ABSTRACT

Geometric Dimensioning and Tolerancing
A Tool for Concurrent Engineering

by

Tapan Kumar S. K. Jain

The concept of Concurrent Engineering recognizes an immediate need for a new design environment and technology and so requires extensive interdisciplinary cooperation and integration of diverse functions of a manufacturing organization such as marketing, design, manufacturing and finance. One of the key factors to achieve successful integration among the departments is better communication and it becomes imperative in cases of varying levels of communication needs, especially in interdepartmental cases.

Concurrent Engineering is a philosophy which provides certain benefits. There are various tools and methods available for implementation of Concurrent Engineering concepts. One of the tools is Geometric Dimensioning & Tolerancing (GD & T), which can be used for indespensible communication of exact part design and its proper execution. Unlike other tools, GD & T concepts emphasize on the integration of various functions in a manufacturing organization.

This thesis discusses the applicability of Geometric Dimensioning and Tolerancing as an integrating tool for related functional departments in the concurrent environment. It also establishes the synchronization between the objectives of the two concepts. Also, it discusses the effect of using GD & T on vendor lead time and manufacturing lead time. The effect on the product quality, the cost economics and the learning curve is also investigated.

Lastly, the thesis concludes that the implementation of GD & T concepts automatically attains the objectives of concurrent engineering. The use of GD & T in industries may lead to widespread implementation of the concurrent engineering concepts globally. Therefore, it can be considered as a medium or tool for Concurrent Engineering.

GEOMETRIC DIMENSIONING AND TOLERANCING
A TOOL FOR CONCURRENT ENGINEERING

by
Tapan Kumar S. K. Jain

A Thesis
Submitted to the Faculty of
New Jersey Institute of Technology
in Partial Fulfillment of the Requirements for the Degree of
Masters of Science in Manufacturing Systems Engineering

Manufacturing Engineering Division
May 1993

APPROVAL PAGE

Geometric Dimensioning and Tolerancing
A Tool for Concurrent Engineering

by

Tapan Kumar S. K. Jain

Dr. Stojan Kotefski, Thesis Adviser — Date
Assistant Professor and Program Coordinator for
Manufacturing Engineering Technology, NJIT

Dr. Raj Sodhi, Committee Member — Date
Associate Professor of Mechanical Engineering and
Director of Manufacturing Engineering Programs, NJIT

Dr. Nouri Levy, Committee Member — Date
Associate Professor of Department of Mechanical Engineering, NJIT

BIOGRAPHICAL SKETCH

Author : Tapan Kumar S. K. Jain

Degree : Master of Science in Manufacturing Systems Engineering

Date : May 1993

Undergraduate and Graduate Education :

- Master of Science in Manufacturing Systems Engineering, New Jersey Institute of Technology, Newark, NJ, 1993
- Bachelor of Science in Mechanical Engineering, Birla Vishwakarma Mahavidyalaya, Gujarat, India, 1989

Major : Manufacturing Systems Engineering

This Thesis is dedicated to
my Mother

ACKNOWLEDGMENT

The author wishes to thank his thesis adviser, Dr. Stojan 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, continuous motivation and sincere concern.

Furthermore, I would like to express my gratitude to Dr. Nouri Levy and Dr. Raj Sodhi for serving as committee members and providing necessary guidance for this work.

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

Lastly, special thanks to Pankaj, Anjali, Deven and all other friends for their unique support and cooperation.

TABLE OF CONTENTS

Chapter Page

LIST OF TABLES

LIST OF FIGURES

Figure	Page

CHAPTER 1
INTRODUCTION

1.1 Introduction

In recent years, the advent of new sophisticated systems, methods, and concepts in manufacturing engineering has led to highly competitive environment. Even though these methods are employed, the performance varies from industry to industry. The reason for these performance varitions may be many depending upon the specific company condition. One of the reasons could be lack of integration of different functional departments of the organization. The methodologies developed so far emphasize on the improvement of one function only and so have limited scope of integrating different functional departments. They do not incorporate the requirements of all the departments. This leads to improvement of individual function but later when they are combined, the complications arise or may lead to infeasibility.

On the contrary, Concurrent Engineering concepts are recent developments, which focus on the integrated approach for design, manufacturing, inspection and support. However, the concepts of concurrent engineering are not new. They have been practiced for a long time but never been thought of using them in a systematic manner. Japanese industries unknowingly have been using some aspects of the Concurrent Engineering. Now their government is helping them to promote it by carrying out research on the feasibility of establishing such an engineering design technology. These concepts are often applied to a small scale industry, where there are few people involved in each department. The facilities are very limited. Therefore, the people have better understanding and interaction among themselves and the departments. But, it becomes dificult in a big organization which have larger departments with many people involved in various activities. The facilities are remote and the communication is not clear & effective. In such cases, a systematic approach is required with concrete regulations to integrate the organizational activities. Concurrent engineering concepts work in this direction.

Recent efforts toward the integration of the product and process design such as ' design for manufacturing', 'design for assembly', 'design for reliability', 'design for automation', attempt to address some of the problems which are generated by job specialization and separation. The new concepts of integration of engineering design and system manufacturing yields significant advantages such as reduction in market lead time, increase in product quality and reliability and reduction in life cycle cost. Despite the fact that many of these trends are important, they lack an overarching rational framework to guide their implementation and they do not always take full advantage of modern managerial and technological capabilities [9]. There requires an approach or media which, while implementing, can take into consideration all the functions.

Offlet, the engineers and scientists are focussing more towards development or employment of new methods or systems. But many of the shortcomings present in the existing systems remain ignored. One of the defects lies in the traditional design language itself. To overcome the problems of traditional drawing, Geometric Dimensioning and Tolerancing (ANSI Y14.5 -1982) has come into effect. This is a design language which is clear, precise and which improves productivity.

1.2 Background

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 carries entirely a different concept than the traditional coordinate dimensioning. It overcomes all the shortcomings of conventional drawing and uniquely supports the objectives of design, manufacturing, tooling, inspection and other related groups.

ANSI Y14.5-1982 is the authoritative document in United States for Geometric Dimensioning & Tolerancing. This standard evolved out of a consolidation of earlier standards namely ANSI Y14.5-1973, USASI Y14.5-1966, ASA Y14.5-1957 and MIL-STD-8C, October 1963. The prime goals for such a consolidation are:

1. to provide a single standard for practices in United States,
2. to update existing practices in keeping with technological advances & 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 United States' desire to be more active, gain greater influence, and pursue a more extensive exchange of ideas with nations in the area of international standard development. [1]

GD & T has gained acceptance in manufacturing environment because it is the link that acknowledges machining capabilities and desired part configurations through the utilization of graphical symbols for form, fit, and function requirements. The GD & T system allows one to maximize tolerance conditions of the parts, while still maintaining interchangeable characteristics. The GD & T technique uses above normal practices in addition to the datum reference, basic dimensions, and various geometric control characteristics, classified in five groups of Form, Orientation, Location, Profile and Runout. These requirements are generally not specified in the standard print specifications, but these additional specifications will further assure product compliance.

The objectives of GD & T are clear and well defined and may lead to effective coordination among the departments. But much less awareness is found in actual practice in the industries. Very few organizations and acedemic institutions provide a formal training or education in the field of GD & T. Majority of them still stick to the conventional drawing methods or they hesitate to employ GD & T approach associating certain myths to it, for example, GD & T raises product costs, GD & T and ANSI Y14.5 are confusing, it is easier to use coordinate dimensioning, Dimensioning and Tolerancing are separate steps, GD & T should be used on critical parts and so on. The GD & T method is used by Military and automative industries.

Concurrent engineering concepts emphasize working together by all the related department representatives around a table. The process may work fine for a short time but then may cause fight among themselves because of a variety of conflicting factors. In such cases, there has to be a systematic methodology or tool which takes care of these factors and should be abide by all the departments.

Although the philosophy of Concurrent Engineering reflects integration of all the activities related to the design of product such as market , sales, finance, engineering, manufacturing and support, it is crucial to achieve this integration during the product design process. An Internal Company studies at Westinghouse, GM's Detroit Diesel Allison Division, Ford, and Rolls Royce, and others indicate that about 70 % of the life cycle cost of a product is determined when it is designed [12] . Design choices determine materials, fabrication methods, assembly methods, and to a lesser degree material handling options, inspection techniques, and other aspects of production system. Production engineers and shop floor workers will consume less time and effort if they are presented with a finished design, and so reduces overall product development cost.

Traditional approaches for product development like Sequential Engineering involves number of recurring activities because of linear characteristics of the process. The information flow is unidirectional from customers need to design to manufacturing to market. The process does have a feed back but its implementation is delayed till each discipline finishes its part of the process. Turino of Logical Solution Technology, Inc. in his paper describes that Concurrent Engineering is an integrated approach that eliminates recurring activities like redesigning and reverification, thereby saving time to market typically between 10 and 25 % and results in a better product. [18, p. 192]

British aerospace study indicates that approximately 65 % of total cost is spent during product conception and validation stages as compared to rest spent in the development, production, operation and support stages of the product development. Therefore, as much of the product's cost is commited early in the design, all the product issues must be considered from begining.

One of the greatest difficulties in organizing multidisciplinary teams, which is an CE concept, is communication among the team members. Markowitz states in his article " Concurrent Engineering journey starts with the first step", that for many organizations, communication may mean collocating employees along project, rather than functional lines [10, p.113]. Otherwise, it is neccessary to provide some communication tool, assuming that project development communications may need a tool beyond that telephone and facimile transmission.

The objectives clearly indicate the intention of using GD & T as to provide a uniform understanding in print reading as regard to part manufacture. Gehrke in his article states [3, p.86] that GD & T is the simplest way to avoid ambiguity in print interpretation. The symbology and concepts used in GD & T do not directly reference to other standards like ISO 9000 series or ABCA (America, Britain, Canada and Australia), but requirements included in these standards effectively mandate the use of GD & T.

1.3 Research Emphasis

In a concurrent environment, where integration is a key word, communication should be accurate and interpretable at all the levels in a uniform way. Today's sophisticated engineering design demands, new and better ways of accurately communicating requirements is one of the reasons for GD & T and this is true in a manufacturing, inspection and tooling enviroments. This is one of the area where importance is given in this thesis.

The thesis compares and analyses the objectives of Concurrent Engineering concepts with Geometric Dimensioning & Tolerancing method of drawing. The implementation of GD & T concepts in different areas such as design, manufacturing, tooling is discussed in details. Later at each area, it will be shown how GD & T concepts interlink all these departments and how if establish the concurrent engineering objectives directly or indirectly. To highlight the importance and accuracy of conditions like MMC, RFS and LMC, besides all the geometric characteristics, emphasis is also given to usage of functional gages. A condition is discussed when LMC features are required to be measured. An alternative method is highlighted for RFS and LMC feature measurements. Moreover, the cost and quality objectives of Concurrent engineering will be established by the use of GD & T methodology. It should also be emphasized that GD & T should be the "key word" for industries and its implementation would automatically lead to realization of concurrent engineering concepts.

CHAPTER 2
GEOMETRIC DIMENSIONING & TOLERANCING

2.1 Introduction

GD & T can be described in its simplest terms as a means of specifying the geometry or shape of a piece of hardware on an engineering drawing. GD & T is one of the three types of dimensions used on engineering drawings. Figure 1 shows how geometric dimensioning fits into the total subject of dimensioning of engineering drawings.

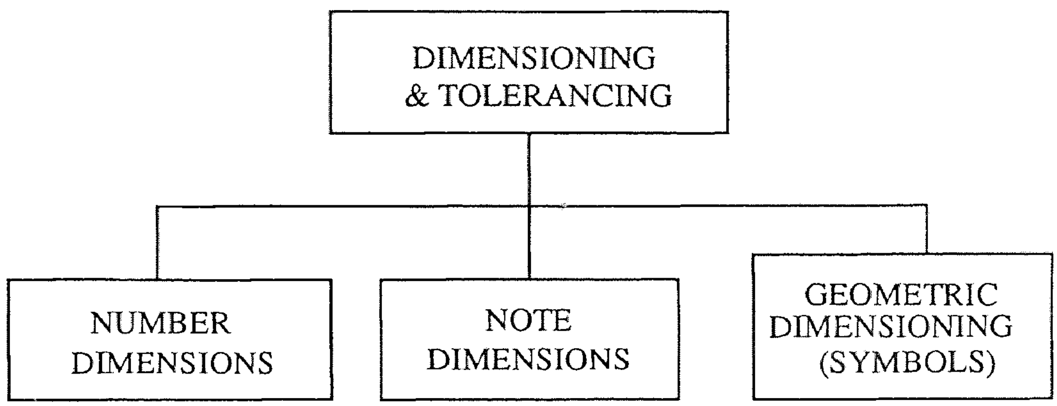


Figure 1. Types of Dimensioning and Tolerancing

GD & T is a dual purpose system. First it is set of standard symbols whicn are used to define part features and their tolerance zones. The symbols and their interpretations are documented by American Standards Institute Dimensioning Standard (ANSI Y14.5M-1982). Secondly, GD & T has philosophy of designing part based on its functions. It is a powerful language which helps a designer in providing with a clear way of expressing design intents and part requirements, which in turn enables the manufacturer to choose the proper method of inspecting and gaging the part, thus protecting the design intent. In this way the marketer the designer, the manufacturer, and inspector with the same standards can

can thus avoid misunderstandings.

The dimensions on a drawing with GD & T also define size and shape of the part in order to function as the design intended. This dimensioning philosophy is a powerful design tool. Also it helps in better communication. As a design philosophy it provides the most liberal tolerances, and thus can provide substantial saving in product costs and company's operating expenses.

Geometric dimensioning & tolerancing is rapidly becoming a universal engineering drawing language & technique that manufacturing industries and government agencies are finding essential to their operation well being.

The authoritive document governing the use of geometric dimensioning and tolerancing in the United States is ANSI Y14.5-1982, "Dimensioning and Tolerancing." This standard evolved out of a consolidation of earlier standards, ANSI Y14.5-1973, USASI Y14.5-1966, ASA Y14.5-1957, SAE Automative Aerospace Drawing Standards and MIL-STD-8C, October 1963. This consolidation has accomplished over years by committee representing military, industrial, and educational interests. The work of the committee has had 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 area of application,
3. to establish a single basis and "voice" for the United States in the interest of international trade, in keeping with United States' desire to be more active, gain greater influence, and pursue a more extensive exchange of ideas with nations in the area of international standard development.

2.2 Advantages

During the past 40 years, the GD & T has matured to become as indespensible tool; it assists productivity, quality, and economics in building and marketing products around the world. The military, the automotive and other industries have been using GD & T for over the years. One of the reason that this subject has become popular is that it saves money. The other advantages of theuse of GD & T can be grouped as following :

1. Improve communications :

GD & T can provide uniformity in drawing specifications and interpretations, and so reduces controversy, guesswork and assumptions. Design, production, and inspection all work to the same language.

2. Better product designs:

The use of GD & T can improve the product designs. First, by providing designer with tools to "say what they mean". Second, by establishing a dimensioning philosophy based on part function. This philosophy, called functional dimensioning, studies product function in the design stage and establishes part tolerances based upon functional requirements.

3. Production tolerances increased :

There are two ways tolerances are increased through the use of GD & T. Firstly, under certain conditions, GD & T provides "bonus" or extra tolerance for manufacturing. This additional tolerance can make a significant savings in production costs. Second, by the use of functional dimensioning, the tolerances are assigned to the part based upon its functional requirements. This often results in a larger tolerance for manufacturing. It eliminates copying existing tolerances, or assigning tight tolerances, because of lack of knowledge to decide reasonable tolerances.

4. Reduced rework and reverification :

Since the drawing indicates the dimensions clearly and unambiguously and how the part is to be manufactured and inspected, the parts are produced exactly as per the design requirements. The rejection quantity is reduced and hence rework and reverification is reduced or eliminated in many cases.

5. Time and Cost saving :

Reduction or elimination of ambiguity, confusions, conflicts, rework and reverification ultimately provides for saving in time and so reduced production costs. And thus the competitiveness of the company.

6. Interchangability :

GD & T is a powerful addition to drafting documentation practice that provides increased design and manufacturing flexibility, and it can ensure 100 % interchangability at optimm cost.

2.3 Disadvantages :

The biggest limitations of GD & T is lack of awareness and training programs availlable in this field. There are very few school and organizations where a formal course on GD & T is offered. Most of the organization employ traditional way of dimensioning which has discrepencies and may leads to either wrong interpretation or incomplete dimensioning specifications. The involvment of people with this subject is because of their personal interest and they developed it by reading articles & books teaching themselves. Another shortcoming is the large number of bad examples of GD & T on drawing today & so lack of uniform interpretation. This makes it extremely difficult, if not impossible for drawing users like manufacturing & inspection departments to correctly interpret drawings where there is no correct interpretation. This leads to much confusion. Usually GD & T is blamed, when really the confusion exist because the dimensions are incorrectly applied.

2.4 Functional Dimensioning

Functional dimensioning is a philosophy of dimensioning & tolerancing a part based on how it functions. When functionally dimensioning a part, the designer performs a functional analysis. A functional analysis is a process in which a designer identifies the functions of a part and uses this information to establish the actual part dimensioning & tolerances. Functional dimensions & analysis are very powerful design tool. Yet, the use of functional dimensioning requires a lot of effort & time even for an experienced designer. The rewards with such types of benefits are :

1. The designer will develop an objective design philosophy.
2. The designer will develop a true understanding of functioning of each part in design.
3. Potential product problems will be identified at the design stage.
4. An objective method for evaluating change requests will be established.
5. Larger tolerance for manufacturing. Tolerances will be based on the "maximum allowable tolerance that will not adveresly affect the product function."
6. Promote better communication between design & development departments.

7. Fewer change requests. In most cases, part tolerances will already be at their maximum value.

2.5 G D & T Terminology

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

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

Angularity : Angularity is the condition of a surfacre, 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 (abbreviated as BSC) is a theoritically 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 bileteral tolerance is a tolerance in which variation is permitted in both directions from the specified dimensions. e.g. 2.50 ± 0.005, where 2.50 is basic dimension and ± 0.005 is bilateral tolerance.

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 intersect by any plane:

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

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

Clearence Fit: A clearence fit is one having limits of size so prescribed that a clearence 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 theoritically 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 theoritically exact axis of 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).

Datum Feature : A datum feature is an actual (physical) feature 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. e.g. | - A - |

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

Datum Reference Planes : 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 fro dimensions for design, manufacture, and measurement. It provides complete orientation for the features involved.

Datum Surface : A datum surface or feature (hole, slot 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 characteristics 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 of GD & T language. 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 ispermitted to vary from true (theoritically 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.

2.6 Geometric Characteristics

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

By controling the form of a feature, there are four characterstics of a part that can be controlled; the flatness of its surface, the straightness of its line element, the roundness of a circular section, or the cylindricity of the part. Whenever the boundaries established by tolerances of size, location, and Rule #1 (MMC conditions) do not supply sufficient control to satisfy part functional requirements, then a form tolerance is applied. Form controls always apply to single features or features-of-size. Form controls are used to define the shape of a feature in relation to itself. Therefore, form controls never use a datum reference.
Flatness of a surface means all the elements of the surface falls in a single theoritical plane. A flatness tolerance is the limit within surface elements are permitted to

vary from a theoritical plane. This is represented in Figure 2. The high and low limits of this surface must lie within the 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 in latitudinal sweeps while the right requires longitudinal sweeps. The net effect is the same as the flatness callout which assumes both sweeps simulteneously.

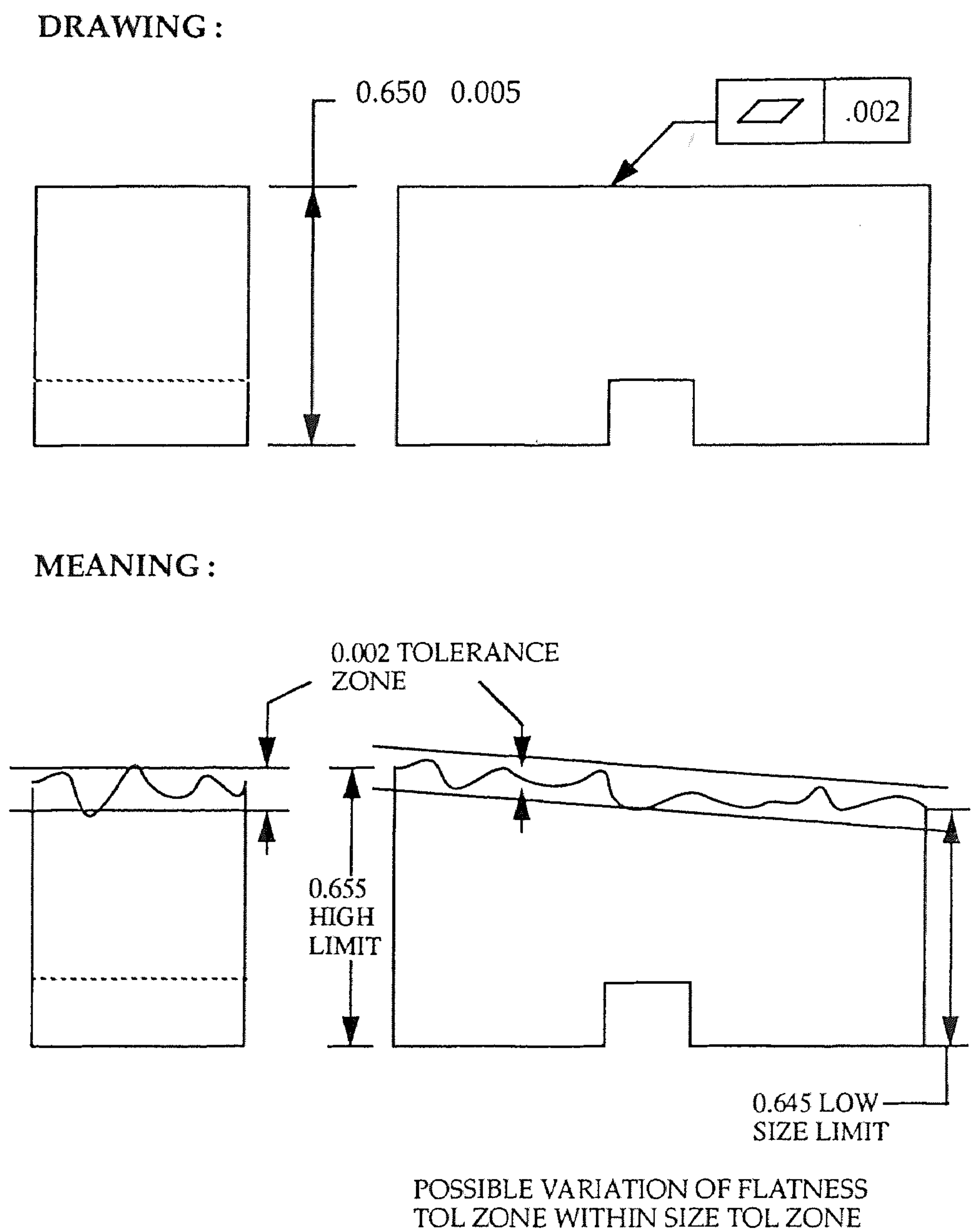
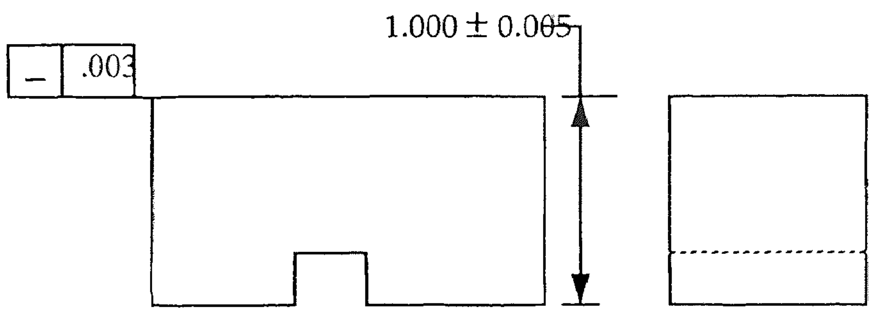


Figure 2 . Flatness

Straightness is a condition which implies linearity or straightness of the object . A straightness tolerance is applied in the view where the elements to be controlled are represented by a straight line. It is typically used as a form control of individual surface such as those on cylindrical or conical surfaces. Since surfaces of this kind are made up of an infinite number of longitudinal elements, a straightness requirement applies to the entire surface as controlled in single line elements in the direction specified. Straightness of size feature (control of axis) is more common and it permits use of Maximum material condition principles. For any size specified within the straightness tolerance range, a straighness of 0.003 must be held. This control of straighness is with in element lines only. The maximum and minimum size can never be violeted. (See figure 3)

DRAWING :



MEANING :

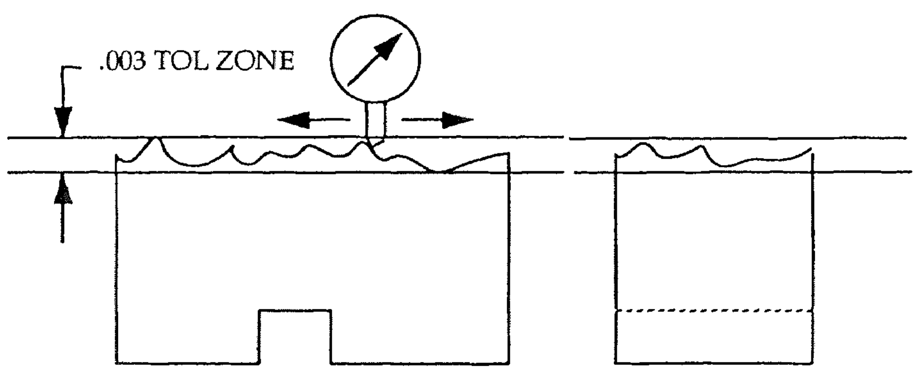


Figure 3 . Straightness

Circularity is the condition on a surface for revolution where:

1. in the case of a cylindrical or cone, all points of the surface intersected by any plane perpendicular to a common axis are equidistant from their axis.
2. in the case of a sphere, all points of the surface intersected by any plane passing through a common center are equidistant from that center.

A circularity tolerance specifies a tolerance zone bounded by two concentric circles within which each circular element of the surface must lie and applies independently at any plane described above. Limits of size exercise control of circularity within the size tolerance. Often this provides adequate control. However, where necessary to further refine form control, circularity tolerancing can be used on any figure of revolution or circular cross section. Figure 4 shows a part

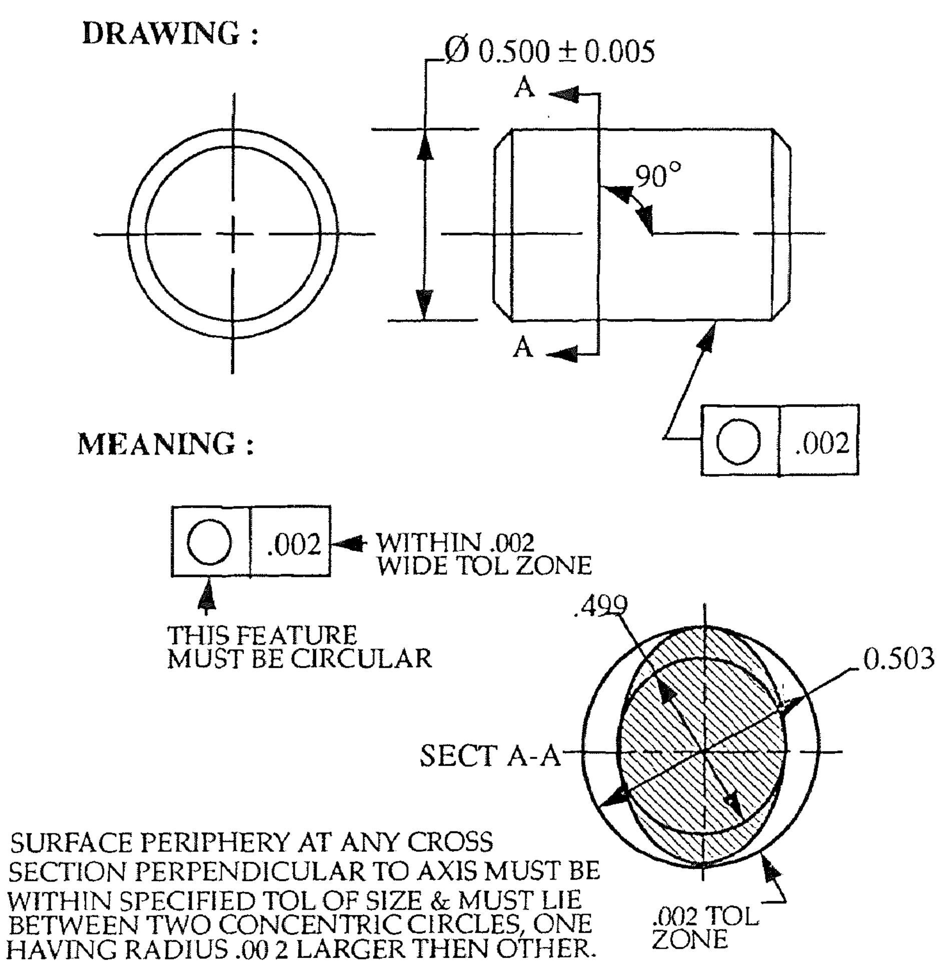
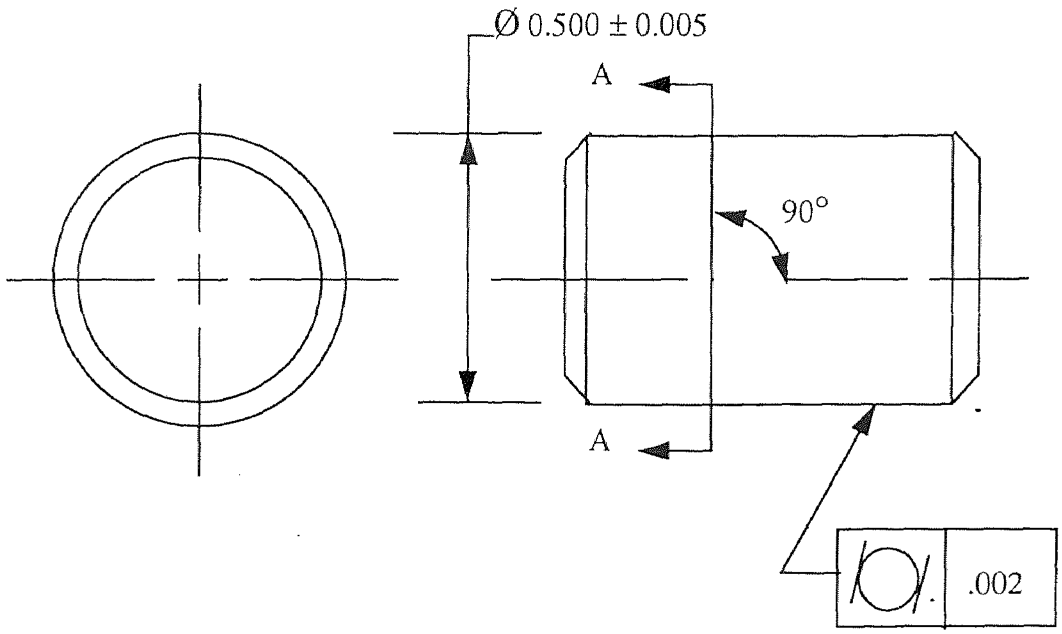


Figure 4 . Circularity

DRAWING :



MEANING :

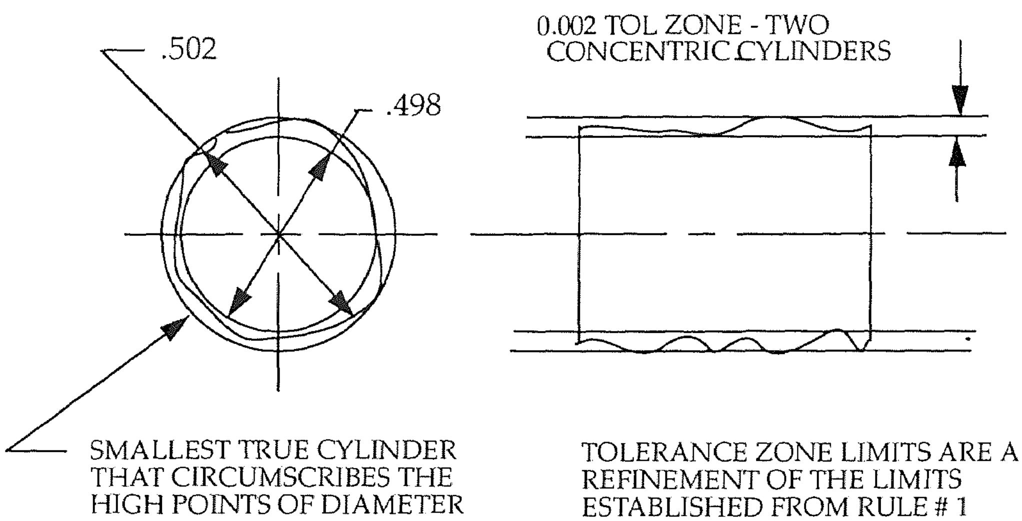


Figure 5 . Cylindricity

with circularity tolerance of 0.002 specified on cylindrical part. The interpretation also indicates how the tolerance zone can be established.

Cylindricity is the condition of a surface of revolution in which all points of the surface are equidistant from a common axis. A cylindricity tolerance specifies a tolerance zone bounded by two concentric cylinders within which the surface must lie.

Limits of size exercise control of cylindricity within the size tolerance. This control is often adequate. However, where more refined form control is required, cylindricity tolerancing can be used. In cylindricity, unlike circularity, the tolerance applies simultenouslyto both circular and longitudinal elements of entire surface.

Figure 5 illustrates a part with a cylindricity tolerance of 0.002. A cylindricity tolerance is interpretated as 0.002 wide tolerance zone defined by two concentric cylinders 0.002 apart. A cylindricity tolerance can be considered circularity tolerancing extended to control the entire surface of a cylinder.

2. ORIENTATION Tolerance : When no orientation controls are specified on a drawing, the orientation (i.e. squareness, angularity, & parallelism) of the part features is controlled by one of the various methods. Lines shown at right angles often have their tolerance controlled by an angular dimensions with a tolerance, or a general note for angular tolerances on the drawing. Features which are shown parallel on a drawing are often controlled by the tolerance limits of the dimension locating the feature surfaces in conjunction with Rule # 1. Orientation controls become necessary when the type of controls mentioned above are inadequate or insufficiently accurate to satisfy the functional requirements.

Orientation controls define the angularity, squareness, and parallelism of part features relative to one another. These are sometimes refers to as attitude controls. There are mainly three orientation controls, namely : Perpendicularity, Parallelism and Angularity.

Perpendicularity is the condition of a surface, or centerplane, or axis being exactly 90° to a datum. A perpendicularity tolerance is the amount which a surface, or axis, or a centerplane is permitted to vary from being perpendicular.

Most perpendicularity applications fall into one of four types of general cases:

1. Perpendicularity applied to a surface or a planer feature-of-size

In this case, the perpendicularity control specifies a tolerance zone defined by two parallel planes perpendicular to a datum plane or axis within which the surface or median plane of the feature must lie. See figure 6 and 7 when it applied to a feature and feature-of-size.

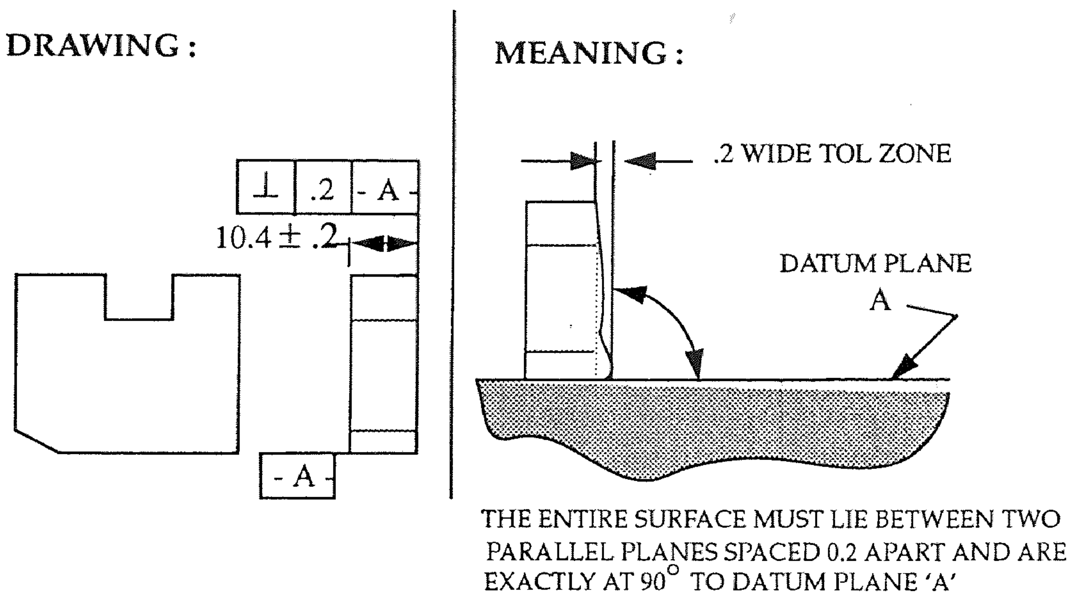


Figure 6 . Perpendicularity Applied to a Plane Surface

2. Perpendicularity applied to a diameter (in one direction only)

In this case, the perpendicularity control specifies a tolerance zone defined by two parallel planes perpendicular to a datum plane or axis within which the axis of the tolerances feature-of-size must lie.

3. Perpendicularity applied to the axis of diameter

In this case, the perpendicularuty control specifies a cylindrical tolerance zone perpendicular to datum plane or axis within which the axis of the considered feature must lie.

4. Perpendicularity applied to a surface line element

In this case, the perpendicularity control defines a tolerance zone of two parallel lines perpendicular to a datum plane or axis.

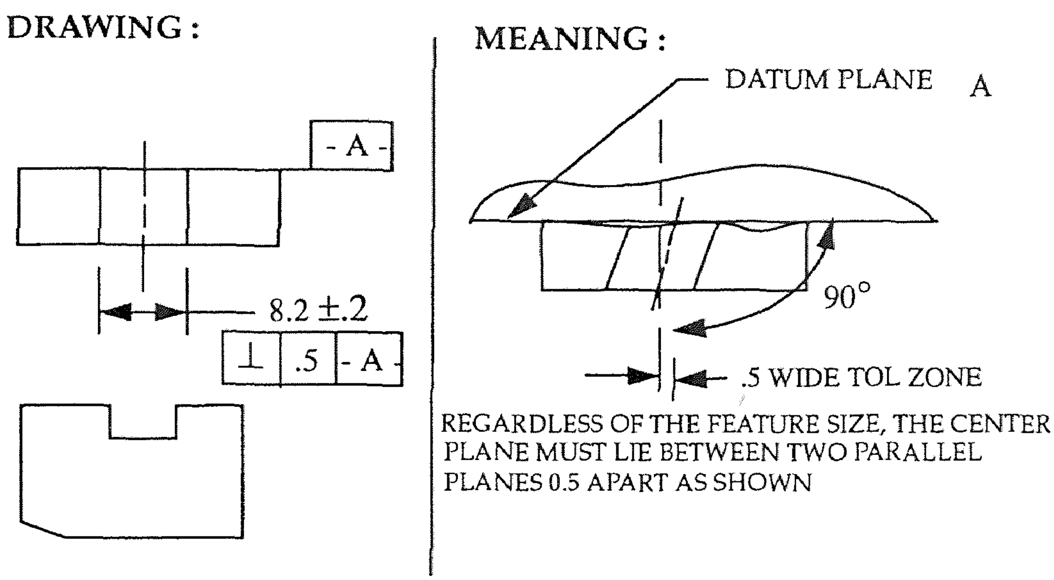


Figure 7. Perpendicularity Applied to a Slot - Centerplane Control

Angularity is the condition of a surface, centerplane, or axis being exactly at a specified angle from a datum. An angularity tolerance is the amount which a surface, centerplane, or axis is permitted to vary from its specified exact angle. Angularity establishes a tolerance zone for a surface, centerplane, or axis which is specified as a basic angle (other than 90°) from the datum plane or axis. An angularity tolerance zone has always two parallel planes.

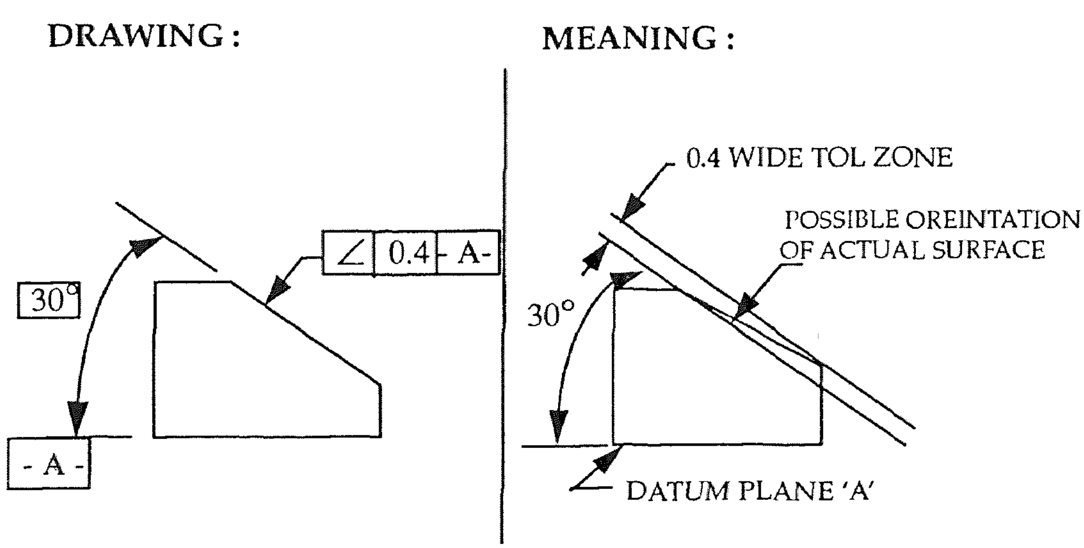


Figure 8. Specifying Angularity for a Plane Surface

There are two main types of applications for angularity :

1. Angularity applied to a surface or a planer feature-of-size

In this case, the angularity control specifies a tolerance zone defined by two parallel planes at the specified basic angle from the datum plane or axis within which the surface or centerplane of the considered feature must lie. See figure 8.

2. Angularity applied to axis

The angularity control specifies a tolerance zone defined by two parallel planes at the specified basic angle from a datum plane or axis within which the axis of the considered feature must lie. See figure 9.

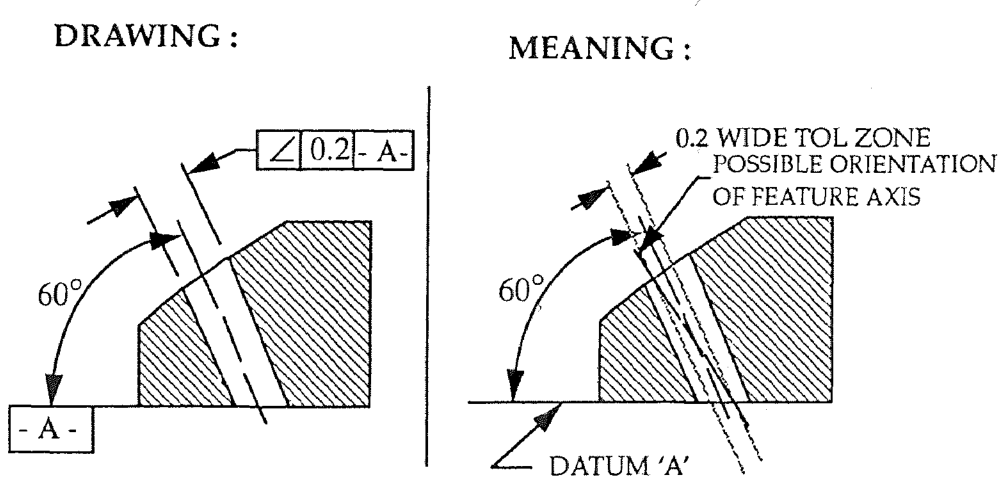


Figure 9. Specifying Angularity for an Axis (Feature RFS)

Parallelism is the condition where all points of a surface, centerplane, or axis are at equidistant from the datum plane or axis. A parallelism tolerance is the amount by which a surface, centerplane, or axis is permitted to vary from the parallel state. A parallelism control establishes a tolerance zone of two parallel planes or a cylinder within which all points of a controlled surface, centerplane. or axis must lie. There are two main applications within which almost all cases can fit. They are

1. Parallel planes as a tolerance zone

As shown in the figure10, this parallelism control specifies a tolerance zone defined by two planes parallel to a datum or axis. The distance between the planes is the tolerance value specified in the parallelism callout. All elements, line elements or axes of the considered feature must lie within these planes.

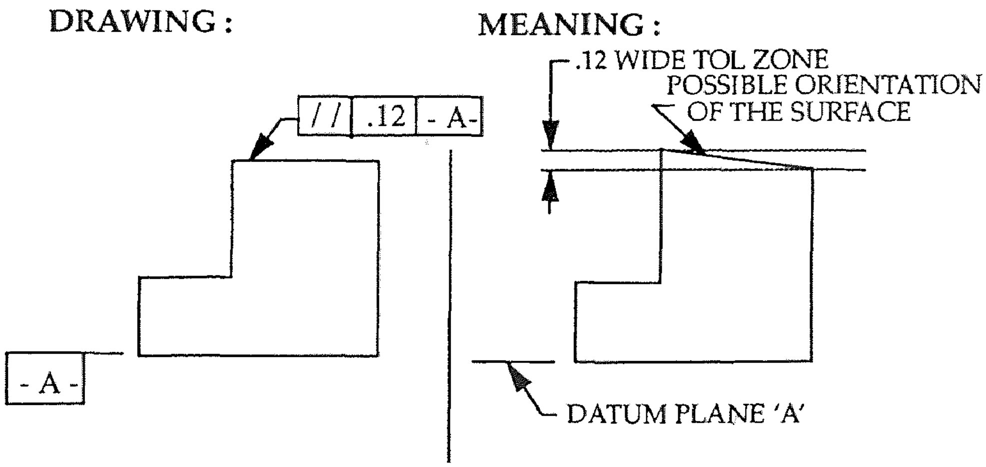


Figure 10. Specifying Parallelism for a Plane Surface

1. A cylinder as a tolerance zone

This parallelism callout specifies a cylindrical tolerance zone parallel to a datum axis within which the axis of the considered feature must lie. This cylindrical tolerance zone is designated by a diameter symbol in the tolerance portion of the feature control frame. The diameter of the tolerance zone is equal to the tolerance value specified in the parallelism callout. (See Figure 11)

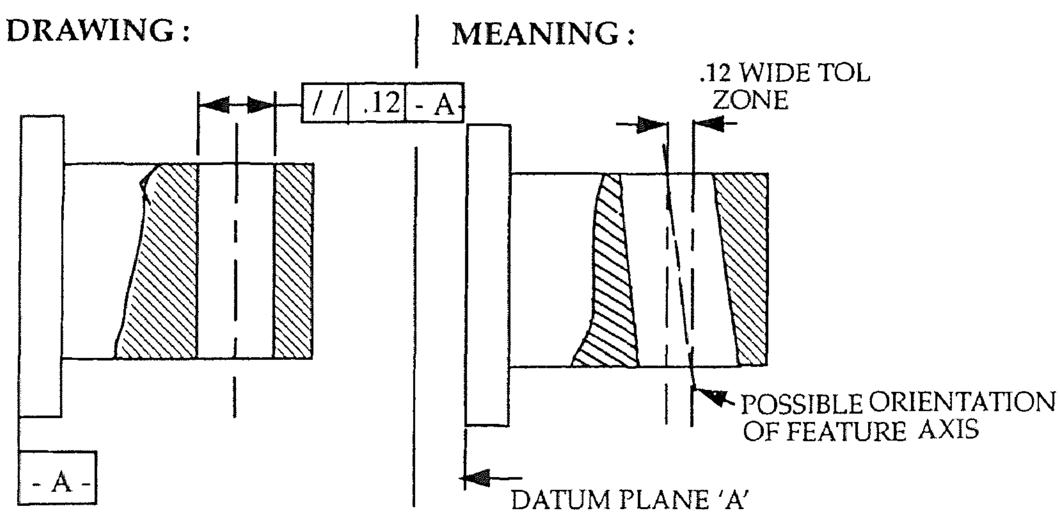
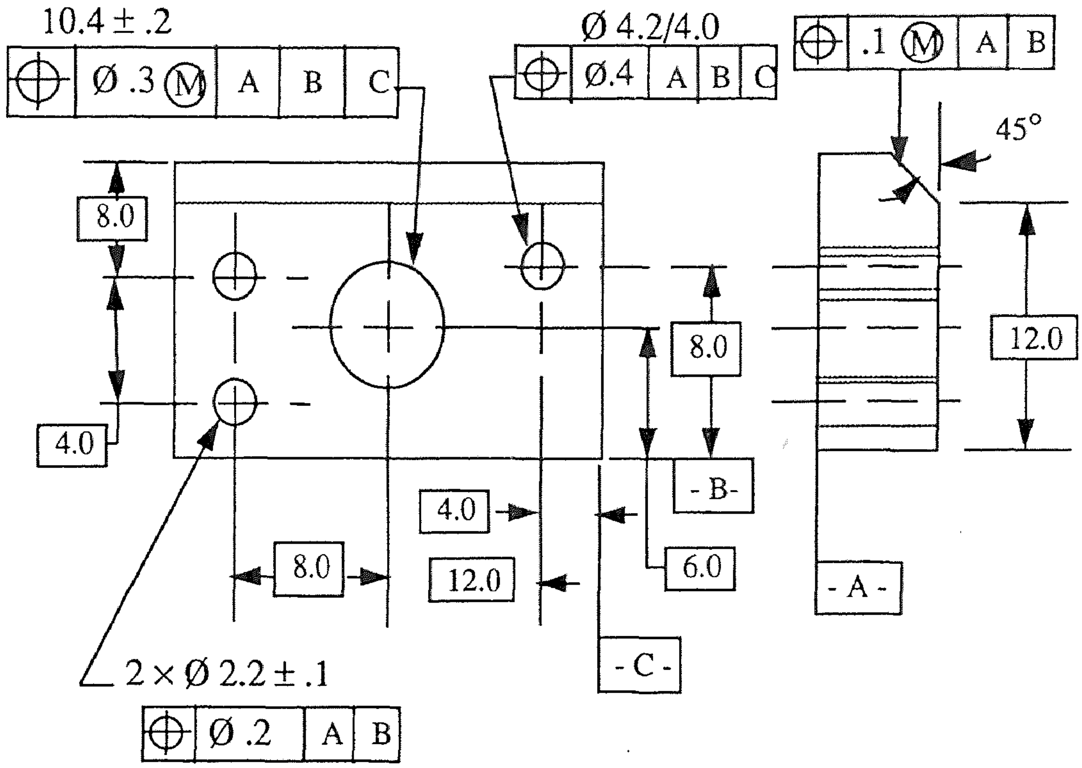


Figure 11. Specifying Parallelism for an Axis (Feature RFS)



POSITIONAL TOL	BASIC REQUIREMENTS 1	2	3	4	COMMENTS
⌖ Ø .3 Ⓜ | A | B | C	YES	YES	YES	YES	GOOD APPLICATION
⌖ Ø .2 | A | B	YES	NO	NO	NO	NEEDS BASIC DIM FROM 'B' NEEDS TERI. DATUM TOL . AT MMC OR RFS ?
⌖ Ø.4 | A | B | C	YES	YES	YES	NO	DOES NOT SPECIFY IF TOL AT MMC OR RFS
⌖ .1 Ⓜ | A | B	NO	YES	YES	YES	POSITIONAL TOL MUST BE APPLIED TO FEATURE OF SIZE

Figure 12. Requirements of Positional Tolerancing Dimensioning

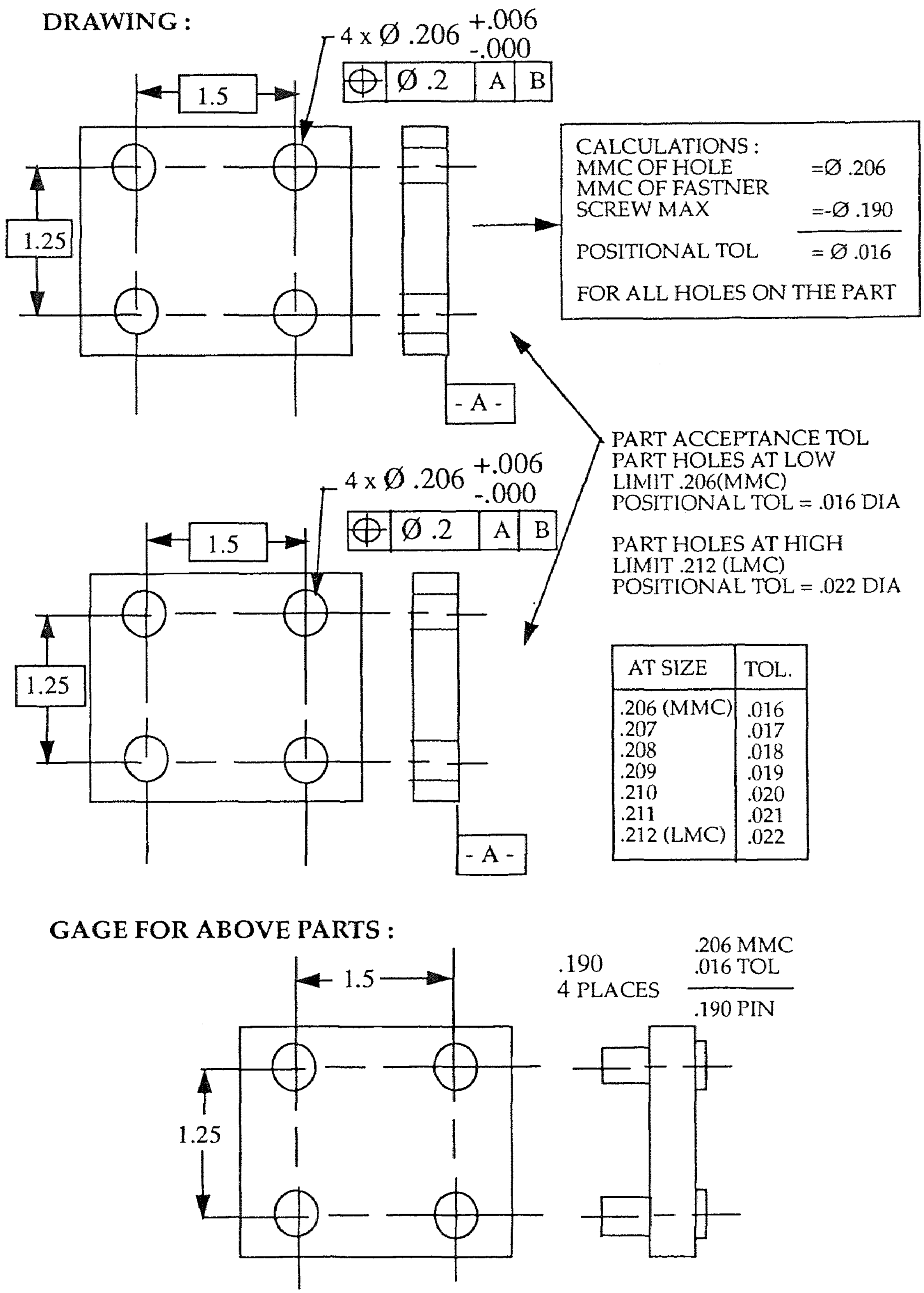


Figure 13 . Floating Fasteners - Positional Tolerances for Mating Parts

3. LOCATION Tolerances : 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. There are two forms in which the locational tolerances can be applied : Positional tolerance and Concentricity. The location tolerances deals with features-of-size only. Therefore, it must be specfied if they are to apply at MMC, RFS, or LMC.

Location tolerances involve features of size and relationships to center plane axes. When function or interchangability of mating part features is involved, MMC principle may be introduced to great advantage. Perhaps the most widely used and best example of the application of this rinciple is position tolerancing. Location tolerances are used to control three types of relationships :

1. Center distance between feature-of-size.
2. Location of a feature-of-size, or a group of feature-of-size relative to a datum or datums.
3. Coaxiality or symmetry of feature-of-size.

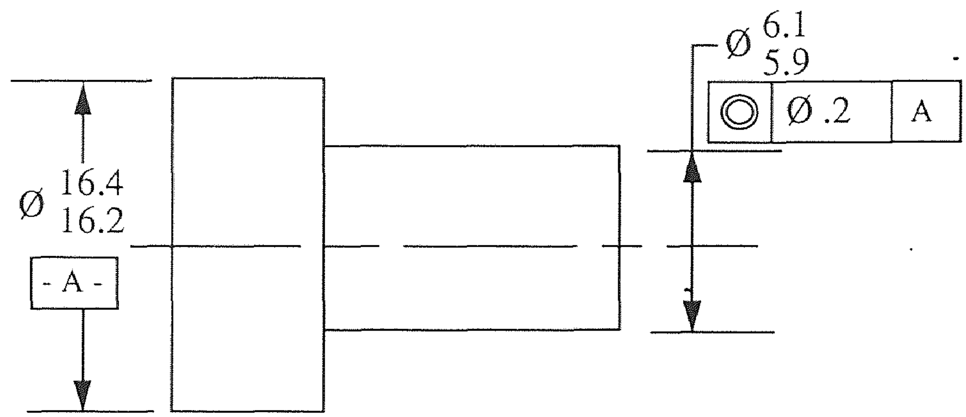
Position is a term used to describe the perfect (exact) location of a part, line, or plane of a feature in relationship with a datum reference or other feature.

Postion tolerance is the total permissible variation in the location of a feature about its exact true position. For cylindrical features the position tolerance is the diameter 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 (e.g. slots, cuts etc.) the position tolerance is the total width of the tolerance zone within which the center plane of the feature must lie, the center plane of the feature being at the exact true position.

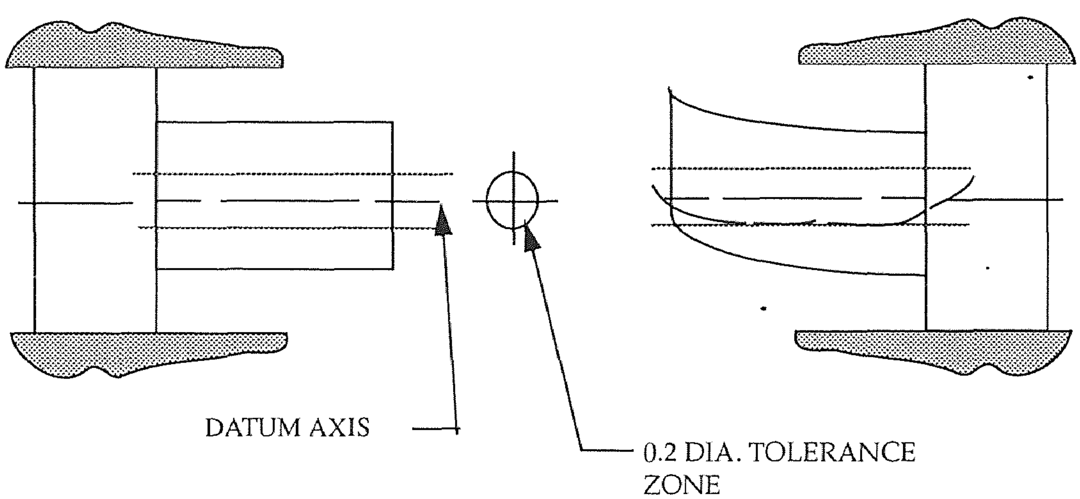
There are four basic requiremen for dimensioning system using positional tolerances :

1. The tolerance of position must be applied to feature of size.
2. Datum references are required and the datum must ensure that repeated measurements of the considered feature can be made.
3. Basic dimensions are used to establish the true location of the feature-of-size from the specified datum and between interrelated features-of-size.
4. MMC, LMC, or RFS must be specified in the feature control frame as prescribed

DRAWING :



MEANING :



EXAMPLE # 1

EXAMPLE # 2

THE DERIVED AXIS OF THE CONSIDERED FEATURE MUST LIE WITHIN THE CONCENTRICITY TOLERANCE ZONE, THIS AXIS IS ESTABLISHED BY ANALYSIS OF THE SURFACE ELEMENTS OF THE CONSIDERED FEATURE .

Figure 14 . Concentricity Tolerance Application

by Rule #2.

If any of these requirements are not fulfilled, the positional tolerance is uninterpretable. An example of positional tolerances are shown in Figure 12. Tolerance of position is used widely because of its ability to describe the requirements of interchangeable components. One of the primary applications of this is related to bolt hole pattern location because no other method describes the functional requirements of mating hole pattern so accurately. The other advantages of the positional tolerances are :

a. Round tolerance zone compared to square zone of coordinate system of dimensioning - 57% larger

b. Permits additional tolerances - bonus and shift

c. Permits use of fixed gages

d. Overcomes tolerance accumulation

e. Protects part functions

f. Lowers production cost

Figure 13 illustrates how the positioning tolerances can be applied to a floating fastner. It also indicates the calculations of tolerances and gage dimensions for checking the holes. More details are explained in chapter 4.

Concentricity is the condition where the axis of a cylinder, cone, square, hex etc. are common to the axis of a datum feature. Concentricity tolerance is the total amount of allowable variation of a feature-of-size to a datum axis. A concentricity tolerance is a cylindrical tolerance zone, whose axis is coincident with the datum axis, within which the axis of the considered feature-of-size must lie.

A concentricity tolerance zone and its datum reference can only be applied on an RFS basis. The size tolerance of a feature-of-size is independent of the concentricity tolerance. The measurement of concentricity tolerance requires that the axis of the considered feature-of-size to be established by detailed analysis of circular elements of the surface. This determines a point of the axis for each circular element checked. All the points of axis must lie within the concentricity tolerance zone. Since irregularities in the form of feature being inspected make it difficult to establish the axis of feature, therefore concentricity tolerances should be avoided whenever possible. When specifying tolerances for coaxial features, considerations should be given to positional or runout tolerances.

A simple illustration of a concentricity is as shown in figure 14.

The following terms apply when using a concentricity callout :

a. Rule # 1 is overridden

b. Rule # 3 applies

c. A datum reference is required

d. The tolerance zone must be RFS

e. The datum references must be RFS.

4. PROFILE Tolerance : Profile tolerance specifies a uniform boundary along the true profile within which the elements of the surface must lie.

A profile tolerance specifies a tolerance zone, always intended and measured normal to the basic profile at all points of the profile, within which the true part surface profile or line profile must lie.

Profile controls can be used to limit the form, size, or orientation of a part feature. The outline of an object in a given plane is referred to as the profile. There are two types of profile tolerances applied to a surface :

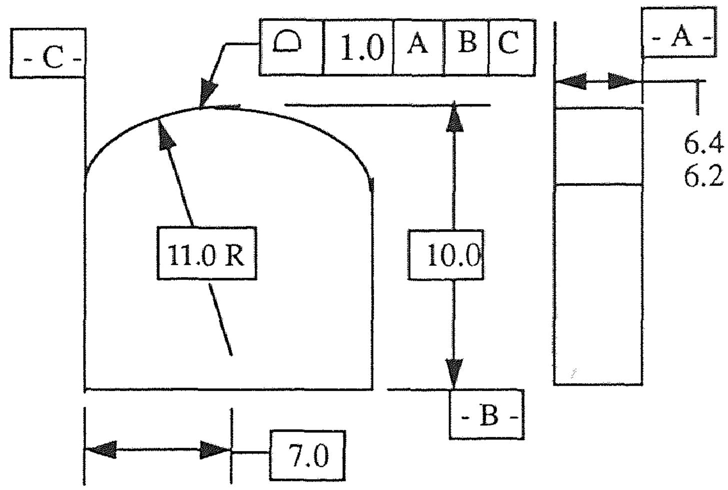
a. Profile of a surface :

The tolerance zone established by the profile of a surface tolerance is a three -dimensional zone or total control across the entire length and width or circumference of the feature, it may be applied to parts having a constant cross section or to the parts having a surface of revolution. Usually profile of a surface requires datum references. Figure 15 indicates the application of profile of a surface tolerances.

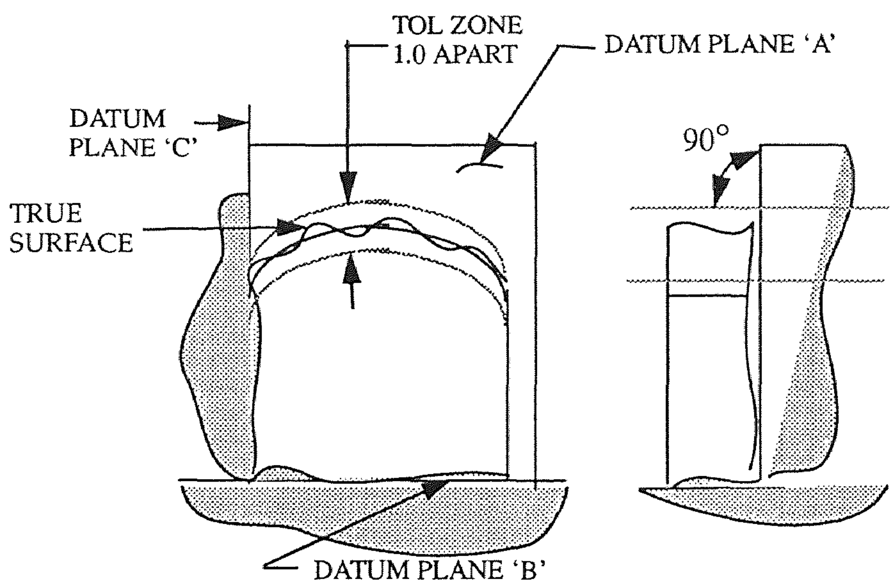
b. Profile of a line :

The tolerance zone established by profile of a line tolerance is a two-dimensional zone extending along the length of the considered feature; it may be applied to the profiles of parts having varing cross section such as propeller, aircraft wing, nose cone and other random cross sections where it is not required to control the entire surface as a single entity. Profile of a line may or may not require the datum references. Figure 16 illustrates the profile of a line application.

DRAWING :



MEANING :



Tolerance zone is equally spaced about true surface

Figure 15. Profile of a Surface

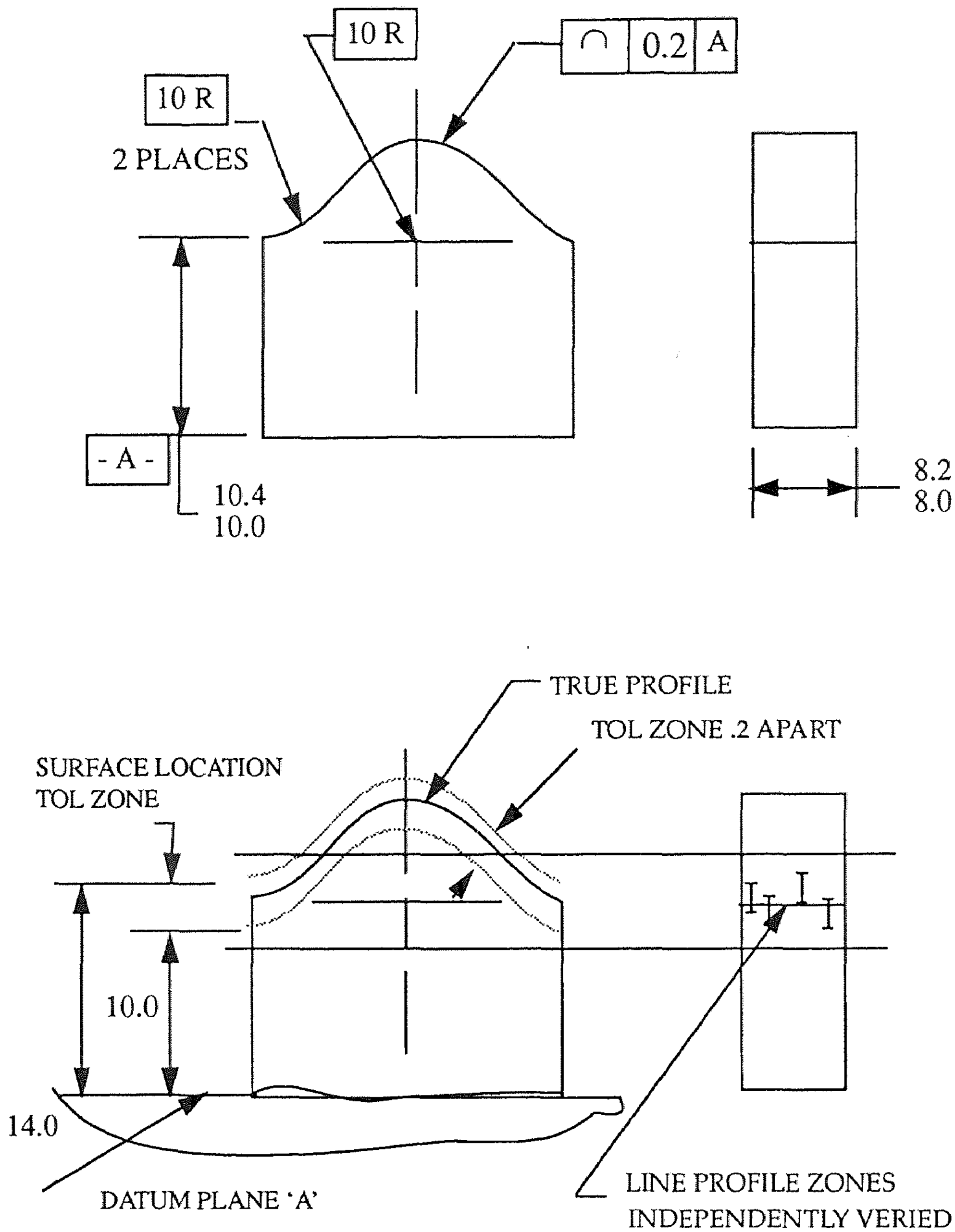


Figure 16. Profile of a Line and Size Control

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

The catagory 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 common axis of rotation. the difference is that the control surface generates the axis of rotation as in concentricity. The reason for runout is that theoritical axis do not have to be located and then there is a large cost difference in terms of manpower and achine requirements between runout and concentricity. Desired features are best controlled by the concentricity callout because it is an axis to axis measurement. It should be noted that concentricity should never be used if position and or runout symbols can be utilized for cost effectiveness.

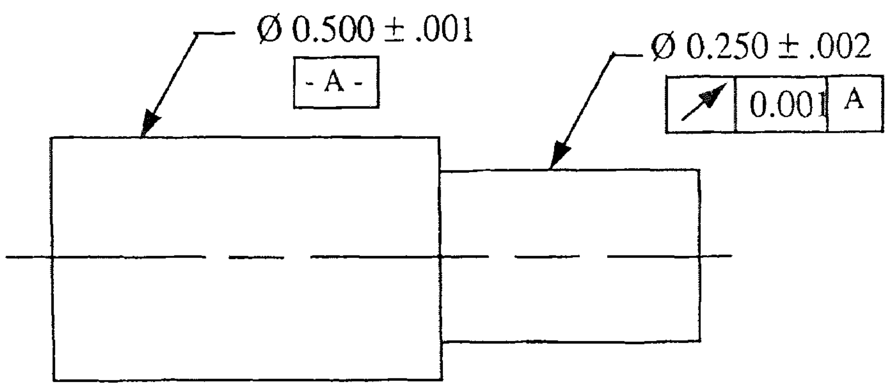


Figure 17. Circular Runout

There are two types of runout callouts : Circular runout and Total runout. As shown in Figure 17, circular runout indicates a out of round condition at a single position perpendicular to a common axis. Total runout is similar to circular runout except rather than a single position it includes the entire surface area. Figure 18 illustrates total runout callout.

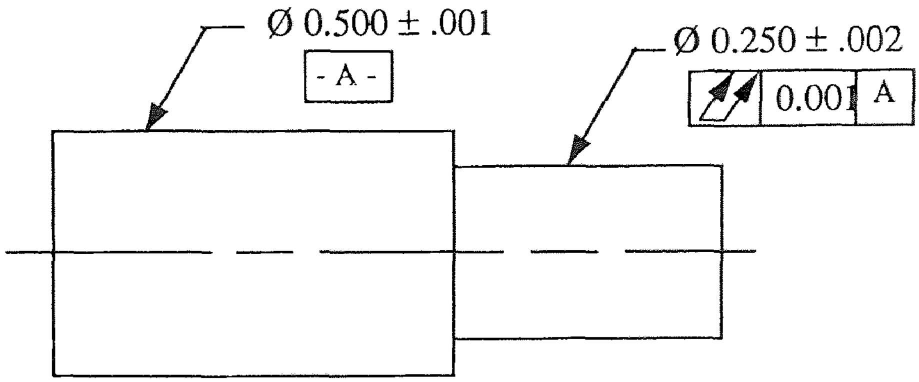


Figure 18. Total Runout

2.7 Rules

There are four important rules to understand in applying GD & T concepts, they are as follows :

(1) Limits of Size Rule : Individual feature of size where only a tolerance of size is specified, the limits of size of an individual feature prescribes 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 (Envelop Principle) - The form of an individual feature is controlled by its limits of size to the extent prescribed in particular conditions. The surface or surfaces of a feature shall not extend beyond a boundary (envelop) of perfect form at MMC. This boundary is the true geometric form represented by the figure 19. 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 that is 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 the perfect form at MMC.

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

(a) Stock such as bars, sheets, tubings, structural shapes, and other items produced to established industry or government standards that prescribe limits for straighness, 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.

(b) Parts subjected to free variation in the unrestrained condition.

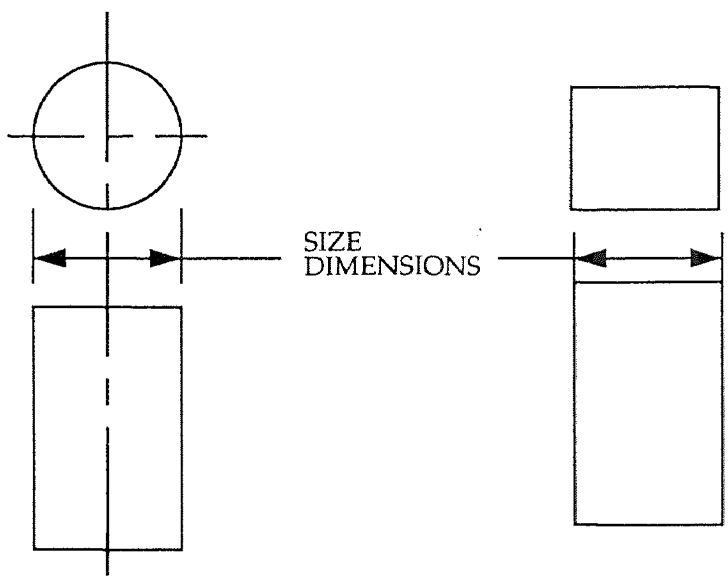


Figure 19. Individual Size Feature

(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 referenc, or both, as applicable.

Other than position tolerance rules : 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 : Eaach tolerance of orientation or position and datum ref-

erence specified for a screw thread applies to the axis of the thread derived form the pitch cylinder. Where an exception to this practice is necessary, the specific feature of the screw thread (such as MAJOR diameter or MINOR diameter). 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 teriary 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 cintrol frame at MMC.

2.8 Virtual Condition

Definition : The virtual condition of a feature is a derived size generated from the collective effect of all profile variations permitted by the specified tolerances. It represents the most extreme condition of assembly at MMC.

Depending upon its function, a feature is controlled by tolerances such as size, form, orientation, and location; MMC or RFS may also be applicable. The virtual condition of a feature is the effective size of the profile that must be considered in determining the clearence between mating parts or features and in establishing gage feature size. When a feature-of-size has no geometric tolerances apllied to it, its virtual condition is equal to its MMC plus the effect of Rule # 1. If a geometric tolerance overrides Rule# 1, then its effect must be considered in determining the virtual condition. The virtual condition concept is used by three groups :

a. Product Designers -

To calculate extreme conditions for analysing mating parts.

b. Inspectors -

To determine extreme conditions for open inspection set-up.

c. Gage designers -

To calculate gage dimensions.

Virtual conditions can be calculated as:

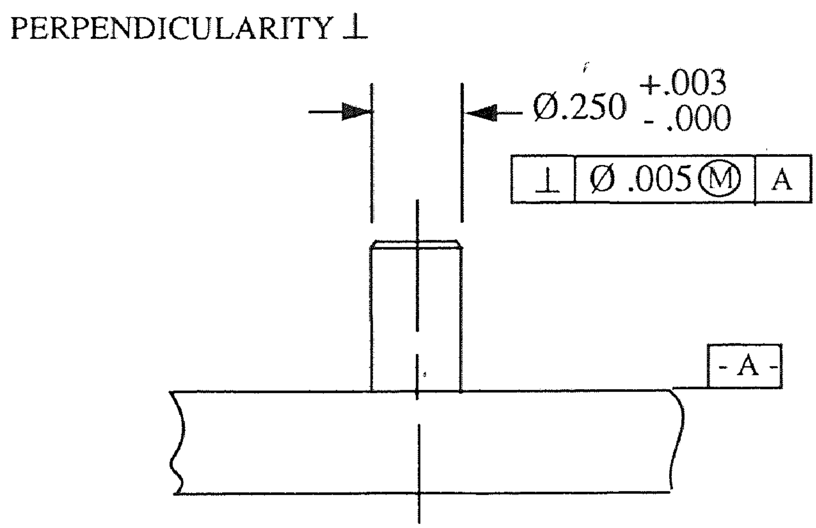
Size + orientation or position control = virtual condition (for shaft)

Size - orientation or position control = virtual condition (for hole)

This net resultant boundary is as illustrated in figure 20 :

Here a perpendicularity tolerance of 0.005 diameter is specified at MMC for a pin whose size limits are 0.250 - 0.253 diameter, the virtual condition boundary is 0.258 diameter and is perpendicular to the datum.

DRAWING :



MEANING :

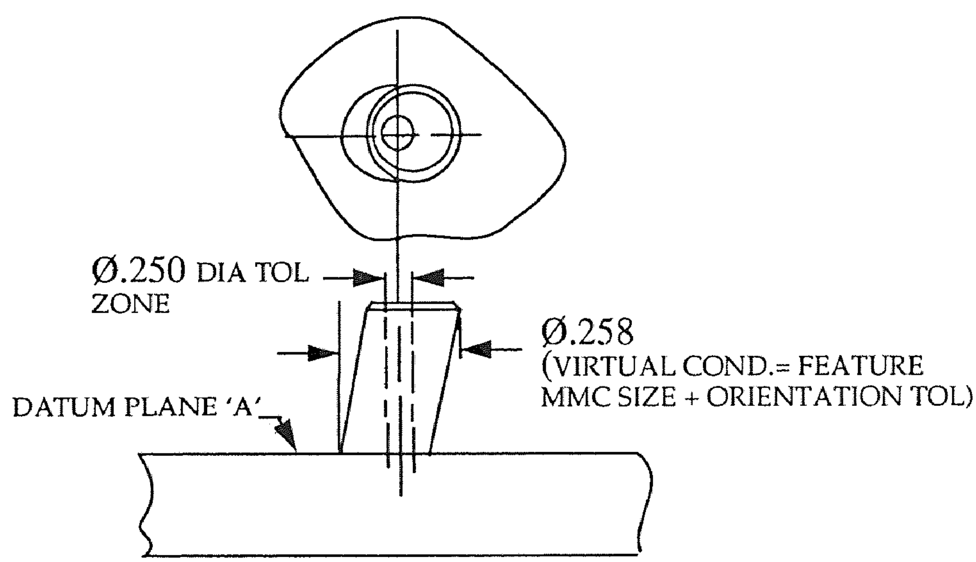


Figure 20 . Virtual Condition

2.9 Modifiers

In addition to the geometric characterstics symbols, there are five modifying symbols used in GD & T. The modifiers specify the conditions on which the other geometric characterstics are applicable. These five modifiers are described in Table 1.

Table 1: Modifiers

TERM	ABBREVIATION	SYMBOL
MAXI. MATERIAL CONDITION	MMC	Ⓜ
LEAST MATERIAL CONDITION	LMC	Ⓛ
REGARDLESS OF FEATURE SIZE	RFS	Ⓢ
PROJECTED TOLERANCE ZONE		Ⓟ
DIAMETER	DIA	Ø

Applicability of MMC, RFS and LMC is limited to features subject to variations in size. They may be datum features or other features whose axes or center planes are controlled by geometric tolerances. In such cases following practices are applied :

a. Tolerance of position (Rule# 2) :

RFS, MMC or LMC must be specified on the drawing with respect to the individual tolerance, datum references, or both as applicable.

b. All other geometric tolerances (Rule# 3) :

RFS applies with respect to individual tolerances, datum reference, or both, where no modifying symbol is specified on drawing where it is required.

The fourth modifier is called projected tolerance zone which means that the theoritical tolerance zone is projected above or below the part as indicated by the callout. The fifth modifier is called Diameter, indicating the dimension as diameter of feature on which it is applied. Each of these modifierare explained in details as following.

2.9.1 Maximum Material Condition (MMC):

Definition : MMC may be defined as the condition in which a feature of size contains the maximum amount of material within the stated limits of size such as minimum hole diameter or 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, & an edge or surface, two holes etc.). Atleast one of the related feature is to be a feature-of-size.
2. The feature to which the MMC principle is to be applied must be a feature-of-size (e.g. - a hole, slot, pin etc.) within axis or center plane.

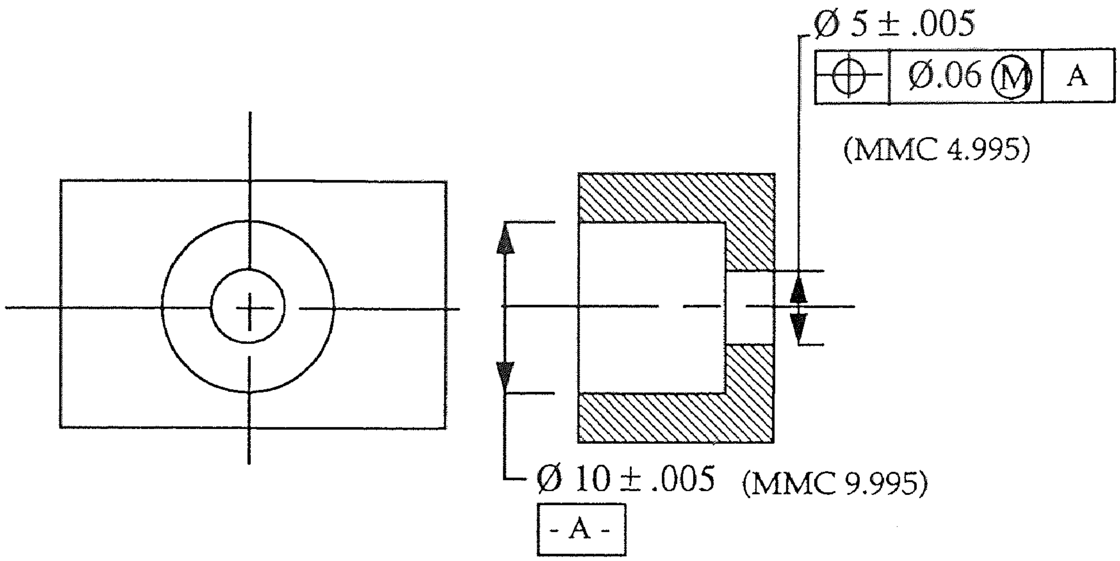
In the conventional method, the MMC condition is described as "worst condition" or "critical condition" etc. , used for the relating mating part features.

The MMC refers to the dimension of a part at which it will contain maximum material i.e. lower limit for a hole and higher limit for a shaft. The Figure 21 indicates the concept of MMC for a internal and externel dimensions.

The MMC size of 10 ± 0.005 diameter hole is 9.995, or its lower limit size. Whenever a hole is at its low size, it retains more material than if it were at its high size on larger size, which will be 10.005 in this example. Similarly, the MMC of 5 ± 0.005 diameter hole is 4.995.

In the same way, it can be seen that for the outer diameter of 9.95 ± 0.005 the higher limit is 9.955. And for 4.95 ± 0.005 dimension the higher limit is 4.955. The two parts, part1 and part2 in the Figure 20 are mating parts. Relative mating part features in this manner ensures their functional relationships. This condition also establishes the criteria for determining necessary form, orientation & positional tolerances.

PART I : (INTERNAL)



PART II : (EXTERNAL)

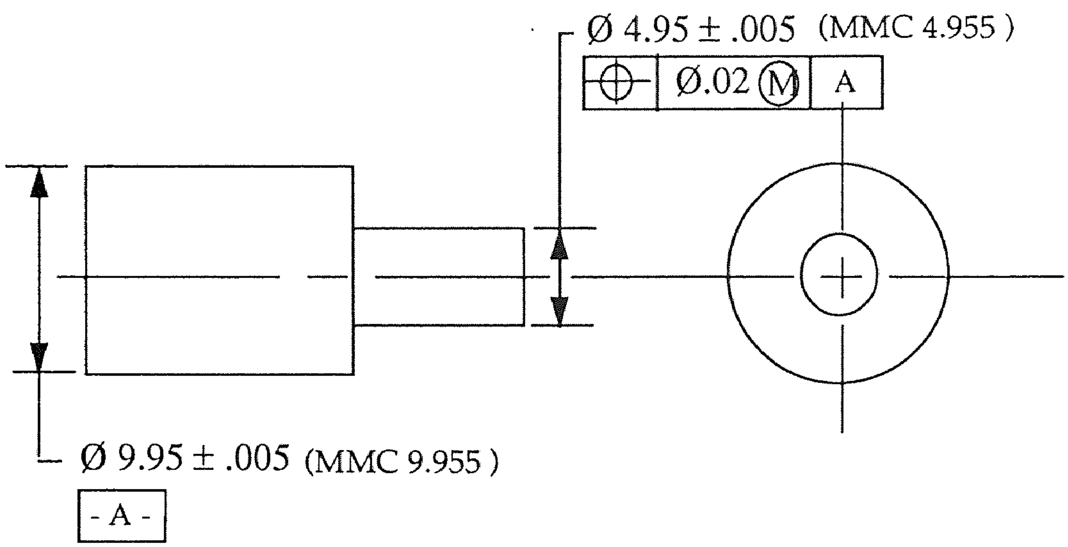
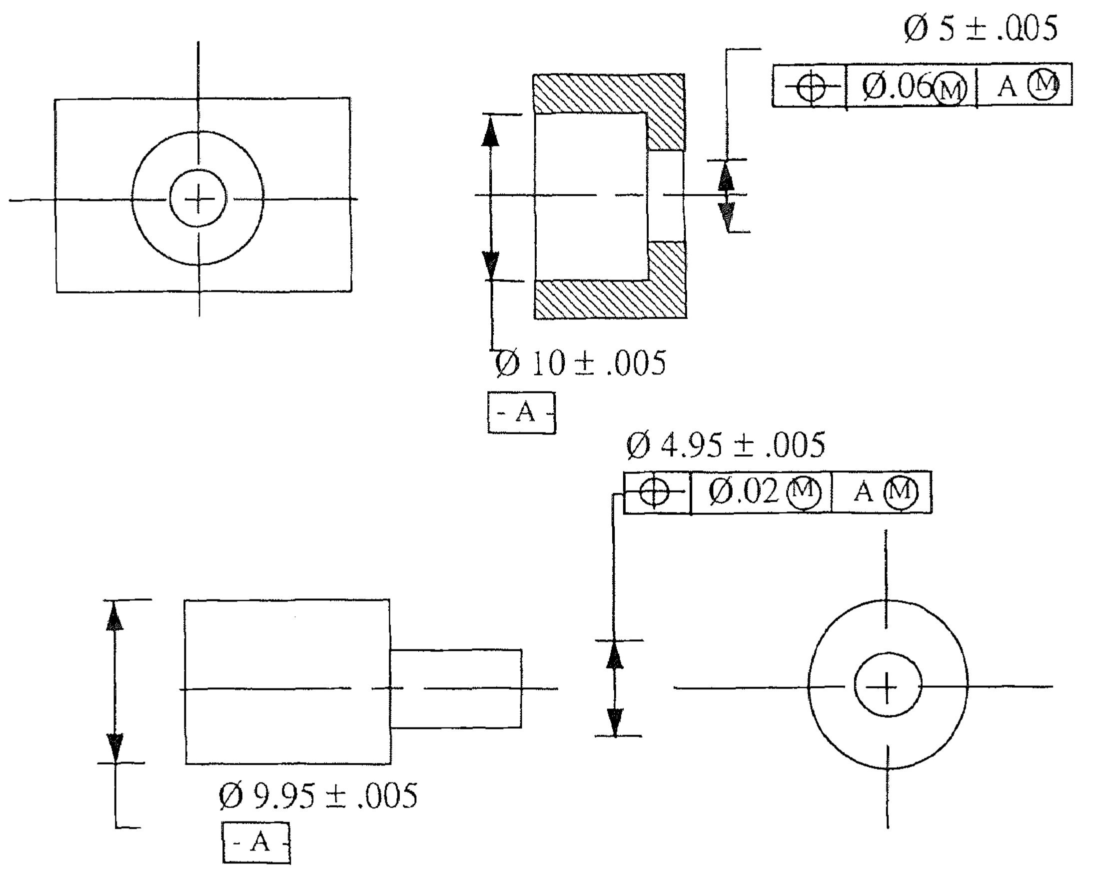


Figure 21 . Maximum Material Condition

The symbol for MMC is the 'M' enclosed in a circle, and the ocasionally used abbreviation is MMC. The symbolic method is to be used with feature control frame only. The MMC permits greater possible tolerances as part features vary from calculated MMC limits. It also ensures interchangability and permits functional gaging techniques.



POSITION TOLERANCE CALCULATIONS :

MMC SIZE HOLE (PART #1) = 4.995

MMC SIZE SHAFT (PART #2) = - 4.955

0.040

MMC SIZE DATUM HOLE (PART #1) = 9.995

MMC SIZE DATUM SHAFT (PART #2) = 9.955

EXTRA TOL FOR EACH PART 0.040

0.040 + 0.040 = 0.080

TOTAL TOL TO BE DIVIDED AS DESIRED TO ESTABLISHED REQUIRED POSITION TOL ON EACH INDIVIDUAL PART CAN BE ANY COMBINATION WHICH TOTALS TO .080 (e.g. 0.06 & 0.02)

PERMISSIBLE HOLE POSITION TOL AS HOLE SIZE DEPARTS (GETS LARGER) FROM MMC :

STATED POSITION TOL WITH HOLE AT 4.995 MMC	= .060
PLUS TOTAL 4.995 HOLE TOL	= .010
POSN TOL WITH DATUM HOLE 9.995 AT MMC	.070
PLUS TOTAL 4.995 DATUM HOLE TOL	+.010
TOTAL POSN TOL WITH BOTH HOLES AT LEAST MAT'L CONDITION (PART # 1)	.080

PERMISSIBLE HOLE POSITION TOL AS HOLE SIZE DEPARTS (GETS LARGER). FROM MMC :

STATED POSITION TOL WITH SHAFT AT 4.955 MMC	= .020
PLUS TOTAL 4.955 DIA TOL	= .010
POSN TOL WITH DATUM SHAFT 9.955 AT MMC	.030
PLUS TOTAL 4.995 DATUM SHAFT TOL	+.010
TOTAL POSN TOL WITH BOTH HOLES AT LEAST MAT'L CONDITION: (PART #2)	.040

Figure 22 Effect and Calculation of MMC

EFFECT OF MMC :

Where a geometric tolerance is applied on an MMC basis, the specified tolerance is interdependent on the size of the considered feature. The tolerance is limited to the specified value if the feature is produced at its MMC limit of size. Where actual size of the feature has departed from MMC, an increase in the tolerance is allowed equal to the amount of such departure. The total permissible variation in the specific geometric characteristic is maximum when feature is at LMC. Also, referencing a datum feature on an MMC basis means the datum is the axis or center plane of the feature at MMC limit. Where the actual size of the datum feature has departed from MMC, a deviation is allowed between its axis or center plane and the axis or center plane of the datum.

Let us consider the previous example as an application using MMC for a tolerence of position (geometric characteristic). Part1 and Part2 (Figure 21) are mating parts and the two features (diameter) of pin must fit over the corrosponding features on cylindrical internal features. The two features on each part are coaxial. For proper assembly the features on two parts must be positioned properly. Therefore, the positioning tolerances must be calculated and distributed on the two parts.

CALCULATION OF POSITIONING TOLERANCES :

Let H = Minimum diameter of clearence hole (MMC limit)

F = Maximum diameter of pin (MMC limit)

T = Positional tolerance diameter

Subscripts will be used where more than one size feature on tolerance are involved.

The two mating parts have two coaxial features where one of these features is a datum for one other (Figure 21). Where it is desired to divide the available tolerance unequally between the parts, the following formula is useful:

$$H1 + H2 = F1 + F2 + T1 + T2$$

For the informations shown in figure 22, and solving for T1 & T2, we get :

$$T1 + T2 = (H1 + H2) - (F1 + F2)$$

$$= (9.995 + 4.995) - (9.955 + 4.955)$$

$$= 0.08 \text{ Total available tolerance}$$

Normally it is easy to produce the tolerances on external features than internal dimensions, therefore larger tolerance is provided on the hole and smaller tolerance on shaft (pin), i.e. T1 = 0.06 and T2 = 0.02. However, these positional tolerances can be distributed as per the design requirements.

These tolerances can be seen in the feature control frame for the part1 and part 2. using these positional tolerances, it can be seen from the tables as shown in Table 2 that how at MMC for hole, the tolerance on diameter could be increased from 0.06 to 0.07 if hole were actually at 5.005. Similarly, Table 3 shows the availability of increased positional tolerance. The virtual condition (VC) for the MMC dimensioned features remains constant and calculated as :

VC = Actual hole size - Positional tol. (for Hole)

VC = Actual pin size - Positional tol. (for Pin)

Since VC is same throughout, any change in size of the hole/pin adjust the positional tolerances (increases) to attain the fixed virtual condition.

Table 2: Effect of Different Hole Sizes on Positional Tolerances (Part 1)

ACTUAL HOLE SIZE	POSITION TOL	VIRTUAL CONDITION
4.995 (MMC)	0.060	4.935
4.997	0.062	4.935
4.998	0.063	4.935
4.999	0.064	4.935
5.000	0.065	4.935
5.001	0.066	4.935
5.002	0.067	4.935
5.003	0.068	4.935
5.004	0.069	4.935
5.005 (LMC)	0.070	4.935

Since most parts are produced somewhere between the high or low size tolerance extremes, the actual position tolerance permissible on part1 would be somewhere between 0.002 to 0.010, say approximately 0.006. It could, however, be 0.010 & still provide proper fit to mating part. The actual positional tolerance permissible for part2 would probably be somewhere between 0.001 to 0.005, say 0.003. It could, however, be 0.005 & still provide proper fit to mating parts.

The advantage in using MMC is that we could change the size of hole to 4.997 and positional tolerance to 0.062, if zero tolerance were used at MMC. This way, a larger toleracne is permitted allowing the tolerance to increase with an increase in the diameter of hole, with no degradation of function. Zero tolerance at MMCs permits the acceptance of more usable parts over the widest possible tolerance range. The acceptance of more usable parts means more production at low cost, which is what positional tolerances tries to achieve.

Table 3: Effect of Different Shaft Sizes on Positional Tolerances (Part 2)

ACTUAL SHAFT SIZE	POSITION TOL	VIRTUAL CONDITION
4.955 (MMC)	0.020	4.975
4.953	0.022	4.975
4.952	0.023	4.975
4.951	0.024	4.975
4.950	0.025	4.975
4.949	0.026	4.975
4.948	0.027	4.975
4.947	0.028	4.975
4.946	0.029	4.975
4.945 (LMC)	0.030	4.975

GAGING WITH MMC :

The tolerance advantages and the possibility of using functional/reciever gages for effective & economic inspection often makes the position technique very derirable under such conditions. However, where the assemblability of mating part feature is involved, position tolerance is always recommonded.

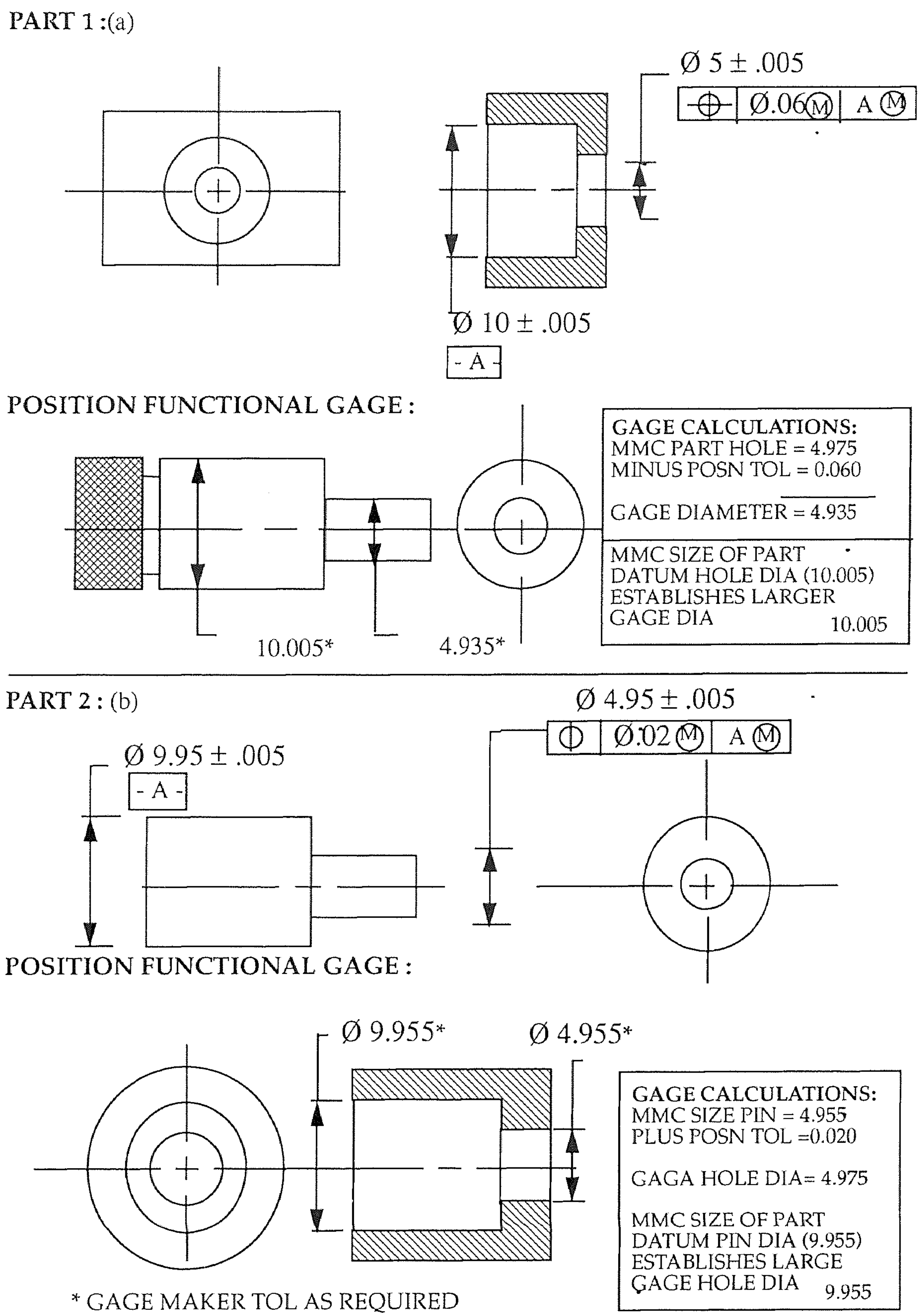


Figure 23 . Gage Dimensions and Calculations at MMC

Functional or reciever gages for coaxial features of mating parts simulate the fit of the actual part features in a manner similar to the hole pattern functional gages. As in the hole pattern functional gages, part tolerance size departure from MMC permits greater part acceptance based on the functional interrelationship of size and positional variations.

To construct a functional gage, tolerance for the gage feature location must be taken from the piece part feature location tolerance. This is commonly refered to as the 10 % rule, which means that up to 10% of part tolerance limits could be rejected by a functional gage if the part were at fringe edge of the acceptable tolerance range. The gage will not, however, ever accept a bad part.

The Figure 23 illustrates functional gage to check the position requirements of each part. Figure 23 (a) shows a gage for functionally checking the position tolerance of holes. The calculations to determine the gage dimensions are shown at right in the figure. As discussed before, gage makers tolerance can also be applied as required. Figure 23 (b) shows part2 and functional gage for checking the position of the shaft diameter. The gage dimensions can be calculated as shown in the figure.

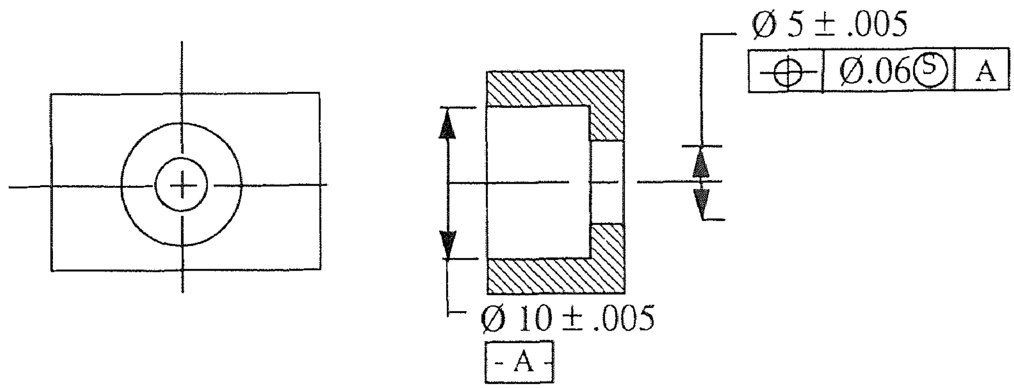
The advantage with MMC is that the virtual condition remains constant. Therefore, the corrosponding mating part (gage) can be constructed with this constant dimension and can be used for all dimensions of part within the tolerance limits. The gages discussed only checks the positional tolerance of the part. Sizes of the associated features requires a separate size checks.

2.9.2 Regardless of feature sizes (RFS) :

Definition : 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.

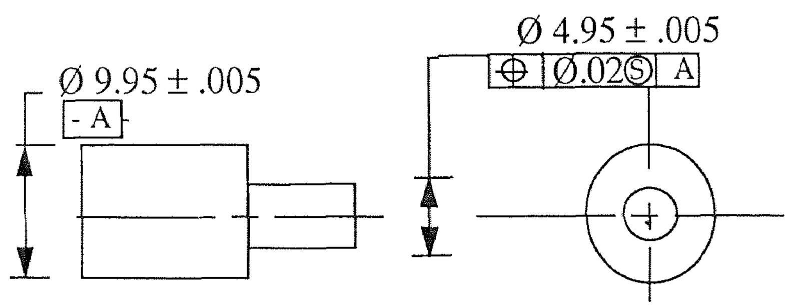
The RFS principle do not allow any additional positional, form or orientation tolerance, no matter to which size the related features are produced. It is actually the independent form of dimensioning & tolerancing which has always been used prior to the introduction of MMC principle. The symbol for RFS is 'S' enclosed in circle. this principle is valid only when applied to feature-of-sizes (for example- hole, slot, pin etc. with an axis or center plane).

PART 1 :



ACTUAL HOLE SIZE	POSITION TOL	VIRTUAL CONDITION
5.005 (LMC)	0.060	4.945
5.003	0.060	4.943
5.001	0.060	4.941
4.999	0.060	4.939
4.997	0.060	4.937
4.995 (MMC)	0.060	4.935

PART 2 :



ACTUAL HOLE SIZE	POSITION TOL	VIRTUAL CONDITION
5.005 (MMC)	0.060	4.945
5.003	0.060	4.943
5.001	0.060	4.941
4.999	0.060	4.939
4.997	0.060	4.937
4.995 (LMC)	0.060	4.935

Figure 24 . Effect of RFS and Calculation

This feature condition actually demands a very tight tolerance, hence it is not flexible for the production department to manufacture the part easily. RFS condition is quite similar to the conventional method of tolerancing i.e. unilateral/bilateral tolerancing, because the positional tolerances could not be increased or decreased as can be done with MMC or LMC.

EFFECT OF RFS :

Where a geometric tolerance is applied on RFS basic, the specified tolerance is independent of the size of the considered feature. The tolerance is limited to the specified value regardless of the actual size of the feature. Likewise, referencing a datum feature on an RFS basis means that a centering about its asis or center plane is necessary, regardless of actual size of feature.

Figure 24 illustrates the previous example but now with RFS condition on the part1 and part2. The tables indicated below each part shows how the positional tolerances are applicable at different hole (or pin) sizes. It can be seen that there is no effect of feature size on positional tolerances, therefore the virtual condition for each feature size within the tolerance zone changes.

GAGING WITH RFS :

Now let us consider, how inspection can be performed when RFS condition is given for positional tolerances. This is illustrated in figure 24 . The virtual condition has also been calculated in figure. The positional tolerance remains the same irrespective of change in size of the hole/ shaft. So, for hole if upper tolerance limit is say 4.995, the positional tolerance will be 0.060, and virtual condition will be 4.935 (4.995 - 0.060). If the feature is perfect i.e. 5.000, still the positional tolerance applicable will be 0.060, thus giving virtual condition as 4.940.

To calculate the size of the functional gage, we need to know the feature size & tolerance specified.

Let F = Feature size

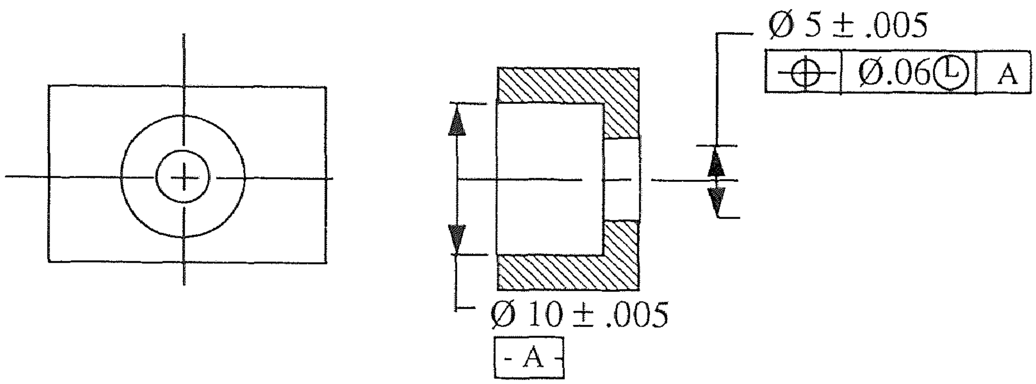
G = Gage size

T = Tolerance

then, $T = F - G$

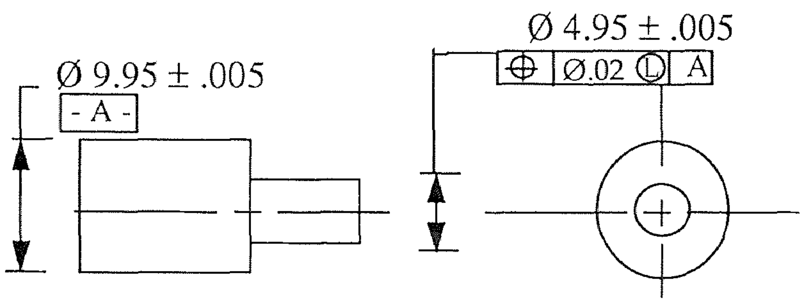
Using this equation, and from figure 24

PART 1 :



ACTUAL HOLE SIZE	POSITION TOL	VIRTUAL CONDITION
5.005 (LMC)	0.060	4.945
5.003	0.062	4.941
5.001	0.064	4.937
4.999	0.066	4.933
4.997	0.068	4.929
4.995 (MMC)	0.070	4.925

PART 2 :



ACTUALPIN SIZE	POSITION TOL	VIRTUAL CONDITION
4.955 (MMC)	0.060	5.015
4.953	0.062	5.015
4.951	0.064	5.015
4.949	0. 066	5.015
4.947	0.068	5.015
4.945 (LMC)	0.070	5.015

Figure 25 . Effect and Calculation of LMC

When $F = 4.999$ & $T = 0.060$

$G = 4.939$

When $F = 5.001$ & $T = 0.060$

$G = 4.941$

When $F = 5.003$ & $T = 0.060$

$G = 4.943$

These calculation indicates that many gages of different sizes are required to measure the feature. The number of gages depends upon the amount of the departure from high size to low size, which can be infinite. The construction of gages are costly & time consuming as they are build with precision, if at all the gages are build for measurement, otherwise this situation appears to be complicated & infeasible for economic production. Therefore, the RFS condition, is not used commonly used besides the feature measurement, it is also difficult for the production department to manufacture tight tolerances provided by this condition.

2.9.3 Least Material Condition (LMC) :

Definition: The condition in which a feature-of-size contains the least amount of material within stated limits of size, for example - maximum hole diameter and minimum shaft diameter.

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 the features for which accuracy of location can be relaxed (position tolerance increased) when the feature leaves LMC & approaches MMC. The amount of increase of positional tolerance permissible is equal to the featuresize departure from LMC.

Whenever least material condition is applied, the positional tolerance applies only when the feature is produced at LMC size. Additional tolerances are permissible but is dependent on, and equal to, the difference between the actually produced feature size (within its size tolerance) and LMC. This is shown in Figure 25.

EFFECT OF LMC :

Where a positional tolerance is applied on as LMC basis, the specified tolerance is interdependent on the size of the considered feature. The tolerance is limited to the specified value if the feature is produced at LMC limit of size. Where the actual size of the feature has departed from LMC, an increase in tolerance allowed is equal to the amount of such departure. The total permissible variation in position is maximum when the feature is at MMC. Likewise, referencing a datum feature on an LMC basis means the datum is the axis or center plane of the feature at the LMC limit. Where the actual size of the datum feature has departed from LMC, a deviation is allowed between its axis or center plane and the axis or center plane of datum.

Sometimes minimum edge distance is the criterion in the hole condition, then at that time use of LMC is most useful. This condition has specific application in aerospace industries because of the breaking strength of the metal.

GAGING WITH LMC :

Figure 25 illustrates the example when LMC is specified. This condition is used under special circumstances. As observed from the calculations as the feature departs from LMC to MMC, the positional tolerance also increases & the virtual condition increases and stays constant at all the feature sizes within the tolerance limit. It can be seen from the calculations in figure, when hole is at LMC i.e. 5.003 and tolerance specified is 0.060, the virtual condition will be 5.065.

Calculating gage dimensions from equation 1.

$G = F - T$ ....Equation 1.

When $F = 5.003$ & $T = 0.062$

$G = 4.941$

When $F = 5.001$ & $T = 0.064$

$G = 4.937$

When $F = 4.999$ & $T = 0.066$

$G = 4.933$

According to equations, the gage size or in other words the virtual condition should vary according above calculations. The virtual condition indicated in Figure 25 is true. It remains constant for all feature sizes and positional tolerance is adjusted as the actual feature dimension vary from its basic sizes.

It can be concluded from the calculations that it is difficult to calculate an appropriate functional gage that complies with the situation. Functional gaging of the hole does not work because of the variations of the axis position with respect to change in feature size. Since there are specific applications for LMC production e.g. aerospace or critical thichness on the part etc., different methods can be employed for such measurements. These methods may be paper gaging, optical comparator, and CMMs. These equipments are sophisticated and costly as compared to functional gages. Additionally, it requires skilled workers for mathematical measurements, which means the loss of time & increase in cost towards payment of high salaries.

2.9.4 Projected Tolerances :

The application of projected tolerance is in the situation where the variation in perpendicularity of treaded or press-fit holes could cause fastners such as screws, studs etc. to interfere with mating parts. An interference can occur where a positional tolerance is applied to the depth of threaded or press-fit holes, and the hole axes are inclined within allowable limits. The attitude of the fastner is restrained by the inclination of the produced hole into which it assembles. Figure 26 illustrates how the projected tolerance zone concept realistically treats the condition. The location & perpendicularity of threaded hole is important only if affects the extended portion of the engaging fastner. When design consideration require a closer control in perpendicularity of a threaded hole, than that allowed by the positional tolerance, a perpendicularity tolerance specified as a projected tolerance zone may be specified.

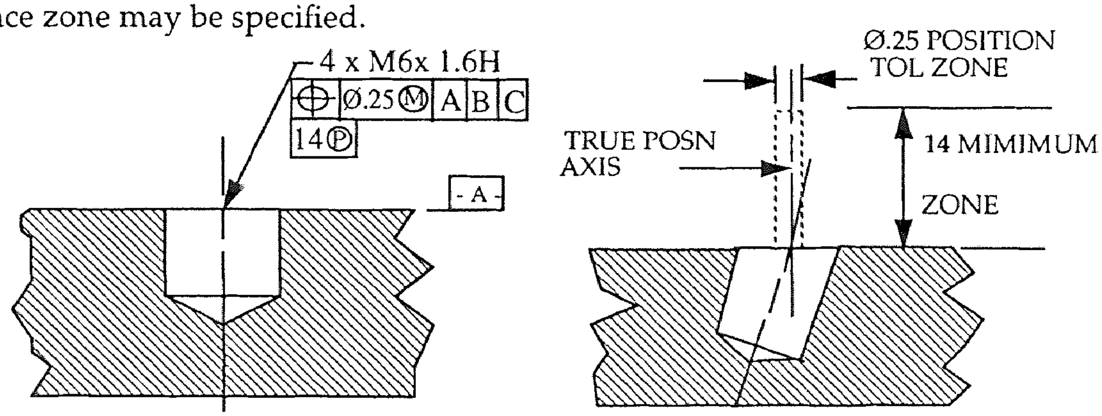


Figure 26 . Projected Tolerances

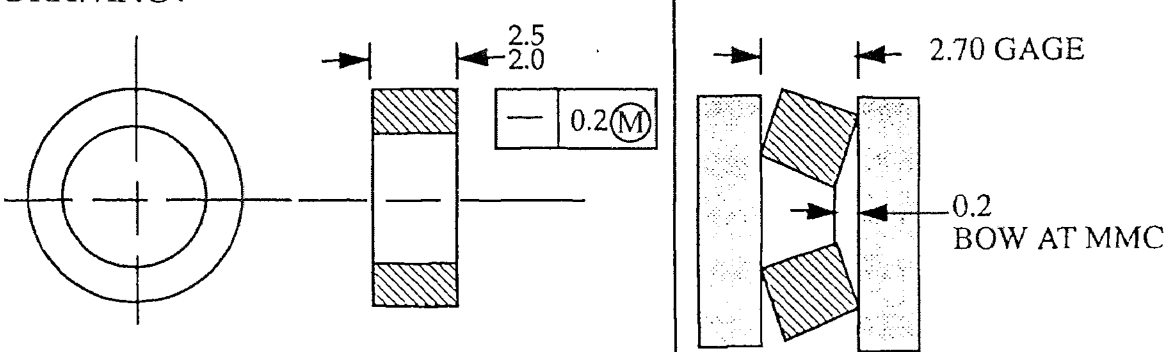
2.10 Bonus Tolerances

Whenever a geometric tolerance is applied to a feature -of-size, and it contains an MMC modifier in the tolerance portion of the feature control frame, a bonus tolerance is possible. When the MMC modifier is used in this fashion, it means that the stated tolerance applies when the feature-of-size is at its maximum material condition. When actual feature-of-size departs from MMC, an increase in the stated tolerance, equal to the amount of departure, is permitted. This increase or extra tolerance is called as bonus tolerance. Figure 27 indicates how the bonus tolerances can be calculated for a straightness application.

The bonus tolerances can also be considered when Rule# 1 is applied to feature-of-size. Rule# 1 states "perfect form at MMC" but when the feature-of-size departs from MMC, a form error equal to the amount of the departure, is permissible.

DRAWING :

INTERPRETATION :



BONUS TOLERANCE CALCULATION :

ACTUALPIN SIZE	POSITION TOL	BONUS TOL	VIRTUAL CONDITION
2.5 (MMC)	0.2	0	0.2
2.4	0.2	0.1	0.3
2.3	0.2	0.2	0.4
2.2	0.2	0.3	0.5
2.1	0.2	0.4	0.6
2.0 (LMC)	0.2	0.5	0.7

Figure 27. Bonus Tolerances

CHAPTER 3

CONCURRENT ENGINEERING

3.1 Chapter synopsis

Concurrent Engineering has become one of the hottest topics in engineering today. Perhaps no other concepts in the past decade has so captured the imagination of design engineers as concurrent engineering (CE). The potential benefits of CE are by now well known namely faster cycle time, better products and more responsive organization. Although the principles that underlie concurrent engineering are conceptually simple, they can be deceptively difficult to implement effectively. They involve a well planned mixture of cutting-edge technology combined with enlightened personnel and resource management policies that cut across departmental lines.

The product life cycle is an important issue in CE. Designers give very little consideration to these issues like product assembly, test, repair, and modification. This is true even though designers are increasingly aware of the need of design product parts so that they can be fabricated economically and still meet performance requirements. While 'Design for Manufacturability' has increased productivity, it is not an integrated approach and thus miss most of the opportunities for productivity enhancement.

This chapter describe the concepts of Concurrent Engineering, its objective, its implementation and complexities during implementation. Also the conceptual design process is with concurrent design (CD) indicating the corrosponding advantages of CD.

3.2 Philosophy of Concurrent Engineering

As the name suggest, Concurrent Engineering means doing the things together or simulteneously. Although this statement can be vaguely interpretated as performing various activities at one time by all areas. Beyond that Concurrent Engineering is an integrated approach of performing various functions of an organization. Concurrent Engineering can mean many things. Normally defined, it is the merging or overlapping of hardware and software design tasks. A broader perspective on CE might more appropriately be called Concurrent product development because it merges all product development disciplines. Among these desciplines are marketing, hardware & software design, purchasing, test, manufacturing, and service.

The philosophy of CE is to reorient product development. Rather than performing past design reviews, testing to correct problems, and constantly reengineering the problems, Concurrent engineering strives to prevent problems and build on existing designs.

Concurent Engineering can be defined in many ways. A formal definition can be given as " Concurrent engineering is designing of a product and process to manufacture that at the same time."

The definition of CE given by two faculty members of Georgia Institute of Technology (Deniel P. Scharge and J. Edward Rogan) is as under :

" Concurrent Engineering is defined as a systematic approach to the integrated, concurrent design of products and their related processes, including manufacturing and support."

Both of these definition place emphasis on integration of all departments involved in the process of product development such as engineering, manufacturing, and marketing in order that they work together during early stages of product development, i.e. conceptual design phase. It is in this phase that the design is fluid, hardware is still remote, few number of people are involved and constraints & restrictions have not become tight. It is this stage that the design team has most latitude to explore alternate possibilities and therefore the greatest opportunities to identify the inherently best product and process to manufacture, thus saving time and money.

A typical manufacturing company occupies a place in long chain of suppliers, fabricators, producers, transporters, wholesalers, and retailers. Raw materials are converted into standard stocks and subsequently to standard components. Materials and standard parts are made into products that are sold to wholesalers & retailers. The management of company seeks to operate within this chain by determining market demands, designing products, scheduling production, pre dicting sales, and deciding upon material orders. The uncertainties surrounding the material availability & price, market demand and the price consumer willing to pay, makes it is difficult to take the decisions. Figure 28 isolates the operating decisions into groups comprising material, production, finished goods, and distribution control. But whatever approach management takes towards operating system, there should always be responding action of marketing, finance, product design and manufacturing departments.

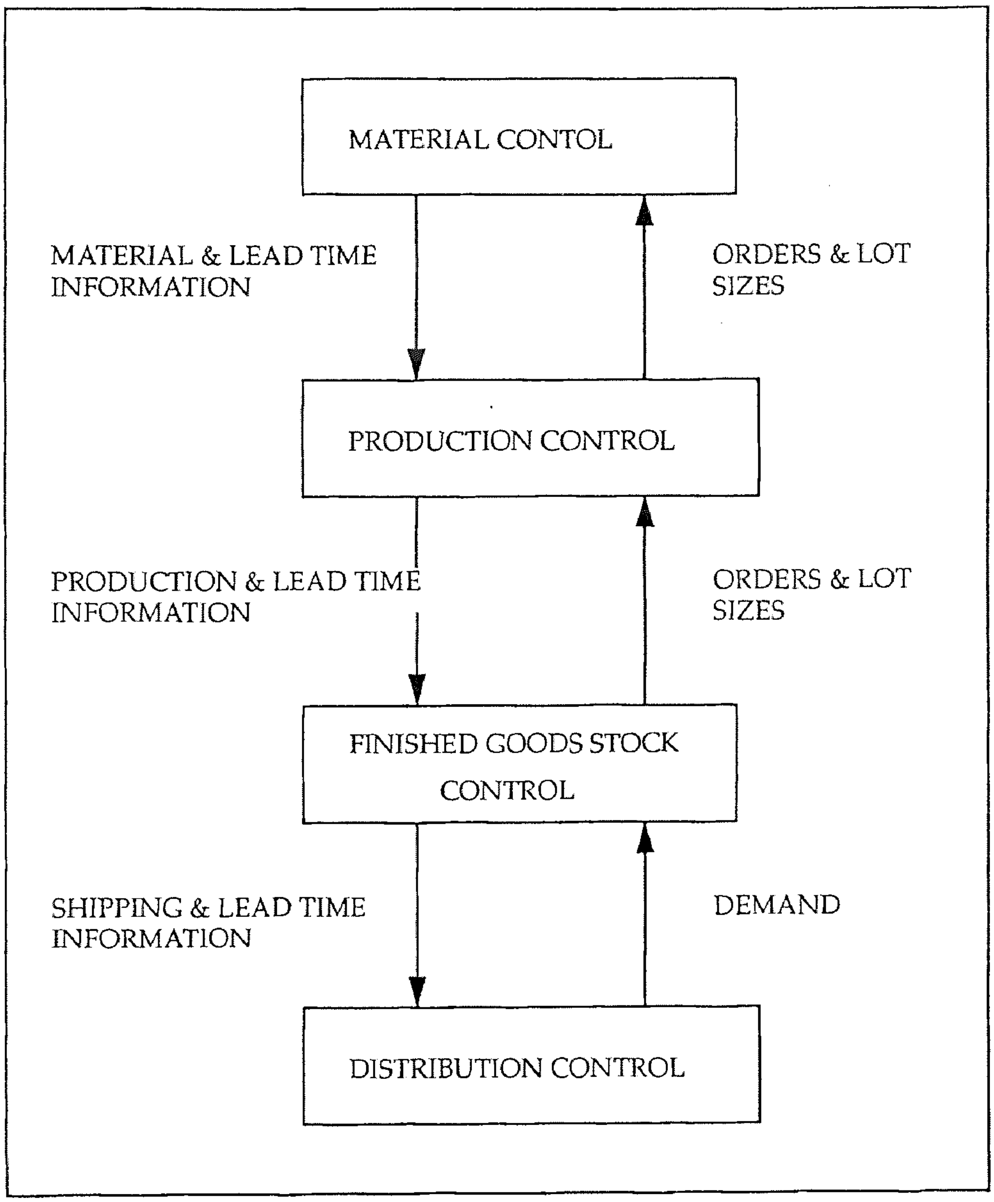


Figure 28 . A Manufacturing Company's Operational Decisions and Their Relationship

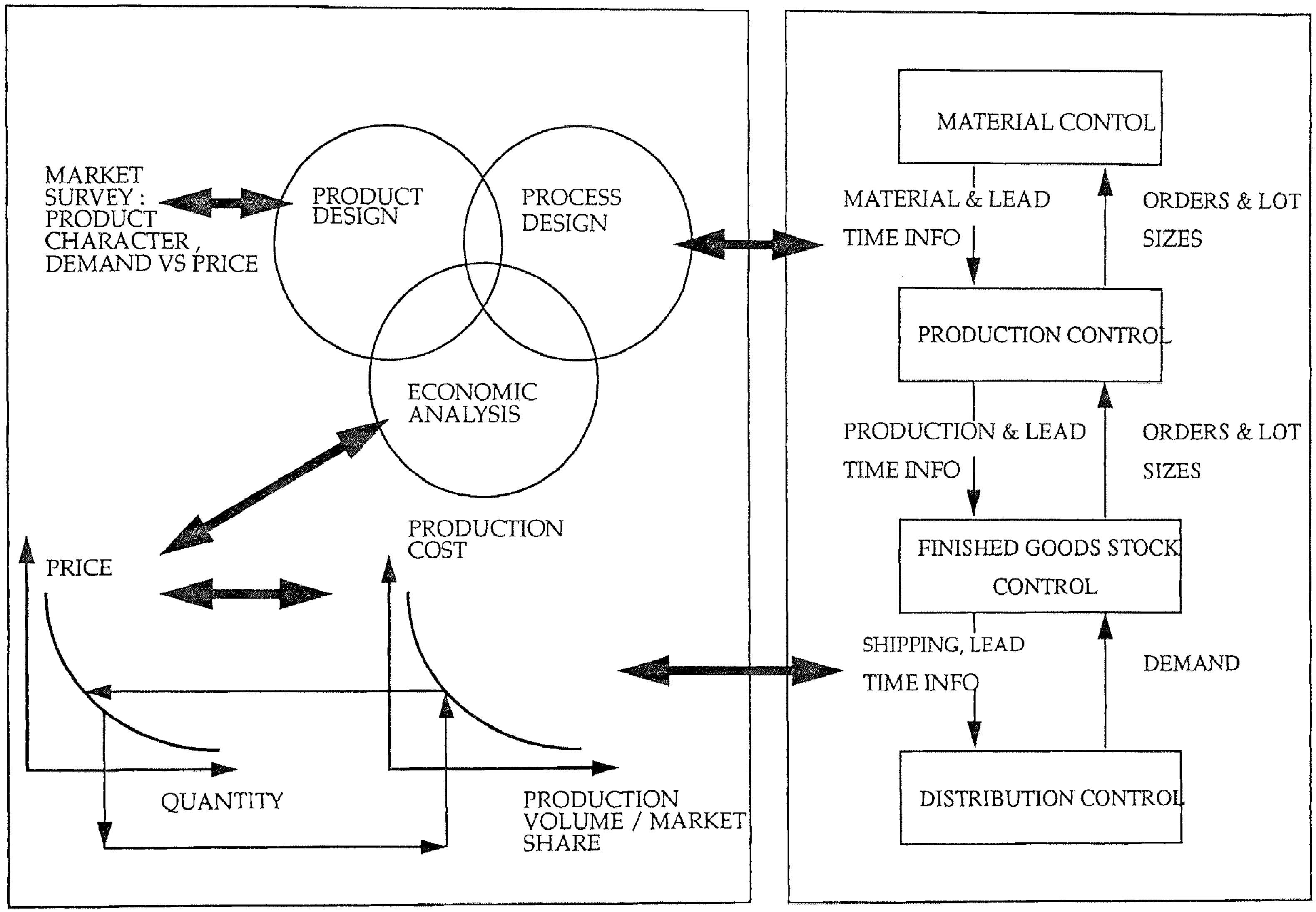


Figure 29 . Integration of Different Functions of an Organization

The problem faced by manufacturing are many like the challenges of competitiveness, the need for better quality, more flexibility and efficiency, and difficulty attaining thesewith manual labor in complex products. Also these problems are long standing. Therefore, an effective mix of technological and institutional advances are required in product development stages so that the above characterstics can be attained. Figure 29 indicates how different functions of an organization can be welded into the coherent team that makes the decisions simultaneously.

The job of a marketing department is to determine the characteristics and features of a potential product and how big its market might be at a certain price. Using a cost model of production cost and possible market share, the marketing department estimates whether a product might be made in the right quantity for the right price.

The product design department seeks to design a product that will meet the market's demand or stimulate a market. Based on the findings of market surveys or stimulated by product research and development, the design department seeks to design a product with a target price that will meet the market's needs or create a new market.

The manufacturing department must be able to make the product as designed and as it is planned to be sold. A manufacturing process must designed to create the product as designed. This process must comprise a strategy for meeting the anticipated market method, including what models will be made, how production will grow and so on.

The finance department must determine if the manufacturing department's plans make sense economically. The overall estimated investment be compared to the market and prospective revenue from selling the product. Also the investment must be compared against alternate investments to determine if the return is adequate.

An integrated approach to manufacturing comprises five interrelated elements. They are :

a. Careful analysis and understanding of fabrication and assembly processes to permit their operation with consistency and quality.

b. Strategic product design, conceived to support a specific strategy for making and selling the product.

c. Rationalized manufacturing system design coordinated with product design.

d. Economic analysis of design and manufacturing alternatives to permit rational choices.

e. Product and system designs that are characterized by robustness and structure.

3.3 Product Life Cycle and C E

Marketplace has become highly competitive. The customer's demand is varied and they need fast introduction of new featured products and at economic prices. In order to be able to compete in global market, the organizations are required to respond quickly to the market by developing new or improved products that fulfill the customer's demand. This has necessitated the companies to reduce their product development cycle time by avoiding recurring and nonproductive activities during the developmental period. It is imperative to replace the 'redo it until it's right' philosophy to 'do it right first time' philosophy with an objective of saving the time and money.

Figure 30 shows how cycle cost of product is determined during various phases of design. Life cycle cost includes cost of materials, manufacture, use, repair, and disposal of a product. The curve does not show how much money is spent in each phase toward creating the product. Rather, it shows how much influence of each phase on final cost . Thus, concept formulation determines about 60 % of the cost, and all activities up to start of full scale development determines 75 %. This means the design decisions made early in the process determine most of the cost, and later decisions make only minor changes to the ultimate total. [4]

Concurrent engineering helps in making early design decisions that will minimize costs over the life of the product. For example, designing a product to fit into an existing manufacturing process rather than requiring a new process (& new capital equipments), can have big impact on cost. While making early design decisions, the manufacturing can be included to propose this cost effective suggestions, whereas alone the designer may not take the extra cost of buying new capital equipment into consideration. Taking some extra design time to ensure error free assembly by using a minimal number of assembly operations can also significantly lower overall product costs.

Figure 31 shows a comparision of times taken in product development for commonly used development technique - Sequential engineering and Concurrent engineering. It can be derived that around 60 - 85 % of overall product cost is determined during the design phase. Product parts assembly, test, & service costs are dictated for more often by the design then by actual manufacturing, testing & service. Also not only the redesign and reverification costs using CE are eliminated, but a considerable amount of time is saved in design verification, test generation, and test because of the efficiency early in the design phase. The saving in time to market typically amount to between 10 and 25 % and results in better product. [1]

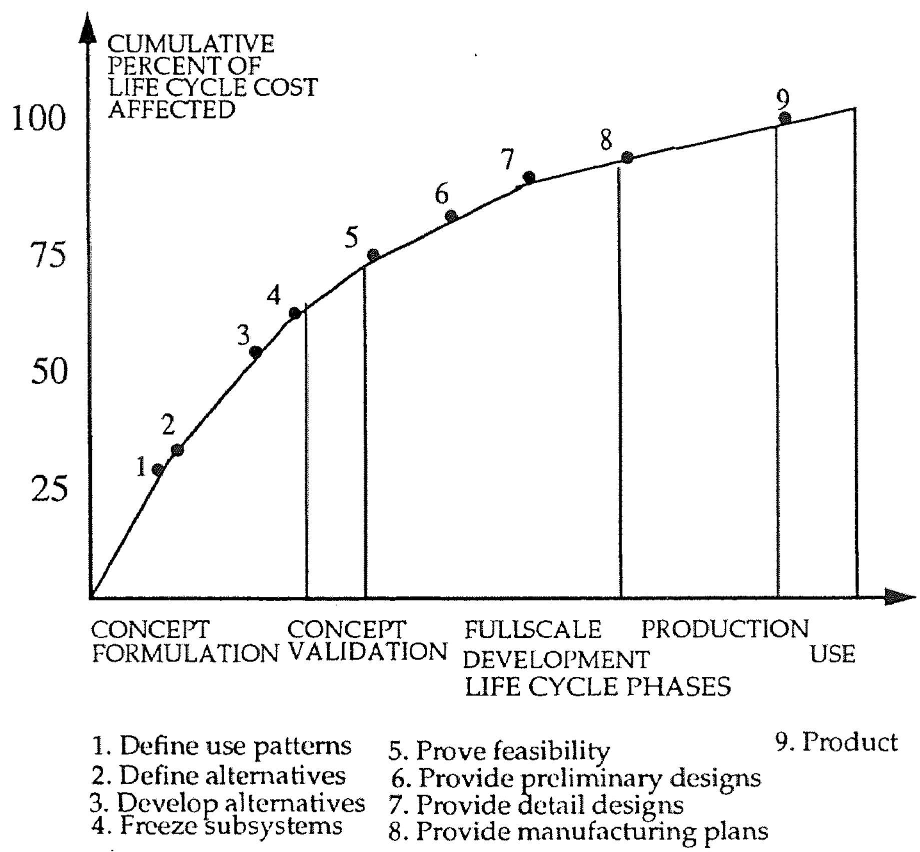


Figure 30 . Product Life Cycle Phases v/s Life Cycle Cost

3.4 Objectives of Concurrent Engineering

1. Improve communication between team members i.e. different departments & thus working on same standards.
2. To make the right decision during nonrecurring activities eliminating potential problems early on.
3. To remove redesigning and reverification at different stages of product development to get the product right at first time.
4. Reduce the time lag between product development and marketing & sales.
5. To satisfy the customer's varied demand at right time.
6. To get competitive edge over competitors by minimizing development and product costs , increasing quality and hence augmenting market share.

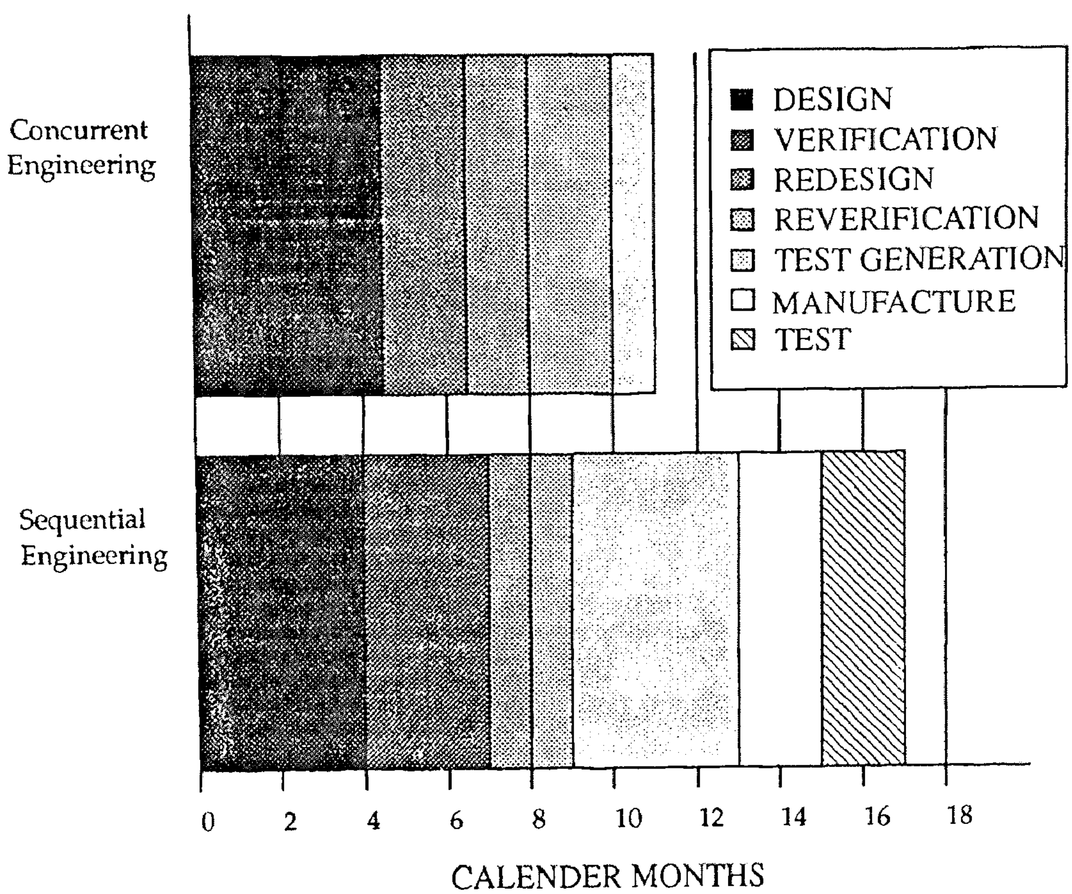


Figure 31 .Product Development Life Comparision

3.5 Traditional Product Development Trend

The Industrial Revolution introduced mass production with standardization and reliance on machine tools that were operated by a widely available, but relatively unskilled labor force. Lately, increasingly complex and yet inexpensive products have resulted from the use of practices instituted at that time. But it is now well recognized that a number of existing practices inimical to today's manufacturing environment of high product mix, but low production volume, find their roots in the very methods that made the Industrial Revolution successful.

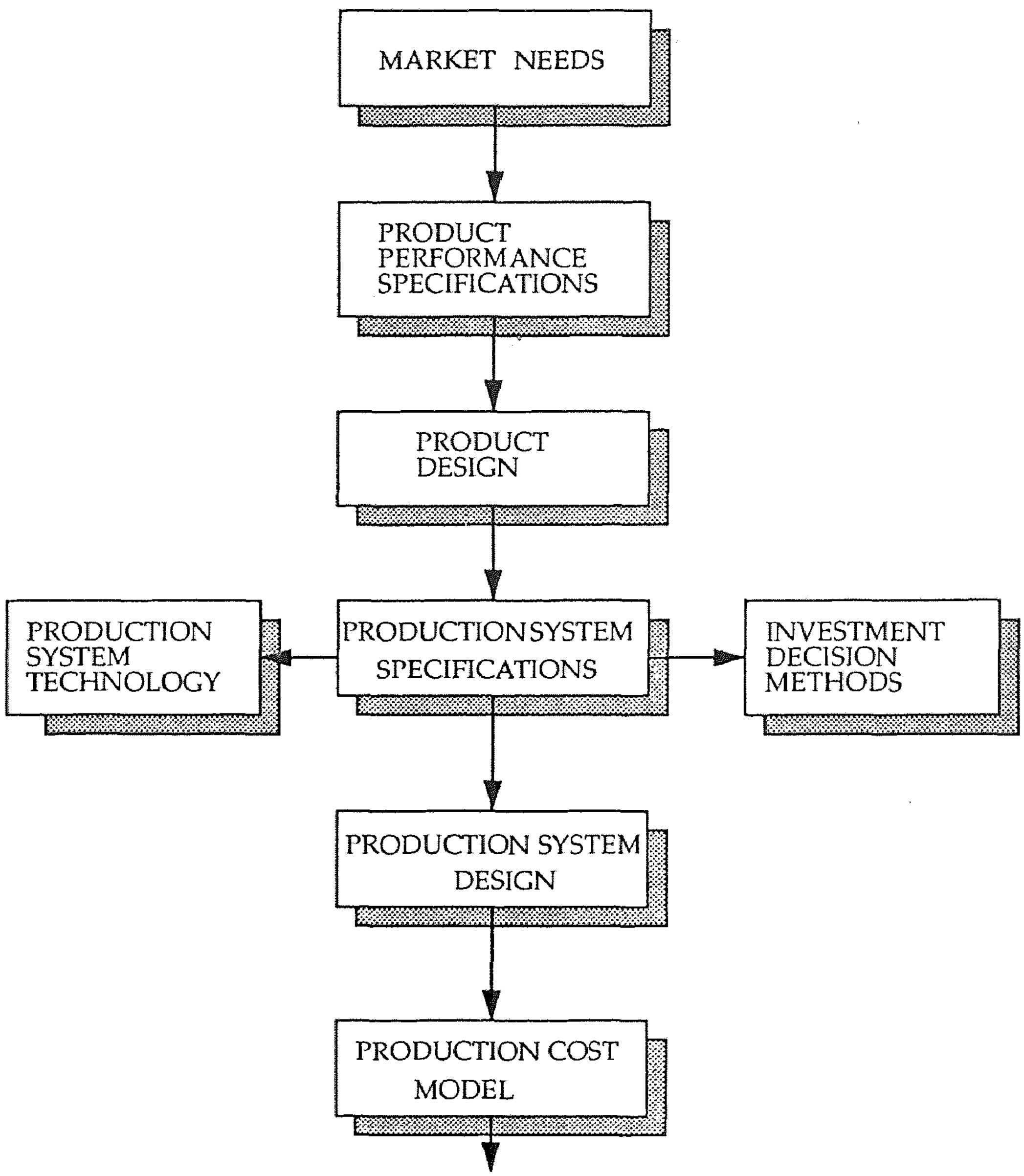


Figure 32 .The Conventional Product -Process Design Method

Figure 32 indicates the conceptual product-process development, the majority of companies are organized along functional lines typically employ greater degree of Sequential Engineering. Engineers are given a technically oriented view which emphasizes determining the need, preparing product specifications, and making trial designs, then prototype for bench tests, then final design, and then manufacturing process plan. These departments can have feedbacks and the problems are uncovered and resolved. But in the main, the process is self-contained from need to final design, with little outside interference. This unidirectional approach to product development is limited by the ability of each group to finish its portion of design. As each functional group finishes its task, it hands the project to the next group in the sequence. This approach is often called 'over-the-wall' engineering.

The greatest disadvantage of Sequential Engineering is its too much linearity. The functions are divided into steps and sequences. It is too compartmentalized, assuming that the design, manufacturing, purchase manager, marketer etc. works in their specified domains. The information flow is unidirectional. Although downstream development departments might contribute useful informations to the design team, no formal mechanism for that data flow exists. The most disadvantages can be independent schedule pressures on each of the functional teams act to inhibit any sponteneous bidirectional information flows. There pressures may lead to change, modify or compromise with project goals such as unquantified ability to test, manufacture, and service of product.

3.6 Integrated or Concurrent Approach to Product Process Development

How the above problems can be avoided is the question? The alternative to Sequential engineering can be Concurrent approach of product development where a multidisciplinary product team can be employed instead of functionally organized teams. These teams consists of representative from the disciplines of engineering and corporate : manufacturing, services, product management, sales, and finance. These teams provides a formal means of bidirectional communication between upstream and downstream operations. These teams can also include representatives from outside the organization. Suppliers representative in the team can often provide an advance look at enhances or superior products in their pipelines that might make the product better. Involving customers at design stage may provide timely market research and feedback on useful or useless features. These outsiders improves its likelihood for success and generally ensures applications

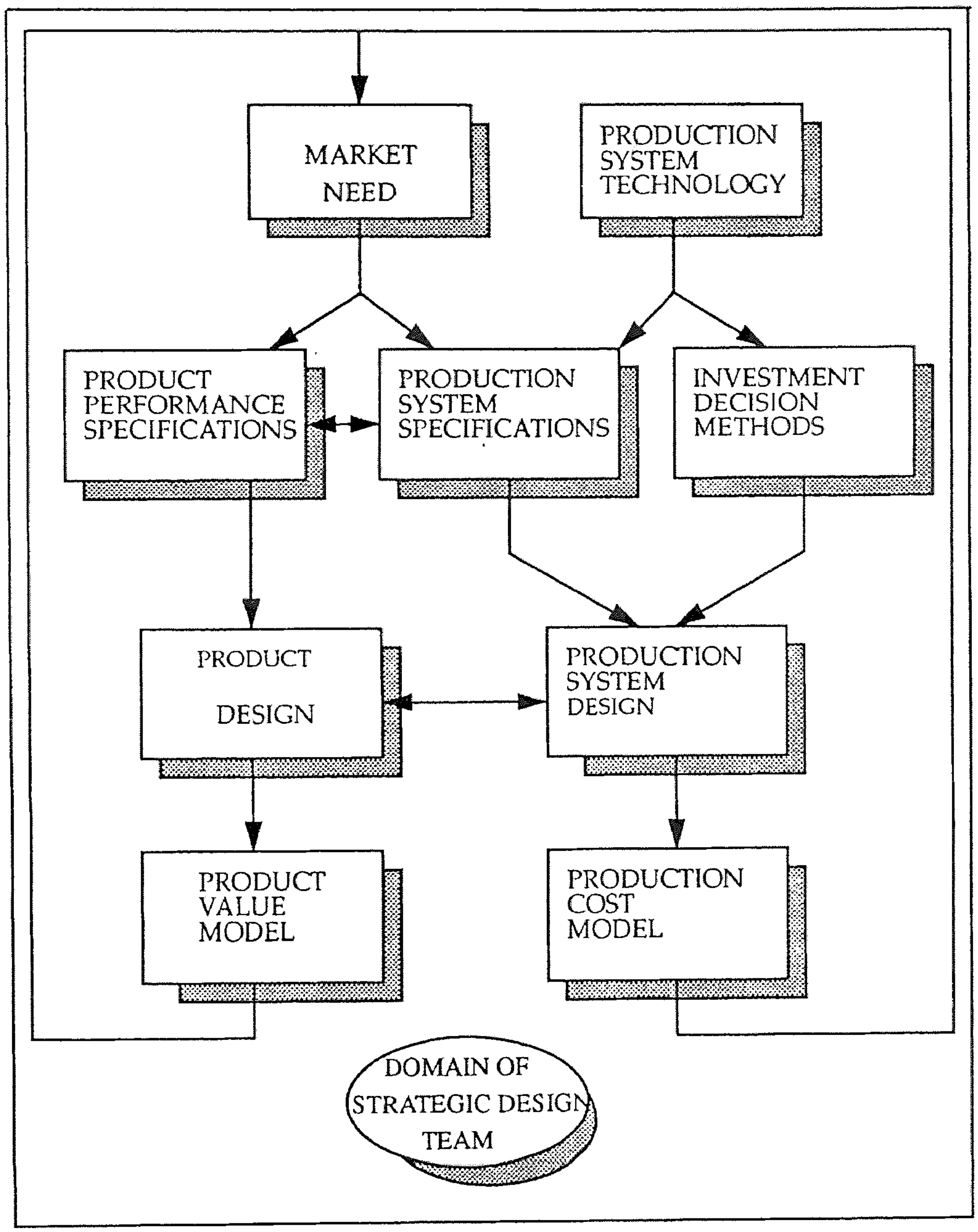


Figure 33 . The Strategic Method of Product and Production System Design

MANAGEMENT	DESIGN ENGINEERING		MANUFACTURING ENGINEERING		QUALITY CONTROL
DETERMINE MARKET NEED :	PRELIMINARY DESIGN	DETAIL DESIGN	MANUFACTURING PLAN	ASSEMBLY PLAN	TEST & INSPECTION PLAN
DESCRIPTION PERFORMANCE COST QUALITY DESIGN SCHEDULE PRODUCTION SCHEDULE	TECHNIQUE PHYSICAL PRINCIPLES AND LAYOUT ENERGY STORAGE AND TRANSMISSION LOAD BEARING SIZES, DIAMETERS , LENGTHS SPEEDS CRITICAL MATERIALS MAJOR COMPONENTS VOLTAGES, PRESSURES, FLOWS	LAYOUT DETAILS TOLERANCES PITCHES DRAFTS CHAMFERS MATERIALS FINISHES FASTENERS SEALS CIRCUIT COMPONENTS	MAKE / BUY MFR EQUIPMENT VENDOR CHOICES MFR. LOCATION SHIFTS LABOR, TRAINING	IN-HOUSE OR OUT ASSY METHODS SHIFTS LABOR TRAINING	WHAT TO CHECK HOW TO CHECK WHEN TO CHECK

GENERAL PRECEDENCE OF DESIGN DECISIONS →

Figure 34 . Precedence of Decisions in Product -Process Design

The methods being used by competitive companies vary according to their traditions and experiences, but they share attributes as per the Figure 33. This diagram emphasizes the degree to which decisions made by different parties affect each other's activity and alter the product's design. A single engineer cannot have knowledge needed to carry out such a comprehensive activity . Also there is no superintelligent computer as such which can design products & manufacturing process. The teams of specialists can contribute their knowledge to create superior products & manufacturing systems. The process of designing do have the iterations, but the iterations between design & production or between production and marketing can be of equal importance. Perhaps this is the cause of traditional time lag between product design and manufacturing system design. The essence of a sophisticated design can depend upon careful choice of tolerances, materials, or new fabrication methods that cannot be seperated from the design of the manufacturing process and therefore it is never too early to begin the process before having engineering prototype in the simulteneous engineering environment.

3.7 Practical Limitation to Concurrent Product Process Design

Although the advantages of Concurrent design are many, it can be difficult to implement because it is difficult to convince people to carry out through. The constraints can be listed as below :

1. One of the engineering limitations include the precedence of decisions. Figure 34 shows this process along with certain precedences that must be observed. Some functions cannot begin until others have been considered. The purpose of CD is not to ignore or obviate these precedences but rather to organize the flow of debate & decisions so that the impact of decisions, especially the influential early ones, is understood by all constituencies.
2. Difficulty in coordinating the activities of the many people involved in the design of complex products. Even if the people in the various departments have similar technical background, they are likely to think & work differently. Thus making the group work more challenging.
3. In organizing multidisciplinary teams, communication can be difficult unless appropriate infrastructure is provided. A procedure or tools can be developed

which will give unique interpretation and understanding at different levels.

4. Ingrained habits and training of engineers and managers and lack of experience in participating in or managing teams. Many people are hesitant in sharing the information with the others thus making the process sluggish and inefficient.

3.8 Tools for Concurrent Engineering

For the effective implementation of Concurrent Engineering, some procedure or tools should be defined that will make this new corporate philosophy to work. Some of these tools can be listed as follows :

1. Design Standardization :

By standardizating the design, a considerable amount of redesigning and reverification time can be saved. The design engineer can frequently search the existing designs and reuse them rather than redesigning the same part. This will also help in interchangability of parts and thus facilitating assembly. The use of computerized database can be helpful in storing and easy retrieval of the standard designs.

2. Taguchi Methods :

These methods are based on mathematical analysis and is intended to aid the designer in creating a product that can be produced within economical tolerances on economical equipments and still function as desired. Taguchi identifies three elements in product design: System design, Parameter design and Tolerance design.

System design means concept design of a product and engineering analysis to determine that the product will function. It also includes the separation of the factors into controllable and uncontrolable ones, or "noise" and then taking suitable actions to eliminate or minimize the noises.

Parameter design is an attempt to deal with unpredictable factors. It uses the statistical methods to identify the probable noise factors and controllable factors associated with the product's function.

3. Quality Function Deployment :

Quality function deployment is a tool for translating customers requirements into the appropriate technical requirements for each of the stages of product development. A marketer can add the customer's attributes in House of Quality to convey the needs to designer to enable him in incorporating these attributes in the design.

4. Process Control Planning :

This method enables the documentation of all the informations regarding the process in a single document for analysis and necessary modification of the processs.

5. Simulation and Analysis :

This relatively new technique permits on-screen "testing" of a system before significant time, capital, and expenses are incurred. The tool provides a means of determining whether or not the system will work and or what is required to make it efficiently.

6. Networks and Data Communication :

Proper communication is vital for successful implementation of Concurrent Engineering project. The flow of information to related departments is important so that further action can be performed efficiently and quickly. The local area networks can make the same information available almost instanteneous and clearly to a large number of users. This allows the rapid transmission of design informations to purchase, sales, inventory control departments etc. Thus facilitating Concurrent Engineering approach.

7. Group Technology :

It is an approach developed to improve the effectiveness of producing similar parts in small batches. A classification and coding scheme is used to identify the similar parts that are already designed and manufactured using similar machine tools and procedures in past. This saves the redesigning of parts and processes to prodece them. Thus saving product development cycle time and cost which are the prime factors in Concurrent Engineering.

8. Value Engineering :

It is defined as the systematic application of recognized techniques that identify the function of the product or service, establish its value and provide the service at the lowest possible cost without diminishing performance or quality. It is the process in which the product design and function are examined critically so as to decide for economic choices of materials, design features and manufacturing processes.

9. Solid Modelling :

A solid model provides a complete geometric representation of a part or an assembly. It includes more information about the part than either a two-dimentional drawing or a 3D wireframe model. It can give a better idea to the representatives from other departments about the concepts included in the design.

10. Design for Manufacturability / Assembly :

This is the tool or process which can optimize the relationship between the materials, technologies discussed, the manufacturing processes, and the costs involved in the design stages.

11. Synchronous Manufacturing :

It is a manufacturing management technique which focuses on the systematic acceleration of materials flow through a production operation. It is also an analytical technique that provides a framework for the systematic application of quality and productivity programs.

12. Project Management :

The project management environment provides the flexibility to assemble the personnel with require expertise to meet specific project requirements. Specific functions in the group can be assigned to the experts of specific field, all working under a leader or evaluation committe in order to have an effective & smooth incorporation of concurrent engineering concepts.

3.9 Advantages and Economy of Concurrent Engineering

If implemented properly, the benefits from CE can be tremendous. Time and money are two important factors that every organization is scarce of and always tries to save or use them in an efficient manner. These two factors can be the summarized advantages by using the concurrent engineering concepts.

Even though, the implementation of CE requires a considerable amount of initial investment to set up the concurrent environment e.g. training, coordinated efforts, computer systems etc., in the long run these costs can be justified by bundle of benefits that CE would provide to the organization.

The following advantages can be obtained by the use of concurrent engineeing:

1. Customer satisfaction :

Marketing point of view, customer satisfaction can be the biggest advantage that an organization can achieve. With the interaction of marketing and sales departments which are in the direct contact of customers, the needs can be directly translated to the designer. This way the designer can incorporate desired features in his design and satisfying customers.

2. Fast time to market :

The overall time for product development cycle is rduced by utilizing the CE concepts, thereby reducing the time it takes to place a new product in the market at economic price. This can give a competitive edge over competitors.

3. Reduction scrap and rework :

Simulteneous product and process design will help a product designer to visualize the facilities available at shop floor and enable him to design a manufacturable product. At the same time, the manufacturing engineer can make valuable suggestions which will be easier and feasible for him to manufacture in existing facilities. By such an interaction the redesigning and reverification time will be saved giving the right product at first time.

4. Cost reduction :

By making early decisions in the design stage, CE minimizes the costs over the life of the products. For instance, a product design has to be done in such a way that it fits into the actual manufacturing process. In this way, the company does not have to invest in new technologies for a new designs, thus reduction in cost. Also the cost for recurring activities, manufacturing and inventory costs can be saved.

5. High Competitiveness :

The reduction in product development cycle time, early introduction of new products in market at economic prices, reduction in overall cast, better quality of products etc. can place company in better position in market over the other competitors. This also improves cpmpany's reputation and share.

CHAPTER 4

GD & T
- A TOOL FOR CONCURRENT ENGINEERING

4.1 Chapter Synopsis

Once the concepts of Concurrent Engineering and GD & T are well defined and the objectives are understood seperately, a great extent of similarities can be observed. This chapters explains about how the GD & T can be applied in design and different fields of manufacturing like tooling, inspection etc. and fulfilling the concurrent engineering objectives. The chapter does this by taking an example and then infering a definate relationship between concurrent engineering and GD & T as a tool.

4.2 Introduction

Geometric Dimensioning and Tolerancing is the means of specifying dimensions & geometry on a drawing with respect to actual 'function' and 'relationship' of the features. 'Functions' and 'Relation' are th key words. By definition, GD & T is a technique which standardizes engineering drawing practices with respect to the function of dimension and tolerances. Thus GD & T can be considered both an engineering design drawing language and a functional production and inspection technique. Uniform interpretation and understanding among design, manufacturing and inspection groups are the major objectives of the system. This will ensure effective communication and result in a reduced potential of misunderstanding.

Concurrent Engineering is one of the recent developing philosophies an organization must adopt and trust in it which can lead to a considerable savings in terms and money. CE as defined by Daniel P. Scharge and J. Edward Rogen of Georgia Institute of Technology, " Concurrent Engineering is a systematic approach to the integrated, concurrent design of products and their related process, including manufacturing, and support."[15, p.34] The concept of CE emphasizes on integrated or team effort where different functions of an organization majorly marketing, design, manufacturing, and finance works together during early stages of product development. The previous researchs in the field of concurrent engineering are discussed in the following paragraphs, which gives an idea about in what areas these concepts can be implemented.

Whenever integration is main goal, communication plays an important role, no matter whether it is a human integration or technological integration. There are many tools available for concurrent concepts applications like Design standardization, Taguchi methods, Value engineering, Design for manufacturability, Design for assembly etc. Apart from DFM and DFA, most of the tools are used in a particular field only. This chapter reports on the application of GD & T in CE stages to show how it can be useful in CE implementation. The chapter discusses the production of a washer and shows how GD & T helps in design, manufacture and inspection of the washers. It compares the conventional and GD & T methods of preparing part prints and establishes GD & T as a tool for successful implementation of Concurrent Engineering concepts.

4.3 The Relevance

Concurrent Engineering emphasizes on the integration of various functions, which does not necessarily mean the technological or computerised integration. Concurrent engineering is the concept where people from different fields are supposed to work simulteneously. It can be called as 'instant feedback' system where changes by any discipline can be is responded or commented immediately by other related departments in their context. For instance, a change in product design would be commented by manufacturer immediately that the new design can be produced or not within the available infrastructure. Thus giving designer, the idea about manufacturability of new design before actual design. In the same way, other departments can also interact with each other at appropriate levels to adhere with concurrent engineering objectives.

The design and manufacturing are the major areas out of all the functional departments on which an organization can concentrate more and can make effective improvements. Marketing is dependent on customers. Once the need enters the organization, it is design & manufacturing departments who are required to fulfill the demand. Also finance departmentcan estimate the price of a product for a particular design and manufacturing process, but if the product's functions and the product cost does not matches with the customer's requirements, it becomes necessary to manipulate with either design or manufacturing processes. So, design and manufacturing are the areas which should be emphasized more for implementation of successful concurrent engineering concepts.

The key points in CE is communication and customer satisfaction in terms of product quality, reliability and economy.

The commonest media of information transfer between design and manufacturing is blue print. A designer must be able to communicate clearly to the manufacturer what he wants to produce. At the same time, the producer must understand exactly same what designer is conveying. That means there should be a design language which indicate one and unique meaning or instruction at different levels. GD & T can be such a tool or medium of instruction which gives designer a clear & consistant method to express design intents and part requirments. The tool also indicates to manufacturer clearly what to make and how to make correctly. At the same time it also indicates the procedure of inspecting and gaging the produced part, thus protecting the design intents. GD & T compels a designer, a manufacturer and inspector to work with same level of standards & understanding.

4.4 Problem Definition

Consider the production of a simple washer. The washer is required in assembly of a precise product. Necessarily, the washer has two main features - the outer diameter and the hole. Close tolerances are provided on these features in order to have a precise assembly. The hole is a functional feature and its dimension and position (location) is extremely important for the assembly purpose. The assembly consists of a pin which is located at the center of a circular cover as shown in the Figure 35. The washer is required to be seated inside the cover while passing through the pin.

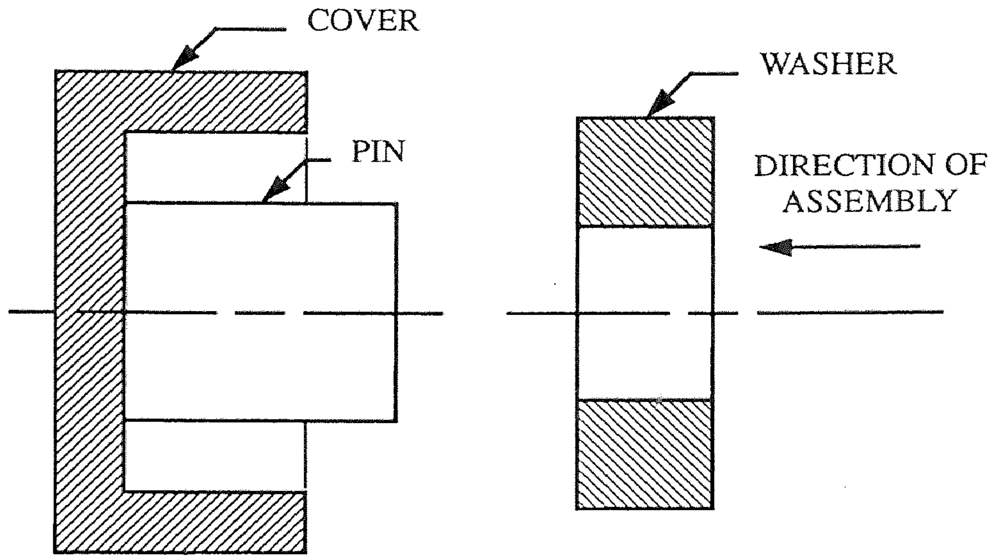


Figure 35 . Assembly Problem

The designer used a conventional method for producing the production drawing for the washer. The conventional method uses coordinate dimensioning system. The tolerances provided are unilateral or bilateral with respect ot basic size of the feature. Figure 36 indicates the production drawing using bilateral tolerances on the two features of the washer.

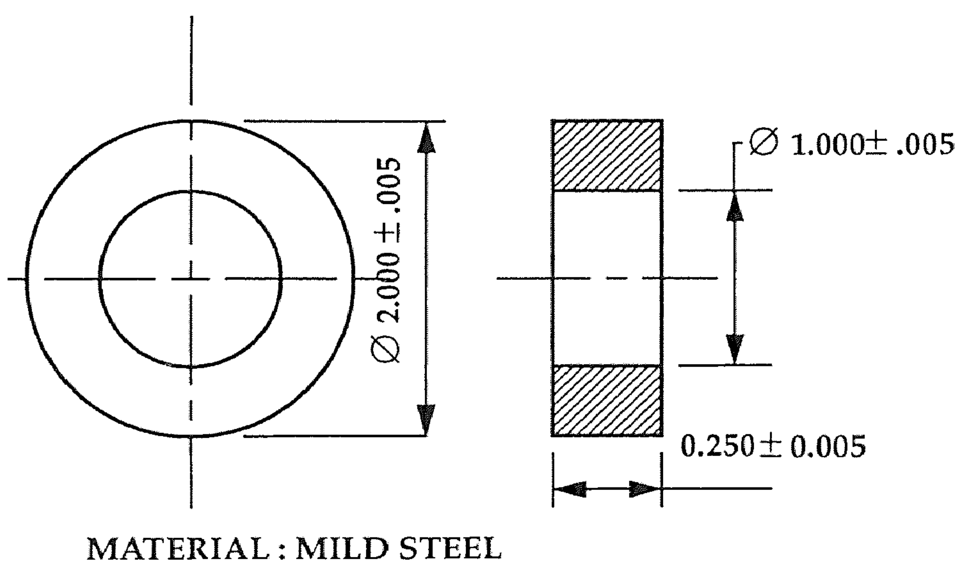


Figure 36. Conventional Drawing of Washer

The production of this washer seems to be simple assuming that the facilities are available to make the washer within the specified tolerances of ±0.005. There may be various sequence of operations and production methods. The washer can be either produced by compound shearing dies or by drilling a hole on the available circular blank. Since precision is required on washer dimensions, consider the later method of production. Also it would help in explaining the concept. Assuming that the blanks are available from some other process and are well within the tolerances. The simplest setup for drilling hole on a drilling machine can be as shown in the Figure 37.

The results of such a manufacturing process may be a hole within perfect dimensions (within tolerances), since the machine used is precise, but on the other hand, the washers might be produced as shown in figure 38.

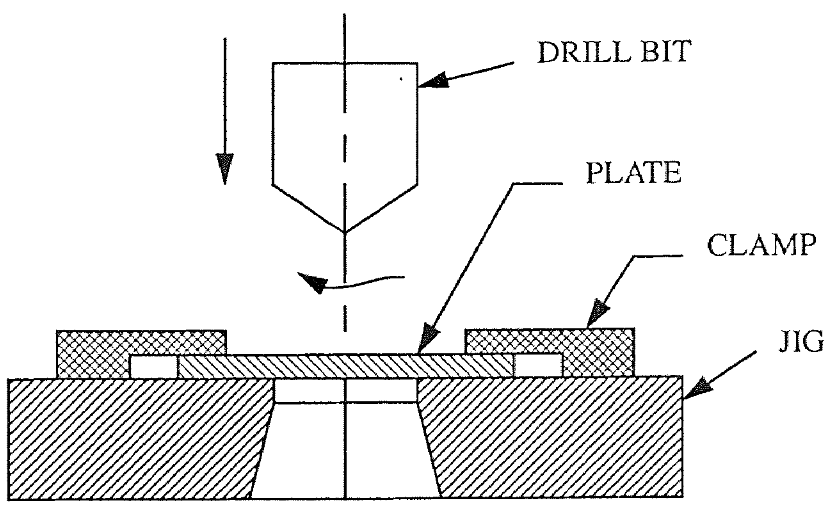


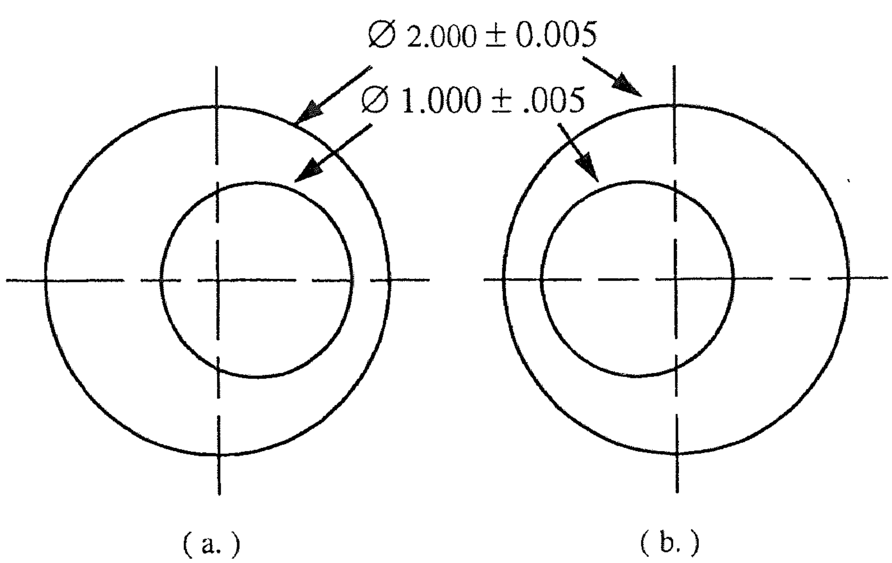
Figure 37. Manufacturing setup with Conventional drawing

Dimensionally, the washers shown in figure 38 a. & b. are with in the specified tolerances. Any washer of such dimensions will pass through the quality checks. Also there won't be any process control since the output product is fitting to required specifications. But such washers can never be assembled to seat on the cover base. The hole does not comply to functional requirements. Eventhough, the outer diameter of washer fits in the cover inside diameter, the misposition of hole will prevent from assembly. These washers can be called as ***non-functional good washers***.

The reasons for such non-functional washers may be many. These washers might be produced at different machines with different setting. The washers might have been manufactured during the different shifts where different people are involved. Different persons can have different interpretations of the existing drawing. The designer do have intention by showing that the hole should be concentric to the outside diameter, but the drawing does not give a solid basis which indicates that the hole should lie at exact center.

On analysing the conventional drawing it can be seen that the two features are independently dimensioned. There is no direct or indirect link between the two dimensions indicated in the drawing, while the assembly procedures does indicate a relationahip between them.

The washer in figure 38 is made with the extreme dimensions , it may not be a good part for assembly. There is a shift in hole with respect to outer diameter which makes the washer non-functional.



(a.) Acceptable Washer with OD Adhering to Left Extermity and Hole Dia. with Right
(b.) Acceptable Washer with OD Adhering to Right Extermity and Hole Dia. with Left

Figure 38. Acceptable Washers

The conventional drawing for washer also lacks in location of the hole. There is a third feature missing which interlocks the two features. There is no control over positioning of the hole with respect to the outer diameter of washer, thus shifting the hole in different directions while producing it dimensionally correct. These discripencies causes the improper positioning of hole & thus making non-functional washer.

Such situations can be handled by using Geometric Dimensioning and Tolerancing practices. The method and explaination is as follows:

It can be concluded from the conventional drawing that the description of hole location with respect to the washer diameter is missing. There are various methods using Geometric Dimensioning & Tolerancing by which the location of the hole can be controlled. It can be located using positioning, concentricity or the hole can be referenced by circular or total runout.

A method of position tolerancing for hole is discussed in this example. Position is a term used to describe the exact location of a point, line, or plane of a feature in relationship with datum reference or other feature. A position tolerance is the total permissible variation in the location of feature (hole) about its exact true position.

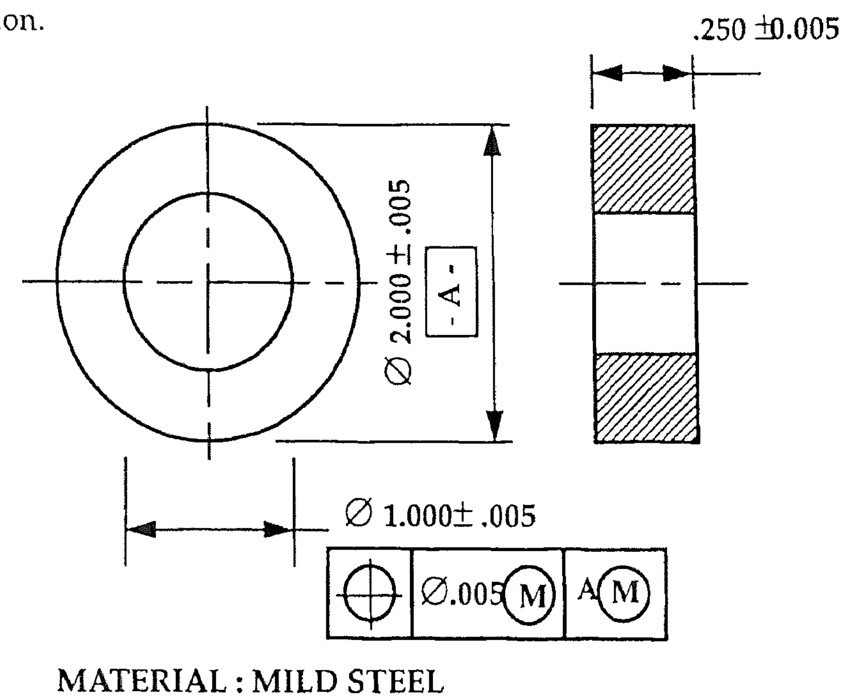


Figure 39. G D & T Drawing for Washer

The production drawing for the washer using Geometric Dimensioning and tolerancing is as shown in Figure 39.

The true position of hole is the center of outer diameter of washer as shown in figure 40, which is denoted as basic dimension (without tolerances). The functional requirement of hole is to be concentric to the washer periphery. Therefore, idealy, the center of hole must coincide with centerl of the washer. The following sections discusses how the GD & T implementation affects on the different functional divisions of a manufacturing organization.

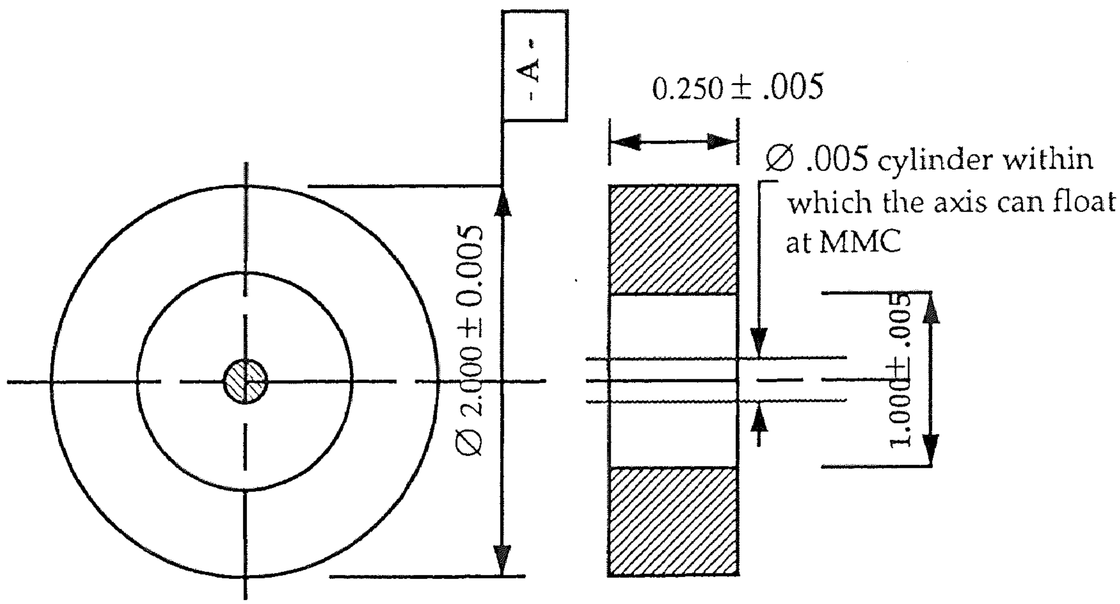


Figure 40. Interpretation of GD & T Drawing

4.5 Manufacturing Engineering Concerns

4.5.1 Design Concerns :

It can be seen from the previous researchs and experiences that while integrating the different departments as an implementation of concerrent engineering, a major emphasis should be given at the early design stages. This is because a majority of costs are committed in this stage. For example a design specifies about material, it gives an idea about the manufacturing or fabrication methods, assembly methods, materials handling options, inspection requireed and other production aspects. All of these fields involves certain cost. This cost is as high as about 70% of life cycle costs of the product.

Since the design contributes to many cost factors, it is imperative to define clearly these factors at design stage only so that the later stages of production can be properly executed and justified. So far as material selection & materials handling are concerned, they affect in a lesser degree because there is no specific equipment setting or executing methods required once the material & handling process is decided. These costs contribution is relatively constant.

The actual problem arises when the part is required to be processes (manufactured) or assembled. A part can be manufactured in different ways with different sequence of operations. Also the part can be manufactured by different workers at different times who will try to interpret the drawing in their own way and produce it as per their knowledge and judgement. Also each machine has inherent error, but assuming this error to be constant for all parts, the parts produced with other variability would be different in dimensions and characteristics. This will create problems in assembly process. Thus will result in defective or non-fuctional parts.

This necessitates need of a tool or media which states all the informations required to produce the part. GD & T can be used as media which indicates almost every information required for production, tooling, and gaging. Since a single drawing drawn with GD & T standards involving various informations and because of unique methodology of GD & T, the drawing is interpreted in one and unique way at different areas of manufacturing like production, tooling and inspection. It creates a link among all these areas so that they can work with same standards.

Let us consider the problem in question. When the washer is drawn with GD & T practices, following points a designer would like to convey :

1. The outer diameter (Ø 2.000 ± 0.005) is considered as the the datum 'A'. As the datum is specified on feature of size, it must be simulated by the modifiers i.e. MMC, LMC or RFS. It can be seen that control frame indicates maximum material condition (MMC) for outer diameter when using as datum 'A'.

2. The tolerance of position is applied to feature of size Ø 1.000 ± 0.005. This per-

mits the hole to vary within the tolerance zone of ± 0.005, but the feature control frame indicates that the hole is to be positioned with respect to datum 'A'. Also the positional tolerance zone is of diameter 0.005 within which the axis of the hole must lie, the center of the tolerance zone being exactly at the true position which is the center of outer diameter. The meaning of GD & T drawing is indicated in figure 40.

3. Maximum material symbols are applied in feature control frame.

Whenever a datum is applied to a feature of size, it must be accompanied with a modifier. A maximum material condition is applied to the outer feature of size which means that the positional tolerances are applied when the datum feature is at MMC i.e. Ø 2.005.

A MMC modifier is also connected with position tolerance which indicates that a tolerance of 0.003 is applied when the hole is at its maximum material condition i.e. Ø 0.995. The modification in positional tolerances when the hole size deviates from MMC is as shown by first two columns of Table 1.

4. A basic dimension (without tolerances) is specified to indicate true location of feature (hole) from the specified datum 'A'.

4.5.2 Tooling Concerns :

Tooling is an essential component of any production system. Product quality depends, to a large extent, on the quality of tools and gages used in the manufacturing and inspection operations. The term tool in manufacturing industry refers to any device that is capable of working a material into a desired shape, holding & locating the material while it is being worked on, or measuring the material after working is finished. Common tools are machine tools, cutting tools, jigs, fixtures, dies, and gages. Tools and gages provides physical means to attaining volume production and interchangeability of component parts.

Before going for actual cutting, the material or work piece is required to be set on the machine properly so that operations can be performed at right place. Setup plays an important role when quality is of prime importance. The set up normally means to the proper location of work in relation to the cutting tool. It may also include sometimes the holding & guiding of tool.

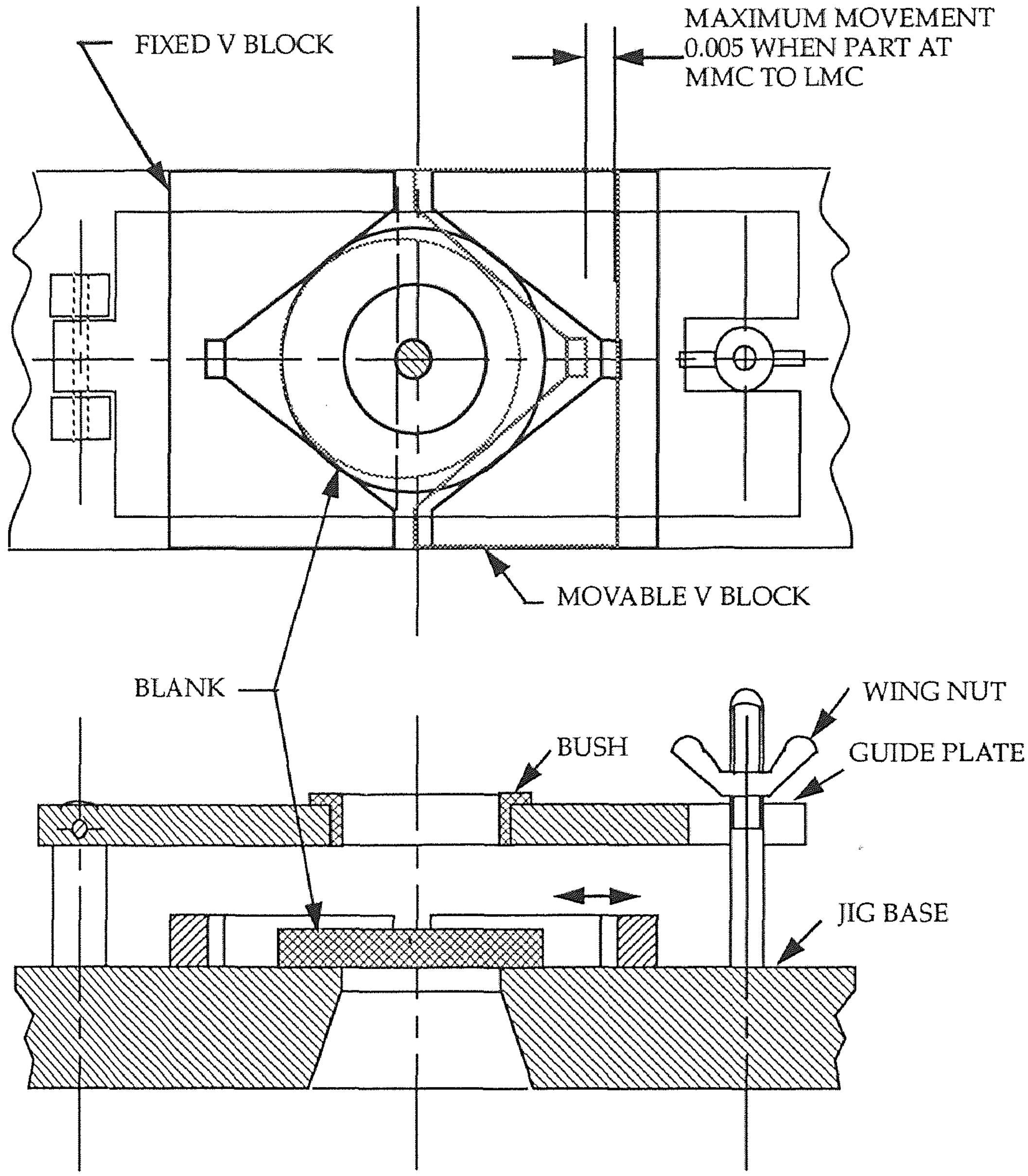


Figure 41 . Drill Jig for Washer Production

The considered example requires a mass production of the washers with precision. Hence, the raw material i.e. blank for drilling hole must be located properly so that the drilled hole has the specified positioning requirements. A drill jig will be required to design for such a purpose which will locate the blank from datum A, the outer diameter. The blank is assumed to be available from previous operation and is within the tolerance limits. The jig should be designed in such a way that it is suitable to locate the blank size from LMC to MMC i.e. Ø 1.995 to Ø 2.005 only. This location would also serve as purpose of datum 'A'.

A possible drill jig design is as shown in the Figure 41. The jig consists of one fixed V blocks and a movable V block, which acts as locators for cylindrical surface i.e. washer outer surface. The movable V block is given freedom in such a way that when it reaches to its maximum position, the circle made by the imaginary three locators touches both the block surfaces and would be equal to the MMC of blank. Therefore, a blank of MMC can be mounted on the jig easily. The blanks more than MMC will not be fit in the jig and hence will be rejected. At the MMC condition, the bush center line, coincides with that of blank at MMC. This will ensure that if blank is at MMC, it will be located perfectly and hole will be drilled exactly at the position. The bush is used to guide the drill bit.

On the other hand, if the blank is at LMC, it would be located by fixed block and minimum position (extreme left) of the movable block. this will ensure that if the blank is below LMC, it is a rejection. When the blank is at LMC i.e. Ø 1.995, its center would still lie on the circumference of tolerance zone, hence making an acceptable positional hole. This is as shown in the Figure 41 by dotted lines.

This jig perfectly follows the 3-2-1 principle of location. The base of the jig gives the effect of three points onwhich the part is mounted. These locators restricts the three degrees of freedom. The fixed V- block provides the two locators and restricts the motion in two directions. The sixth degree of freedom will be stopped by the moving V-block, which will act as a single locator. It can be seen that this jig design locates the washer successfully and at the same time allows the outer dimension variation of the washer.

The drill bit will be located and guided at the center of blank when it is at MMC (true position). This will ensure that if blank is at MMC, it will be located perfectly and hole will be drilled exactly at true position. If the blank deviates from MMC the hole position will still be ensured within the axis tolerance zone of Ø 0.005.

Thus it can be seen that GD & T also tells a tool design department as how to produce a tool for an interchangeable part production. It has also incorporated a tool design division and thus pursuing the integrated approach of CE implementation.

4.5.3 Production Concerns :

By providing additional informations, GD & T has dramatized product engineering to a large extent. When using a coordinate system and producing & inspecting a prototype following problem arises :

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 and production tends to blame each other. But the truth is that the language or the design intent was not clear, hence communication of how the part has made was a failure, though the 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. 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 procedures are 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.

It can be seen from the GD & T drawing for the washer that it delivers the exact sequence of operation for the manufacturing department. By doing so it maintains the uniformity in the manufacturing of all the washers irresective of where it is made i.e. at different machines or at vendor etc. and who makes it. As can be seen from the positional tolerances on the hole, it is positioned with respect to the datum 'A' (the outer diameter of blank). This means before going for drilling operation, the blank outer diameter must satisfy all the requirements to act as a datum. In the example, the outer diameter is required to be within the tolerance of ± 0.005. Once the blank satisfy this condition, it will be used as datum for the nesxt operation. There may be other conditions associated with this datum feature (OD) e.g. perpendicularity of outer diameter etc.. In this case, this first condition must be preceded by any other operation in sequence.

Once datum is fixed, the next operation is to drill a hole of Ø 1.000 ± 0.005, but a positional tolerance of Ø 0.005 is associated with it., which means that the axis of the hole is permitted to vary within the cylindrical tolerance zone of Ø 0.005 as shown in figure 41. To attain at this manufacturing situation, a drill jig is designed as discussed in previous section. It can be observed that the blank is located from outer diameter and jig design takes care of the variation in blank diameter. The following shows the extra tolerances calculations for hole. It indicates the permissible hole position tolerances as hole size departs from MMC. This is as shown in Figure 42.

Stated position tolerance with hole at 0.995 MMC	= 0.005
Plus total 0.995 hole tolerance	= 0.005
Position tolerance with datum at 2.005 MMC	= 0.010
Plus total tolerance with 2.005 datum hole	=0.005
Total position tolerance for hole	= 0.015

One more advantage, the manufacturer can achieve by using GD & T is getting "bonus tolerances". The bonus tolerance value comes from feature-of-size tolerance. It is equal to the amount that the feature-of-size departs from MMC. This bonus tolerance makes the part easily manufacturable and economical. In the above example, the positional tolerances are provided at hole maximum material conditions. This means that when hole is at Ø 0.995 size, a positional tolerance of 0.005 is applicable. The virtual condition of hole would be 0.990, which is the theoritical minimum diamension of the hole. The calculation of bonus tolerances at different hole sizes in as indicated in Table 1.

It can be observed that each and every information provided by designer in GD & T print guides the manufacturer in deciding the steps in production of the part. In this way both designer and manufacturer has same interpretation and execution of the drawing without any ambiguity and conflicts.

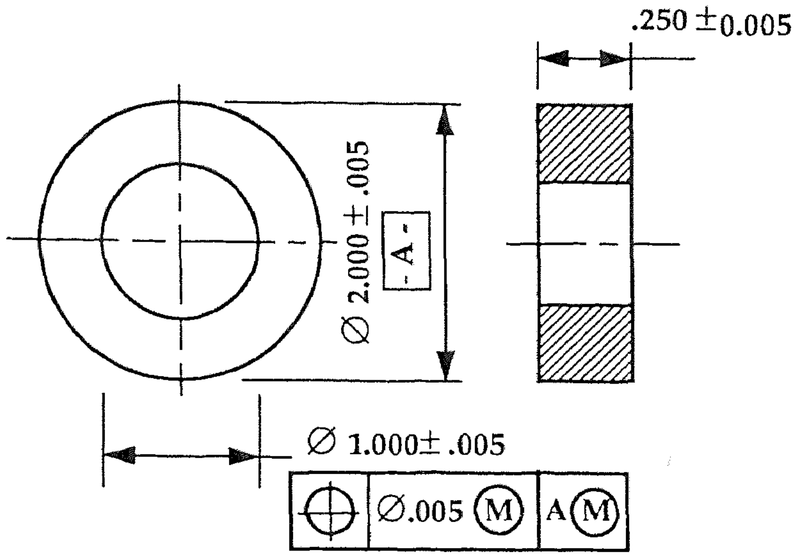
4.5.4 Inspection :

Marketing and sales are the departments, which bring the need of customers to design stages. These needs incorporate quality as well as economic requirements. Quality planning operations are initiated by consumer quality requirements. Quality control operations are directed and implemented from cost-criterion basis. Economic decisions underlie the development of specification tolerances, control procedures and inspection plans.

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 mechanical industries. There are two types of inspection processes that can be involved :

1. Inspection/ Checking : This process of inspection involves the measurement of variables or attributes on the part. Basically, it checks the dimensional accuracy and other defects present on the part. It is primarily concerned with determining the degree to which production output conformed to the established technical specifications for the product. The result of such checks can be used for product and process controls. It can also be used for preparing quality audits to generate feedback informations to the quality planning operations and upper level management sections.

2. Performance check or Testing : This kind of inspection is used when product is finished after assembly. It actually checks the function of product what the product is supposed to do. An assembly consists of more than one part and it may be possible that the parts do have movements relative to each other. Therefore, it must be ensured that parts are dimensioned and manufactured properly so that, when assembled, they achieve the desired relative motion and thus giving desired functions.



EXTRA TOL FOR HOLE :
PERMISSIBLE HOLE POSITION TOL. AS HOLE SIZE DEPARTS FROM MMC :

STATED POSITION TOL. WITH HOLE AT 0.995 MMC	= 0.005
PLUS TOTAL 0.995 HOLE TOLERANCE	= 0.005
POSITION TOL. WITH DATUM HOLE AT 2.005 MMC	= 0.010
PLUS TOTAL TOL. WITH 2.005 DATUM HOLE	= 0.005
TOTAL POSITIONAL TOLERANCE FOR HOLE	= 0.015

Figure 42. Effect and Calculation of MMC on Hole Positioning

Table 4 : Positional Tolerances and Bonus Tolerances Applicable at Hole Sizes

Hole size	Positional tol.	Bonus tolerance	Virtual condition
0.995 (MMC)	0.005	0.0	0.990
0.997	0.007	0.002	0.990
0.998	0.008	0.003	0.990
0.999	0.009	0.004	0.990
1.000	0.010	0.005	0.990
1.001	0.011	0.006	0.990
1.002	0.012	0.007	0.990
1.003	0.013	0.008	0.990
1.005 (LMC)	0.015	0.010	0.990

In such cases, functional gages can be used. These gages check the dimensions and tolerances on the parts as well as ensure that part would function when assembled. GD & T gives all the information about dimensioning and functionality of part and thus help a gage designer to design and produce the specific gage for specific part feature.

Functional gages are economic to manufacture and use as well. These gages give only two types of answer i.e. accept or reject. Also a functional gage has fixed dimensions and is suitable for one or few dimension checks. In other words, the functional gage can only be applicable to a part with constant virtual condition. Hence in the cases where virtual condition changes with the part dimensions other methods, for instance comparators should be used to determine the variation and accept or reject the part depending upon the permissible dimensional limits.

In the example in question (washer), the positional tolerance is applicable when the hole becomes Ø 0.990. As the hole size departs from MMC, there is additional positional tolerance available for manufacturing, but still the virtual condition remains same for all sizes. This indicates that a functional gage can be constructed for an MMC condition of hole. The gage dimensions and design is discussed in the next section.

But if the positional tolerances are applied with LMC, the situation would be different. With the change in hole size from LMC to MMC, the positional tolerance would also change. So, when virtual condition is calculated, it changes with different hole sizes. This is shown in the Table II. This implies that there should be different gages mede for each different size of hole, which is uneconomical, because to construct a functional gage requires precision and is also costly affair. In such cases, the other methods can be employed. For example, A comparator or Dial Indicator. A full indicator movement method setup is as shown in Figure 43. Variations at single cross section will be observed on the dial indicator and if it varies within the limits, the washer would be accepted. The same process should be used at all the cross section of the washer. This method has an advantage that the dial indicator can be used for any washer irrespective to the hole size or material conditions. But it is a slow and tedious process of measurement.

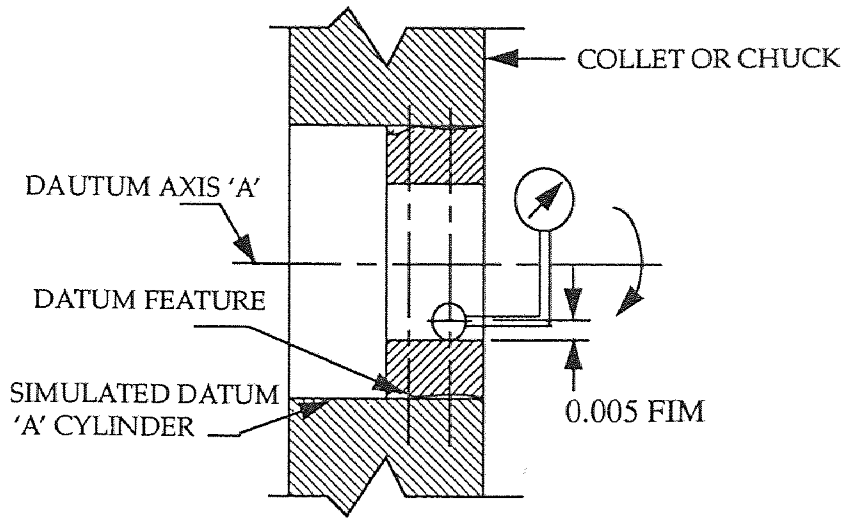
4.5.5 Gaging :

Transfer of a length standard from the National Bureau of Standards to a manufacturing plant is accomplished by means of gage blocks. A gage is a reference piece, either square or round in cross section, with two faces end surfaces or cylindrical surfaces. These surfaces are measurement surfaces. That is, the end faces are flat parallel surfaces where separation has been established to tight wave precision and accuracy.

When a GD & T drawing is presented to an inspection department, it gives the exact picture of how the part was produced by manufacturer. It will give an idea about which features were placed as datum and which features are made with reference to the datum. Accordingly, the functional gage can be prepared for gaging a particular feature on the component.

In the above example, the possible functional gage can be as shown in the Figure 44. The Ø 0.990 size gage pin is determined by the MMC size of the hole, 0.995, minus the stated positional tolerance of 0.005. Functional gages are, ofcourse, not required with positional applications, but they do, however, provide an effective method of evaluation where desired. The inner diameter of the gage is produced at MMC (Ø 2.005) which checks the outside diameter of the washer.

Once the outside diameter of washer is ensured, the next step is to check the dimension and location of hole. The Figure 44 illustrates the mating part situation represented by a functional or fixed pin. The gage pin size as calculated above represents the maximum permissible offset of the hole within its stated positional tolerance when hole is at MMC size of 0.995. The figure shows the hole at MMC offset to the maximum permissible limits of the 0.005 positional tolerance zone. The hole is within tolerance and, as can be seen, would satisfactorily pass the simulated mating part condition as represented by the gage pin. It can be noted that that a functional pin gage illustrated here explains that it can be used only to check the positional location of the hole. Hole size tolerances, however, must be held within the tolerances specified on the drawing and must be checked separately from the positional check.



THE FEATURE MUST BE WITHIN THE SPECIFIED TOL . SIZE AT ANY MEASURING POSITION. EACH CIRCULAR ELEMENT OF THE SURFACE MUST BE WITHIN 0.005 FIM WHEN THE PART IS ROTATED ONE FULL ROTATION ABOUT THE SPECIFIED DATUM AXIS WITH THE INDICATOR FIXED IN POSITION NORMAL TO THE SURFACE.

Figure 43. FIM Setup when Positional Tolerances at Hole is at LMC

Table 5 : Positional Tolerances with LMC Applicable at Different Hole Sizes

Hole size	Positional tol.	Virtual condition
1.005 (LMC)	0.005	1.000
1.003	0.007	0.996
1.002	0.008	0.994
1.001	0.009	0.992
1.000	0.010	0.990
0.999	0.011	0.988
0.998	0.012	0.986
0.997	0.013	0.984
0.996	0.014	0.982
0.995 (MMC)	0.015	0.980

4.5.6 Vendor Concerns :

A key to the success of the Concurrent Engineering project is the inclusion of suppliers as members of the team. The manufacturing engages in long-term relationships with selected vendors who demonstrate the ability and desire to be flexible and work as a integral part of the team. Whereas in the past a supplier would bid on a specification for a system or a component, now a vendor is developing the specifications for products that he/ she will be awarded on a sole source basis.

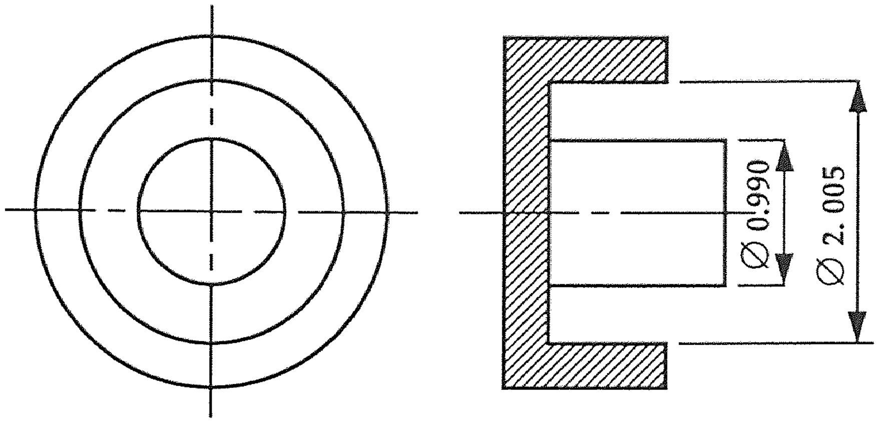
An GD & T drawing can translate the exact needs of the company to the vendor. And at the same time vendor have clearcut idea about what is to be manufactured, How to make it and how it functions. Also the inspection procedures are well defined. This will lead to reduction in the inspection points e.g. inspection at vendor's end may eliminate the inspection at company. At the same time, the vendor has all the chances to provide the right product at first time. Thus, while concurrent concepts tells to include the suppliers as a team member, the GD & T makes the link tighter in the sense it gives supplier a credibility and confidence to keep the contract on long term basis.

4.6 Inferences

The example explains the implementation of GD &T and compares this method of dimensioning with conventional methods. When analysing closely, it can be established that procedures and concepts in GD & T directly or indirectly tends to adhere with the concurrent engineering objectives. The interrelations are as discussed below :

(a) One of the objectives of concurrent engineering is to achieve a team effort among the functional groups in an organization and in turn improve the communication within the departments.

The above example illustrates how using a GD & T drawing , the designer communicates with the manufacturer, tool designer and inspection. The GD & T print includes enough information so as to guide them in their operations. For example, the datum, dimensions and locational tolerances between two features of washer gives the precedence of operation to actual producer. At the same time, the drawing tells the inspection how to gage the washer. This way a single GD & T drawing provides one & unique interpretation to different departments thus improving communication and ensuring a standardized operation on different



Gage

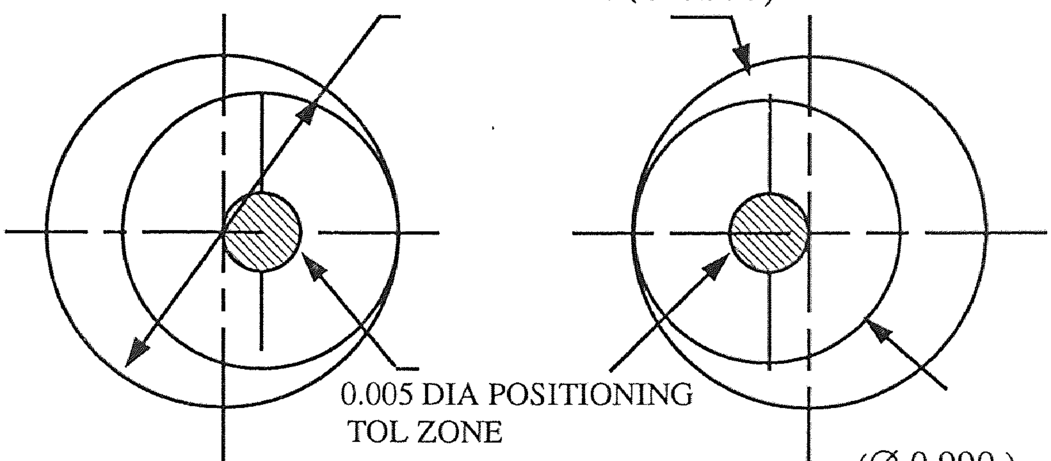


Figure 44. Mating Part Situation Represented by Gage Pin

GAGE CALCULATIONS :

MMC size hole	= Ø 0.995
Minus permissible position tolerance	= (-) 0.005
Gage size	= Ø 0.990

The datum outer diameter is treated as datum when it is at MMC i.e. Ø 2.005

levels.

(b) As discussed before, the concurrent engineering approach is effective in conceptual design phase. In this phase the design is fluid, hardware is remote and few people are involved, therefore the interactions and constraints are less. It is the stage when the designer has maximum latitude to explore the alternatives and the a chance to identify best product and process concepts for manufacture.

The concept of GD & T compels a designer to include the manufacturing aspects of the part in his design. Also GD & T tries to consolidate the design, manufacturing and inspection in very early stages of product development. It is often difficult for a designer to know about manufaturing aspects of product. Hence, before designing a GD & T drawing, the designer has to consult with the manufacturer and inspection department. This improves the team effort and also ensures that the design will be implementable successfully at later stages of product development cycle.

(c) Another concurrent engineering concept is to eliminate or reduce redesigning and reverification time and related costs. The objective is to make the right decision during the non-recurring activity. Making good decisions in early stages creates maximum leverage in terms of investing a little time and money during product design to achieve larger profits over life of product.

With the intense competition, designers often are under pressure to bring the product in market as quickly as possible. There is always an argument that there is no time for concurrent engineering practices. But time to market a product is not only a design time. Time to market is the time it takes to get a product into the customer's hands at a competitive prices. Also there is a time involved in manufacturing, testing and other process before product is marketed.

Concurrent Engineering helps speed the products actual time to market, even if that means spending little more time making sure the design is flawless in its performance and making sure it is manufacturable, testable and serviceable.

Since GD & T provides unique interpretation to different people and also considers production and inspection aspects early at design stages, it ensures that the right product will be made at first time thus eliminating or reducing the possibility of recurring design and verification.

(d) Learning curve : The time taken to produce first unit is always more than the subsequent units. Ultimately, the curve becomes horizontal at particular time which can be called as standard time to produce the unit. GD & T can help in steeping down this learning curve. Much of the time is spent in interpretation of drawings and then setup for manufacturing. If the drawing provides a clearcut

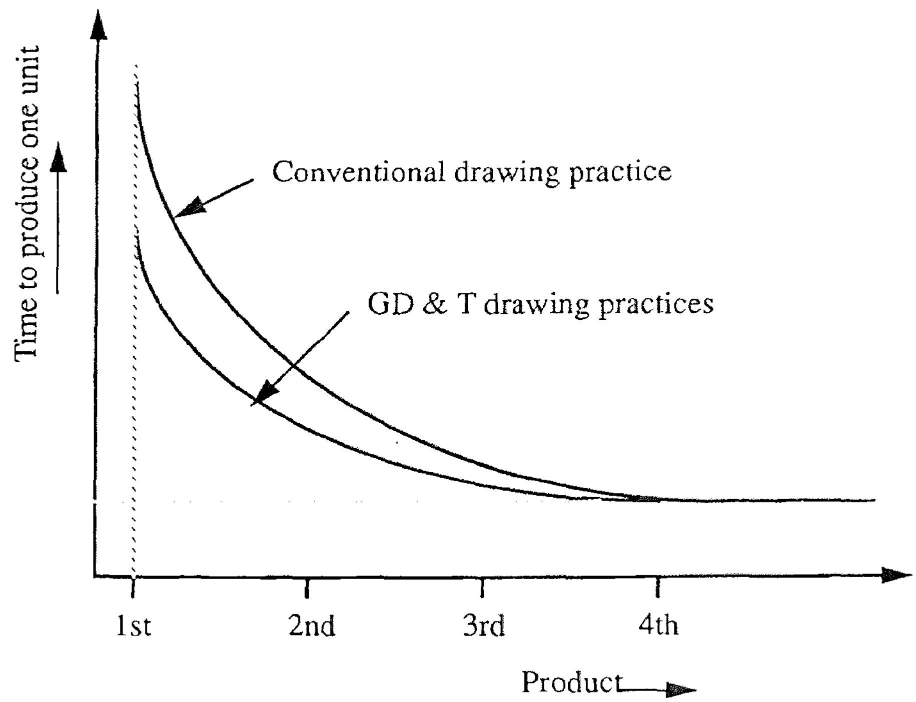


Figure 45 Effect on Learning Curve

idea about what to produce and how to produce, the curve can even start from lesser time as compared to traditional drawing methods. Also it can reach to standard time for unit manufactring very early, thus saving time. Figure 45 shows the learning curve with conventional and GD & T application.

(e) Concurrent engineering concepts also leads to achieve a competative edge over competitors by minimising developmental and product costs, increasing quality and hence augmenting market share.

GD & T has indirect contribution to this goal. Certainly uniformity in print interpretation among different functional groups makes the learning curve steeper. Much of the time is spent in understanding the drawings & then setup for manufaturing. If the drawing provides a clearcut idea about what to produce and how to produce, the product developmental time is reduced and also quality product is produced. This also reduces the time lag between product development and marketing & sale. Thus GD & T gives both quality as well as marketing advantages to an organization.

CHAPTER 5
CONCLUSION

5.1 Conclusion

Concurrent engineering is a philosophy or management practice which leads to a definite benefit of faster cycle time, better products, better communication and more responsive organization. But successful implementation of these concepts requires a physical tool, the use of which would lead to common advantages as of concurrent engineering. Geometric Dimensioning and Tolerancing can be considered as a tool or medium, the concepts of which comply directly or indirectly with concurrent engineering objectives.

As discussed in chapter 4, using geometric dimensioning and tolerancing language provides the manufacturing with more benefits than the traditional drawing system. It can be observed from the inferences how GD & T can help in realising the goals of concurrent engineering. The first and major accomplishment that could be achieved is a uniform communication at different levels. It is also a media which integrates the related departments by providing full and clear informations to all. Moreover, a proper use and interpretation of GD & T eliminates the recurring activities like redesigning, reinspection and performance checks etc. This leads to saving in product development time and costs. Also it allows an early introduction of products in the market. This also improves the product manufactured quality. Thus gaining a competitive edge over other competitors.

It can be seen that all of the above achievements tend to represent the concurrent engineering objectives in one way or other. Therefore, it can be considered as a tool or media for implementing the concurrent engineering philosophy in a manufacturing industry.

The use of GD & T does not give a solution by itself. The process of product development is always sequential and it applies in a specific order. But, GD & T captures the interdependece relationships among the departments and at the same time maintains the order of operations.

The greatest shortcoming of GD & T system is lack of awareness and training available. There are very few educational institutions and organizations which provide a formal education for this dimensioning system. The myths and barriers for practicing GD & T concepts must be broken and adequate training and instructions must be spread among actual users so as to achieve the benefits of Geometric Dimensioning & Tolerancing.

ISO 9000 and ABCA standards do not refer to GD & T directly, but GD & T does comply with the recommondations of these standards. Therefore, the use of GD & T in industies may lead to widespread implementation of Concurrent Engineering concepts globally.

5.2 Future Research

A great potential lies in improving and evolving easier techniques for implementing GD & T at the unskilled level. The current system is an effort of three decades of research by a committee action representing military, industrial and educational interests. This is a result of three other standards which were used separately.

As discussed before an immediate need in this area is to increase the awareness about geometric dimensioning and tolerancing methodology at all levels of users, from shop floor worker to designer. A definite scope lies in the development of a computerized system, which incorporate CAD with all tolerancing and geometric features. Besides this it should coordinate with Concurrent Engineering requirements. There are some software in market integrating the design and manufacturing aspects, but lacks in emphasizing the tooling, inspection and financial goals of concurrent engineering.

REFERENCES

(1) Foster, Lowell W. *Geo-Metrics II*, 1986 Edition, Addison- Wesley Publishing Company, Inc., Massachusetts 01867.

(2) Garrett, Ronald W. "Eight Steps to Simultaneous Engineering." *Manufacturing Engineering*, November (1990) : 41-47.

(3) Gehrke, T. "Blueprints Never Lie", *Machine Design*, November 12 (1992) : 83.

(4) 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.

(5) Krulikowski, A. *Geometric Dimensioning and Tolerancing, A self study Workbook*, Effective Training Incorporated, Westland, MI-48185.

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

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

(8) Kotefski, S. "Geometric Dimensioning and Tolerancing." To be published.

(9) Kuo, Way, and Hsu, J. P. "Update : Simultaneous Engineering Design in Japan." *Industrial Engineering*, October (1990) : 23 -26.

(10) Markowitz, Michael C. "Concurrent Engineering Starts with First Step." *EDN*, July 18 1991, 110 -114.

(11) Mcknight, Seldon W., and Jackson, Jerry M. "Simultaneous Engineering Saves Manufacturing Lead Time, Costs and Frustration." *Industrial Engineering*, August (1989) : 25-27.

(12) Nevins, James L. and Whitney, D. E. *Concurrent Design of Products and Processes*, 1989 Edition, Mcgraw Hill, Inc., USA.

(13) Neuman, Al. "Geometric Dimensioning and Tolerancing." *Carbide and Tool Journal*, July/August (1990) : 9-15.

(14) Requicha, A. A. G. " Toward a Theory of Geometric Tolerancing." *The International Journal of Robotics Research*, No.4 Winter (1983) : 45-60.

(15) Scharge, D.P. and Rogan, J. E. "Impact of Concurrent Engineering.", *Aerospace America,* September (1991) : 34.

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

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

(18) Turino, J. "Concurrent Engineering Speeds Development Time, Lower Costs." *EDN*, May 7 1992, 191-195.

(19) Valaer, P. "The Problems with Geometric Dimensioning and Tolerancing." *Machine Design,* August 23 (1990) : 129-133.