department give tight tolerances that is very difficult to manufacture. The department give tight tolerances that is very difficult to manufacture. The time and hence cost to manufacture the same product increases. If a product is made under tight tolerances, the chances of making the right part on the first attempt is poor. So if the same part has to be produced again and again, it only indicates more time to make the product and the cost of the material, labor increases exponentially.

On the other hand if a relaxed tolerance is specified, the product is made but has problems when inspected. It is particularly true for mating parts. Apart from these problems, the traditional drafting language itself is a not a clear language to follow. Accumulation of tolerance is a good example and is illustrated in figure (12). The tolerance has been specified but it does not indicate the reference point. Hence chances are that tolerances are accumulated to one side, if the specified tolerances are all in their low limit. This is also true when the tolerances are all in their high limit.

As we have discussed, we observe some shortcomings particularly in the design language. With the help of specific symbols and datum references, GD&T helps to convey the message to the production department more clearly. This problem is discussed in section 3.4 with the help of figures (10), (11),and (12). Diversity of the product line and manufacture makes considerably more stringent demands of the completeness, uniformity, and clarity of drawings, which is been provided by GD&T, thereby reducing controversy and guesswork.

GD&T describes the form of the product or part clearly and describes the part with respect a datum. This is described and defined in section 2.2. Mating parts produced using traditional language always had problems during inspection, most of the parts were rejected even though the tolerances were kept under control using traditional drafting procedures. This is where

GD&T steps in to rectify the problem with the help of Location and Orientation characteristics, among others like Datum referencing.

When it comes to inspect the quality of the product, Functionally gaging the part using Functional gages of the physical kind is the most common and the simplest means of employing the technique. It is a popular method because, it represents the mating part and requires literally no skilled laborers to operate the same. When GD&T techniques are specified like MMC, RFS and LMC, functionally gaging the part is known to be the best. Still, using functional gages of the physical kind for these conditions is a problem, because of the specific nature of the condition and is discussed in chapter (6). The description and meaning of the conditions (LMC, MMC, RFS) are explained in chapter (4).

1.4 Research Emphasis

The intricacies of today's sophisticated engineering design demand, new and better ways of accurately and reliably communicating requirements is one of the reasons for GD&T and this is true in a manufacturing environment. This is one of the areas where importance is given in this research work.

To highlight the importance and accuracy of conditions like MMC, RFS and LMC, besides Perpendicularity and the concept of Bonus Tolerance (which will be discussed in detail in subsequent chapters), emphasis is also laid on in the usage of functional gages, their advantages and their shortcomings. An alternative method will be discussed to overcome this handicap like Paper gaging. Cost effectiveness of using GD&T will be discussed, also cost effectiveness of using Functional gages of the physical kind will be discussed. Moreover the variation in the cost will be analyzed when an alternative method is chosen to overcome certain peculiar when an alternative method is chosen to overcome certain peculiar situations like LMC and RFS. It will also be emphasized that it should be the 'spoken word' throughout industry, the military, and internationally on engineering drawing documentation.

CHAPTER TWO
TERMINOLOGY

2.1 GD&T Terms and Definition

To get a clear view of the concepts of GD&T, an understanding of its terms and definitions are important. These terms are used throughout, either using a symbol associated with the term or using a short term. Most of the terms described in the chapters, are defined below with some illustrations.

Actual size: An actual size is the measured size of the feature.

Angularity: Angularity is the condition of a surface, axis, or center plane which is at a specified angle (other than 90°) from a datum plane or axis.

Basic Dimension: A dimension specified on a drawing as BASIC (or abbreviated BSC) is a theoretically exact value used to describe exact size, profile, orientation, or location of a feature or datum target. It is used as the basis from which permissible variations are established by tolerances in feature control frames or on other dimensions or notes.

Bilateral Tolerancing: A bilateral tolerance is a tolerance in which variation is permitted in both directions from the specified dimension, 1.500±.005.

Center Plane: Center plane is the middle or median plane of a feature.

Circular Runout: Circular runout is the composite control of circular elements of a surface independently at any circular measuring position as the part is rotated through 360°.

Circularity: Circularity is the condition on a surface of revolution where all points of the surface intersected by any plane

1. Perpendicular to a common axis (cylinder or cone) or

2. Passing through a common center (sphere) are equidistant from the center.

Clearance Fit: A clearance fit is one having limits of size so prescribed that a clearance always results when mating parts are assembled.

Coaxiality: Coaxiality of features exists when two or more features have coincident axes, i.e., a feature axis and a datum feature axis.

Concentricity: Concentricity is a condition in which two or more features (cylinders, cones, spheres, hexagons, etc.) in any combination have a common axis.

Cylindricity: Cylindricity is a condition of a surface of revolution in which all points of the surface are equidistant from a common axis.

Datum: A theoretically exact point, axis, or plane derived from the true geometric counterpart of a specified datum feature. A datum is the origin from which the location or geometric characteristics of features of a part are established.

Datum Axis: The datum axis is the theoretically exact axis of the datum feature (a center line on the drawing) and the axis of the actual datum feature when its surface is in contact with the simulated datum; the smallest circumscribed cylinder (for external features) or largest inscribed cylinder (for internal features).

DatumFeature: A datum feature is an actual (physical) of a part used to establish a datum.

Datum Feature Symbol: The datum feature symbol contains the datum reference letter in a drawn rectangular box.

Datum Line: A datum line is that which has length but no breadth or depth such as the intersection line of two planes, center line or axis of holes or cylinders, reference line for tooling, gaging, or datum target purposes.

Datum Reference Plane: A datum reference frame is a set of three mutually perpendicular datum planes or axes established from the simulated datums in contact with datum surfaces or features and used as a basis for dimensions for design, manufacture, and measurement. It provides complete orientation for the features involved.

Datum Surface: A datum surface or feature (hole, slot, diameter,etc) refers to the actual part, surface,or feature coincidental with, relative to, and/or establish a datum plane.

Dimension: A dimension is a numerical value expressed in appropriate units of measure and indicated on a drawing and in other documents along with lines, symbols and notes to define the size or geometric characteristic (or both) of a part or part feature.

Feature: A feature is the general term applied to a physical portion of a part and may include one or more surfaces such as holes, pins, screw threads, profiles, faces, or slots. A feature may be individual or related.

Feature Control Frame: The feature control frame is a rectangular box containing the geometric characteristic symbol and the form, orientation, profile, runout, or location tolerance. If necessary, datum references and modifiers applicable to the feature or the datums are also contained in the box.

Geometric Characteristics:: Geometric characteristics refer to the basic elements or building blocks which form the language of GD&T. Generally, the term refers to all the symbols used in form, orientation, profile, runout and location tolerancing.

Position Tolerance:: A position tolerance (formerly called true position tolerance) defines a zone within which the axis or center plane of a feature is permitted to vary from true (theoretically exact) position.

Runout:: Runout is the composite deviation from the desired form of a part surface of revolution during full rotation (360°) of the part on a datum axis.

Virtual Condition (*Size*): Virtual condition of a feature is the boundary generated by the collective effects of the specified MMC limit of size of a feature and any applicable geometric tolerances.

2.2 Geometric Characteristics

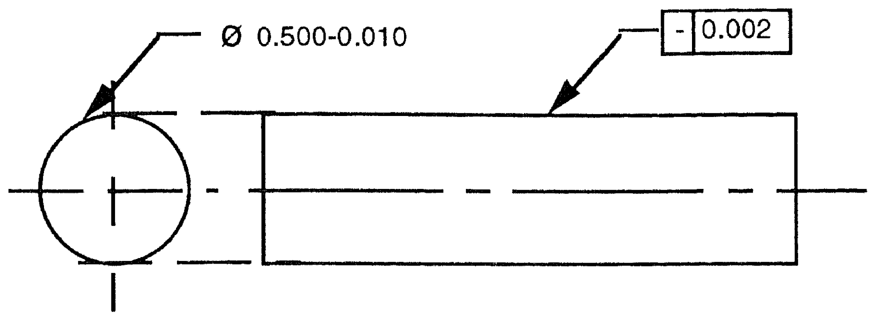
Geometric Dimensioning and Tolerancing controls particular desired features through the use of characteristic symbols. These characteristics is grouped for simplicity and similarity and also based on functionality. They are Form, Profile, Orientation, Runout and Location. These characteristics are described below.

1. FORM Tolerance: A form tolerance states how far an actual surface or feature is permitted to vary from the desired form implied by the drawing.

In this category, there are four representations for a component feature;

Straightness is a condition where form (shape) of a object is linear (straight). In establishing a linear condition controls can be established to monitor this condition. An example is shown in figure (1). Straightness of a size feature (control of axis) is more common and permits use of Maximum Material Condition principles. For any size specified within this range (as in figure (1) a straightness of 0.002 must be held. This control of straightness is in element lines only. The minimum and the maximum sizes can never be voilated.

Flatness measures planer properties. It is very similar to straightness. This is represented in figure (2), and the high and low limits of this



Meaning;

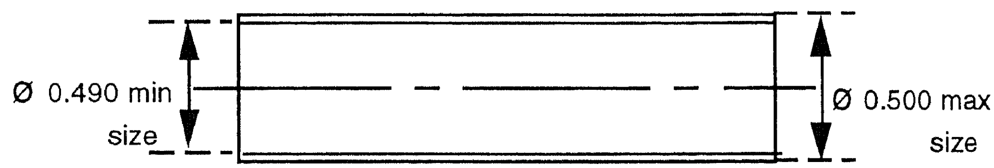


Figure 1 Straightness

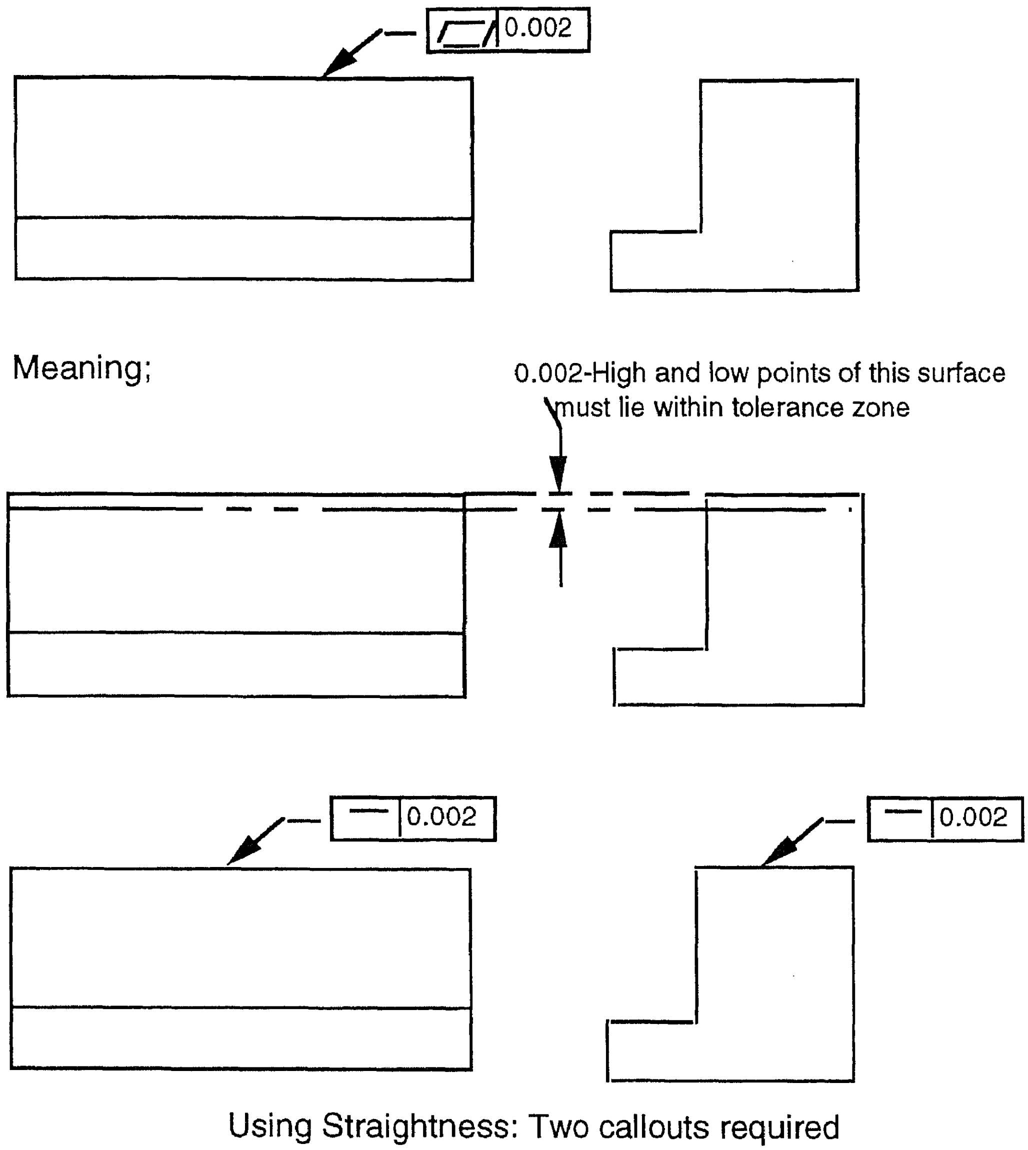


Figure 2 Flatness

surface must lie within tolerance zone. To represent the identical flatness condition using straightness, two callouts are required, as shown in the bottom of the figure. The left side view for straightness (let's say) in latitudinal sweeps while the right requires longitudinal sweeps. The net effect is the same as the flatness callout which assumes both sweeps simultaneously.

Circularity is a surface condition of cylinders, spheres, and cones. The surface condition is measured with respect to the circumference at a position that has a specific location and is perpendicular to the center of axis. The symbol for circularity is shown in table (1). An example is illustrated in figure (3).

Cylindricity is similar to circularity with the addition of taking length into account. Cylindricity can be related to total runout because it is concerned with the variances of a circular surface to that of a common axis. As illustrated in figure (3) the maximum and minimum sizes can never be violated. Any size between 0.248 and 0.252 are acceptable as long as cylindricity is within 0.001 inch per side.

2. ORIENTATION Tolerance: An orientation tolerance states how far an actual surface or feature is permitted to vary relative to a datum or datums.

   In the category of orientation, features such as perpendicularity, angularity, and parallelism is controlled. Orientation at the machinist level represents the requirements of tool and fixture calibration. It may indicate location of X, Y coordinates or indication of a central axis. Examples of Perpendicularity, Angularity and Parallelism are shown in figure (4). The *Angularity* feature is merely a linear movement about a common vertex and datum plane or axis. As shown in figure (5) surface

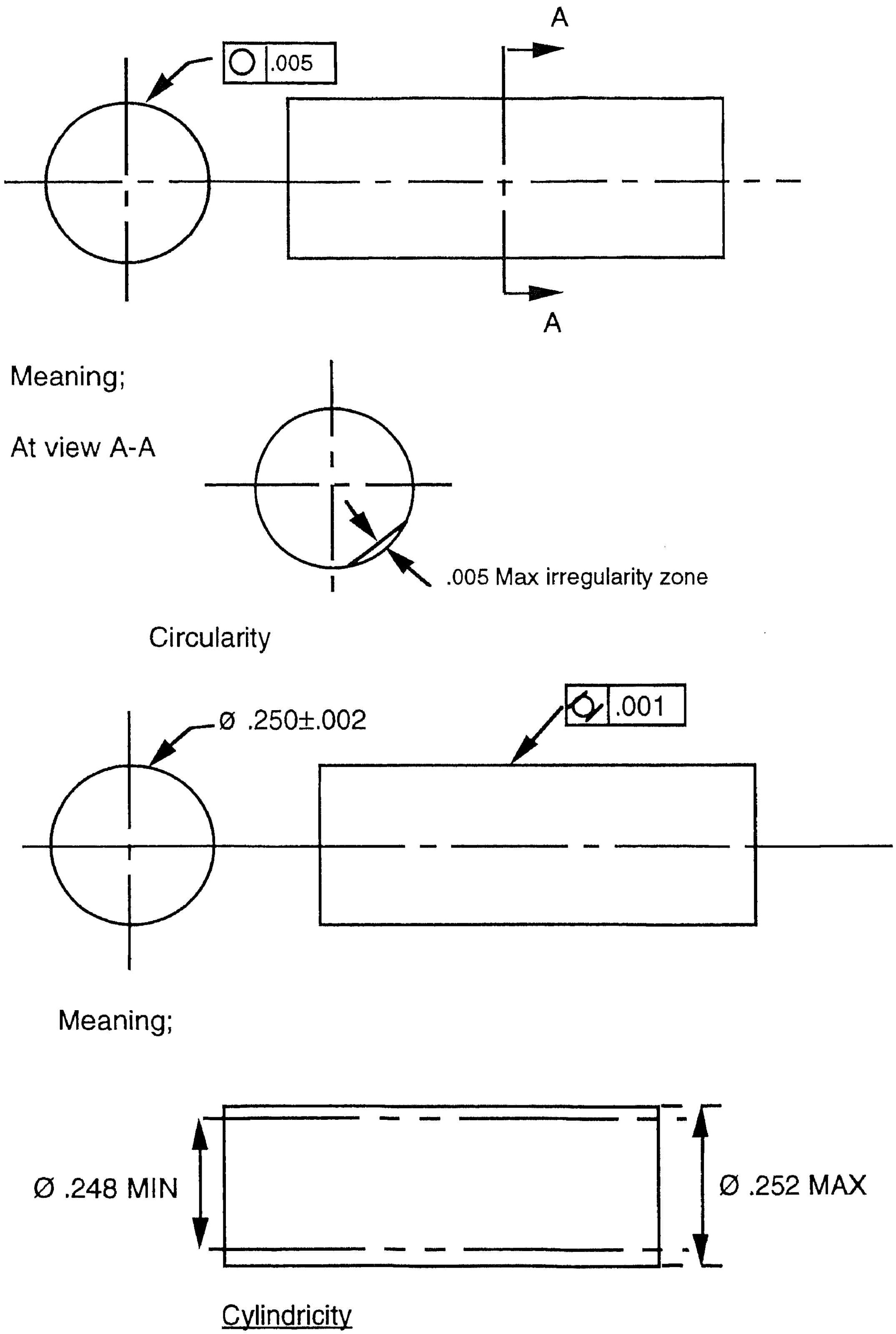


Figure 3 Circularity and Cylindricity

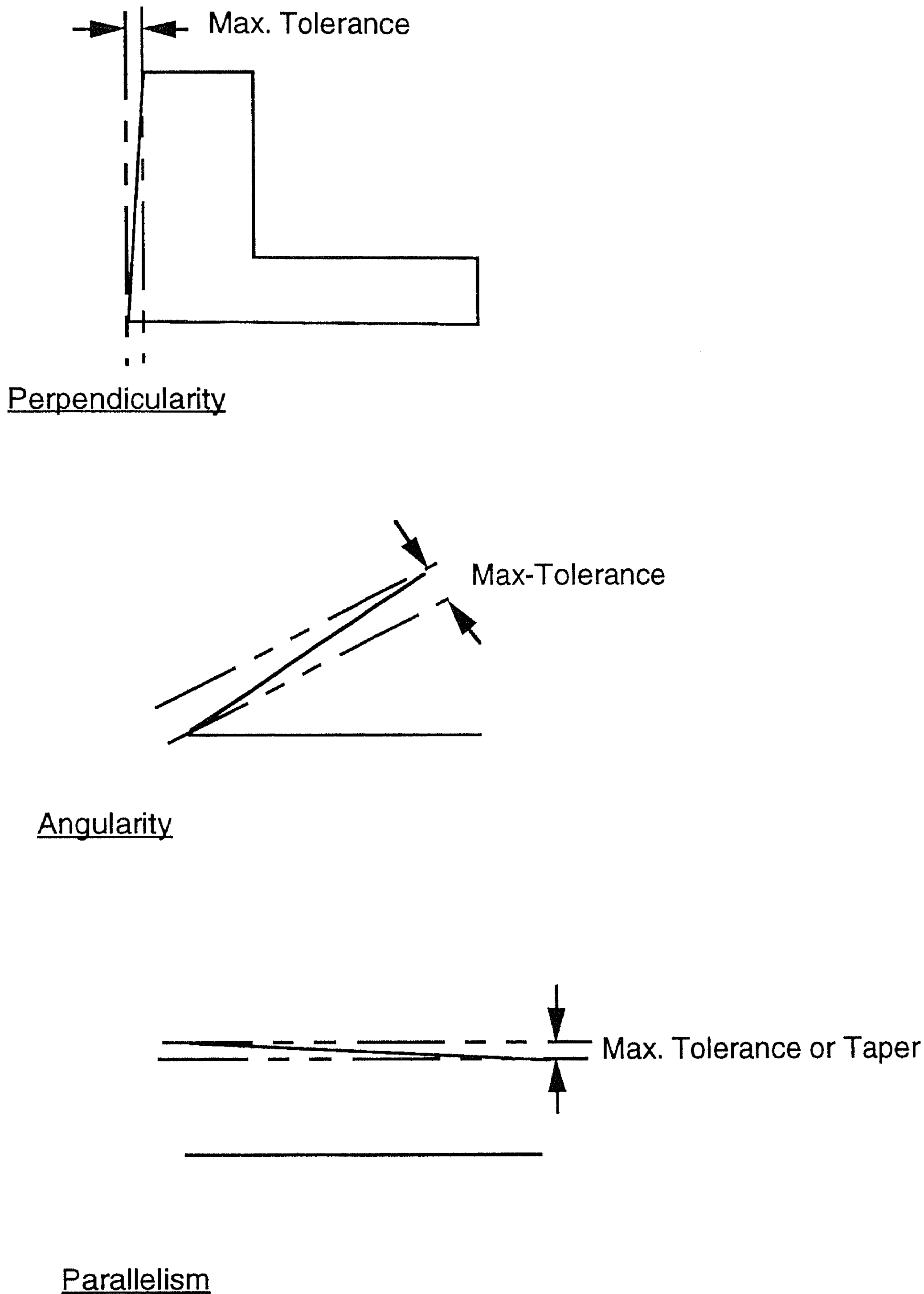


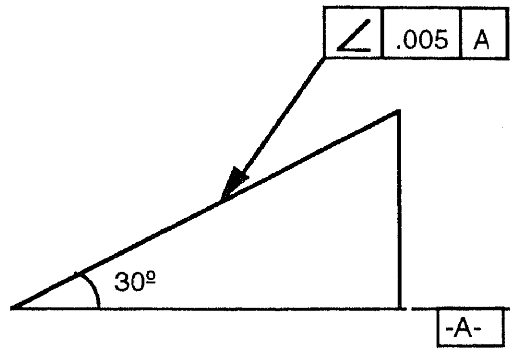
Figure 4 Orientation Characteristics

of the object must lie between phantom lines of 30° maximum/minimum ranges. *Perpendicularity* is a feature condition of a line or plane that is at a theoretical 90° to another datum line or plane. This feature is used to control *squareness* or *angularity* aspects of a component-- Very similar to angularity, except that the intended angle is limited to a theoretical value of 90°. *Parallelism* is the feature condition of having a line, axis or plane. This relationship generates orientations from datum surfaces so that proper calibrations can be created from imperfect surface areas. It can also be used for flatness control as in the illustration shown in figure (4). In the figure the surface area has a maximum taper allowance of 0.005 inches with respect to datum surface. one more point has to be noted that the parallelism is planer and not linear.

3. PROFILE Tolerance: A profile tolerance states how far an actual surface or feature is permitted to vary from the desired form on the drawing and/or relative to a datum or datums. The profile feature is a control of shape configurations. A profile is a condition of points, lines, and circles which can be controlled for considerations such as perpendicularity, concentricity, parallelism, angularity, and such.

   There are two types of profile features; *Profile of line* - which monitors the profile in single linear plane elements. Similar to cross sections. Profile of a surface - which monitors the entire profile surface desired for features. Figure (6) shows examples for both the cases.

4. RUNOUT Tolerance: A runout tolerance states how far an actual surface or feature is permitted to vary from the desired form implied by the drawing during full (360^0) rotation of the part on a datum axis.



Meaning;

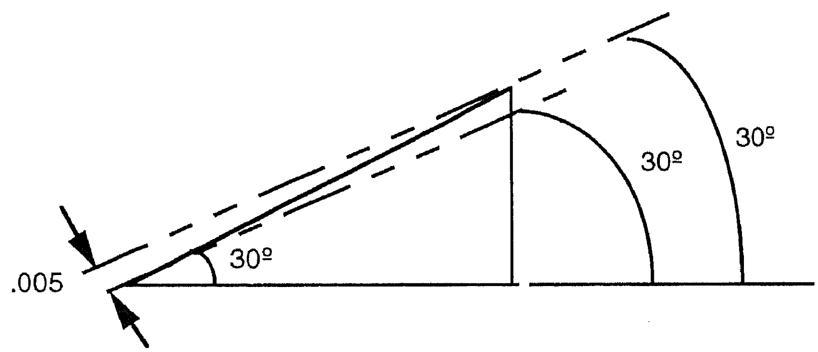
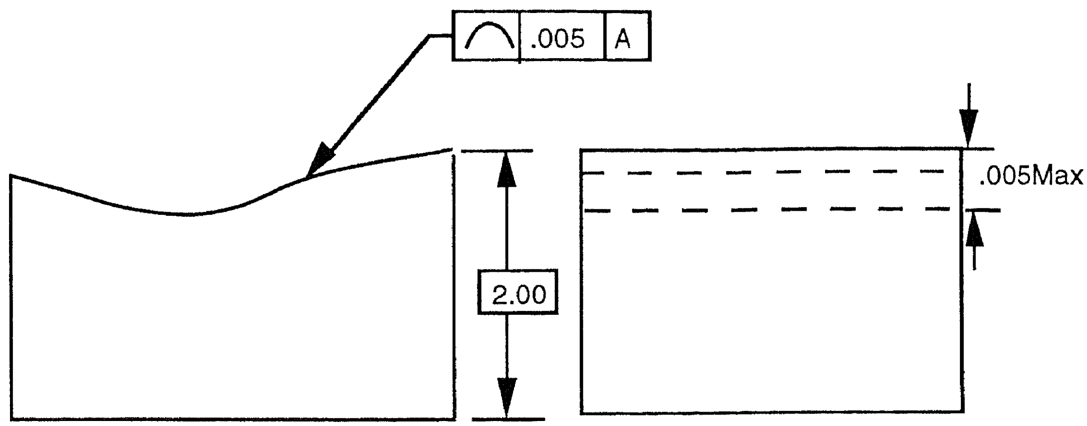
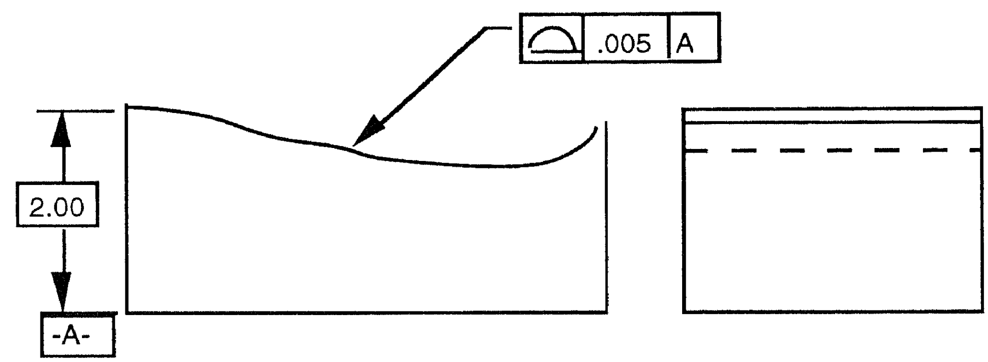


Figure 5 Angularity



Profile of a line



Profile of a surface

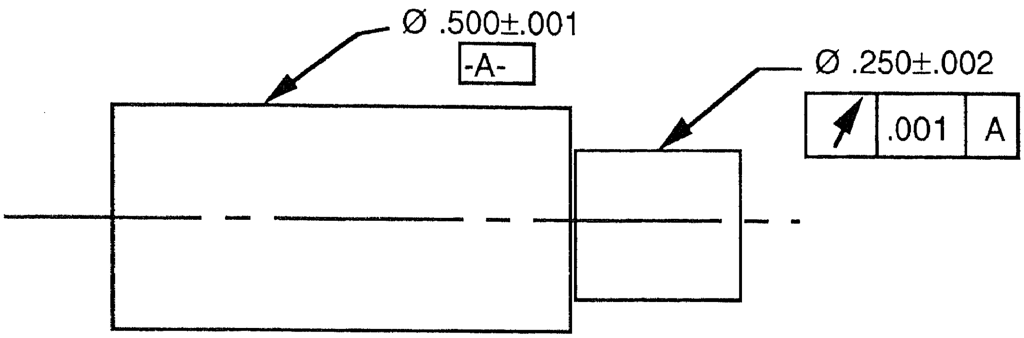
Figure 6 Profile Classification

The category of runout examines how circular an actual surface is with respect to its axis, in which the axis is generated from a control surface. In comparing the two variables, one can conclude that it is similar to a concentricity measurement with respect to a common axis of rotation. The difference is that the control surface generates the axis of rotation as in concentricity. The reason for run-out is that theoretical axes do not have to be located and then there is a large cost difference in terms of manpower and machine requirements between run-out and concentricity. Desired features are best controlled by the concentricity call-out because it is a axis to axis measurement. It should be noted that concentricity should never be used if either position and/or runout symbols can be utilized, for reason of cost effectiveness.

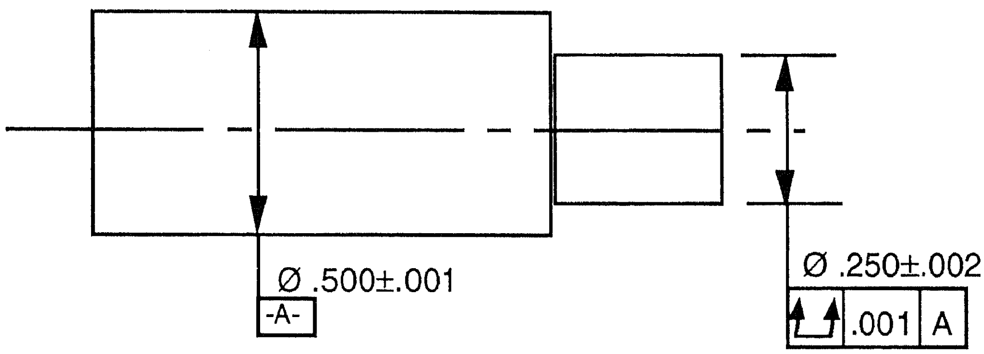
There are two runout call-outs that exist: Circular runout and Total runout. As shown in figure (7) circular runout indicates a out of round condition at a single position perpendicular to a common axis. Total runout is similar to circular run-out except rather than a single position it encompasses an entire surface area. This is illustrated in figure (7).

5. LOCATION Tolerance: A location tolerance states how far an actual size feature is permitted to vary from the perfect location implied by the drawing as related to a datum, or datums, or other features.

   GD&T indicates location of a dimension in two forms; *Position* and *Concentricity.* Position (both linear and circular) define a theoretical location from an axis or center. Once having established this theoretical location, variances can be measured from this ideal location. Reality has mandated the position tolerance which is a variation zone from the



Circular Runout



Total Runout

Figure 7 Runout Types

ideal condition. In the linear position, the location is a starting surface, line or point while in concentricity it is a center of axis.

It is important to note that concentricity measures axis to axis relationships. Concentricity is ideally applied under conditions where rotating parts require balancing and other dynamic considerations. Unfortunately the center of the axis is a difficult feature to locate and measure from; this is why runout callouts are prefered.

2.3 Kinds of Feature

The geometric features are also divisible into three kinds features to which a particular characteristic is applicable:

1. INDIVIDUAL feature: A single surface, element, or size feature which relates to a perfect geometric counterpart of itself as the desired form; no datum is proper nor used.

   All the form characteristics like Flatness, Straightness, Circularity, Cylindricity are grouped under this feature. As it is observed all these features relates to a perfect geometric form of itself as the desired form.examples are shown in figures (1), (2), (3).

2. RELATED feature: A single surface or element feature which relates to a datum, or datums, in form or orientation.

   Orientation, Runout, and Location characteristics are related kind of feature. A size feature (for e.g. hole, slot, pin, shaft) which relates to a datum, or datums, in form, attitude (orientation), in other words these are additional constraints to explain the situation in which it has to be produced. It is also very helpful for the inspection department to inspect the part. Here it is particularly critical since the inspection department has to know where to start their measurements from. In chapters (5), (6), and (7), a lot has been discussed about positional tolerancing. This positional tolerance is understood very well with the help of datums and other parameters such as run-out, since the position feature is related and measured from from these reference points. The symbols for all these characteristics are shown in figure (8).

3. INDIVIDUAL or RELATED Feature: A single surface or element feature whose perfect geometric profile is described which may, or may not, relate to a datum, or datums.

   Profile of a line and profile of a surface are examples of a feature being individual or related; i.e. that these two features can be independant or related to some datums or other parameters. These profiles are not a very key item during inspection, besides it can be easily manufactured and measured. Profile of a line is the condition permitting a uniform amount of profile variation, either unilaterally or bilaterally, along a line element of a feature. The profile of a surface is the condition permitting a uniform amount of profile variation, either unilaterally or bilaterally, on a surface.

2.4 Rules:

There are four important rules to understand in applying GD&T concepts, they are; (1) Limits of Size Rule, (2) Position Tolerance Rule, (3) Pitch Diameter Rule, and (4) Virtual /Datum Condition Rule. These are defined and described Below.

(1) Limits of Size Rule: Individual Features of Size- Where only a tolerance of size is specified, the limits of size of an individual feature

CHARACTERISTIC	SYMBOLS
Straightness	—
Flatness	⏥
Angularity	∠
Perepndicularity	⊥
Parallelism	//
Concentricity	◎
Position	⌖
Circularity	○
Symmetry	⌖
Profile of a line	⌒
Profile of a surface	⌓
Circular runout	↗
Total runout	⌰
Cylindricity	⌭
Datum Feature	-A-
Maximum Material Condition (MMC)	Ⓜ
Regardess of Feature Size (RFS)	Ⓢ
Least Material Condition (LMC)	Ⓛ

Figure 8 Geometric Characteristics and Symbols

prescribe the extent to which the variations in its geometric form as well as size are allowed.

Variations of Size- The actual size of an individual feature at any cross-section shall be within the specified tolerance of size.

Variations of Form (Envelope Principle)- The form of an individual feature is controlled by its limits of size to the extent prescribed in particular conditions. As seen in the figure the surfaces, or surfaces, of a feature shall not extend beyond a boundary (envelope) of perfect form at MMC. This boundary is the true geometric form represented by the figure (9). No variation is permitted if the feature is produced at its MMC limit of size.

Where the actual size of a feature has departed from MMC toward LMC, a variation in form is allowed equal to the amount of such departure.

There is no requirement for a boundary of perfect form at LMC. Thus, a feature produced at its LMC limit of size is permitted to vary from true form to the maximum variation allowed by the boundary of perfect form at MMC.

When perfect form at MMC does not apply:

The control of geometric form prescribed by limits of size does not apply to the following:

- Stock such as bars, sheets, tubing, structural shapes, and other items produced to established industry or government standards that prescribe limits for straightness, flatness, and other geometric characteristics. Unless geometric tolerances are specified on the drawing of a part made from these items, standards for these items govern the surfaces that remain in the "as furnished" condition on the finished part.

- Parts subject to free state variation in the unrestrained condition.

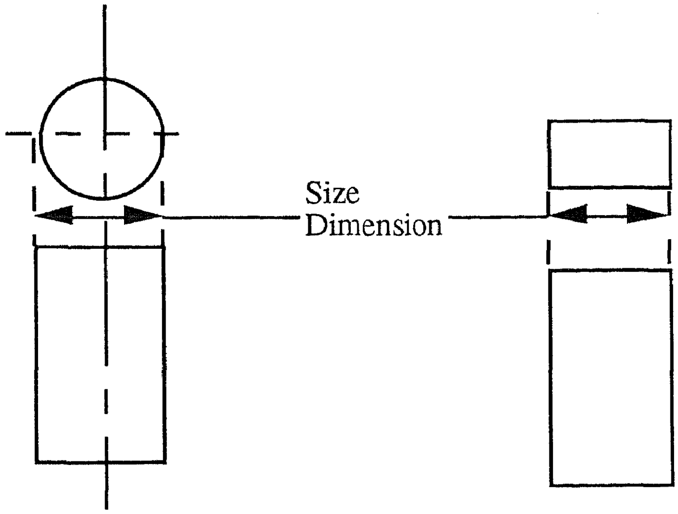


Figure 9 Individual Size Features

(2) Position tolerance rule: For a tolerance of position, MMC, LMC, or RFS must be specified on the drawing with respect to the individual tolerance, datum reference, or both, as applicable.

Other than position tolerance rule: For all applicable geometric tolerances, other than position tolerance, RFS applies with respect to the individual tolerance, datum reference, or both, where no modifying symbol is specified. MMC must be specified on the drawing where it is required.

(3) Pitch diameter rule: Each tolerance of orientation or position and datum reference specified for a screw thread applies to the axis of the thread derived from the pitch cylinder. Where an exception to this practice is necessary, the specific feature of the screw thread (such as MAJOR ø or MINOR ø) shall be stated beneath the feature control frame or beneath the datum feature symbol, as applicable. Each tolerance of orientation or location and datum reference specified for gears, splines, etc. must designate the specific feature of the gear, spline, etc. to which it applies (such as pitch ø, PD, MAJOR ø, or MINOR ø). This information is stated beneath the feature control frame or beneath the datum feature symbol.

(4) Datum/Virtual condition rule: Depending on whether it is used as a primary, secondary, or tertiary datum, a virtual condition exists for a datum feature of size where its axis or center plane is controlled by a geometric tolerance. In such a case, the datum feature applies at its virtual condition even though it is referenced in a feature control frame at MMC.

CHAPTER THREE
GD&T - A SUPERIOR LANGUAGE

3.1 GD&T - A Superior Language

This standard is a time proven element of our drafting language. Applied knowledgeably, GD&T is a powerful addition to drafting documentation practice that provides increased design and manufacturing flexibility, and it can ensure 100% interchangeability at optimum cost.

The ability to define and express the virtual condition within the GD&T language enables the engineer or designer to define the true functionally related maximum limits of production variability, while ensuring design integrity and, thereby, optimizing cost. By giving the designer the means to clearly express design intent and part requirements, GD&T enables the manufacturer to choose the proper way to produce a part. Eliminating tolerancing errors can help a company decrease scrap, rework, changes, confusion, and downtime.

GD&T ensures the design dimensional and tolerance requirements, as they relate to actual function are specifically stated and thus carried out. GD&T is considered as a superior language for it provides uniformity and convenience in drawing delineation and interpretation, thereby reducing controversy and guesswork. The use of datums, Form characteristics like Perpendicularity and parallelism make this language superior.

The large concepts in GD&T are solid. Some small refinements continue to be made in the language, as in the evolution of any language. But these refinements will not cause revolutionary changes in how GD&T is currently applied in designing a part and transmitting its functional requirements to the shop floor.

3.2 Modifications and Improvement

The below stated are some of the salient Features of GD&T. There are many differences between conventional drafting and GD&T, here emphasis is given to only to a few that is relevant to our case.

- In the conventional drafting procedure a square tolerance is provided, unlike for the GD&T where a circular tolerance is ensured. It is obvious from this point that more tolerance is provided in geometric dimensioning. Estimated increase in tolerance is 57%. This is shown in figures (15) and (16). Figure (15) represents the traditional 'plus minus' Tolerances and figure (16) represents the positional Tolerances.
- The problem in using co-ordinate dimensioning is that it is not able to meet the level of precision demanded by technologies such as computer aided design (CAD), computer aided manufacturing (CAM) and electronic gaging. This problem is being rectified by using GD&T.
- GD&T allows a product to be tested on paper rather than in the prototype form unlike in the conventional form of tolerancing. This is because GD&T is a more specific language and it tells us how and where to measure from with the help of datums and other characteristics, unlike the regular drafting procedure.
- GD&T's drawings are unambiguous, i.e. the rules govern size, location, orientation and form expressions for each part. In co-ordinate dimensioning the drawings are uncertain.
- GD&T is a powerful addition to documentation practice that provides increased design and manufacturing flexibility, and it can ensure 100% interchangeability at optimum cost. In the regular drafting, this problem is evident.

- GD&T uses datums, basic dimensions and geometric controls: which link tolerances to the size of the feature, define a virtual condition which is a key element of nearly every design. The virtual condition is frequently viewed as the combination of all worst cases of part variability for assembly.
- The technique that GD&T utilizes above normal drafting practices, is the datum reference including perpendicularity, flatness, parallelism etc. besides these, the concept of Bonus Tolerance makes the production department to manufacture parts comfortably and hence ensures zero rejection by the inspection department. This is been discussed in detail in chapter two.

3.3 Lapses in the Traditional Drafting

The main problem with conventional tolerancing using regular drafting practices is in the language itself. Many a times the designer likes to specify some things, but he does not have the words or symbols to say so. This is where Geometric tolerancing makes all the difference, It is a superior language by the use of datum references, basic dimensions and various geometric control characteristics including perpendicularity, flatness, parallelism and such as displayed in the symbol chart.

Datum reference: A Datum reference is a datum feature and the resulting datum plane or axis.

Basic dimension: A dimension specified on a drawing as BASIC (or abbreviated BSC) is a theoretically exact value used to describe the exact size, profile, orientation, or location of a feature or datum target. It is used as the basis from which permissible variations are established by tolerances in feature control frames or on other dimensions or notes. A basic dimension is symbolized by boxing it.

Perpendicularity: Is the condition of a surface, axis, or line which is 90 Degrees from a datum plane or a datum axis This condition is discussed in detail in section (4.1).

Flatness: Flatness is the condition of a surface having all elements in one plane.

Parallelism: Parallelism is the condition of the surface, axis or line which is equidistant at all points from a datum plane or axis.

Besides, Geometric tolerancing ensures flexibility and more tolerance, by the use of positional tolerancing.

Besides the language and symbols, the Tolerance that is specified in the drawing in reality is a square Tolerance. This is due to the traditional 'plus minus' Tolerances. Hence instead of a Diametrical Tolerance, as in positional Tolerancing, we will have to be satisfied with a square Tolerance, undergoing a loss of 57%. This is been discussed in section 4.5-Bonus Tolerance.

The inspection department has also encountered a heap of problems in measuring and checking the part for accuracy when traditional drawing parts are put forth. Functionally gaging these parts were also difficult. With the loss of Tolerance as indicated earlier in this section increases the cost per item. The 'plus minus' way of Tolerancing does not ensure interchangeability of mating parts at assembly. All the conditions and the characteristics of GD&T assure product compliance.

3.4 Rectification Using GD&T

In engineering practice, the focus of tolerance dimensioning is in the measurement of the finished piece. The questions that usually arise are What are the actual dimensions? Is perpendicularity true? Are parallel surfaces parallel? Are flat surfaces flat? Are cylindrical surfaces cylindrical? If there is a drilled hole, where is it and how big is it?

As an example of a problem in measurement, consider a flat surface. From where do you begin to measure? The flat surfaces may not be flat, the plane surfaces may not be a plane, and right angles may not be true right angles. These conditions are illustrated in figures (10)., and (11).

Another problem with dimensioning is tolerance accumulation. If several dimensions are in series, all of them consistently oversized or undersized, the accumulation of these tolerances all in one direction could make the part unusable. This is shown in figure (12).

As an example of the problems raised in conventional tolerancing, consider the three-holed part in the figure (13).
In particular, the tolerance zone for the geometric center of the upper right hole must lie within a square, 0.1mm on aside. The maximum deviation of the true position of the center of the hole would be 0.07mm, one half the length of the diagonal. If a through bolt were placed in this hole and through its mating hole, the allowance between the bolt and a hole would have to account for this maximum deviation. That is if the center of the hole were at the lower left corner of the tolerance zone, and the center of the mating hole were at the upper right corner of the tolerance zone, the bolt would just pass through both holes without interference. We would allow for a difference in the true position of the centers of the holes of 0.14mm.

Let us now assume that we have allowed for this variation in position in position and the hole diameters are 20.00. The maximum diameter of the of the through bolt is held to 19.86mm. But we only make use of this generous allowance if the centers of the mating holes are located on a diagonal. If the centers of the two mating holes are separated by 0.14mm but

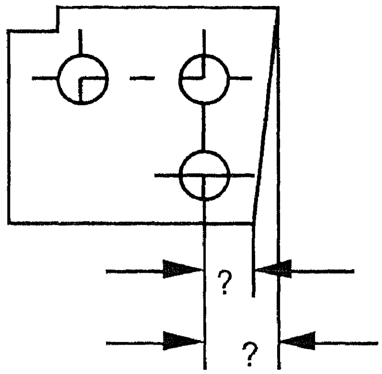


Figure 10

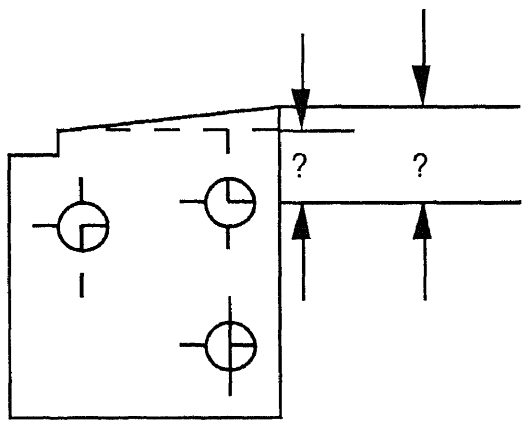


Figure 11

Figures 10 and 11 Common Problems in Measurement

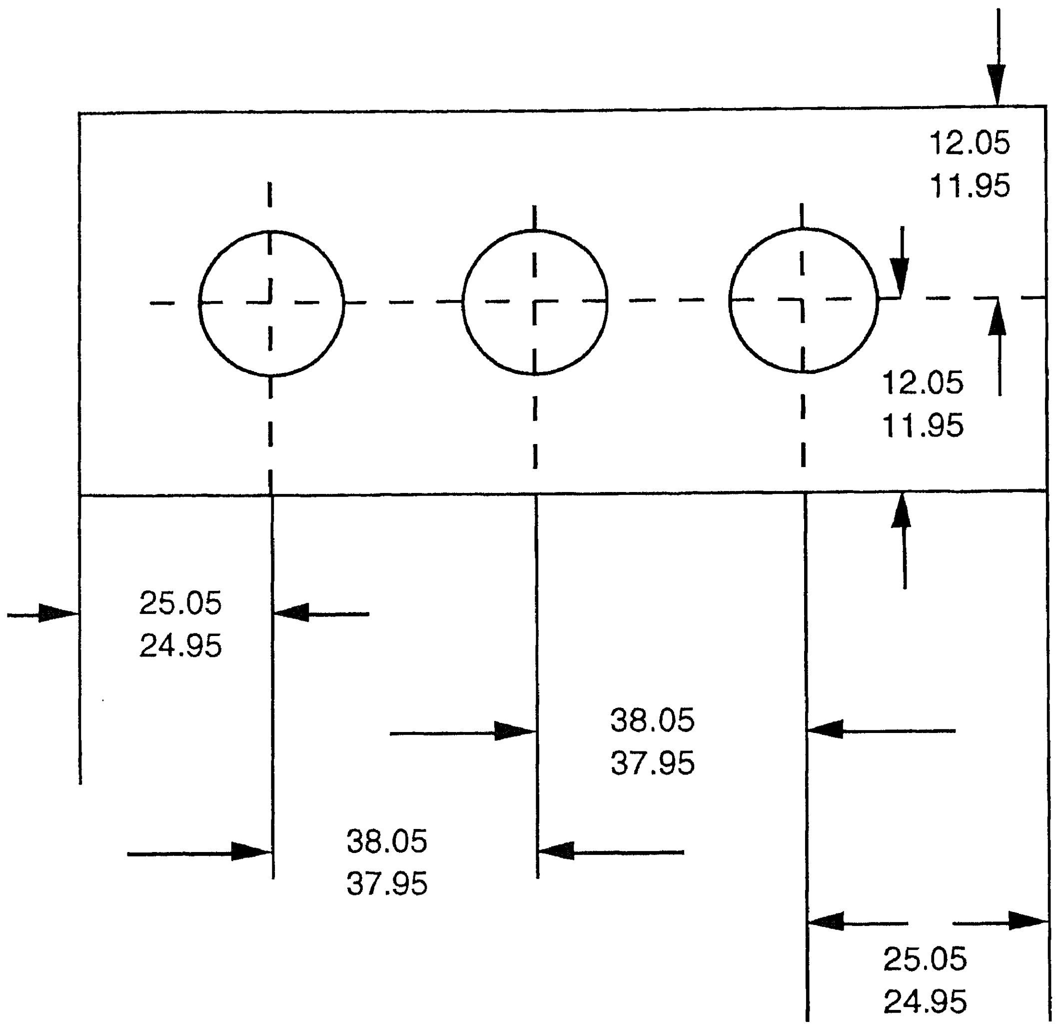


Figure 12 Accumulation of tolerance

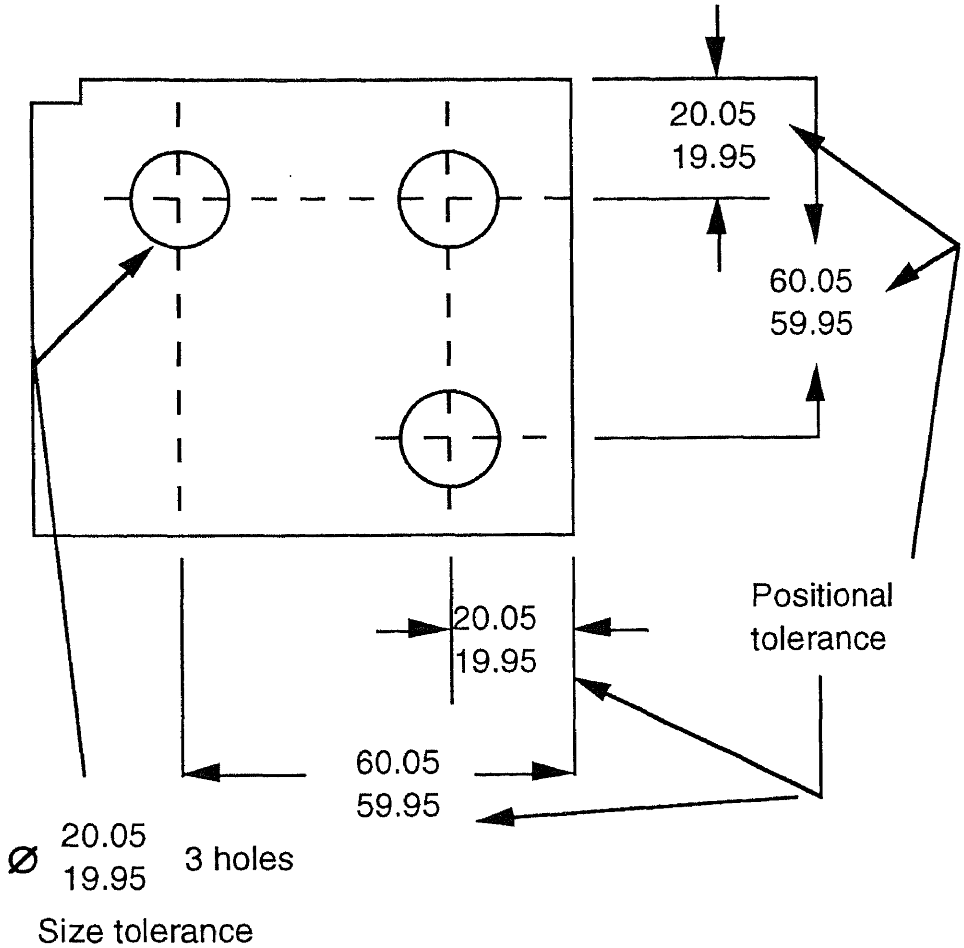


Figure 13 Problems Raised in Conventional Tolerancing

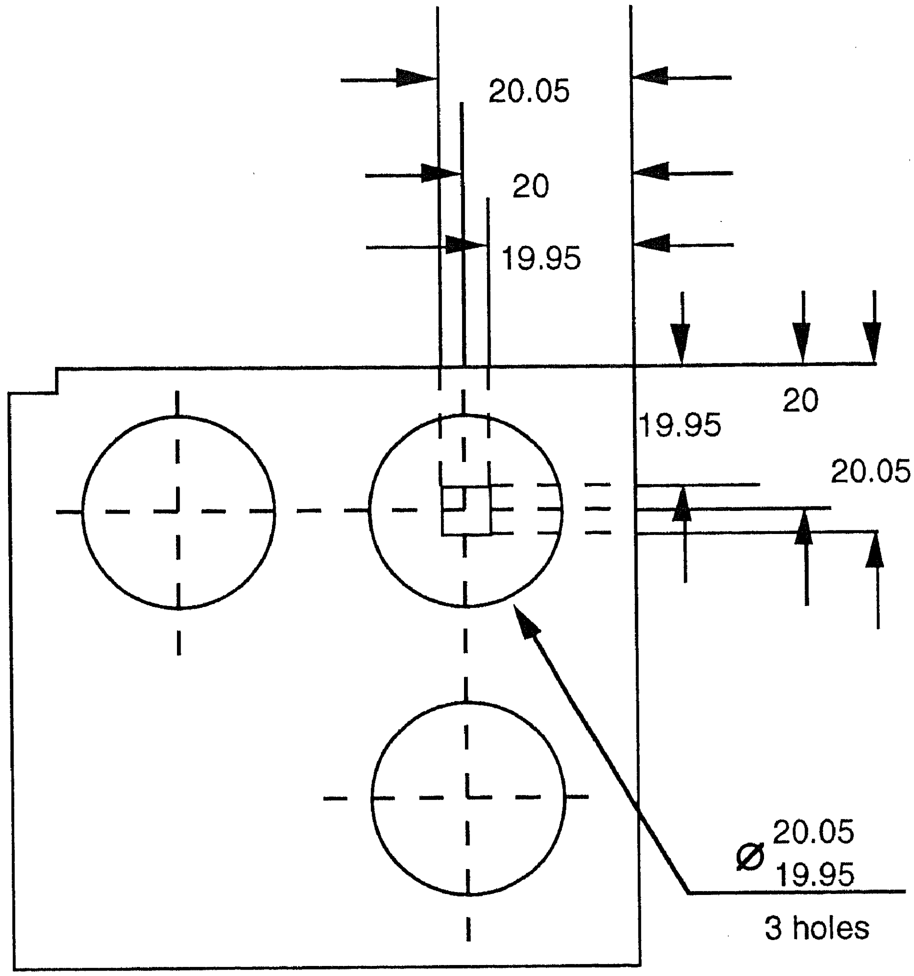


Figure 14 Description of a Square Tolerance

Let us now assume that we have allowed for this variation in position in position and the hole diameters are 20.00. The maximum diameter of the of the through bolt is held to 19.86mm. But we only make use of this generous allowance if the centers of the mating holes are located on a diagonal. If the centers of the two mating holes are separated by 0.14mm but are located on aline other than the diagonal, the parts would be rejected, even though the two holes would mate and receive the bolt. We have used a square tolerance zone, and the center to center distance between the two holes would lie outside the allowed tolerance zone except when the two holes line up on a diagonal. The figure (14). shows this situation.

We are now in the ridiculous situation of rejecting a part that would perform the service for which it was designed simply because the working drawing says that it should be rejected. This is not supposed to happen. Our ability to communicate design intent has been lost! An unacceptable part should not be usable.

This problem is corrected by *Positional tolerancing,* which locates the theoretically exact position of a feature, as established by basic dimensions. The use of position tolerancing results in a circular tolerance zone, and a circular tolerance zone is 57 percent larger than a square tolerance zone. More parts can be accepted.

In fig.(15), the location tolerance and the size tolerance for the circular hole are separated. All the details are given in a rectangular box. This is how GD&T states it. Measuring and inspecting a finished part to check it against the stated dimension in another problem.

Positional tolerancing (as shown in figure (16).) also removes the uncertainty about the origin of measurements. From where are

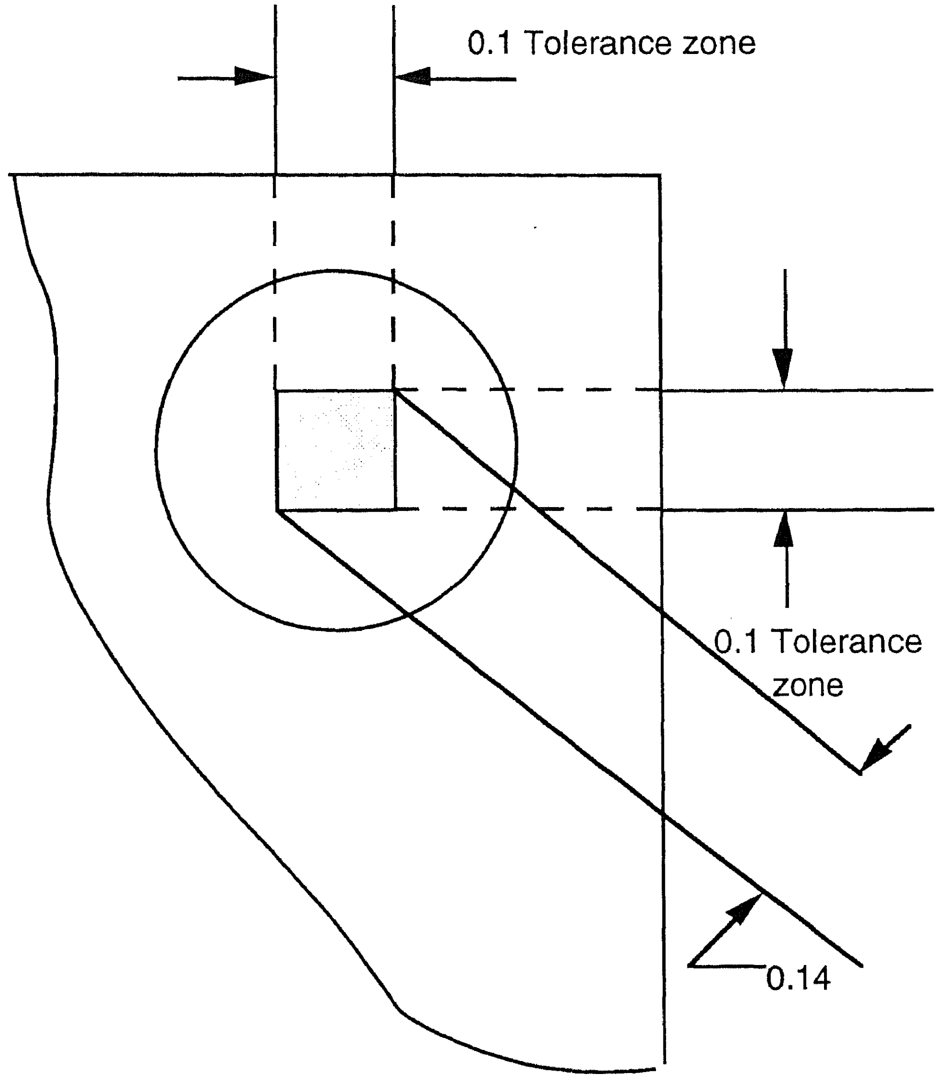


Figure15 Size Tolerance

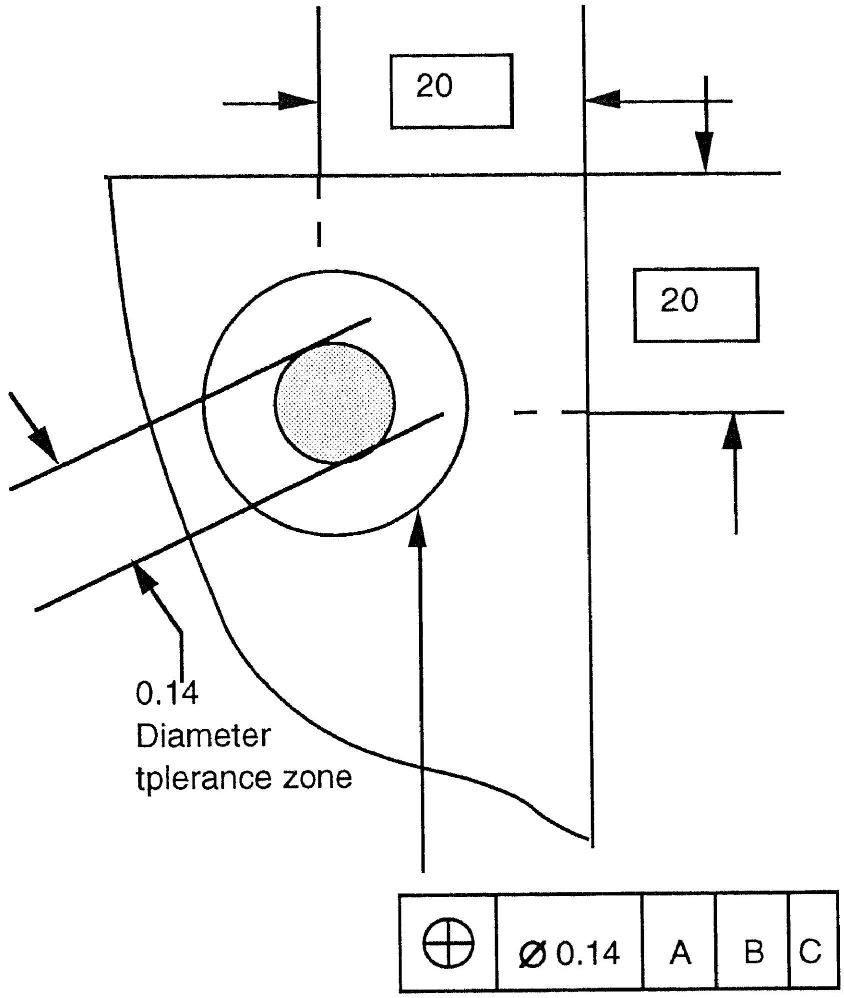


Figure 16 Positional tolerancing

measurements to be made? With conventional tolerancing, the origin is subject to interpretation, and different people interpret differently. Position tolerancing ties down the co-ordinates for measurement by specifying the datums from which measurements are made.

CHAPTER FOUR
PRODUCT DEVELOPMENT EMPHASIS ON:

4.1 Perpendicularity (Orientation)

Perpendicularity is a feature condition of a line or plane that is at a theoretical 90^0 to another datum line or plane. This feature is used to control "Squareness" or "Angularity" aspects of a component - very similar to angularity, except that the intended angle is limited to a theoretical value of 90^0. This condition is picturized in figure (17). The surface that is specified in the figure must be within the specified Tolerance of size and must lie between two parallel planes (.005) apart) which are perpendicular to the datum plane. Note that the perpendicularity tolerance applied to a plane surface controls flatness if a flatness tolerance is not specified (that is, the flatness will be atleast as good as the perpendicularity.

When perpendicularity tolerancing is critical, it may be necessary to limit the tolerance deviation to an amount equal to the feature size deviation from MMC. This assumes that the part form must be perfect at MMC size and that the virtual condition (size) can be no greater than that at MMC. The only permissible form tolerance must be acquired from the variation in part size in the increase of the feature hole size.

As seen in figure (18) Noncylindrical feature at MMC, datum a plane, the feature median plane must be within the specified tolerance of location. When the feature is at Maximum Material Condition (.500) the maximum perpendicularity tolerance is 0.005 wide. Where the feature is larger than its specified minimum size, an increase in the perpendicularity tolerance is allowed.

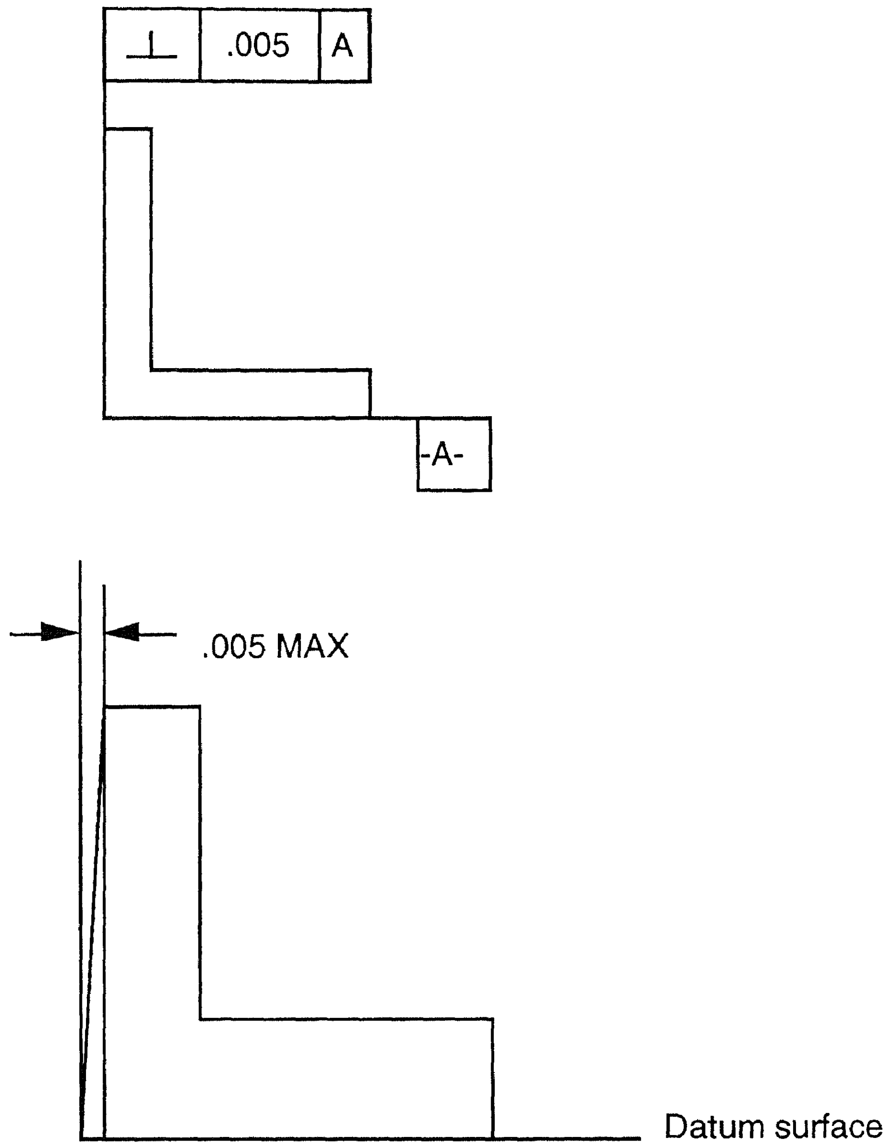
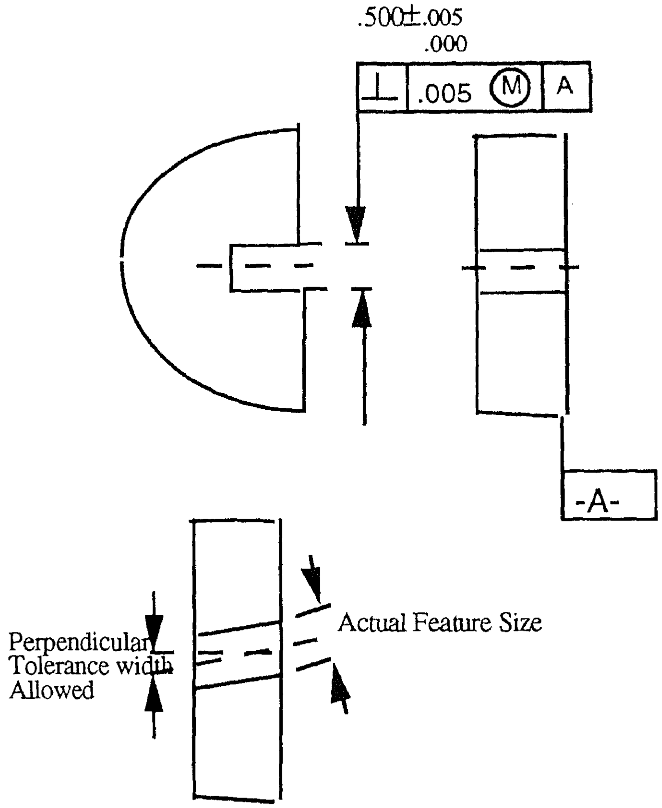


Figure 17 Perpendicularity



Actual Feature Size	Perpendicularity Tolerance Width Allowed
0.500 MMC	.005
0.501	.006
0.502	.007
0.503	.008
0.504	.009
0.505 LMC	.010

Figure18 Noncylindrical feature at MMC, Datum a Plane

4.2 Maximum Material Condition(MMC):

Maximum Material Condition may be defined as the condition in which a feature of size contains the maximum amount of material within the stated limits of size for example, minimum hole diameter and maximum shaft diameter. The MMC Principle is normally valid only when both of the following conditions exist:

1. Two or more features are interrelated with respect to location or orientation. (Example - a hole and an edge or surface, two holes etc.). Atleast one of the related features is to be a feature of size.
2. The feature to which the MMC principle is to apply must be a feature of size (Example - a hole, slot, pin etc.) with an axis or center plane.
3. MMC might also be considered as a " new" term for an "old" situation, such as the familiar terms worst condition, critical size etc., used in the past for relating mating part features. It is one of the most important concepts in GD&T. A thorough understanding of its meaning is essential.

Note in the figure (19)., that the MMC size of the 2.250+/-0.01 diameter hole is 2.240, or its low limit size. Whenever a hole is at its low limit size, it retains more material than if it were at its high limit or larger size, which will be 2.26 in our case.

Now it is also understood that a pin of 2.235 +/- 0.01 will be in MMC when the pin is at its high limit i.e. 2.245. This condition establishes the criteria for determining necessary form, orientation and positional tolerances. The symbol for MMC, the M enclosed in a circle and occasionally used abbreviation MMC are shown. The symbolic method of denotation is to be used with feature control frames only. Generally the use of MMC principle

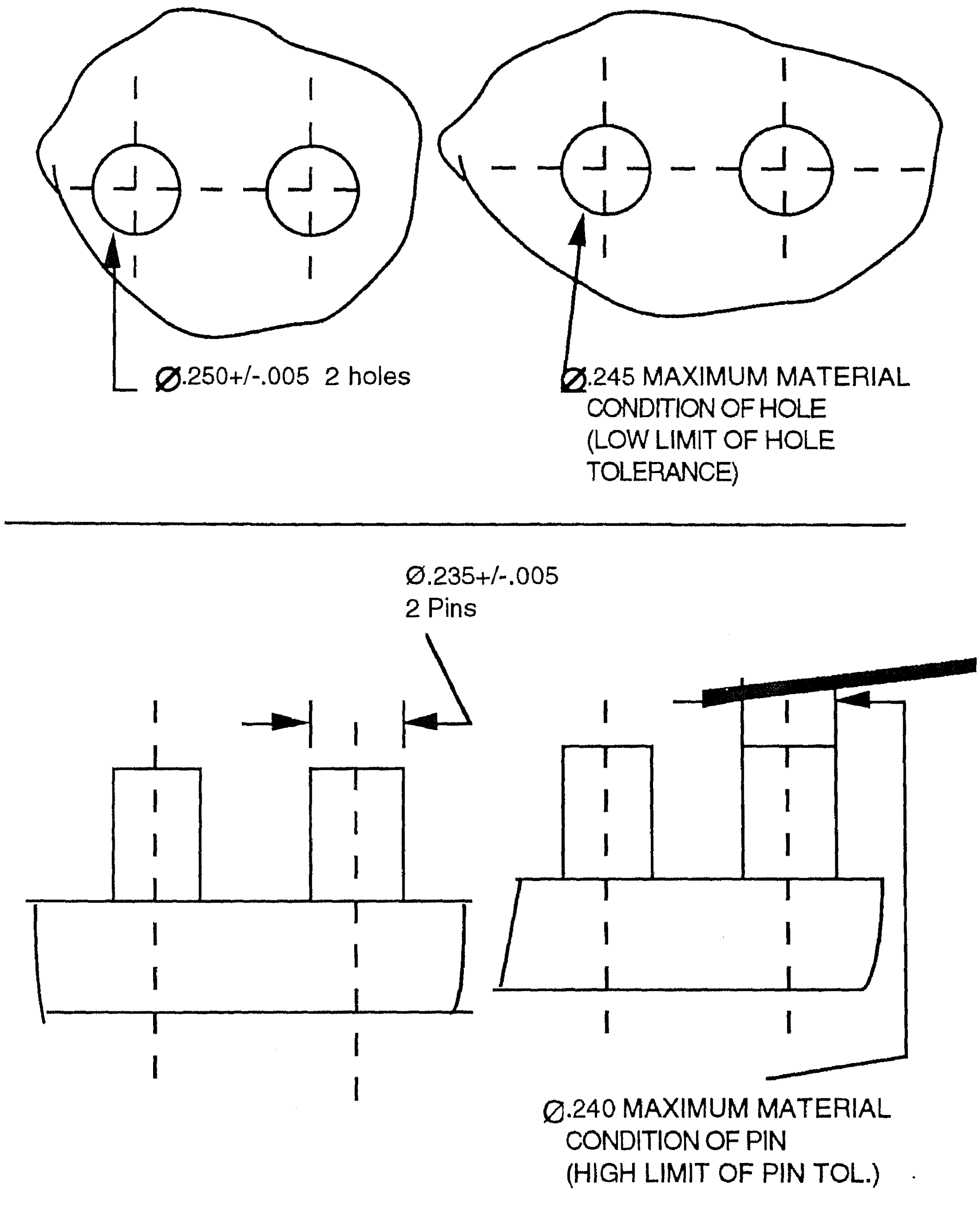


Figure 19 Maximum Material Condition

permits greater possible tolerance as part features vary from calculated MMC limits. It also ensures interchangeability and permits functional gaging techniques.

Now let us consider an application using the Maximum Material Condition for a tolerance on position. A bracket with two holes must fit over two mating cylindrical pins (Figure (20a). Figure (20b). shows a conventionally toleranced drawing. The Maximum Material Condition would be when the maximum size of the pins, at the maximum separation distance, must fit within two minimum holes. If the hole sizes are larger, the positional tolerance could be increased. This condition is shown in figure (20c). Using Maximum material Conditions for the hole, the tolerance on diameter could be increased from 0.02mm to 0.06mm if the holes were actually 5.10mm in diameter. What is more interesting is that we could change the size to 5.07mm, and the tolerance to 0.03mm, if zero tolerance were used at the Maximum Material Condition! We have now permitted a larger tolerance and permitted the tolerance to increase with an increase in the diameter of the hole, with no degradation of function (see figures (20d) and (20e).). Zero tolerance at Maximum Material Conditions permits the acceptance of the parts over the widest possible tolerance range. The acceptance of more usable parts means more production at less cost, which is what positional tolerancing is all about.

4.3 Regardless of Feature Size (RFS) :

RFS is defined as "the term used to indicate that a geometric tolerance or datum reference applies at any increment of size of the feature within its size tolerance".

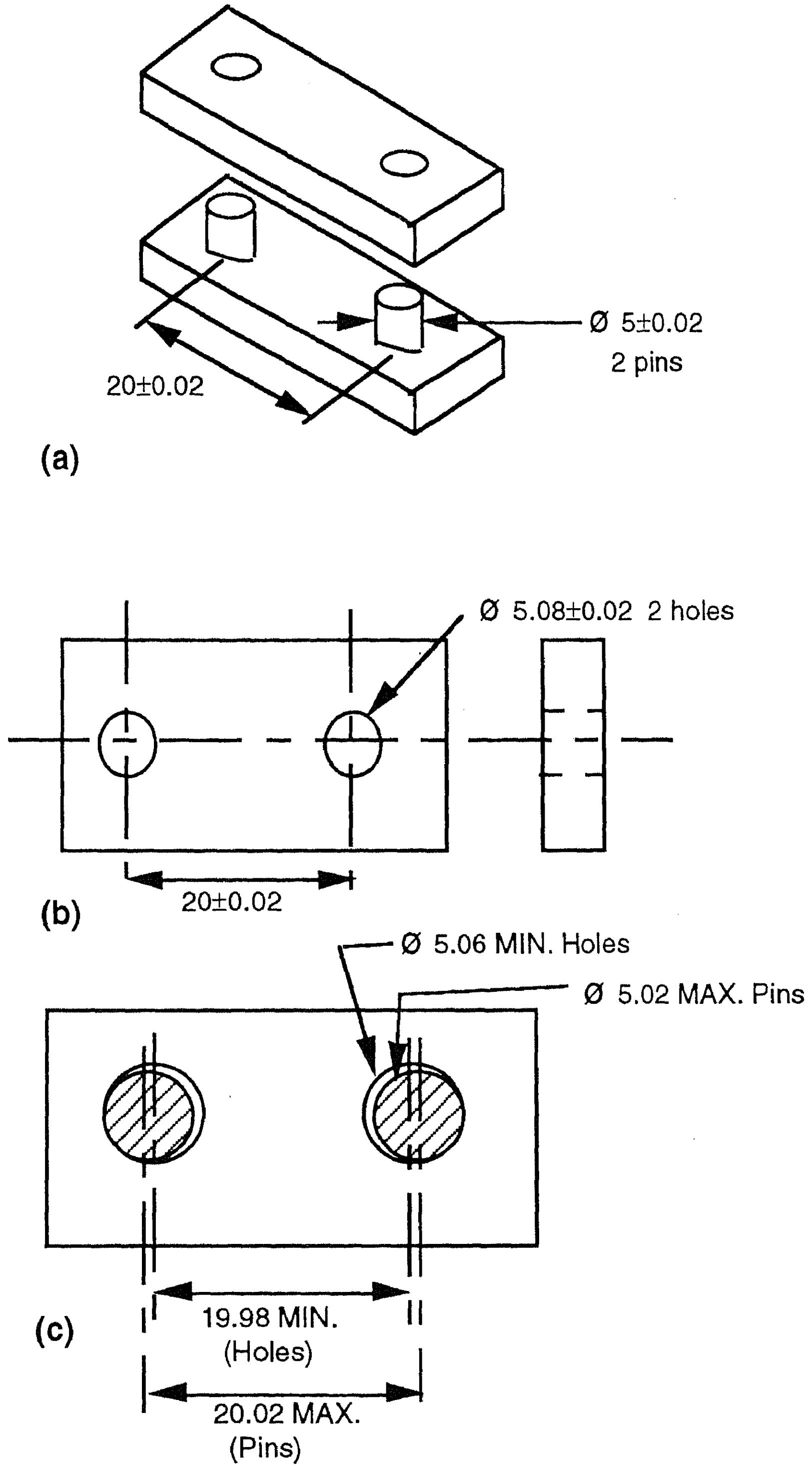
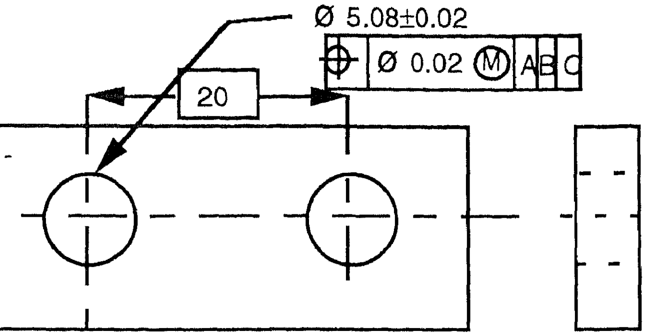


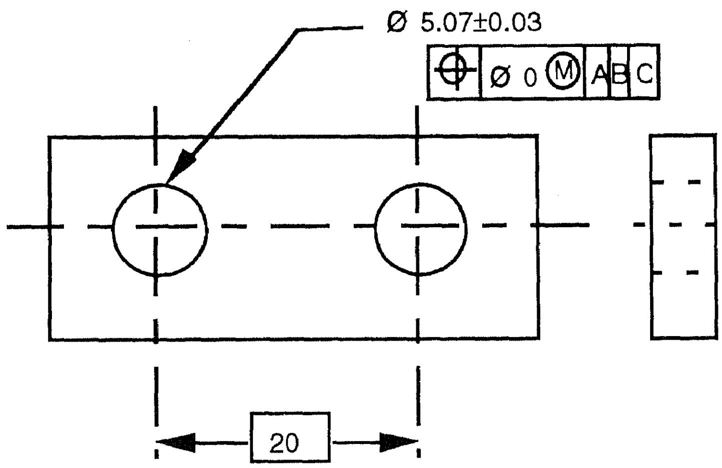
Figure 20 (a), (b), (c) Maximum Material Condition

(d.)



Actual Hole Size	Tolerance
5.06	0.02
5.07	0.03
5.08	0.04
5.09	0.05
5.10	0.06

(e.)



Actual Hole Size	Tolerance
5.04	0
5.05	0.01
5.06	0.02
5.07	0.03
5.08	0.04
5.09	0.05
5.10	0.06

Figure 20 (d.) (e.) Maximum Material Condition

RFS is another principle of GD&T, which unlike MMC, permits no additional positional, form or orientation tolerance, no matter to which size the related features are produced. It is really the independant form of dimensioning and tolerancing which has always been used prior to the introduction of the MMC principle. The symbol for RFS is an "S" enclosed in a circle. The RFS principle is valid only when applied to features of size (for example - a hole, slot, pin etc., with an axis or center plane). The size connotation cannot be applied to a feature which does not have "size".
This feature actually demands a very tight tolerance, in other words it is not very flexible for the production department to manufacture the part easily.

RFS condition is very much comparable to the traditional 'plus minus' tolerances, in the sense that the positional tolerance could not be increased or decreased as related to MMC and LMC conditions. This emphasizes that the tolerances are tight and are not flexible. Still RFS has an edge over the traditional way because of the circular tolerance. This condition is depicted in figure (32).

4.4 Least Material Condition (LMC) :

The condition in which a feature of size contains the least amount of material within the stated limits of size. For example - maximum hole diameter and minimum shaft diameter. LMC is opposite to MMC. For example - a shaft is at its LMC when it is at its low limit of size and a hole is at LMC when it is at its high limit of size.

This method is applicable to special design requirements that will not permit MMC or that do not warrant the exacting requirements of RFS. It can be used to maintain critical wall thickness or critical center locations of features for which accuracy of location can be relaxed (position tolerance increased) when the feature leaves the least material condition and approaches MMC. The amount of increase of positional tolerance permissible is equal to a feature size departure from least material condition.

Whenever LMC is specified on a drawing, the positional tolerance applies only when the feature is produced at its LMC size. This is depicted in figure (21). Additional positional tolerance is permissible but is dependant on, and equal to, the difference between the actually produced feature size (within its size tolerance) and LMC. This is shown in figure (22). It may be noticed from figure (23). that, tolerance zone increases as the feature size departs from LMC towards MMC.

Sometimes minimum edge distance is the criteria in the hole condition, then at that time the use of LMC condition is most useful. This is emphasized when particular metal is used in aerospace industries, this is because of the breaking (cracking) strength of the metal. This is depicted in figure (21). This situation is also discussed with respect to functional gaging in figure (33).

Functional gaging of the physical kind cannot be employed for the LMC condition, because of the variation in positional tolerance due to the size feature variation . As it will be discussed in section 6.2 the virtual condition will remain the same i.e. the functional gage diameter will remain the same, which creates problems while measuring for quality and product compliance.

4.5 Bonus Tolerance

With the introduction of positional tolerancing in GD&T, a revolution has taken place. Position is a term used to describe the perfect (exact) location of a

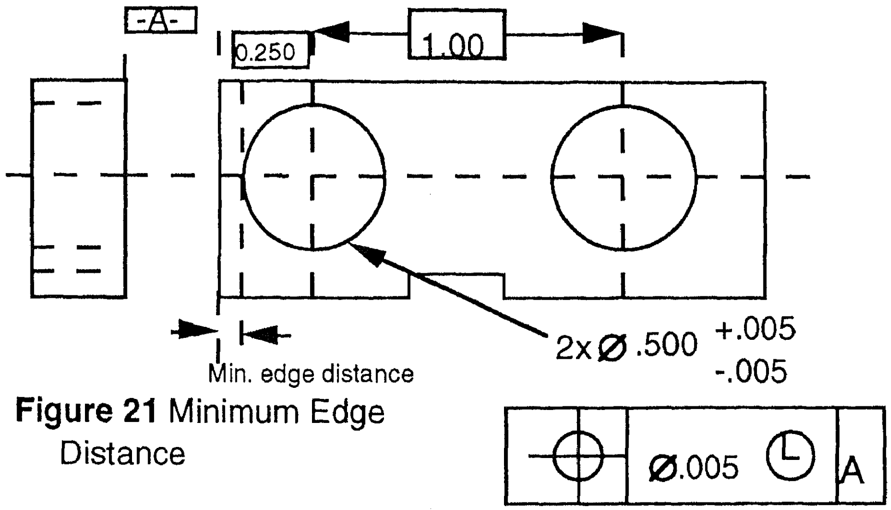


Figure 21 Minimum Edge Distance

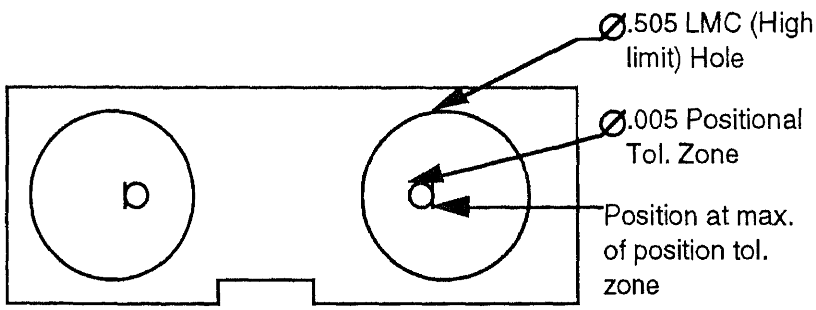


Figure 22 LMC When High Limit

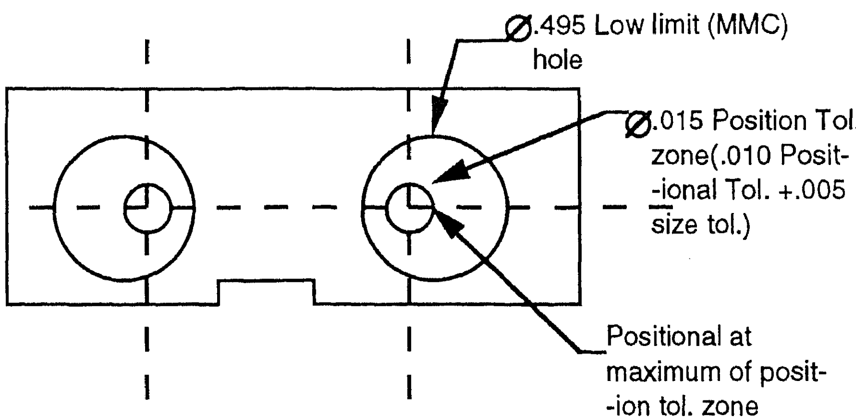


Figure 23 LMC When Low Limit

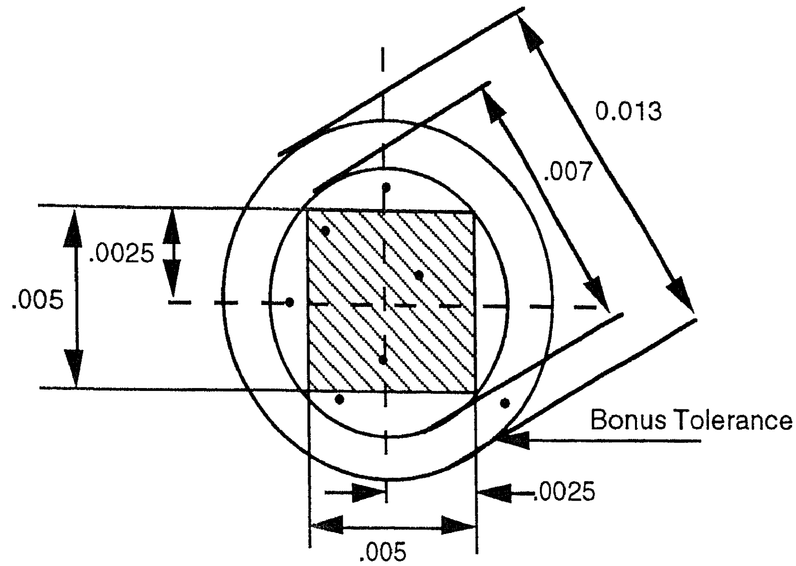


Figure 24 Concept of Bonus Tolerance

point, line or plane of a feature in relationship with a datum reference or other feature.

A position tolerance is the total permissible variation in the location of a feature about its exact true position. For cylindrical features (holes and bosses) the position tolerance is the diameter (cylinder) of the tolerance zone within which the axis of the feature must lie, the center of the tolerance zone being at the exact true position. For other features (slots, tabs, etc.) the position tolerance is the total width of the tolerance zone within which the center of the feature must lie, the center plane of the zone being at the exact true position.

To understand better of bonus tolerance we will have to understand the position theory a typically applied to a part for purposes of function or interchangeability. We shall also compare the position system with the coordinate system. Let us investigate in figure (24). a part with four holes in a pattern which must line up with a mating part to accept pins etc, to accomplish assembly, or four holes pattern to accept the pins, or studs of a mating part to accomplish assembly.

The top portion of the figure shows the part with a hole pattern dimensioned and toleranced using a coordinate system. Comparing the two approaches, we find the following differences:

1. The derived tolerance zones for the hole centers are square in the coordinate system and round in the position system.
2. The hole center location tolerance in the top part of the figure is part of the coordinates (the 2.000 and 1.750 dimensions). In the bottom figure, however, the location tolerance is associated with the hole size dimension and is shown in the feature control frame at the

right. The 2.000 and 1.750 coordinates are retained in the position application, but are stated as BASIC or exact values.

For this comparision, the 0.005 square tolerance zone has been converted to an equivalent 0.007 position tolerance zone. The two tolerance zones are superimposed on each other as shown in the figure. The black dots represent possible inspected centers of this hole on eight separate piece parts. We see that if the coordinate zone is applied, only three of the eight parts are acceptable. However, with the position zone applied, six of the eight parts appear immediately acceptable.

The position diameter shaped zone can be justified by recognizing that the 0.007 diagonal is unlimited in orientation. Also, a cylindrical hole should normally have a cylindrical tolerance zone. A closer analysis of the representative black dots and their position with respect to the desired location clearly illustrates the fallacies of the coordinate system when applied to a part such as that illustrated.

The dot in the upper left diagonal corner of the square zone and the dot on the left outside the square zone are in reality at nearly the same distance from the desired exact center. However, in terms of the square tolerance zone, the hole on the left is unacceptable by a wide margin, whereas the upper left hole is acceptable. Note than the hole produced off center under the coordinate system has greater tolerance if the shift is on the diagonal and not in the horizontal or vertical direction.

Thus the 0.007 position tolerance of the example would normally be based on the MMC size of the hole(.247). As the hole size deviates from the MMC size, the position of the hole is permitted to shift off its true position beyond the original tolerance zone to the extent of that departure. The BONUS tolerance of 0.013 illustrates the possible position tolerance should

CHAPTER FIVE

MANUFACTURING ENGINEERING CONCERNS.

5.1 Effect on Design

When the design language is clear, the understanding by the production and inspection department is better. The product could be made easier besides less rejection, when the intent is known and thus productivity is ensured. This additional tool is been provided by GD&T. GD&T is a powerful addition to drafting documentation practice that provides increased design and manufacturing flexibility. It can ensure 100% interchangeability at optimum cost.

GD&T uses such factors as datums, basic dimensions and geometric controls which link tolerances to the size of a feature. It is also used to define a virtual condition, a key element of nearly any design. The virtual condition is frequently viewed as the combination of all worst cases of part variability for assembly. The ability to define and express the virtual condition within the GD&T language enables the engineer/designer to define the true functionally related maximum limits of production variability while ensuring design integrity and thereby optimizing costs.

In contrast the old co-ordinate system of dimensioning cannot define clear, constant and functionally related virtual conditions at all. The more designers work with the GD&T language, the more sense it makes as a design tool. With reference to figure (25).

As seen in the block diagram, the design stage is the most critical stage. The designer's job is to design the product and convey the message clearly to the production department. According to the conventional type of product engineering a Prototype of the product is made first and tested for quality. If it

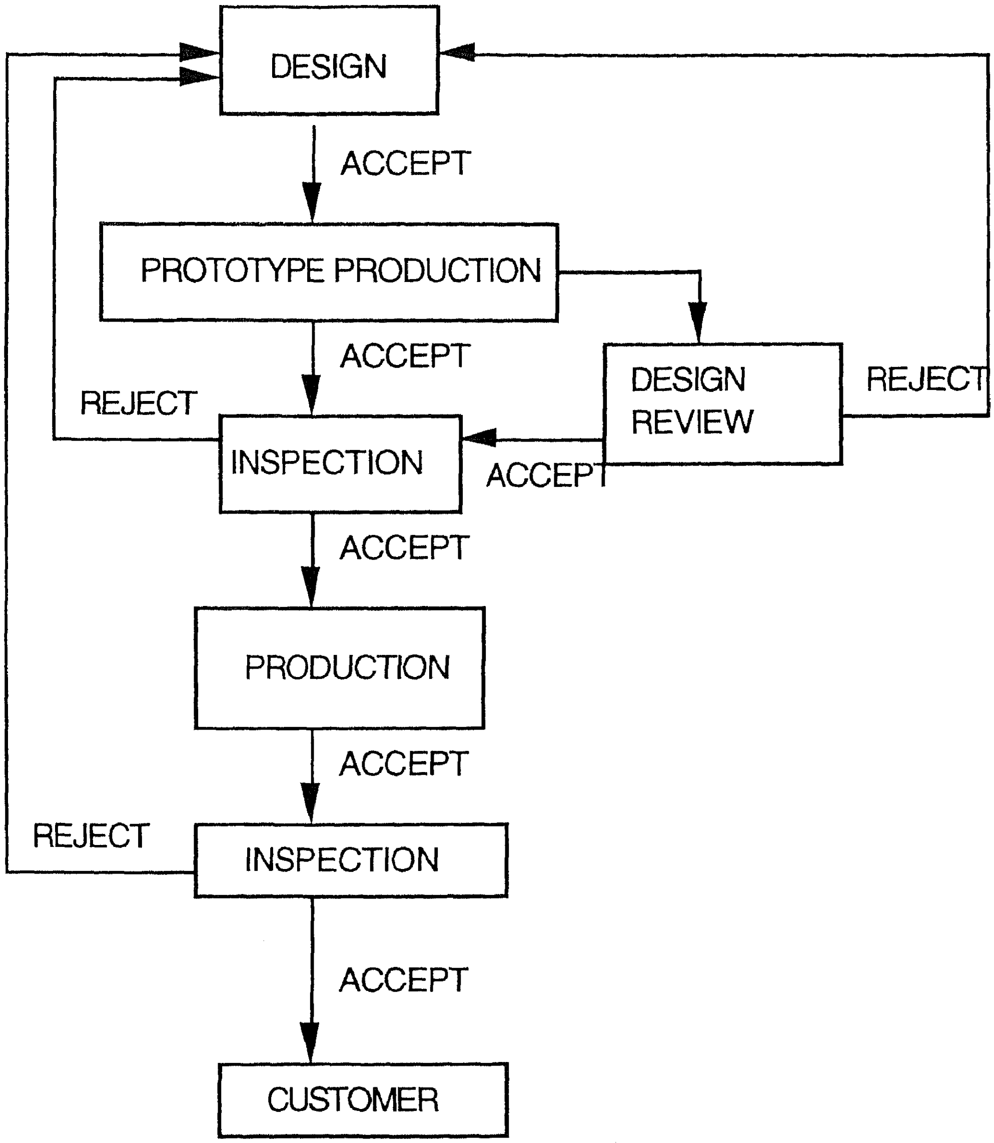


Figure 25 Block Diagram Effect on Design

is acceptable then production is envisaged. Even after that an inspection is conducted and then sent to the customer after it has been accepted.

But GD&T eliminates out many uncertainties and hence prototype production. This procedure eliminates out the cost linked with the regular process and the time. Above all it gives the Production department more work area to work with. Many a times the designer knows what he wants to convey, but he has problems in communicating in the regular drafting procedures. This has been eliminated with the use of GD&T.

The traditional drawings provides only the barest minimum of explanation. For example the drawing might indicate size and location but ignore the interrelationship of features or connecting parts. GD&T eliminates this problem through a system of symbols that do not leave detail open to interpretation. This system can be applied to any design application although it is critical in instances where part features are instrumental to the functionality or when the parts have to be interchangeable.

GD&T also addresses the issue of material condition, a critical element in the design of functional parts. Dimensional accuracy and tolerances are particularly important to integrated manufacturing since they affect the manufacturability, time, cost and quality of a product. It also provides the machinist more work area in which to produce an acceptable part. That means design ensures manufacturability.

The main Motive of Design for Manufacture (DFM) is to reduce the number of the parts in the design of a part, consequently reducing cost, complications, and savings in material and time. To employ this kind of concept in the manufacturing field it needs the support of the production department. The production department has to comply with the design requirements and specifications and this can not be done without the help of

GD&T. It is always easy to produce a product in different components, but when they are assembled they do not fit. This is because either the design is not clearly interpreted or by giving tight tolerances, this problem will be evident if the traditional drafting procedure is used. Whenever part features are critical to function or interchangeability, the 'plus minus' kind of tolerancing does not work good in ensuring quality products. This is where GD&T steps in, and hence all these concepts like DFM needs to rely on for cost savings and productivity.

5.2 Impact on Product Engineering

GD&T has also dramatized product engineering to a large extent. Previously when using the coordinate system a Prototype of the product was produced. This Prototype went through a series of inspection and the design of the product was reviewed. Two problems lie here.

1. Sometimes the product made according to the design specification with great difficulty, was rejected by the quality control and the inspection department. The design department pointed their finger at the production people and the production people pointed their finger at the design department. But the truth is that the language or the design intent was not clear, hence communicating of how the part has to be made was a failure, thought he specification was in tolerance.
2. Many a times the problem lied in making the product itself under the specified tight tolerances. The machinist was under great pressure to produce the part which is difficult to make under design specifications.

These two problems have been solved by GD&T. In the first case GD&T ensures 57% more tolerance zone compared to the square tolerance. This is

done by providing a circular tolerance zone. This ensures that not many parts are rejected. This is illustrated by a bar chart in figure (26).
With the help of GD&T's symbologies and system of representing features and other items described earlier, helps the machinist or the production department to understand the drawing better and the intent of the drawing is clear.

In the second case, positional tolerancing in GD&T provides bonus tolerance. Bonus tolerance means more room for the machinist to work on the product., hence better the part will be in quality. As seen in figure (27). as the condition departs from MMC, the machinist is given more positional room. This means two things, one is the machinist is more relaxed in work creating the product with more positional tolerance. The other thing is that the product or part has the scope of rework, with the increase in positional tolerance. To rework the part within the tolerance specifications, is one of the greatest advantages using GD&T. This cannot be done with the regular drafting practices.

5.3 Tooling

Product quality depends, to a large extent, on the quality of the tools and gages used in the manufacturing and inspection operations. The term *tool* in the manufacturing industries refers to any device that is capable of working a material into the desired shape, holding the material while it is being worked on, or measuring the material when the work has been completed. Common tools are machine tools, cutting tools, jigs, fixtures, press dies, and gages. A jig is a device for holding the material being machined while an operation is performed, at the same time guiding the tool that performs the operation. A

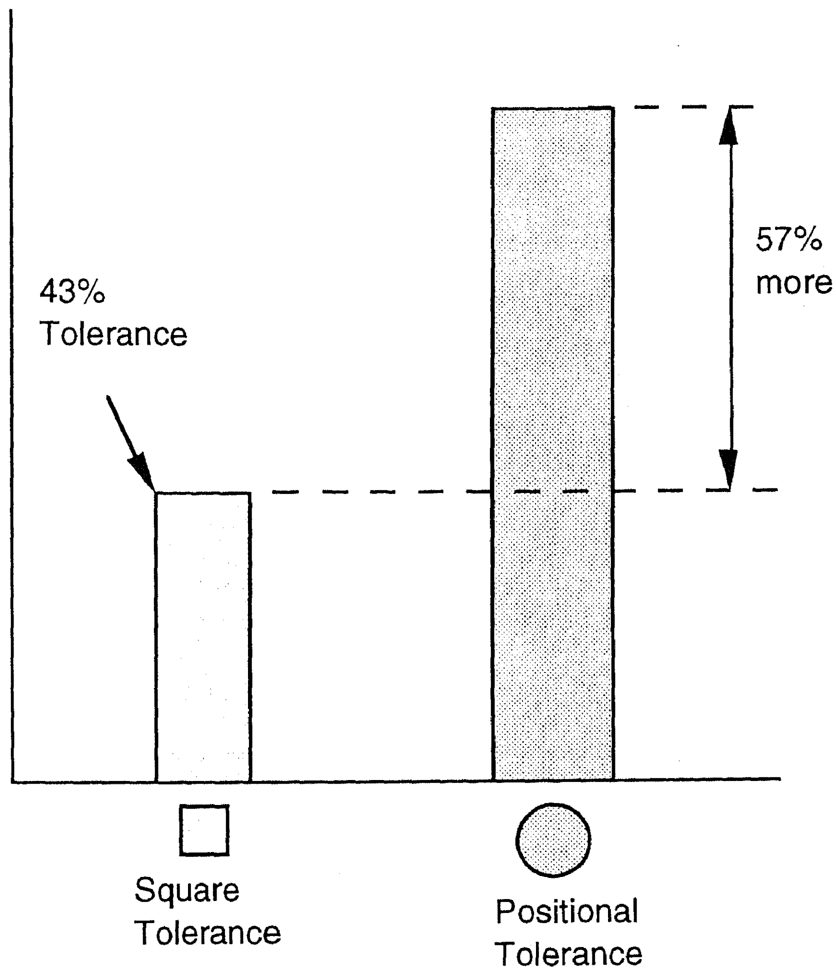
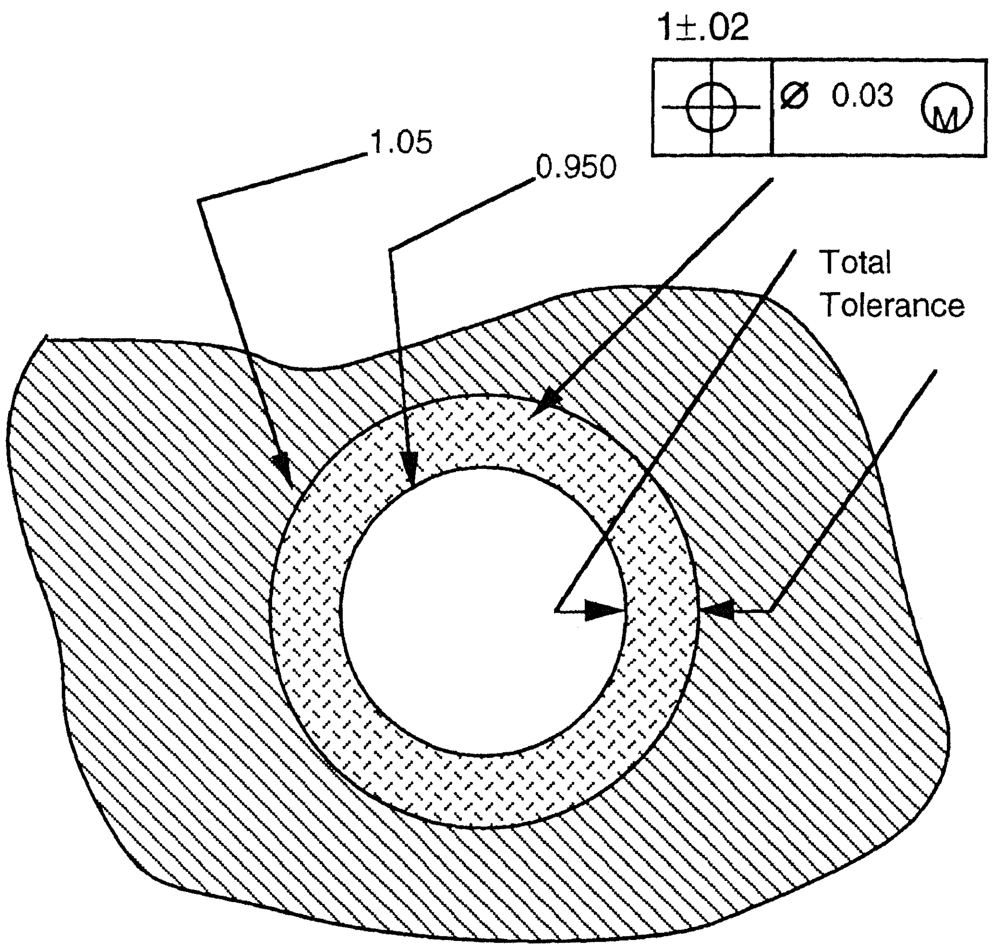


Figure 26 Comparision of Tolerance Zones



Max. Size	Tol.	Virtual condition
MMC 0.98	0.03	0.95
0.99	0.04	0.95
1.00	0.05	0.95
1.01	0.06	0.95
LMC 1.02	0.07	0.95

Figure 27 Bonus Tolerance as per the Production Department

fixture is a device for holding the material while an operation is performed. A gage is a device for measuring the quality characteristic to check its conformance to the technical specification.

Tools and gages provide the physical means of attaining volume production and interchangeability of component parts. Tools and gages are subject to constant wear and deterioration. Thus, it is essential that a system of tool and gage control be established to maintain the quality of the tools and gages. Another reason for strict tool control is that frequently tools are designed and used to control the dimensional quality characteristics of the product without the benefit of inspection. Quality control of the product is indirect-that is, the tool controls the product characteristic and scheduled inspections of the tool replace product-parts inspection.

5.4 Inspection:

Cost is a prime criterion at every level of the quality system. Quality planning operations are initiated by consumer quality requirements. A basic consumer consideration is cost. Quality control operations are directed and implemented from a cost-criterion basis. Economic decisions underlie the development of specification tolerances, control procedures, and inspection plans.

Since economics is the core of the quality-decision process, serious consideration should be given to the accuracy of the input to cost models and cost computations. An important input factor is accuracy of measurement of quality characteristics. Consider the process variability value, $6\sigma_x$ being equal to 0.005 in. What proportion of the 0.005 in. is attributable only to process variation, and what proportion to measurement error? A mean sample measurement is 0.507 in. and the corresponding point on the $\bar{x}$ chart indicates measurement is 0.507 in. and the corresponding point on the $\bar{x}$ chart indicates an out of control condition. Perhaps the sample mean is truly 0.504 in.and the process is in control. Measurement error has generated a decision error.

In production situations, specification requirements for modern products are so restrictive that measurement error becomes most serious problem facing the quality control and inspection staff. This is particularly true in many mechanical industries and especially true in the aerospace industry. In these cases 2 elements become critical quality determinants. These elements are :

1. Tool and gage control, and
2. Inspection and test.

Several classifications of inspection are possible. One classification, based on the method of measurement, is variables and attributes inspections. Another classification, dependent on the number of product items examined, is 100% inspection (called screening or detailing) and sampling inspection.

Based on the purpose of the inspection operation, 100% inspection is either operational sorting or corrective sorting. Similarly, regarding purpose, sampling inspection is either acceptance sampling or control sampling. The question of whether or not to sort product is an economic problem involving an estimate of the estimate of the cost generated by failure to detect defectives as they occur in the manufacturing system.

Inspection is primarily concerned with determining the degree to which production output conformed to the established technical specifications for the product. The resulting inspection information is used for two purposes:

1. To control manufacturing operations and product quality characteristics, and

2. To prepare quality audits to generate feedback information to the quality-planning operations and upper level management sections.

The inspection operation may be classified in two categories based on the method of measurement-variables inspection and attributes inspection. Variables inspection includes any inspection operation where the gage indicates, on a continuous scale, deviations from the technical specification. For example, a dimensional specification may be 0.501, 0.502, 0.503, etc. (or, 0.499, 0.498, etc.). With attributes inspection, the gage merely classifies the product into discrete categories. For example, the gage may classify product as being effective or defective. Another common classification is undersize, oversize, and within the specification limits. The categories into which the product is separated are discrete and usually few in number.

5.4.1 The Measurement Problem:

Modern tolerancing systems recognize four basic product conditions to be controlled by tolerance specifications:

1. Size
2. Form
3. Location
4. Function - conditions of assembly, operation.

In practice, these conditions interrelate to define quality characteristics and the problem of measuring quality characteristics to evaluate conformance to specifications becomes complex.

A physical factor, which makes it difficult to define and control quality characteristics, is lack of true geometric perfection. Shapes into which material is to be fabricated are defined by geometric terms. The geometric definition assumes a perfect form. However, perfect forms cannot be produced. Thus, variations from perfect form must be defined and controlled if a specific quality is to be maintained. These geometric variations are controlled macro errors. Figure (28) indicates a simplified example. a perfect form is defined by the specified one-inch square in (a) Possible departures from perfect form are (b) nonparallelism, (c) not square, and (d) rounded corners.

The interrelationships of size, form, and location conditions required to define quality characteristics, coupled with production variations due to geometric form and rigidity errors, lead to a variety of complex measurement problems involving sophisticated gaging methods. Figure (29) summarizes these product conditions and error factors.

5.5 Gages:

In 1875, a length standard was established by the International Bureau of weights and measures at sevres, France. The standard is a platinum - iridium bar with three microscope lines engraved at each end. The distance between the central lines in each group of three lines defines the International Prototype Meter. Thirty-one meter bar duplicates were constructed and distributed to the principal nations as standards. The United States received Meters no. 21 and 27, which have been retained as standards by the National Bureau of Standards.

5.5.1 Gage Blocks:

Transfer of a length standard from the National Bureau of Standards to a manufacturing plant is accomplished by means of gage blocks. A gage block is

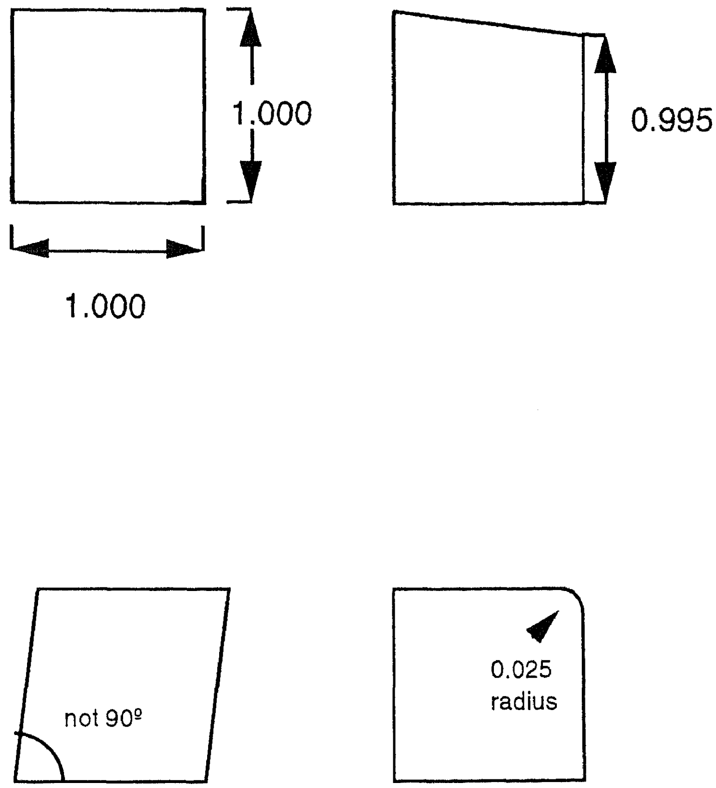


Figure28 Macro Errors

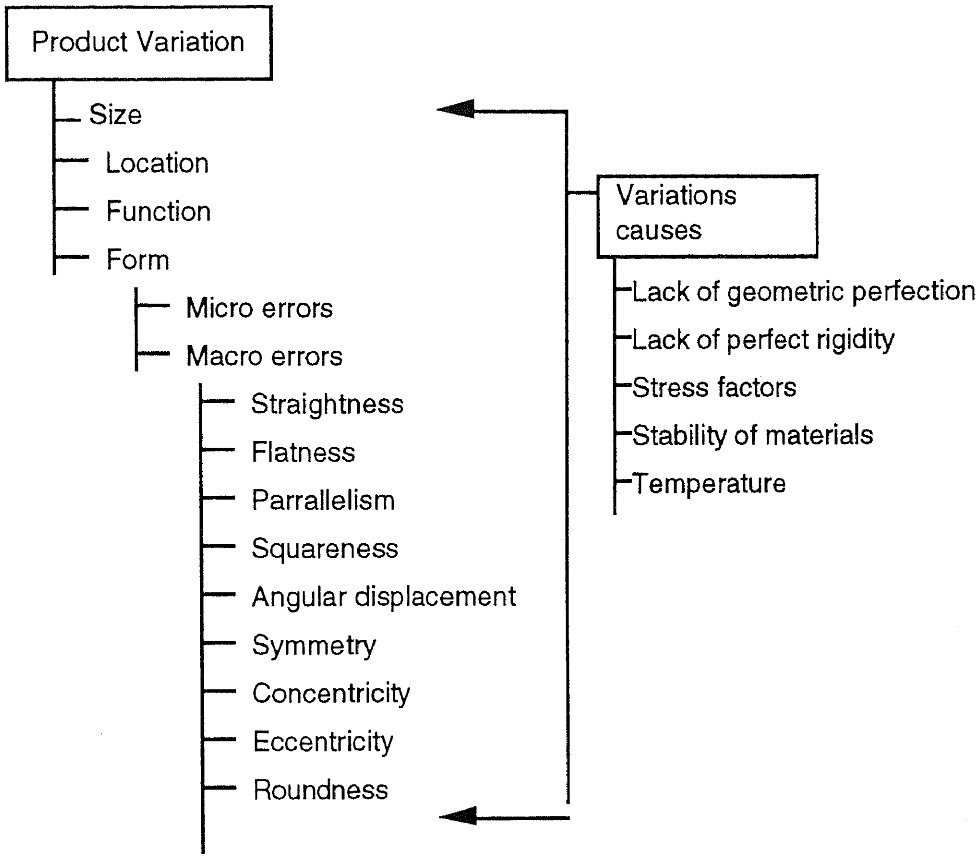


Figure 29 Product Variations

a reference piece, either square or round in cross section, with two end faces which are the measurement surfaces. That is, the end faces are flat parallel surfaces whose separation has been established to light wave precision and accuracy.

Gage blocks are made from SAE52100 alloy steel, tungsten carbide stainless steel, chrome-plated steel, and chromium carbide.
The most important criteria for judging gage block integrity is its degree of flatness and parallelism since these factors bear directly on one's ability to measure its length reliably.

Gage block sets are available in a wide variety of sizes, depending on measurement requirements. Accessories make it possible to use the blocks for production measurements and thus eliminate a possible source of error from an intermediate gage calibrated by the blocks. However, the primary purpose and use of gage blocks is to calibrate other gages used in the manufacturing plant. For example, the entire plant's production performance depends on the master set of gage blocks for that plant.

There are three general classes of gages:

1. Working gages
2. Inspection gages and
3. Master gages.

The classification is based on the use of the gage. Working gages are used by process operators and process set-up people.

5.5.2 Criteria for Selecting Gaging Equipment:

Gage requirements are implied by the three terms-Accuracy, Precision and Reliability. Accuracy is a relative matter. It is a comparison of desired results with undesired results. Relative to gaging, accuracy refers to the ratio of to incorrect readings. It is frequently called the quality of conformity.

Precision is a measure of the variability of instrument readings. Precision can be expressed either in terms of the range or standard deviation of the distribution readings. The smaller the range or standard deviation, the higher is the precision of the gage.

Reliability means the probability of a reading occuring in a specified interval bisected by the true reading. The meaning of reliability corresponds to that of a confidence interval in statistics.

All these three terms are dependent.on each other.

Some of the principal criteria for selecting gaging equipment are-

Amplification (or magnification) is the ratio of the indicator displacement along the gage scale to the input dimensional displacement.

Discrimination (or resolution) is the ability of the gage operator to visually separate scale divisions. Clearly, amplification facilitates discrimination and increases the precision of the gage.In selecting a gage for a given inspection job, a compromise is made between amplification and range of the indicator scale. For a fixed scale size, higher amplification decreases the range of the scale, and conversely.

Calibration accuracy (or linearity) describes how well readings at various points on the gage scale correspond to the true dimensions being measured. This refers to the full working range of the gage and is expressed either as a specific number or as percent of full scale.

Repeatability (repeat accuracy) refers to how closely the gage indicates the same reading over a series of trials using one or more test standards. How well an instrument retains its calibration setting over a period of time is called stability (or drift) . This is usually expressed as percent error in a given number of hours. This criterion is not absolute. That is, gage stability required to measure a large run of production parts would not be important when gaging just a few pieces.

Sensitivity is the smallest dimensional input to the gage that produces a readable change on the gage scale. This is usually expressed as a number, such as millionths inch. Although high sensitivity is a desirable gage property, it can be wasted if the repeat accuracy is poor or if resolution is not adequate.

Mechanical gages amplify input dimensional displacement by some means of producing a mechanical advantage, such as gear train or reed mechanism.

In electronic gaging systems, an input dimensional displacement at the gage measuring point produces an electrical output (e.g., voltage, current, resistance, reactance). Like reed type gages, most electronic gages are comparators. Advantages of electronic gaging systems are high amplification, variety of amplifications in a single instrument, and fast measurement speed.

Air gaging systems measure size by monitoring the difference in flow or pressure of an air stream. The gage is first zeroed against a reference master of known size. Measurement is made by metering the pressure loss between the product -part surface and the master.

Air gages are especially useful in measuring small hole diameters, long holes, and various geometric conditions as out-of-roundness,taper and so-forth.

Optics is being increasingly used in modern gaging devices. Typical optical gages are simple hand-held magnifiers, microscopes with optical scales and micrometer stages. The optical comparator is a widely used inspection method for checking linear and angular measurements, thread forms, gear teeth, and contours of all types.

Fixed type of gages are the most economical means of inspecting product parts on a mass production basis. A plug gage, for example, can check a hole specification in a matter of seconds. The cost of savings from using a fixed a gage , instead of an indicating gage, are due to

1. Cost of the gage, and
2. Speed of the gaging operation.

One disadvantage, however, is that fixed gages can discriminate only to 0.0001 to 0.0002 in.

5.6 Functional Gages:

A functional gage is in essence the "reconstruction" of the mating part from the requirements indicated on the design. It describes, as well, a representing mating part (or mating situation) which simulates the two parts in assembly. it also represents a worst case part which remains an acceptable part. The functional gaging approach is not required on such parts; it is an available option.

A functional gage would never accept a bad part but could reject a border line good part. This is because in the standard method of allocating gage making tolerances, some part tolerance is utilized for the gage.

To build a functional gage, tolerance for the gage features location must be taken from the piece part feature location tolerance. This is commonly refered to as the 10% rule (or 5% to 10% rule), which means that up to 10% (sometimes sightly more) of the part tolerance limits could be used for gage tolerance. Therefore, a part of borderline (extreme limit of acceptable tolerance) conditions could be rejected by a functional gage if the part were at the fringe edge of the acceptable tolerance range. The gage will not, however, ever accept a bad part.

the fringe edge of the acceptable tolerance range. The gage will not, however, ever accept a bad part.

The advantages of a functional gage are:

1. It minimizes time and resources involved to verify parts.
2. It represents functional interface of the concerned features.
3. It recognizes the subtle composite effects of size, orientation, and position as a 'go,' 'no go' result.
4. Provides a 'hard' tool which can be utilized by anyone with reasonable technical skill; does not require a highly skilled inspector.
5. Provides alternate methods for verification from surface plate, open set-up, coordinate measuring, etc.
6. It will never accept a 'bad' part.

There are some disadvantages of a functional gage:

1. Could reject borderline good parts.
2. Must be reworked if the part is revised.
3. Requires gage-maker's tolerance taken from piece part tolerances (up to 10% usually).
4. Costs for building, storage, and maintenance.
5. Does not quantify results (it's 'go' or 'no go')

Functional gaging principles can also be utilized without a functional gage. Alternative methods such as graphic analysis, Paper gaging, "scatter grams," etc., using various tools, including the computer, are at our disposal. For example, the results of a co-ordinate measuring machine (CMM) or comparable method can be used to simulate functional gaging. Further, a mathematical solution from data derived from a CMM operation can be determined with the assistance of calculators and computer programs.

Functional gaging principles can be achieved in three different ways.

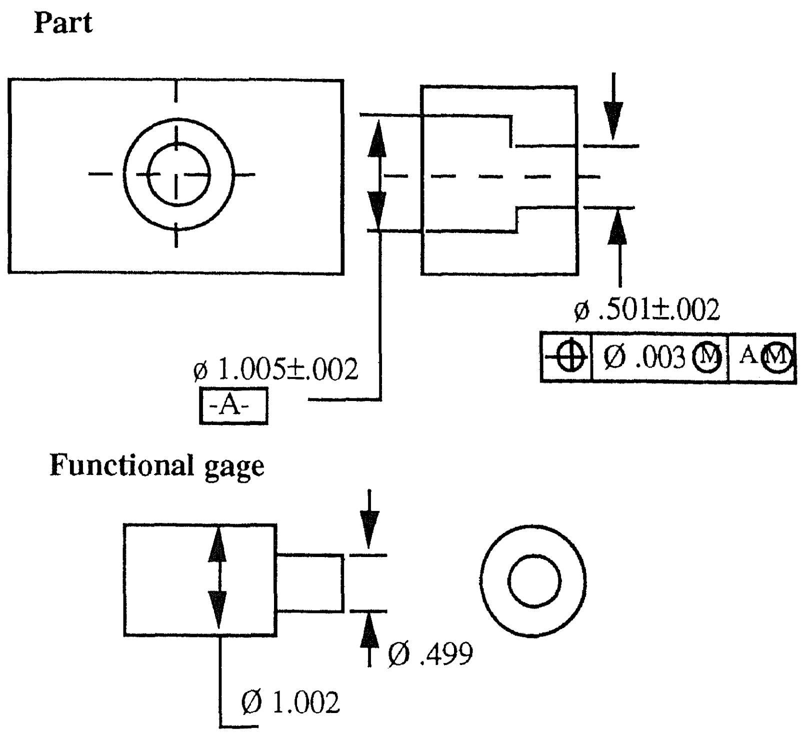
1. A functional gage.

2. Physical graphic analysis (e.g., Paper gaging), or

3. Mathematically Using a calculator or computer programs. Establishment of the computer programs, however, requires a superior knowledge of the technical principles involved.

The graphic or mathematical methods may be necessary where the precision of the part may not permit functional gaging (insufficient tolerance can be derived for the gage build), where parts are rejected by a functional gage and are suspected as borderline good parts, where RFS specifications to the features controlled prevent use of a functional gage, where the functional gage is not justified, etc. Further, it should be noted that the mathematical (calculator/computer) methods may be used to bypass functional gaging and graphic analysis completely, usually with greater accuracy as well as more rapidly.

Functional gaging techniques, familiar to a large segment of industry through many years of application, are fundamentally based on the MMC position concept. It should be clearly understood, however, that functional gages are not mandatory in fulfilling MMC position requirements.



Gage Calculations
MMC Part Hole Size =.501
Stated Position Tol.= -.002
Gage Diameter = .499

Figure 30 Functional Gages

CHAPTER SIX
PROBLEM

6.1 Statement

Functional gages of the physical kind cannot be used, when Least Material Condition (LMC) is specified in GD&T under special circumstances. This is the concern of the inspection department. In other words only Maximum Material Condition (MMC) can be gaged very effectively and economically.

6.2 Experimentation and Analysis

Functional gages are very popular with the inspection department. They are widely used for checking mating parts. These gages actually represent a part and Hence make things easier for the inspection department. A functional gage of the physical variety cannot be used in inspecting a product under conditions specified as LMC. It has also some problems if RFS is used to specify a feature. The functional gages are very handy when MMC condition is specified.

The problem of gaging with respect to some conditions (MMC, LMC, RFS) is illustrated with respect to figures (31)., (32)., (33). Figure (31). explains the condition when MMC is specified., RFS is the condition when the positional tolerance does not vary when the size varies according to tolerance specifications. This is emphasized in figure (32). Figure (33). shows the reader with the help of calculations that the functional gages is not practicable.

As the statement specifies the functional gages take a back seat for example in Aerospace industries, where the unusual but useful condition of LMC is applied. To understand this situation the part in figure (31). consists of four of diameter 0.25, with a size tolerance of 0.020. It also has a positional tolerance of 0.007. as indicated in a frame with Maximum Material Condition. In the frame it is indicated that the hole's position could be off by 0.007 diametrically at MMC with respect to datums A, B, C. Maximum Material condition means that the hole's diameter is smaller i.e. 0.230 the positional tolerance will be 0.007. So the virtual condition of the hole will be 0.230-0.007=0.223.

Now if the hole size is 0.250 the positional tolerance is 0.027, the virtual condition is 0.223. Similarly if hole is at LMC i.e.at 0.270, the positional tolerance is 0.047 and the virtual condition is 0.223. The point to note is that here virtual condition remains the same, but positional tolerance increases as the feature departs from MMC to LMC. The increase in tolerance will be equal to the amount of departure.

Now consider a functional gage, which simulates mating parts at their worst condition. The worst condition in our case will be ø 0.230 and 0.007 positional tolerance, i.e. the virtual condition will be

$T = F - G$ Equation (1)

Where T = Tolerance

F = Feature size

G = Gage size

or $G = F - T$. Equation (2)

so if F = 0.230

T = 0.007

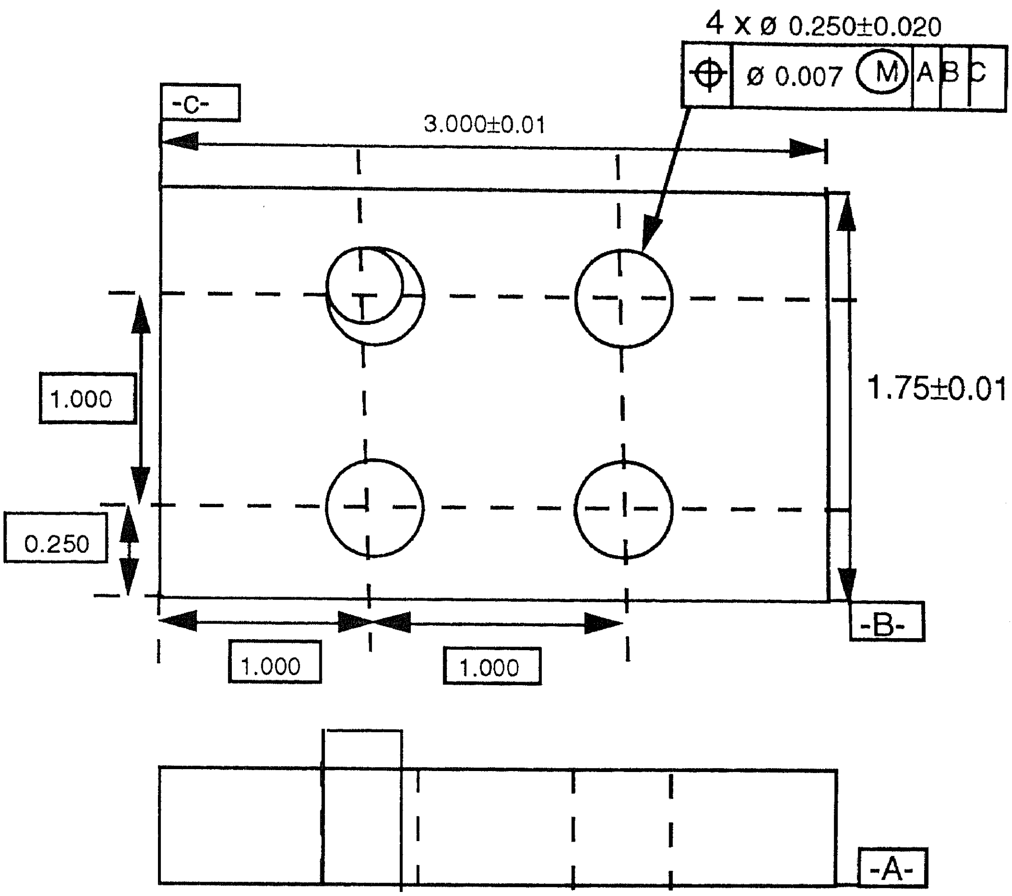
G = 0.230-0.007

= 0.223

similarly when F = 0.250

T = 0.027 - known from the figure 31.

Therefore G = 0.250-0.027=0.223



Feature	Position	Virtual
MMC 0.230	0.007	0.223
0.240	0.017	0.223
0.250	0.027	0.223
0.260	0.037	0.223
LMC 0.270	0.047	0.223

Figure31 Gaging-MMC Condition

Also when F = 0.270

T = 0.047

G = 0.223

We observe that the same gage holds good because of positional tolerancing and when the MMC condition is specified. This is true and useful since we will be using only one type of gage and even the size of the gage will not vary. This will make the job of the inspection department easier and productivity is retained.

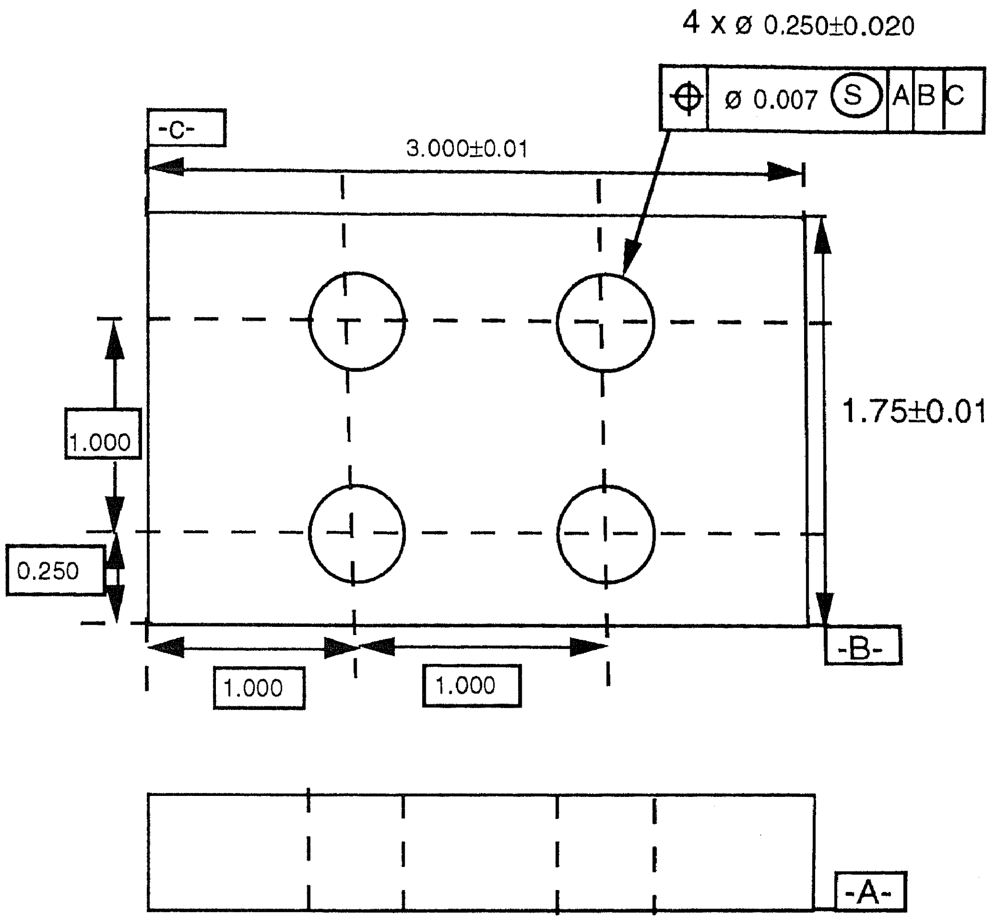
Here actually the inspection department has to check the feature size separately since the true variations remain to be in the feature size. Only after the size tolerance is checked then the positional tolerance is checked with the help of a functional gage, this procedure assures product quality.

Now we will discuss when the situation is same but when Regardless of Feature Size (RFS) is specified. This has been illustrated in figure (32). The virtual condition has also been calculated in the figure. The positional tolerance remains the same irrespective of the change in size of the hole. So if the hole is in the upper tolerance limit say 0.270 the positional tolerance will be 0.007 and the virtual condition will be 0.263. If the feature is perfect i.e. 0.250 hole the positional tolerance will remain 0.007 and the virtual condition changes to 0.243. Also if it is the lower side 0.230 the positional tolerance will not change and thereby the virtual condition changes to 0.223.

To calculate the size of the functional gage, we need to know the feature size and the tolerance specified. This illustration is clearly shown in figure (32).

By applying equations (1) and (2)

T = F - G.



Feature	Position	Virtual
RFS 0.270	0.007	0.263
0.260	0.007	0.253
0.250	0.007	0.243
0.240	0.007	0.233
0.230	0.007	0.223

Figure32 Gaging-RFS Condition

when F= 0.270 and T=0.007

G = 0.263

when F = 0.250 and T = 0.007

G = 0.243

when F = 0.230 and T = 0.007

G = 0.223

This indicates that we will be needing many gages of different sizes to measure that one feature. The number of the gages depend on the amount of the departure from high size to low size. The number of gages, if at all we are going to use will be an costly affair and also time consuming. The RFS condition, therefore is not used commonly, not only for measurement problems but also it is difficult for the production department to manufacture tight tolerances. In a way RFS condition is similar to the traditional 'plus minus' tolerance.

Now we will consider figure (33). which illustrates the part when LMC is specified. This is a strange situation but useful under special circumstances. As observed from the calculations as the figure departs from LMC to MMC, the positional tolerance also increases and the virtual condition increases and stays constant irrespective of the change in position. As seen from the calculations in the figure, when the hole is LMC i.e. 0.270 and tolerance specified 0.007, the virtual condition will be 0.277 which means

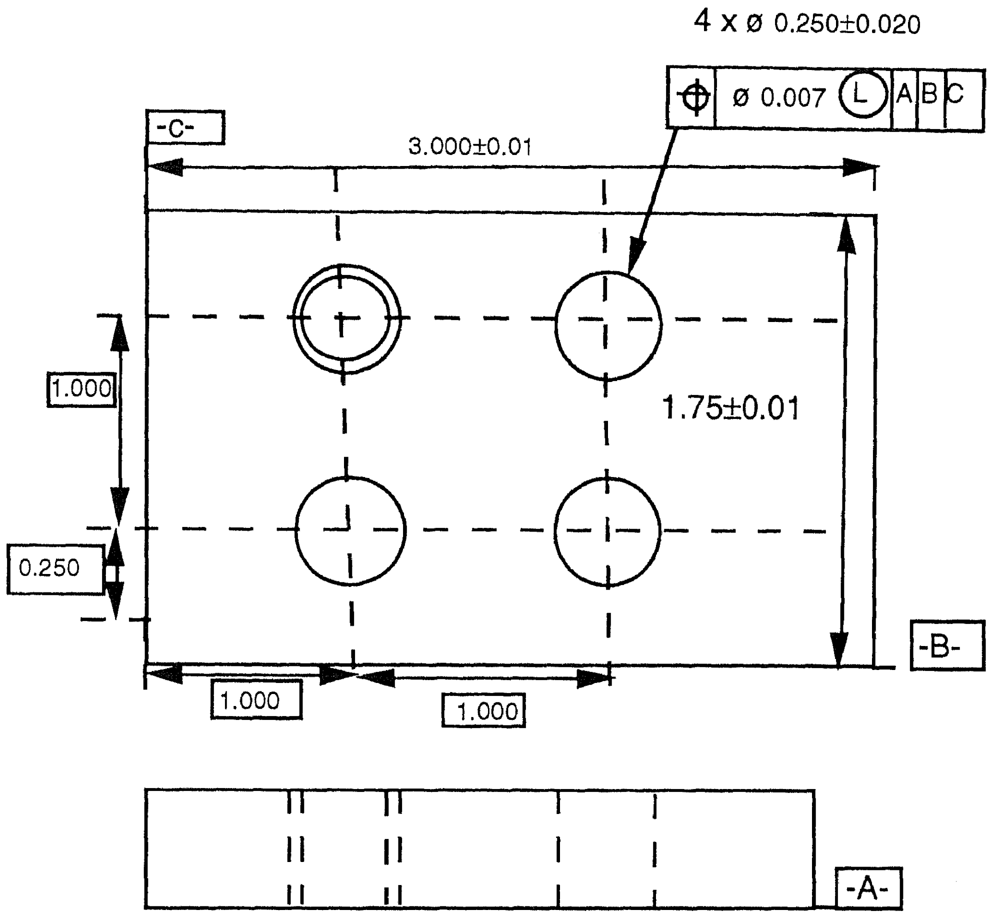
from the Equations (1) and (2) T = F - G

or G = F - T

If F = 0.270 and T = 0.007

G= 0.270 - 0.007

G= 0.263



Feature	Position	Virtual
LMC 0.270	0.007	0.277
0.260	0.017	0.277
0.250	0.027	0.277
0.240	0.037	0.277
MMC 0.230	0.047	0.277

Figure 33 Gaging-LMC Condition

If F = 0.250 and T = 0.027

G= 0.250 - 0.027

G= 0.223

If F = 0.230 and T = 0.047

G= 0.230 - 0.047

G= 0.183

According to the Equations, the gage size or in other words the virtual condition should vary as per the calculations shown above. But it is not true, practically the positional tolerances add on as shown correctly in the figure. That is if the feature size is 0.270, the positional tolerance will be 0.007 and virtual condition becomes 0.277, and if the feature is in its high size i.e. 0.230 the positional tolerance is 0.047, but the virtual condition remains 0.277. This itself is due to the peculiar nature of the condition itself. By following this rule we can ensure the minimum edge distance requirements (described in section 4.4) for product integrity. This condition is used rarely and in situations where high precision is required (Aerospace industries is a good example).

By looking at the calculations above, it is difficult to calculate an appropriate functional gage of the physical kind to comply with the situation. Whereas it could be measured by other methods discussed in the next paragraph, functionally gaging the hole does not work because of the variations of the axis position with respect to change in feature size. Any other method other than the functional gages of the physical kind, points out in increase in expenditure towards either buying sophisticated machines or requires skilled labors to mathematically measure. This also indicates lóss of time and increase in cost towards payment of high salary.

Alternate to functional gaging is paper gaging, optical comparators and Computer Measuring Machine (CMM). In this section we will discuss about Paper Gaging. Paper gaging is accomplished through plotting an enlarged scale of coordinately measured feature positions onto a piece of standard graph paper and then plotting the resulting differentials (actual position versus true position) to a selected scale (e.g., one square = 0.001) with a dot on the graph. An overlay chart (gage) of tracing paper or other transparent material containing a series of graph -scale circles of desired increments is placed over the graph to depict the position tolerance zones. Note that the paper gaging method simulates part function and functional gaging. However, the individual tolerance zones are each assumed to be represented by the one exact (true) position on the graph. The exact (basic) dimensions of the pattern are assumed as 0 in the X and Y directions.

Paper gaging simulates hard gaging and part function and thus is an effective technique. The best advantage of this method is that it gives a permanent record. The disadvantage of this method is that it requires some time to do the procedure and needs an skilled inspector to do the calculations efficiently.

CHAPTER SEVEN
CONCLUSION

7.1 Conclusion

As discussed through the chapters of this thesis, using Geometric Dimensioning and Tolerancing language provides the manufacturing industry with more benefits than the traditional drawing system. It eliminates the communication problems between the design and production team, by specifying clearly as to what is required and how the part should be manufactured Inspecting the part for product compliance using functional gages of the physical kind is very common, economical, less time consuming, simple and cost effective.

As discussed in chapter six functional gaging of the physical kind is simple and an effective method to use. It practically dictates little or no training to conduct the inspection. The same gage could be used for all the parts. Time is saved dramatically and the cost which is always linked to the time variable also drops. Initially it would be necessary to invest some amount of money towards the manufacture of the gage, but the use of GD&T and the benefits derived from the same outweighs the other disadvantages that the system encounters. As we have observed in the previous chapter functional gaging of the physical kind for a special condition like MMC is useful and advantages. But when conditions like LMC and RFS is used this method does not seem to work, forcing the inspection department to adopt other methods, in the process losing money and time. This handicap is not because of a faulty gage, but because of the unusual condition of the terms in itself. Concluding on this part there are two things, either the terms (LMC and RFS) should be avoided or a different type of technique should be involved.

An alternative to the method that we have discussed has been defined in chapter six. In the other method discussed i. e. paper gaging on the contrary requires some precise training as to conduct the experiment. Besides the second factor, it is time consuming and mistake prone. So if we take more time to do a job, the cost (which is hidden variable) increases - like the wages will be more for the inspector, instead of measuring many parts a day, we measure only a few ones (depending on the type of the method used). This delays the process of supply to the customer. Besides functional gages never ever accept a bad part, also it literally represents functional interface of the concerned features. Moreover with the utilization of GD&T properly the gain is more and quality of the products is ensured.

7.2 Future Research

Tremendous potential had always lied in the improvement of Geometric Dimensioning and Tolerancing language itself. The current system is an effort of three decades of research by an committee action representing military, industrial, and educational interests. In fact this standard actually evolved out of three different standards. Now in our case to remove the handicap in measuring LMC and RFS conditions, it will be very difficult to produce a single functional gage (physical kind), which will vary as and when required - like a special metal.

The scope lies in eliminating out RFS and LMC conditions altogether or else combining them to form a new system and introduce some new means to specify this situation, which the designer could clearly communicate the idea to the production department. Moreover the machinist must also be given more room to work the part with. That is the machinist should be given a part with the right amount of tolerance.

REFERENCES

[1] St. Charles, David P. "Little-used System can Solve Manufacturing Problems." *Quality Progress.* February (1990): 32-33.

[2] Gossard, D.C., Zuffante, R.P., and Sakurai, H. "Representing Dimensions, Tolerances and Features in MCAE Systems." *IEEE Computer Graphics and Applications* . March (1988): 51-59

[3] Foster, Lowell W. "The Application of Geometric Tolerancing Techniques." *Geo-Metrics 11.* (1986).

[4] Johnson, R.H., and Associates. "Dimensioning and Tolerancing." *Final Report.* R-84-GM-02.2. CAM-1. May (1981)

[5] Kirkpatrick, Elwood G. *Quality Control for Managers and Engineers.(1970)*

[6] Kotefski, Steve. "Geometric Dimensioning and Tolerancing." To be Published (1991)

[7] Krulikowski, Alex. "Nine Myths of Geometric Dimensioning and Tolerancing." *Machine Design.* June 6 (1991): 86

[8] Krulikowski, Alex. "The Seven Deadly Sins of Dimensioning and Tolerancing." *Machine Design.* November 8 (1990): 60-61.

[9] Requicha, A.A.G. "Toward a Theory of Geometric Tolerancing." *The International Journal of Robotics Research.* 4 Winter(1983): 45-60.

[10] Shepherd, Don, W. "Applying Geometric Dimensioning Successfully." *Machine Design.* May 12 (1988): 112.

[11] Thompson, Daniel, C. "Tighter Tolerances at Lower Costs." *Mechanical Engineering.* September (1988): 36-42.

[12] Utpal, R. "Feature-Based Representational Scheme of a Solid Modeler for Providing Dimensional and Tolerancing Information." *Robotics and Computer - Integrated Manufacturing,* 3/4 (1988): 335-345.

[13] Valaer, Paul. "The Problems with Geometric Dimensioning and Tolerancing." *Machine Design..*August 23 (1990): 129-133.