

Introduction

As more and more manufacturers become immersed in the global economy, standardization plays a critical role in their success. Geometric dimensioning and tolerancing (GD&T) provides a set of standardized symbols to describe parts in a way that is meaningful to manufacturers and customers around the world. This course will provide some general information about GD&T. At the end of this course, you will be able to meet the objectives outlined below.

Objectives

- Define GD&T.
- Describe the scope of GD&T standards.
- Distinguish between a feature and a datum.
- Distinguish between GD&T and traditional tolerancing.
- Define the Datum Reference Frame (DRF).
- Describe how the DRF and the part are related.
- List the major categories of geometric tolerances.
- Explain the straightness tolerance.
- Define the flatness tolerance.
- Explain the circularity tolerance.
- Define the cylindricity tolerance.
- Explain the profile of a line tolerance.
- Describe the profile of a surface tolerance.
- Explain the angularity tolerance.
- Define the perpendicularity tolerance.
- Explain the parallelism tolerance.
- Describe the position tolerance.
- Explain the concentricity tolerance.
- Define the symmetry tolerance.
- Explain the circular runout tolerance.
- Describe the total runout tolerance.
- List the material conditions modifiers.
- Explain how bonus tolerance is applied to a hole.
- List the contents of the feature control frame.
- Describe the advantages of GD&T.

SECTION

1

THE GD&T PROCESS

Objectives

- Define GD&T
- Describe the scope of GD&T standards
- Understand when and why GD&T is used
- Understand commonly used GD&T symbols & terms
- Distinguish between a feature and a datum.
- Distinguish between GD&T and traditional coordinate tolerancing.
- Define the Datum Reference Frame (DRF).
- Describe how the DRF and the part are related.

What is GD&T

Parts manufactured in a shop must meet specific design requirements shown on engineering drawings. GD&T is a way of specifying engineering design and drawing requirements with particular attention to actual function and relationship of the part features. The best method for describing how the parts should fit together and how they function should be one that is understood by people in all stages of the process. GD&T can be thought of as an engineering design drawing language and a functional production and inspection technique. It aids manufacturers in sophisticated engineering designs as well as meeting demands for more completeness, uniformity, and clarity.

This unique system uses standard, international symbols to describe parts in a language that is clearly understood by any manufacturer that is familiar with the standard.

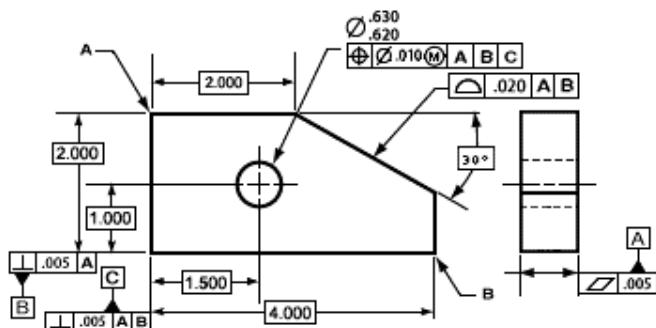


Figure 1-1. A GD&T drawing uses standard symbols to describe a part.

Figure 1-1 shows a side and end view of a simple part and contains many of the symbols that define the characteristics of a work-piece. Traditional drawings often contained handwritten notes that required translation for manufacturers in different countries. The GD&T symbols substitute for those notes and greatly reduce the chance of mistakes.

GD&T represents a significant improvement over traditional dimensioning methods in describing the form, fit, and function of parts. GD&T is considered to be a mathematical language that is very precise. It describes each work piece in three “zones of tolerance” that are then described relative to the **Cartesian coordinate system**.

A Little History

The Cartesian coordinate system was developed by Rene Descartes (pronounced “day-kart”), a French mathematician, philosopher & scientist.



Figure 1-2. Rene Descartes

Descartes (Renatus Cartesius) was born in 1596, in France, and died in 1650. During his life, he formed much of the thought about the order of things in the world and established three precepts about the method by which we should examine all things. The most important influence was the first precept, which states, in Descartes words, *“never to accept anything for true which I did not clearly know to be such”*. This new idea of skepticism influenced many to start finding out things for themselves rather than relying solely on authority. The idea as such may have been the starting point for the development of modern science.

That idea of examining everything in relation to what should be “exact and perfect” led to Descartes’ development of the Cartesian coordinate system. During an illness, as he lay in bed sick, Descartes saw a fly buzzing around on the ceiling, which was made of square tiles. As he watched he realized that he could describe the position of the fly by the ceiling tile it was on. After this experience he developed the coordinate plane to make it easier to describe the position of objects.

GD&T has developed as a method to question and measure the “truth” about the **form**, **orientation**, and **location** of manufactured parts. Like any other language, GD&T uses special punctuation and grammar rules, and it is important to use them properly in order to prevent misinterpretations. It takes time, practice, and patience to become familiar with the GD&T system. It is comparable to learning a new language.

The Background of GD&T

The American Society of Mechanical Engineering and the International Organization for Standardization have worked together to create a system for part design that can be understood and used around the world. ASME Y14.5M and ISO 1101 are the actual written standards that define the GD&T standard.

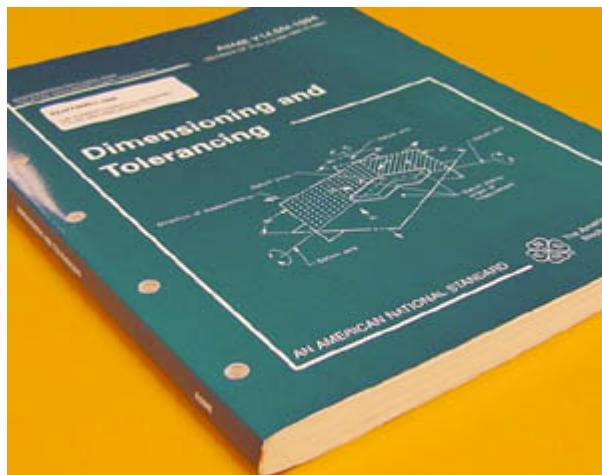


Figure 1-3. The ASME Y14.5M standards book.

The GD&T drafting standard is not meant to be a standard for inspection, but rather a standard for describing the design of a part. However, GD&T does give an inspector a clear understanding of what the designer intended. This information can help the inspector determine how to form the best measurement standard. The standards dictated by ISO & ASME provide a world wide system to design and create parts.

When Should GD&T Be Used & Why:

Generally speaking, there are many instances that call for GD&T to be used. Some of these instances are listed below:

- When part features are critical to function or interchangeability.
- When functional gauging techniques are desirable.
- When datum references are desirable to ensure consistency between manufacturing and gauging operations.
- When computerization techniques in design and manufacture are desirable.
- When standard interpretation or tolerance is not already implied.

There are many obvious reasons why GD&T makes sense in the manufacturing environment. For example:

- It saves money.
- Provides for maximum producibility of a part through maximum production tolerances.
- Ensures that design dimensional and tolerance requirements, as they relate to the actual function, are specifically stated and thus carried out.
- Adapts to, and assists, computerization techniques in design and manufacturing.
- Ensures interchangeability of mating parts at assembly.
- Provides uniformity and convenience in drawing delineation and interpretation, thereby reducing controversy and guesswork.

Advantages of GD&T:

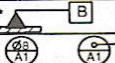
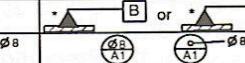
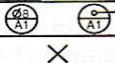
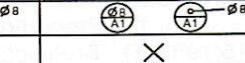
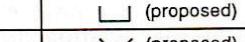
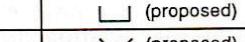
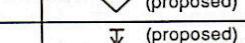
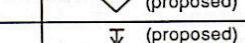
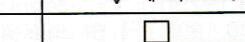
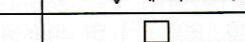
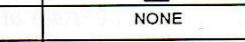
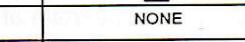
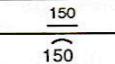
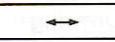
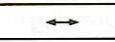
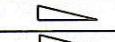
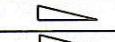
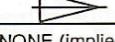
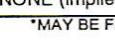
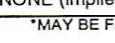
GD&T instructions are a significant improvement over the traditional methods. GD&T is a compact language that can be understood by anyone who has learned the symbols and it replaces the numerous notes that were often used to describe the part.

In accordance with the guidelines of ASME Y14.5M and ISO 1101 standards, GD&T offers greater design clarity, improved fit, better inspection methods, and more realistic part tolerances. By emphasizing how features relate to each other, manufacturers can better control the design, fit and function of parts. This process ensures that good parts pass inspection and bad parts are caught and rejected before they reach the customer.

Many geometric tolerances require strict inspection methods beyond the capabilities of basic calipers or micrometers. A coordinate measuring machine is best suited for inspecting most features and their relationships.

Commonly Used GD&T Terms & Symbols:

GD&T symbols are known universally as a method of specifying requirements without using notes or words on the drawing. The symbols are created to look like the requirement they identify. Symbols can specify things such as repetitive features, diameters, radius, spotfaces, and counterbores. Most of the symbols used between ASME and ISO are identical. However, there are a few differences. The chart below outlines some of the most common symbols and their appearance for both ISO and ASME. There are a few symbols that are used in the ASME Y14.5M, 1994 Standard that are being proposed for the ISO standards. The symbols marked with an "X" are new or revised from the previous Y14.5M, 1982 standard.

SYMBOL	ASME Y14.5M	ISO
FEATURE CONTROL FRAME	$\oplus \phi .030 \text{ (M) A B C}$	$\oplus \phi .030 \text{ (M) A B C}$
DIAMETER	ϕ	ϕ
SPHERICAL DIAMETER	$s\phi$	$s\phi$
AT MAXIMUM MATERIAL CONDITION	(M)	(M)
AT LEAST MATERIAL CONDITION	(L)	(L)
REGARDLESS OF FEATURE SIZE	(S)	NONE
PROJECTED TOLERANCE ZONE	(P)	(P)
X FREE STATE	(F)	(F)
X TANGENT PLANE	(T)	(T) (proposed)
X STATISTICAL TOLERANCE	(ST)	NONE
X RADIUS	R	R
X CONTROLLED RADIUS	CR	NONE
SPHERICAL RADIUS	SR	SR
BASIC DIMENSION (theoretically exact dimension in ISO)	50	50
X DATUM FEATURE		 or 
DATUM TARGET		
TARGET POINT	X	X
DIMENSION ORIGIN	$\phi \rightarrow$	$\phi \rightarrow$
REFERENCE DIMENSION (auxiliary dimension in ISO)	(50)	(50)
NUMBER OF PLACES	8X	8X
COUNTERBORE/SPOTFACE		 (proposed)
COUNTERSINK		 (proposed)
DEPTH/DEEP		 (proposed)
SQUARE		
ALL AROUND		NONE
DIMENSION NOT TO SCALE		
ARC LENGTH		
X BETWEEN		NONE
SLOPE		
CONICAL TAPER		
ENVELOPE PRINCIPLE	NONE (implied)	(E)

*MAY BE FILLED OR NOT FILLED

Figure 1-4. GD&T symbols.

The drawing below shows several of the most common symbols applied to a drawing.

TYPE OF TOLERANCE	CHARACTERISTIC	SYMBOL	DATUM REFERENCE
FORM	STRAIGHTNESS	—	INDIVIDUAL
	FLATNESS	/\	
	CIRCULARITY (ROUNDNESS)	○	
	CYLINDRICITY	∅	
PROFILE	PROFILE OF A LINE	⌞	INDIVIDUAL OR RELATED
	PROFILE OF A SURFACE	⌞⌞	
ORIENTATION	ANGULARITY	∠	RELATED
	PERPENDICULARITY	⊥	
	PARALLELISM	//	
LOCATION	POSITION	⊕	RELATED
	CONCENTRICITY	○○	
	SYMMETRY	≡	
RUNOUT	CIRCULAR RUNOUT	↗	RELATED
	TOTAL RUNOUT	↗↗	

Figure 1-5. Common tolerance symbols

In order to form a good understanding of GD&T, it is necessary to understand several basic terms and symbols. These terms are outlined below and will be discussed in more detail in subsequent sections:

- **Radius** - Two types of radii can be applied in GD&T. The radius (R) and the controlled radius (CR) are used to distinguish general applications (R) from those which require further restrictions (CR) on the radius shape.
- **Statistical Tolerancing Symbol** - Tolerances are sometimes calculated using simple arithmetic. These mathematic calculations are used to assign various features of a part to the total assembly. Statistical tolerancing can be applied to a part to increase tolerances and decrease the manufacturing cost. If a part is designated as being statistically toleranced, it must be produced using statistical process controls.
- **With Size** - When a feature is said to be “with size” it is associated with a size dimension. It can be cylindrical or spherical or possibly a set of two opposing parallel surfaces.
- **Without Size** - A feature that is without size is a plane surface where no size dimensions are indicated.
- **Feature Control Frames** - This is potentially the most significant symbol in any geometric tolerancing system. It provides the instructions and requirements for the feature to which it is related. Only one

requirement is contained in a feature control frame. Multiple features require multiple feature control frames. The first frame contains one of the 14 geometric characteristic, the second contains the total tolerance for the particular feature, the third and subsequent compartments of the feature control frame contain specified datums. It is important to note that the feature control frame controls the surface of a flat feature and the axis or median plane of a feature of size.

- **Material Condition Modifiers** - Often it becomes necessary to refer to a feature in its largest or smallest condition, or it may be necessary to refer to a feature regardless of feature size. These conditions are designated as the maximum material condition (MMC), the least material condition (LMC) and regardless of feature size (RFS). For example, MMC would be used to express the largest pin or the smallest hole. LMC would be used to describe the smallest pin or the largest hole. RFS might be used to show that a geometric tolerance applied to any increment of feature size of any feature within its size tolerance.

Datums and Features

All manufactured parts exist in two states; the **imaginary**, geometrically perfect design and the **actual**, physical, imperfect part. Before learning the principles of GD&T, you have to understand the difference between **datums** and **features**. The design of a part consists of many datums, each of which is a geometrically perfect form. These datums can be a straight line, a circle, a flat plane, a sphere, a cylinder, a cone, or a single point. Datums are imaginary. They are assumed to be “exact” for the purpose of computation or reference as established from actual features, or from which the location or geometric relationship of other features of a part may be established.

By utilizing datums for reference, the tolerances take on new meaning. Now, features can have a tolerance relationship to each other both in terms of form and also location. *Datums on an engineering drawing are assumed to be perfect.*

Features are the real, geometric shapes that make up the physical characteristics of a part. Features are the specific component portions of a part which may include one or more surfaces such as holes, screw threads, profiles, faces or slots. Features can be individual or they may be interrelated. As shown in Figure 1-6, any feature can have many imperfections and variations.

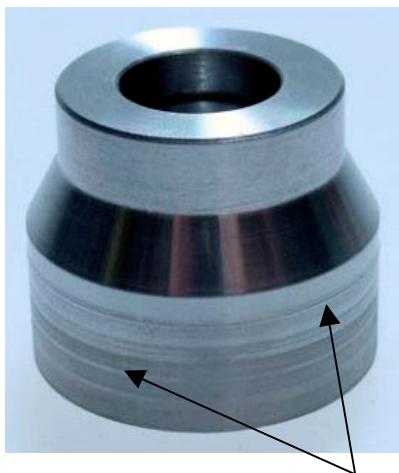


Figure 1-6. Imperfections and variations are visible on the surface of this part.

The **tolerances** in a design tell the inspector how much variance or imperfection is allowable before the part must be considered unfit for use. The tolerance is the difference between the maximum and the minimum limits on the dimensions of the part. Since parts are never perfect, a **datum feature** is used during inspection, to substitute for the perfect datum of the drawing. The datum feature can be a cylindrical hole, a straight edge, a flat surface, a corner, etc. You may see datum features simply referred to as datums.

The Datum Reference Frame

The GD&T system positions every part within a **datum reference frame** (DRF). The DRF is, by far, the most important concept in the geometric tolerancing system. It is the skeleton of the system, or the frame of reference to which all requirements are connected. The lack of understanding of datums is usually what makes the concepts of *position* and *profile* difficult for most people to grasp. Let's examine the Datum Reference Frame.

Engineering, manufacturing, and inspection all share a common three plane concept. These three planes are mutually perpendicular, perfect in dimension and orientation, and are exactly 90 degrees to each other. In geometric tolerancing we call this concept the **datum reference frame**.

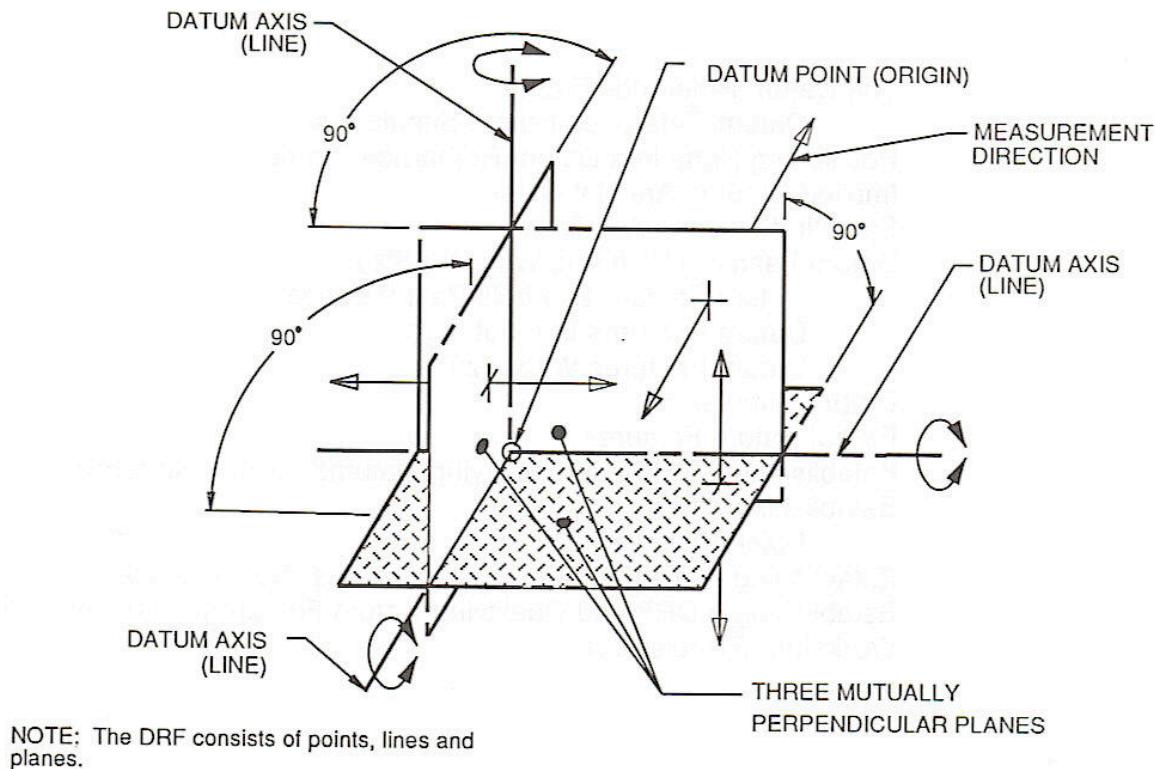


Figure 1-7. Datum Reference Frame

The DRF consists of a system of features as a basis for dimensions for manufacture and inspection. It provides complete orientation or a skeleton to which the part requirements are attached. The three main features of the datum reference frame are the **planes, axes, and points**.

The DRF consists of three imaginary planes, similar to the perpendicular X, Y, & Z axes of the traditional coordinate measuring system. Existing only in theory, the planes of a DRF make up a perfect, imaginary structure that is mathematically perfect. These imaginary planes must be mapped onto actual physical parts to permit inspection. All measurements in the datum plane originate from simulated datum planes and not the datum feature or part surface. Datums are listed in the feature control frame in the order that the part is loaded in the DRF.



Figure 1-8. A granite surface plate and angle plate.

A flat granite surface plate, such as one used with a Coordinate Measuring Machine (CMM) can substitute for an imaginary horizontal plane. The flat side of an angle plate, shown in Figure 1-8 above, substitutes as a second plane perpendicular to the surface plate.

The datum reference frame will accommodate both rectangular and cylindrical parts.

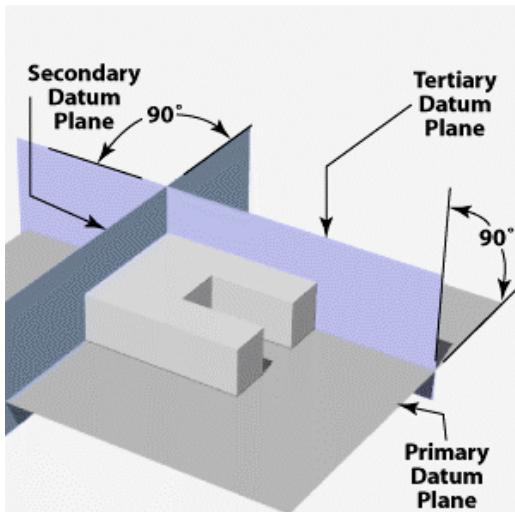


Figure 1-9. The datum reference frame with a rectangular part.

As shown above, a rectangular part fits into the corner represented by the intersection of the **three datum planes**. The datum planes are imaginary and therefore perfect. They are considered an absolute reference base. There is a primary, secondary, and tertiary datum plane for each part. The part will vary from these planes, even though the variation may not be visible to the naked eye.

A cylindrical part, seen in Figure 1-10, rests on the flat surface of the primary plane and the center of the cylinder aligns with the vertical **datum axis** created by the intersection of the other two planes.

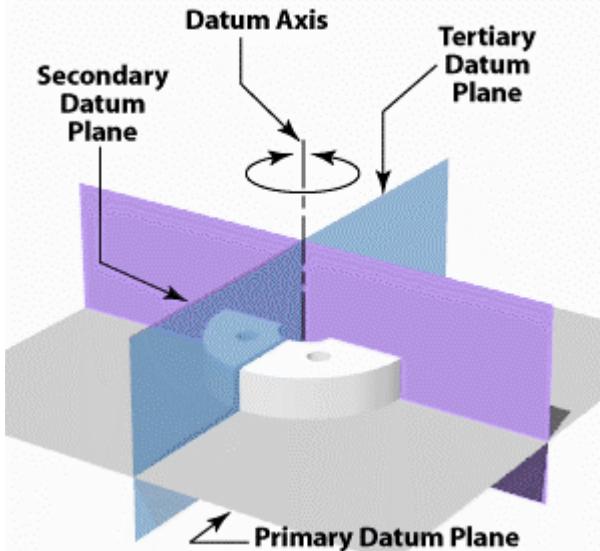


Figure 1-10. A cylindrical part is aligned at its center axis.

Implied Datums

The order of precedence in the selection and establishment of datums is very important. The picture below shows a part with four holes located from the edges with basic dimensions. The datums are not called out in the feature control frame. Rather, they are **implied** by the dimensions and by the edges from which those dimensions originate. Thus, we imply that these edges are the datums.

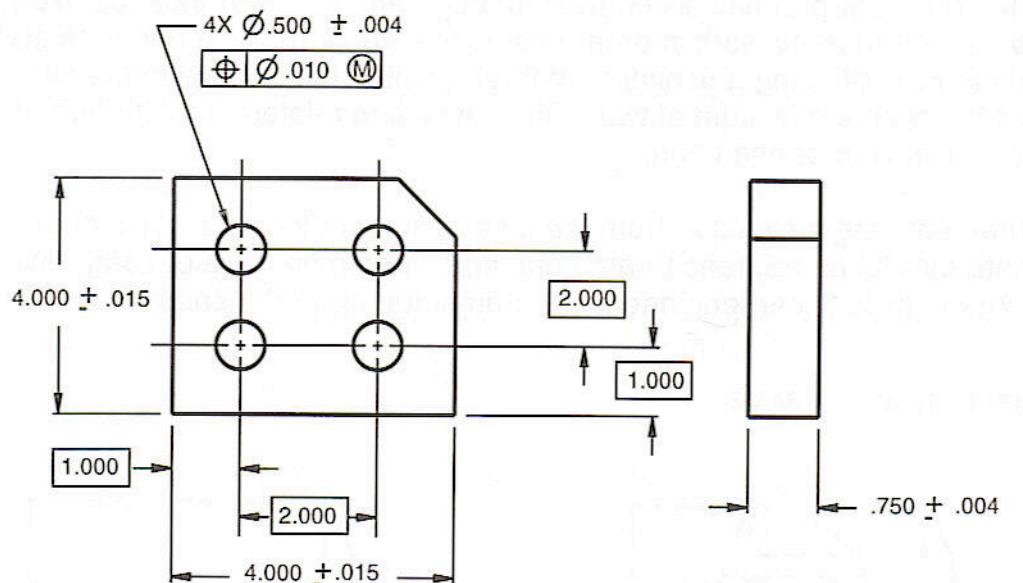


Figure 1-11. Angled block with holes

The problem with implied datums is that we do not know the order in which the datums are to be used. We know that the datum reference frame is perfect, but the parts are not perfect; we cannot build perfect parts. None of the edges are perfectly square, but the part is still good because it is within acceptable limits for size and squareness. The 90° corners also have a tolerance limit. In theory, even if the corners were out of perpendicularity by only .0001, the part would still “rock” back and forth in the “theoretically perfect” datum reference frame.

The Order of Datums

GD&T instructions designate which feature of the part will be the **primary datum**, **secondary datum** and **tertiary datum** references. In what order do we load an imperfect part into a perfect reference frame. Is the bottom edge the secondary datum? Is the large surface the primary datum? Which datum is the tertiary? It is not clear.

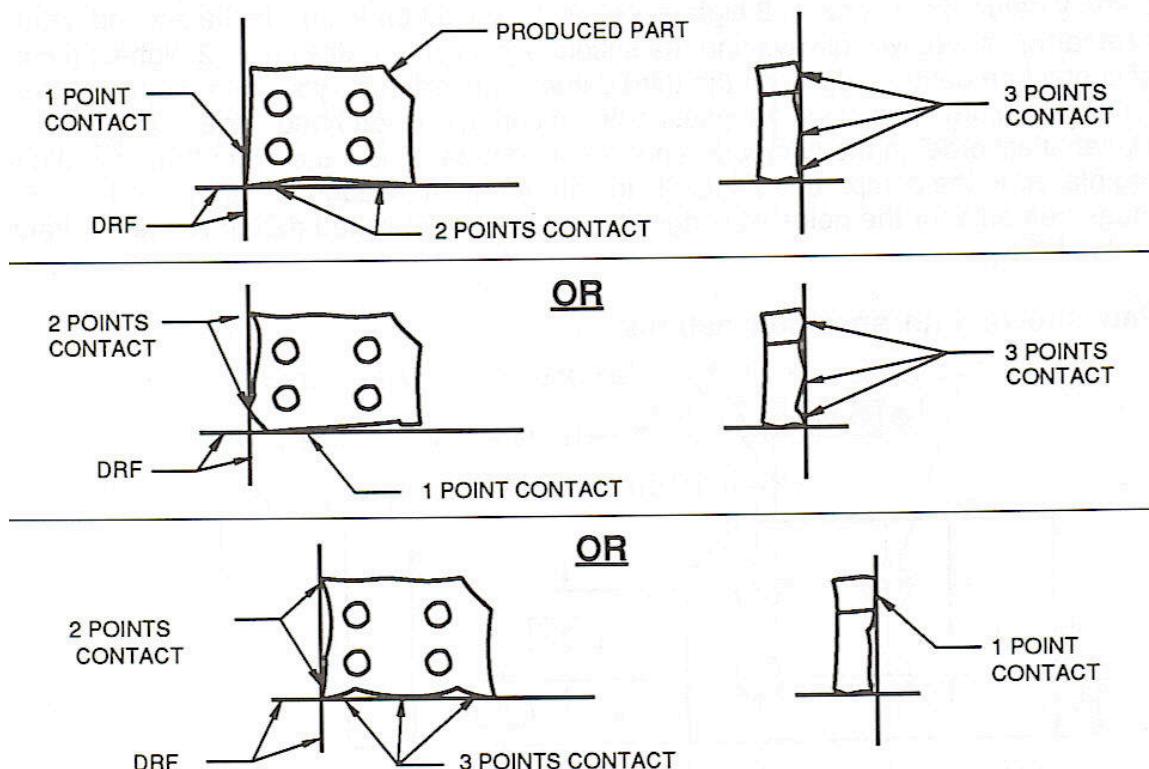


Figure 1-12. Different positions affect accuracy of measurements.

Engineering, manufacturing, and inspection may all have different interpretations of the order this part should be loaded into the DRF. This will result in different interpretations as to where the holes must lie on the part.

These first, second, and third datum features reflect an order of importance when relating to other features that don't touch the planes directly. Creating a datum reference frame is mandatory in order to achieve interchangeable parts.

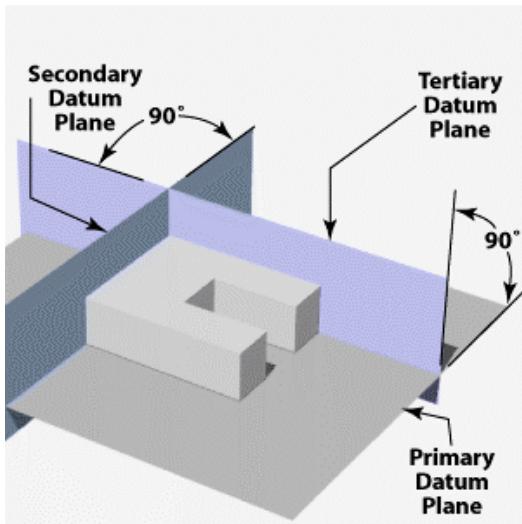


Figure 1-13. The datums describe the proper order for positioning parts.

These datum references are important because, as shown in Figure 1-14 below, the same part can be inspected in several different ways, each way giving a different measurement. Parts which have been produced by casting, forging, and molding may not have flat surfaces to establish datum planes.

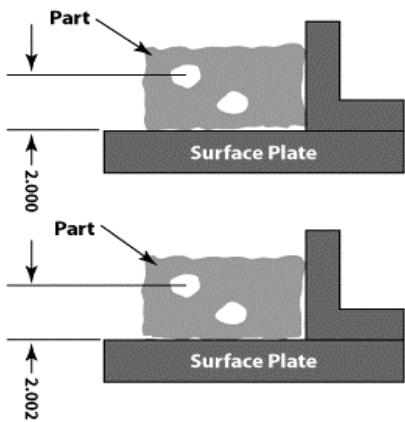


Figure 1-14. The same feature can be measured differently, depending on the positioning of the part.

Improper positioning could result in measurement errors unless the preferred positioning in the **inspection fixture** is indicated in the drawing. The primary datum feature must have at least three points of contact with the part and contacts the fixture first. The secondary has two points of contact and the tertiary has three points of contact with the part. This process correctly mirrors the datum reference frame and positions the part the way it will be fitted and used.

SECTION**2**

HOW THE GEOMETRIC SYSTEM WORKS

There are several important factors that must be understood in order to have a good understanding of GD&T. This section will introduce the geometric system and how it works. After completing this section, team members will have a basic understanding of the following objectives.

Objectives:

- Understand plus/minus tolerancing
- Understand geometric tolerance zones: position, profile, orientation, and form
- Compare geometric and limit tolerancing
- Understand Material Condition Modifiers: MMC, LMC, RFS
- Understand modifier rules
- Define the rules for screw threads, gears, and splines
- Understand when to use modifiers

Plus/Minus Tolerancing:

Plus/Minus Tolerancing, or Limit Tolerancing, is a two dimensional tolerancing system. When considering the bearing housing drawing pictured below, the dimensions on the drawing are perfect.

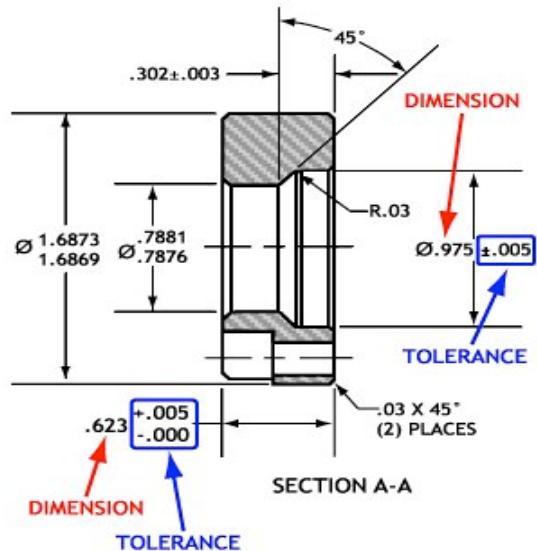


Figure 2-1: Plus & Minus tolerancing of a bearing housing.

When the product designer, using their drafting or CAD equipment, draws the part, the lines are straight, angles are perfect and the holes are perfectly round. However, when the part is produced in a manufacturing process, inevitably, there will be errors because nothing can be built to the perfect, imaginary dimensions of the drawing. There will be variations in the corners and surfaces of the part. The variations will be undetectable to the human eye, but can be picked up using precise measuring instruments such as a CMM machine. This creates a breakdown of sorts in the plus/minus tolerancing system. Parts are three dimensional and this tolerancing system is only two dimensional. It is simply high and low limits and is not oriented to specific datums.

In a plus/minus tolerancing system, the datums are implied and therefore, are open to varying interpretations. Plus/minus tolerancing works well when you are considering individual features. However, when you are looking at the relationship between individual features, plus/minus tolerancing is extremely limited. With the dawn of CAD systems and CMMs, it has become increasingly important to describe parts in three dimensional terms, and plus/minus tolerancing is simply not precise enough.

Geometric Tolerance Zones:

A geometric tolerancing system establishes a coordinate system on the part and uses limit tolerancing to define the form and size of each feature. Dimensions are theoretically exact and are used to define the part in relation to the coordinate system. The two most common geometric characteristics used to define a feature are position and profile of the surface.

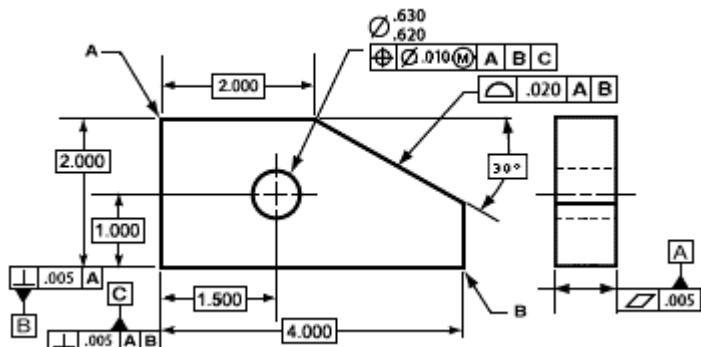


Figure 2-2: Drawing with feature control frame.

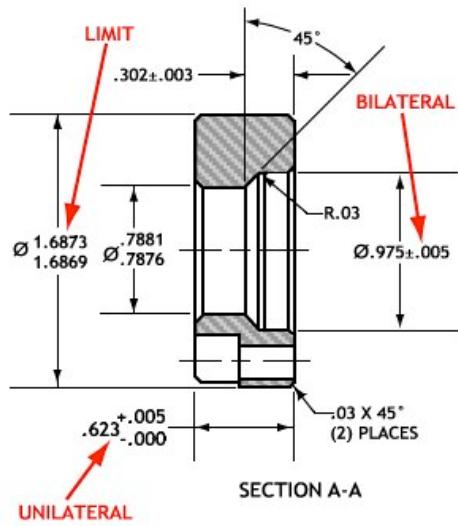
Referring to the angle block above, **position tolerance** is located in the first block of the feature control frame. It specifies the *tolerance* for the location of the hole on the angle block. The “boxed dimensions” define what the exact location of the center of the hole should be. 1.000×1.500 . The position tolerance block states that the center of the hole can vary **no more than .010** inches from that perfect position, under Maximum Material Condition. The position tolerance zone determines the ability of the equipment used to produce the part within limits. The tighter the position tolerance is, the more capable the equipment. Position tolerance is merely a more concise manner in which to communicate production requirements.

Profile tolerance (half-circle symbol) is specified in the second block of the feature control frame. It is used to define a three dimensional uniform boundary that the surface must lie within. The tightness of the profile tolerance indicates the manufacturing and verification process. Unimportant surfaces may have a wide tolerance range, while important surfaces will have a very tight profile tolerance range.

Form tolerance refers to the flatness of the part while **orientation** tolerance refers to the perpendicularity of the part specified on the datums. These two tolerances are chosen by the designer of the part in order to match the functional requirements of the part. Form and orientation tolerances control the instability of the part.

Geometric vs. Bilateral, Unilateral & Limit Tolerancing:

The difference between **geometrically tolerated** parts and **limit tolerated** parts is quite simple. *Geometric tolerances* are more precise and clearly convey the intent of the designer of the part using specified datums. It uses basic dimensions which are theoretically exact and have zero tolerance. *Limit tolerancing*, on the other hand, produces a part that uses implied datums and larger, less exact tolerances that fall into three basic categories - Bilateral, Unilateral and Limit dimensions.



- **Bilateral tolerances** specify the acceptable measurements in two opposite directions from a specified dimension.
- **Unilateral tolerances** define the acceptable range of measurements in only one direction from a given dimension.
- **Limit dimensions** give the acceptable measurements within two absolute dimensions.

Figure 2-3: Different types of tolerances

Material Condition Modifiers:

Material Condition Modifiers are used in geometric tolerancing. They have a tremendous impact on the stated tolerance or datum reference. These modifiers can only be applied to features and datums that specify size. Examples of these features are holes, slots, pins, and tabs. If modifiers are applied to features that are without size, they have no impact. However, if no modifier is specified in the feature control frame, the default modifier is RFS or *regardless of feature size*. There are three material condition modifiers.

- **Maximum Material Condition - (MMC)** - This modifier gives room for additional position tolerance of up to .020 as the feature departs from the maximum material condition. This is a condition of a part feature wherein, it contains the maximum amount of material, or the minimum hole-size and maximum shaft-size.
- **Least Material Condition - (LMC)** - This is the opposite of the MMC concept. This is a part feature which contains the least amount of material, or the largest hole-size and smallest shaft-size.
- **Regardless of Feature Size - (RFS)** - This is a term used to indicate that a geometric tolerance or datum reference applies at any increment of size of the feature within its size tolerance.

Bonus Tolerance

Material condition modifiers give inspectors a powerful method of checking shafts and holes that fit together. Both MMC and LMC modifiers allow for **bonus tolerance**. The hole in Figure 2-4 has a certain position tolerance, but at MMC (maximum material condition), the hole is smaller, tighter, and exhibits a perfect cylindrical form.

Bonus tolerance is possible with both the MMC and LMC modifiers, but if the RFS modifier is specified, the stated tolerance applies and, in all cases, there is no bonus tolerance.

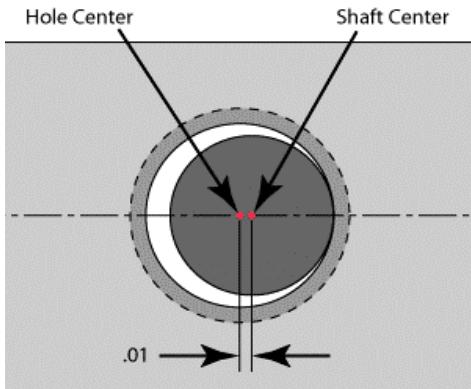


Figure 2-4. MMC of the hole presents the tightest fit.

As more material is removed from around the hole, the space is larger and provides a looser fit for the shaft. Therefore, the position tolerance for the hole can be increased, and both the shaft and the hole will still fit. This increased tolerance is called the **bonus tolerance** of the hole and changes as the size of the hole increases.

MMC decreases the cost of manufacturing and inspection and is very effective for parts that require assembly.

Certain fits require increased precision and greatly affect the part's function. The RFS modifier is stricter, but is necessary for those parts.

The Feature Control Frame

GD&T instructions contain a large amount of information about the features being described. Each feature is given a **feature control frame** that reads from left to right, like a basic sentence.

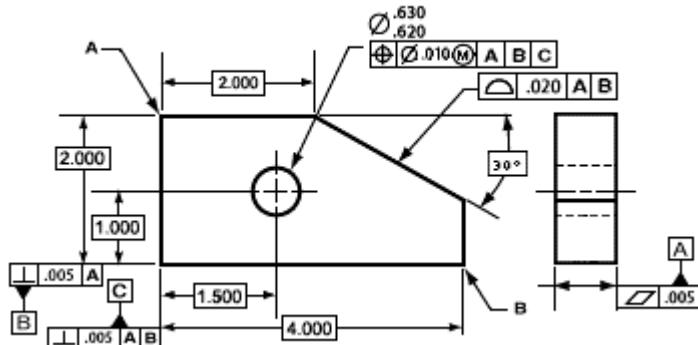


Figure 2-5. Each part feature contains a feature control frame.

The feature control frame organizes the GD&T instructions into a series of symbols that fit into standardized compartments.

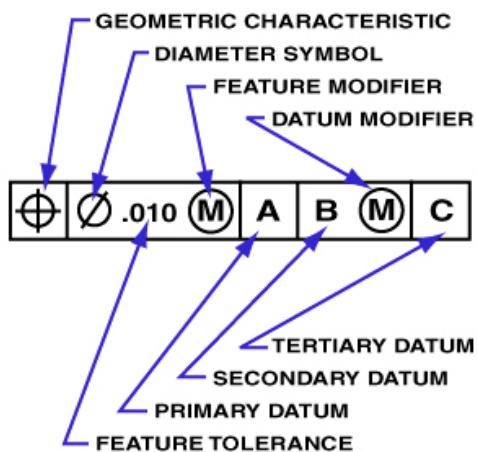
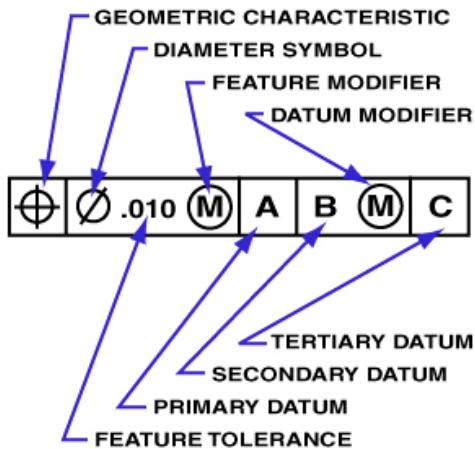


Figure 2-6. Standardized information in feature control frame.

- The first compartment defines the geometric characteristic of the feature, using one of the 14 standard geometric tolerance symbols (⊕ means “position”). A second feature control frame is used if a second geometric tolerance is needed.
- The second compartment contains the entire **tolerance for the feature**, with an *additional diameter symbol* to indicate a cylindrical or circular tolerance zone. No additional symbol is needed for parallel lines or planes. If needed, material condition modifiers would also appear in the second compartment.
- The third compartment indicates the **primary datum** which locates the part within the **datum reference frame**. Every related tolerance requires a primary datum but independent tolerances, such as form tolerances, do not.



- The fourth and fifth compartments contain the secondary and tertiary datums. Depending on the geometric tolerance and the function of the part, secondary and tertiary datums may not be necessary.

The primary, secondary and tertiary datums do not have to be designated as A,B, or C. A part can have several datums and may be labeled as D, E & F or G,H, & I, etc. Whatever its designation, the most important factor is its placement in the feature control frame. If "G" is shown in the third compartment, that makes it the primary datum and is therefore the first plane to be aligned with the DRF.

SECTION

3

STRAIGHT AND CYLINDRICAL TOLERANCES

Objectives

- List the major categories of geometric tolerances.
- Explain the straightness tolerance.
- Define the flatness tolerance.
- Explain the circularity tolerance.
- Define the cylindricity tolerance.
- Explain the profile of a line tolerance.
- Describe the profile of a surface tolerance.

Types of Tolerances

Part features are defined using a range of GD&T tolerance types. They are divided into the following five major categories:

- The **form tolerance** includes flatness, circularity, cylindricity, and straightness. They define features individually and are relatively simple.
- The **profile tolerance** includes the profile of a surface and the profile of a line. These two powerful tolerances control several aspects of a feature.
- The **orientation tolerances** define parallelism, perpendicularity, and angularity.
- The **location tolerances** determine concentricity, symmetry and position, with position being the most common.
- **Runout tolerances** are used only on cylindrical parts. They are *circular runout* and *total runout*.

TYPE OF TOLERANCE	CHARACTERISTIC	SYMBOL	DATUM REFERENCE
FORM	STRAIGHTNESS	—	INDIVIDUAL
	FLATNESS	/\	
	CIRCULARITY (ROUNDNESS)	○	
	CYLINDRICITY	◎	
PROFILE	PROFILE OF A LINE	⌞	INDIVIDUAL OR RELATED
	PROFILE OF A SURFACE	⌞⌞	
ORIENTATION	ANGULARITY	∠	RELATED
	PERPENDICULARITY	⊥	
	PARALLELISM	//	
LOCATION	POSITION	⊕	
	CONCENTRICITY	◎◎	
	SYMMETRY	≡≡	
RUNOUT	CIRCULAR RUNOUT	↗	
	TOTAL RUNOUT	↗↗	

Each type of tolerance is either an **individual tolerance**, a **related tolerance**, or both.

An individual tolerance is not related to a datum.

A related tolerance must be compared to one or more datums.

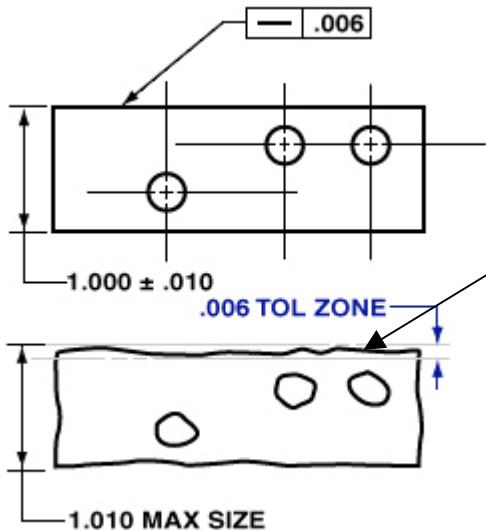
These tolerances are explained in more detail in the following lessons.

Figure 3-1. The five categories of geometric tolerances.

Straightness and Flatness

Straightness and **flatness** are two types of form tolerances that are relatively simple and control the shape of feature. Since they are form tolerances, flatness and straightness define a feature independently.

Straightness is a **two-dimensional tolerance**. An edge must remain within two imaginary parallel lines in order to meet a straightness tolerance. The distance between these two lines is determined by the size of the specified tolerance. Many objects can have a straightness tolerance. Most rectangular parts have one, but the edge or center axis of a cylinder may also have a straightness requirement.



The variations in straightness of this part are greatly exaggerated to illustrate how the surface must stay within the tolerance zone. No reasonable machining process would produce a part that looked like this, but the actual variations may not be visible to the naked eye.

Figure 3-2. Straightness variations are limited between two, imaginary parallel lines.

Flatness is a three-dimensional version of the straightness tolerance. Instead of using just two lines, flatness requires a surface to be within two imaginary, perfectly flat, perfectly parallel planes. Like a piece of lunchmeat between two slices of bread, only the surface of the part, not the entire thickness, is referenced to the planes.

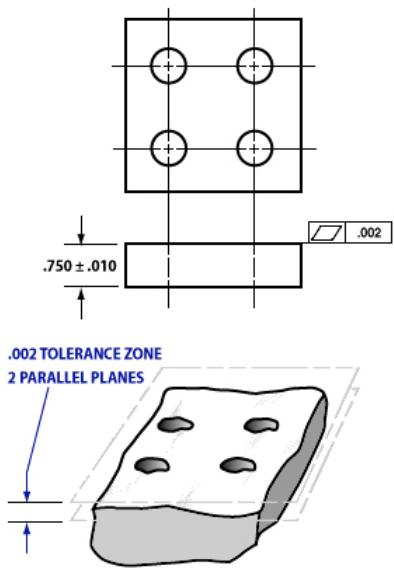


Figure 3-3. Flatness is limited between two planes.

If a flat surface is to be used as the primary datum, its tolerance must be specified in the drawing.

Circularity and Cylindricity

In addition to straightness and flatness, **circularity** (often called roundness) and **cylindricity** complete the list of “form tolerances”. These tolerances define the shape of round features and they are referenced independently from any other part feature.

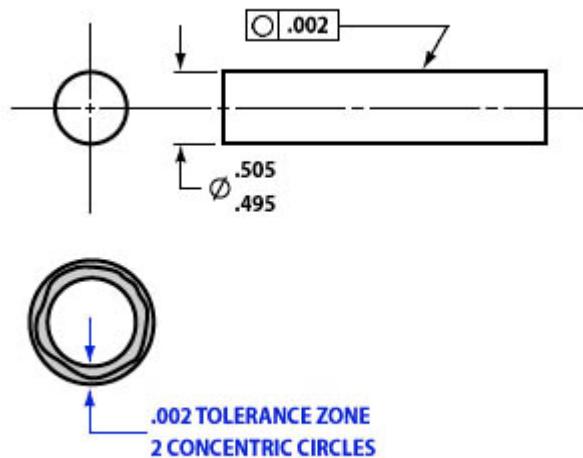


Figure 3-4. Circularity defines roundness between two concentric circles.

The two-dimensional tolerance of circularity, although most often used on cylinders, can also apply to cones and spheres. Circularity demands that any two-dimensional slice, or **cross-section**, of a round feature must stay within the tolerance zone created by two concentric circles. Most inspectors will check multiple cross sections, but each section must meet the tolerance on its own.

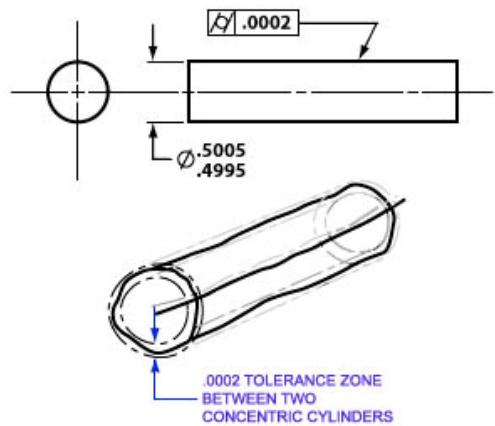


Figure 3-5. The cylindricity tolerance zone is created by locating an imaginary concentric cylinder inside a larger cylinder of the same length.

The inner and outer walls of a section of steel pipe are a good example of the cylindricity tolerance zone. Although the pipe could not be used to verify cylindricity, the thickness of the pipe wall is a good illustration of the zone.

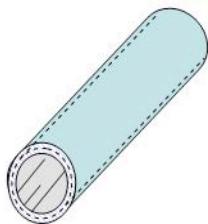


Figure 3-6. The thickness of the wall of a pipe represents the cylindricity tolerance zone.

All cross sections of the cylinder feature must be measured together, so the cylindricity tolerance is only applied to cylinders. No other shape would fit the zone.

Circularity and cylindricity cannot be checked by simply measuring various diameters with a micrometer. The part must be rotated in a high-precision spindle of a machine that measures roundness. The best method would be to inspect the part using a **coordinate measuring machine (CMM)**.

Profile of a Line and Surface

The two versions of profile tolerance are the **profile of a line** and the **profile of a surface**. These two powerful tolerances can be used to control features such as cones, curves, flat surfaces, irregular surfaces, or cylinders. Each of these features has a profile. The profile is simply an outline of the part feature in one of the datum planes. The line and surface profile both compare the actual profile to the imaginary, “perfect” profile specified in the design. They control the orientation, location, size, and form of the feature.

The profile of a line, seen here in Figure 3-7, is a two-dimensional tolerance that controls any straight line or contour. It requires the profile of the feature to fall within two imaginary parallel lines that follow the actual profile of the feature. The tolerance is indicated by the space between the two lines.

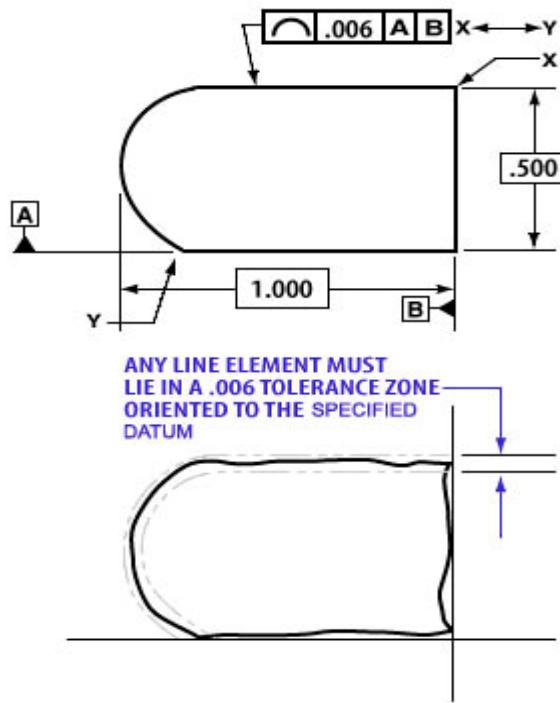


Figure 3-7. Profile of a line.

The **profile of a surface** is a three-dimensional version of the line profile. It is often applied to complex and curved contour surfaces such as aircraft and automobile exterior parts. As shown in Figure 3-8 below, the tolerance specifies that the surface must remain within two three-dimensional shapes. These shapes follow the true profile of the feature.

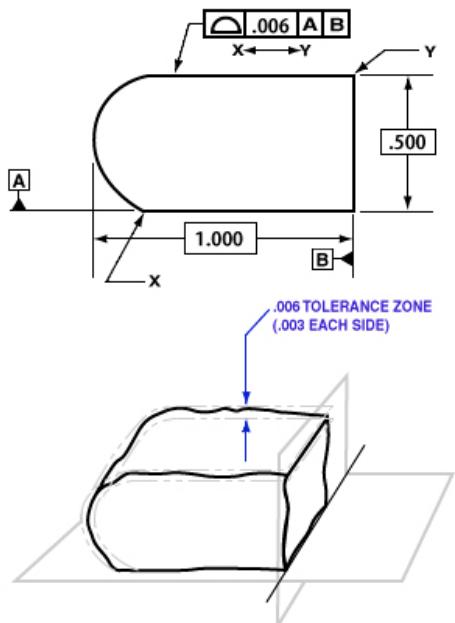


Figure 3-8. The three-dimensional surface profile.

SECTION**4****ORIENTATION AND
LOCATION TOLERANCES****Objectives**

- Explain the angularity tolerance.
- Define the perpendicularity tolerance.
- Explain the parallelism tolerance.
- Describe the position tolerance.
- Explain the concentricity tolerance.
- Define the symmetry tolerance.
- Explain the circular runout tolerance.
- Describe the total runout tolerance.

Angularity, Perpendicularity, and Parallelism

So far, we have dealt primarily with the flat, straight features of parts. Now, we will examine other relationships that surfaces have with each other. Three of these relationships are called orientation tolerances. They are **angularity**, **perpendicularity**, and **parallelism**. The following three figures show how these tolerances define the angle that must be formed. They define how one feature is oriented to another by specifying how one or more datums **relate** to the primary “toleranced” feature, and so, are called **related tolerances**.

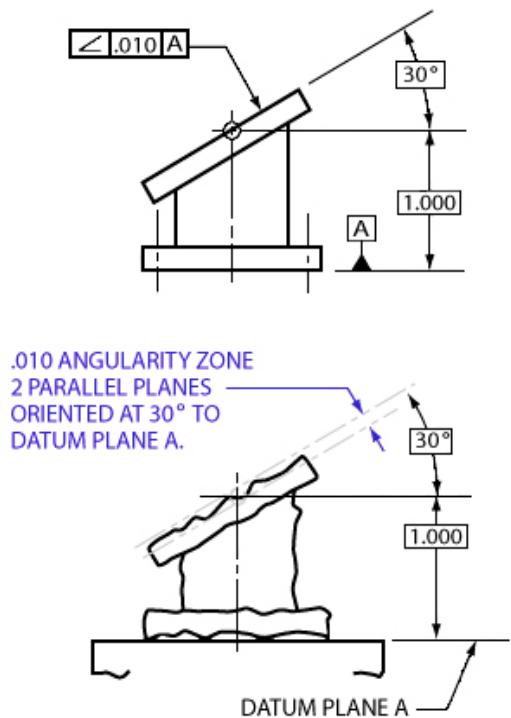


Figure 4-1. Angularity defines the specified angle between a related datum and the angled feature.

The angularity tolerance is three-dimensional and the shape of the tolerance zone depends on the shape of the feature. If applied to a flat surface, the tolerance zone becomes two imaginary planes, parallel to the ideal angle, and spaced apart at the prescribed distance. If angularity is applied to a hole, it is referenced to an imaginary cylinder that exists around the ideal angle and the center axis of the hole must stay within that cylinder.

Perpendicularity and **parallelism** are also three-dimensional tolerances and use the same tolerance zones as angularity. The difference is that parallelism defines two features that must remain parallel to each other, while perpendicularity specifies a 90-degree angle between features.

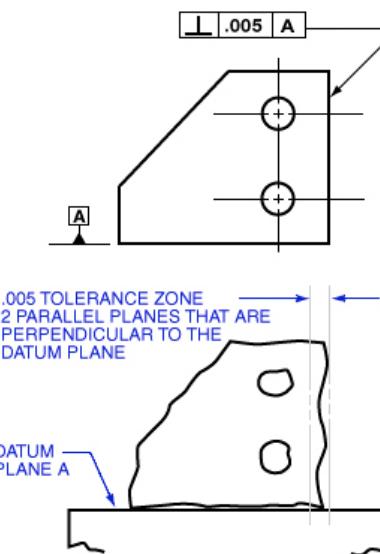
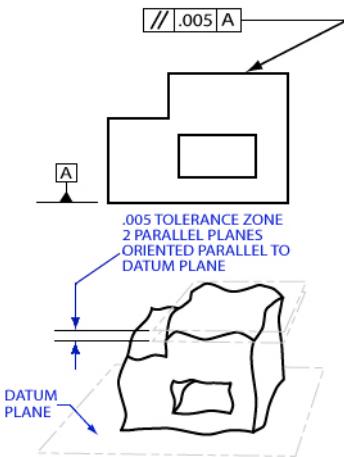


Figure 4-2. Perpendicularity defines variation from a 90-degree angle.



Flatness and parallelism are sometimes confused with each other. However, flatness is not related to another datum and it looks at the feature independently.

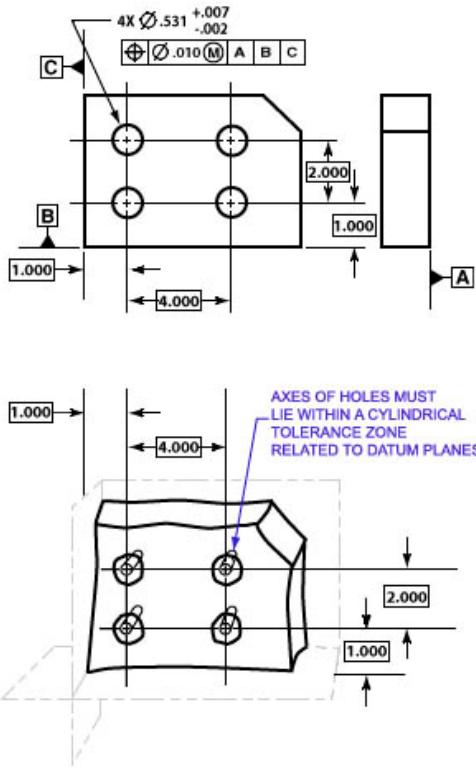
Parallelism relates the inspected feature to another datum plane.

Whenever an orientation tolerance is applied to a flat surface, it indirectly defines the flatness of the feature.

Figure 4-3. Parallelism defines a feature that is parallel to the reference datum plane.

Position

The **position** tolerance is likely one of the most common of the location tolerances. The ideal, exact location of the feature is called the **true position**. The actual location of the feature is compared to the ideal true position and involves one or more datums to determine where that true position should be. The position tolerance is also a three-dimensional, related tolerance.



As you can see from Figure 4-4, position has nothing to do with the size, shape, or angle of a feature, but rather “where it is”. In the case of the holes shown here, the tolerance again involves the center axis of the hole and must be within the imaginary cylinder around the intended true position of the hole.

If the tolerated feature is rectangular, the tolerance zone involves two imaginary planes at a specified distance from the ideal true position.

The position tolerance is easy to inspect and is often done with just a functional gage, like a “go/no-go” gage.

Figure 4-4. Position tolerance defines deviation from the ideal true position.

Concentricity and Symmetry

Concentricity is another one of the location tolerances and is three-dimensional, but unlike position, it is not commonly measured. It too, relates a feature to one or more other datum features. As in Figure 4-5 below, concentricity is used to compare two or more cylinders and ensure that they share a common center-axis.

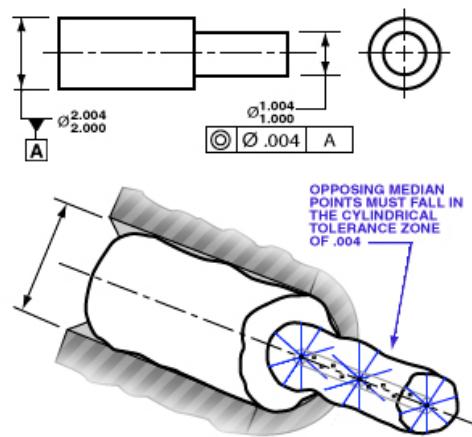
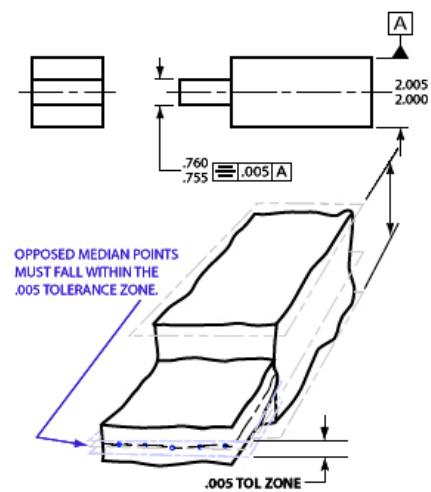


Figure 4-5. Concentricity measures the median points of multiple diameters within the cylindrical tolerance zone.

The process must be performed several times and each median point must independently fall within the center axis tolerance zone.

Symmetry is much like concentricity, except that it controls rectangular features and involves two imaginary flat planes, much like parallelism.



Both symmetry and concentricity tolerances are difficult to measure and increase the cost of inspection.

However, when a certain characteristic of a part, such as balance, is a primary concern, these tolerances are very effective.

Figure 4-6. Median points between two planes control symmetry.

Circular and Total Runout

Runout tolerances are three-dimensional and apply only to cylindrical parts, especially parts that rotate. Both circular and total runout reference a cylindrical feature to a center datum-axis, and simultaneously control the location, form and orientation of the feature.

A part must be rotated to inspect **circular runout**. A calibrated instrument is placed against the surface of the rotating part to detect the highest and lowest points. As shown in Figure 4-7, the surface must remain within two imaginary circles, having their centers located on the center axis.

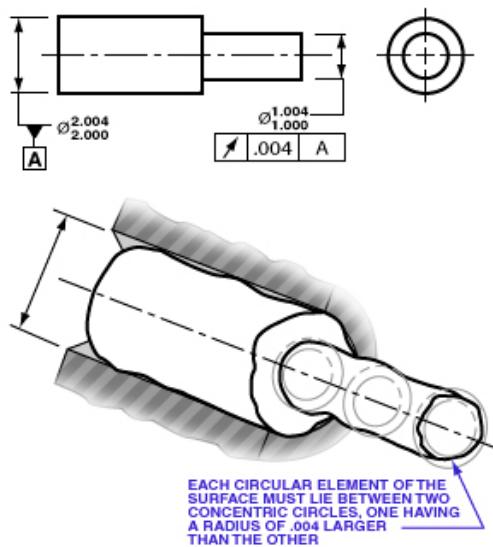


Figure 4-7. Circular runout measures a series of circular cross sections.

Total runout involves tolerance control along the entire length of, and between, two imaginary cylinders, not just at cross sections.

By default, parts that meet the total runout tolerance automatically satisfy all of the circular runout tolerances.

Runout tolerances, especially total runout, are very demanding and present costly barriers to manufacturing and inspection.

Figure 4-8. Total runout controls the features along the full length of the surface.

SECTION**5**

General Rules of GD & T

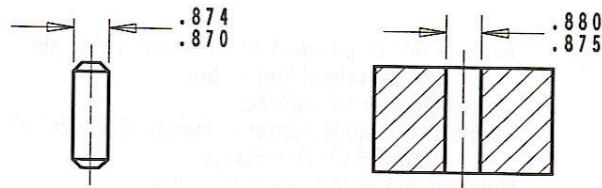
Geometric dimensioning and tolerancing is based on certain fundamental rules. Some of these follow from standard interpretation of the various characteristics, some govern specification, and some are General Rules applying across the entire system.

Rule #1 – Limits of Size:

Rule #1 is the Taylor Principle, attributed to William Taylor who in 1905 obtained a patent on the full form “go-gage”. It is referred to as Rule #1 or “Limits of Size” in the Y14.5M, 1994 standard. The Taylor Principle is a very important concept that defines the size and form limits for an individual feature of size. In the international community the Taylor Principle is often called the “envelope principle”.

**EXTREME VARIATIONS OF SIZE AND FORM ALLOWED ON AN INDIVIDUAL FEATURE OF SIZE
BY THE LIMITS OF SIZE**

THIS ON THE DRAWING



MEANS THIS

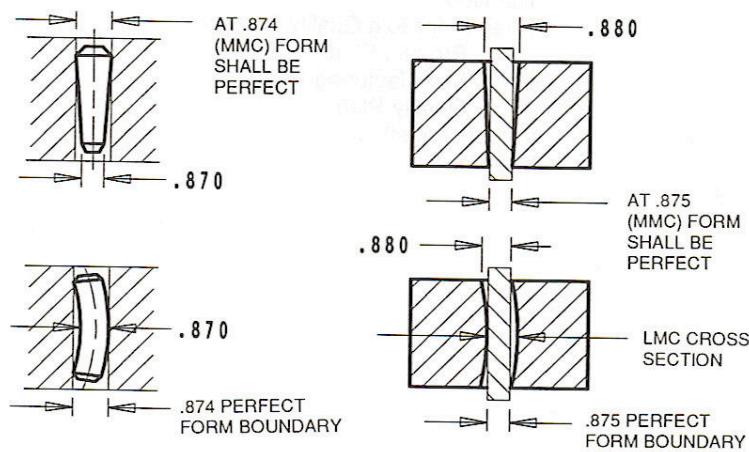


Figure 5-1: Variations of size.

Figure 5-1 illustrates the variations in size that are possible while still keeping within the perfect boundaries. The limits of size define the “size” (outside measurements) as well as the “form” (shape) of a feature. The feature may vary within the limits. That is, it may be bent, tapered, or out of round, but if it is produced at its maximum material condition, the form must be perfect.

a. Individual Feature of Size:

When only a tolerance of size is specified, the limits of size of an individual feature prescribe the extent to which variations in its geometric form as well as size are allowed.

b. Variation of Size:

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

c. Variation of Form:

The form of an individual feature is controlled by its limits of size to the extent prescribed in the following paragraph and illustration.

1. The surface or surfaces of a feature shall not extend beyond a boundary (envelope) of perfect form at Maximum Material Condition (MMC). This boundary is the true geometric form represented by the drawing. No variation is permitted if the feature is produced at its MMC limit of size. (*Plain English- If the part is produced at Maximum Material Condition, it shall not be bigger than the perfect form of the drawing.*)
2. Where the actual size of a feature has departed from MMC toward LMC, a variation in form is allowed equal to the amount of such departure.
3. There is no requirement for a boundary of perfect form at LMC. Thus, a feature produced at LMC limit of size is permitted to vary from true form to the maximum variation allowed by the boundary of perfect form at MMC.

Rule #2 - Applicability of MMC, LMC, & RFS :

In the current ASME Y14.5M-1994, Rule # 2 governs the applicability of modifiers in the Feature Control Frame. The rule states that “Where no modifying symbol is specified with respect to the individual tolerance, datum reference, or both, then RFS (Regardless of Feature Size) automatically applies and is assumed. Since RFS is implied, it is not necessary to include the symbol. Therefore, the symbol  has been eliminated from the current standard. MMC and LMC must be specified where required.

Rule #3 - Eliminated:

Rule #4 & #5 - Eliminated:

What is Virtual Condition ?

Depending upon its intended purpose, a feature may be controlled by tolerances such as form, size, orientation and location. The collective (total) effects of these factors determine the clearances between mating parts and they establish gage feature sizes. The collective effect of these factors is called “virtual condition”.

Virtual condition is a constant boundary created by the total effects of a “size” feature based on its MMC or LMC condition and the geometric tolerance for that material condition.

For example, the pin in the illustration below has two virtual sizes.

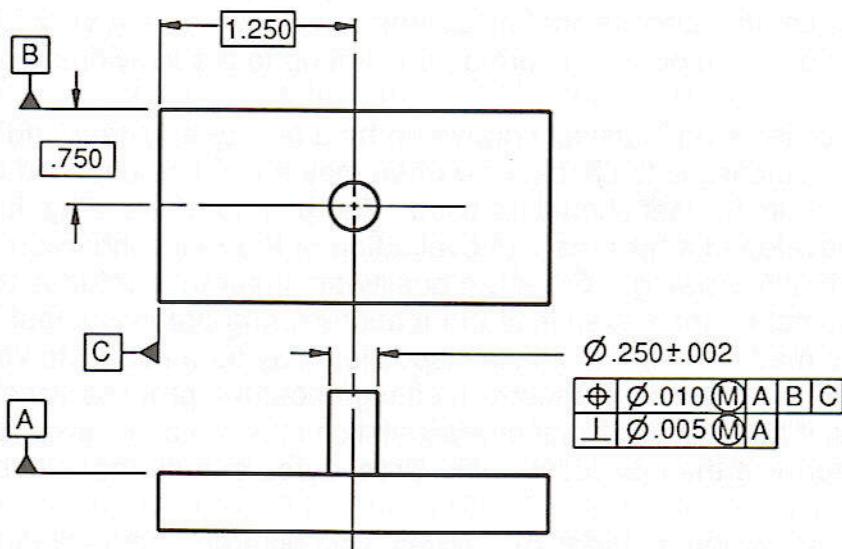


Figure 5-2. Alignment Pin on a flat block.

The size tolerance for the pin ($.250 \pm .002$) and the location and perpendicularity tolerances listed in the Feature Control Frame combine to create two possible virtual sizes. First, regardless of its position or angle, the pin must still lie within the $.002$ boundary specified for its width. However, the tolerance for perpendicularity allows a margin of $.005$. So, if the part were produced at MMC to $.252$ and it deviates from perpendicularity by the $.005$ allowed, the total virtual size of the pin can be considered to be $.257$ in relation to datum A.

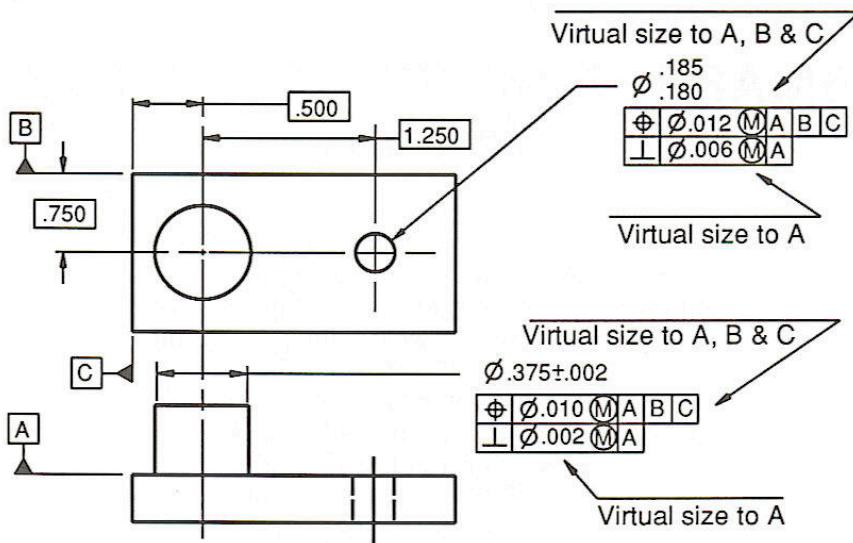
Second, the position tolerance of $.010$ combined with the size tolerance of $.002$ would produce a virtual size of $.262$ in relation to datums A, B and C.

This means that an inspection gage would have to have a hole of $.262$ to allow for the combined tolerances, even though the pin can be no more than $.252$ diameter. Therefore, three inspections would be necessary in order to check for size, perpendicularity, and location.

Virtual Size of a Hole

When calculating the virtual size of a hole, you must remember the rule concerning Maximum Material Condition (MMC) and Least Material Condition (LMC) of holes. Recall that when machining a hole, MMC means the “most material that can remain in the hole”. Therefore, a hole machined at MMC will be smaller and a hole machined at LMC will be larger. It is important to read the Feature Control Frame information carefully to make sure you understand which feature is specified and what material conditions are required.

On the drawing below, calculate the virtual sizes for the identified features.



SECTION

6

Limit Tolerancing (Plus / Minus) vs. Geometric Tolerancing

Limit tolerancing is rather restricted when it comes to inspecting all of the features of a part and their relationship to each other. Plus/minus tolerancing is basically a two dimensional tolerancing system, or a caliper / micrometer type measurement. It works well for individual features of size but does not control the relationship between individual features very well. Limit tolerancing can be used, but it is important to remember its limitations. Consider the limit tolerancing applied to this angle block drawing.

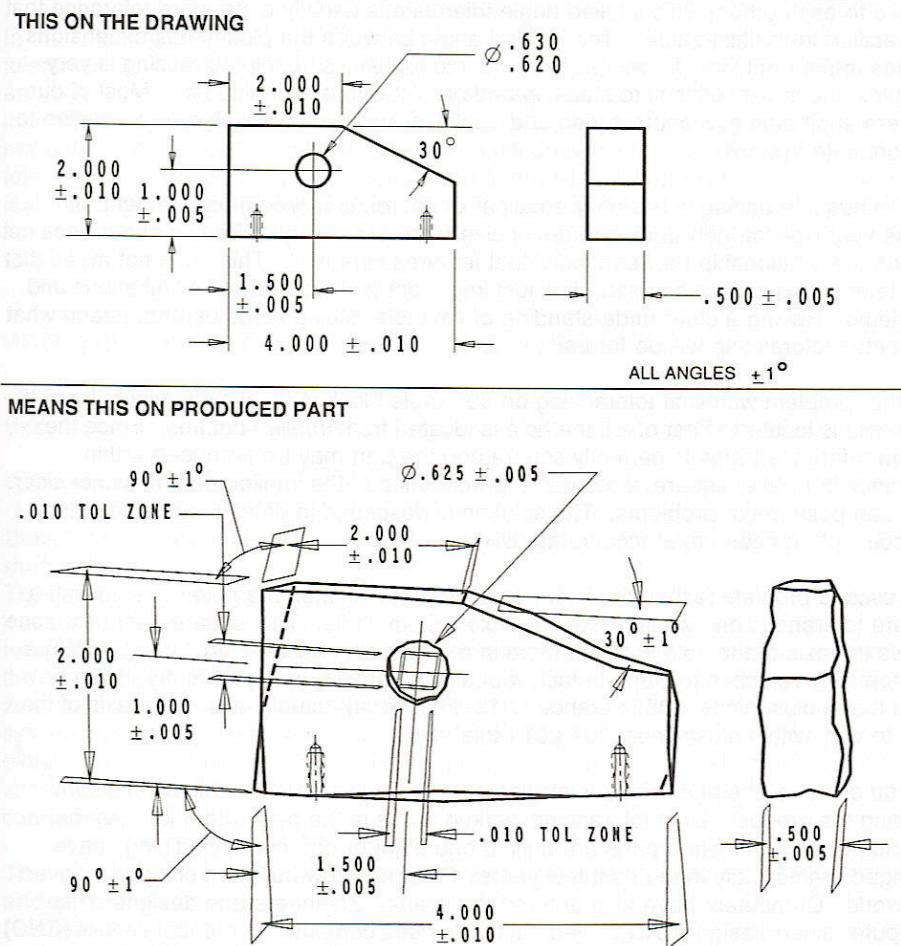


Figure 6-1. Angle block using only plus/minus tolerancing.

The product designer uses CAD equipment to draw the picture very straight and square (top drawing). The part is then produced in the manufacturing process and because of the imperfection of the equipment, will have inherent

variations. Visually, the block will look straight and square. The variations will be so small that they are undetectable with the human eye. However, when the parts are inspected using precision measuring equipment such as a CMM, the angle block starts to look like the bottom drawing (greatly exaggerated).

The block is not square in either view. The surfaces are warped and not flat. The hole is not square to any surface and it is not round. It is at this point that the limit system of tolerance breaks down. Plus/minus tolerances are two dimensional; the actual parts are three dimensional. Limit tolerances usually do not have an origin or any location or orientation relative to datums. The datums are usually implied. Most of our modern engineering, manufacturing and quality systems all work square or relative to a coordinate system. Parts must be described in a three dimensional mathematical language to ensure clear and concise communication of information relating to product definition. That is why we need geometric tolerancing.

Applying Geometric Tolerancing

In the following illustration the same angle block is now shown with geometric

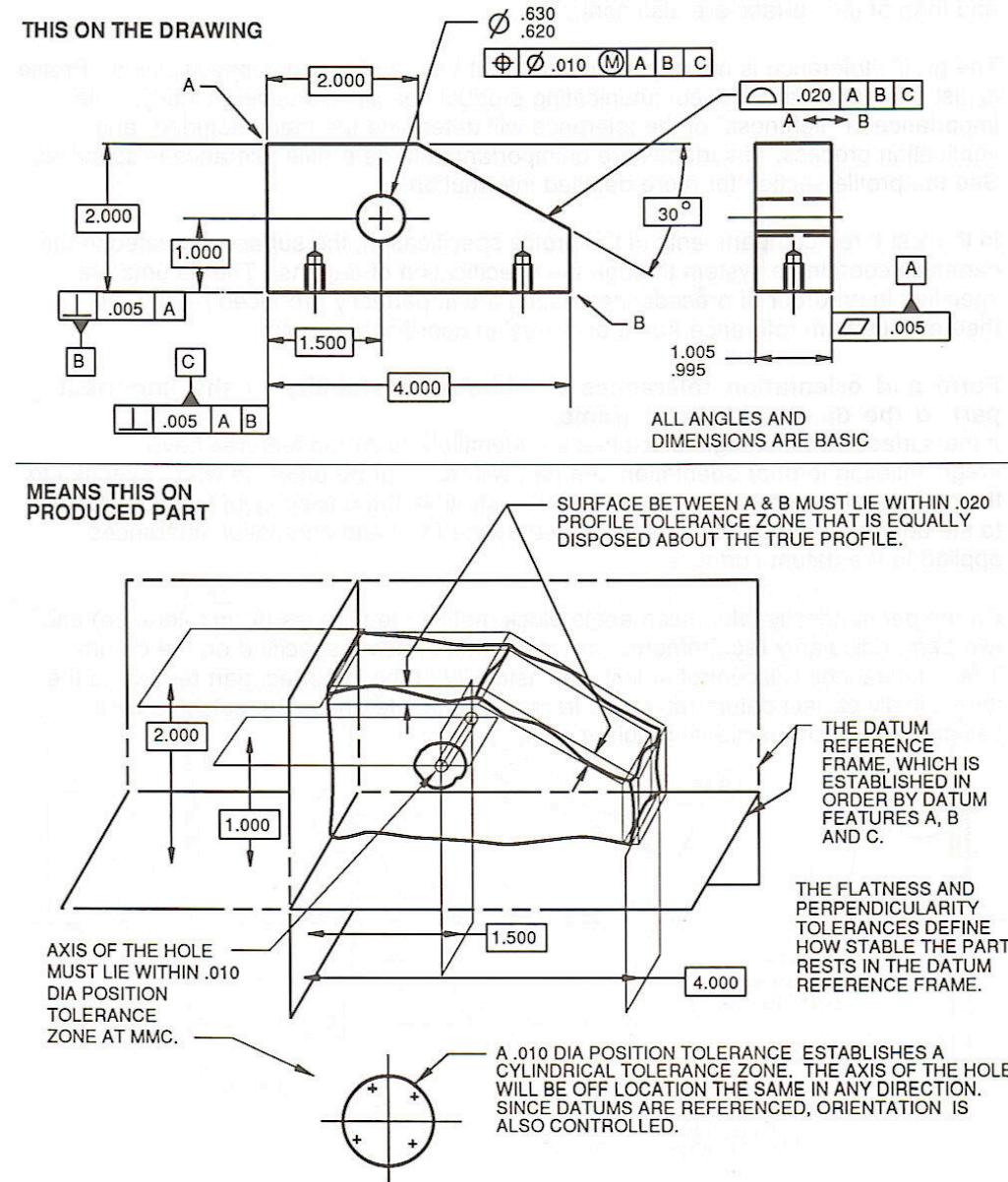


Figure 6-2. Angle block using geometric tolerancing.

Notice that datums A, B and C have been applied to features on the part establishing a X, Y and Z Cartesian coordinate system.

Geometric tolerancing is a very clear and concise three dimensional mathematical language for communicating product definition. In the example in Figure 6-2, the angle block hole and surfaces are clearly defined with geometric tolerancing. Form and orientation tolerances establish the stability of the imperfect part to the datum reference frame.

A close-up look at the angle block shows how the features are controlled. For example, the hole location is controlled by the feature control frame shown below.

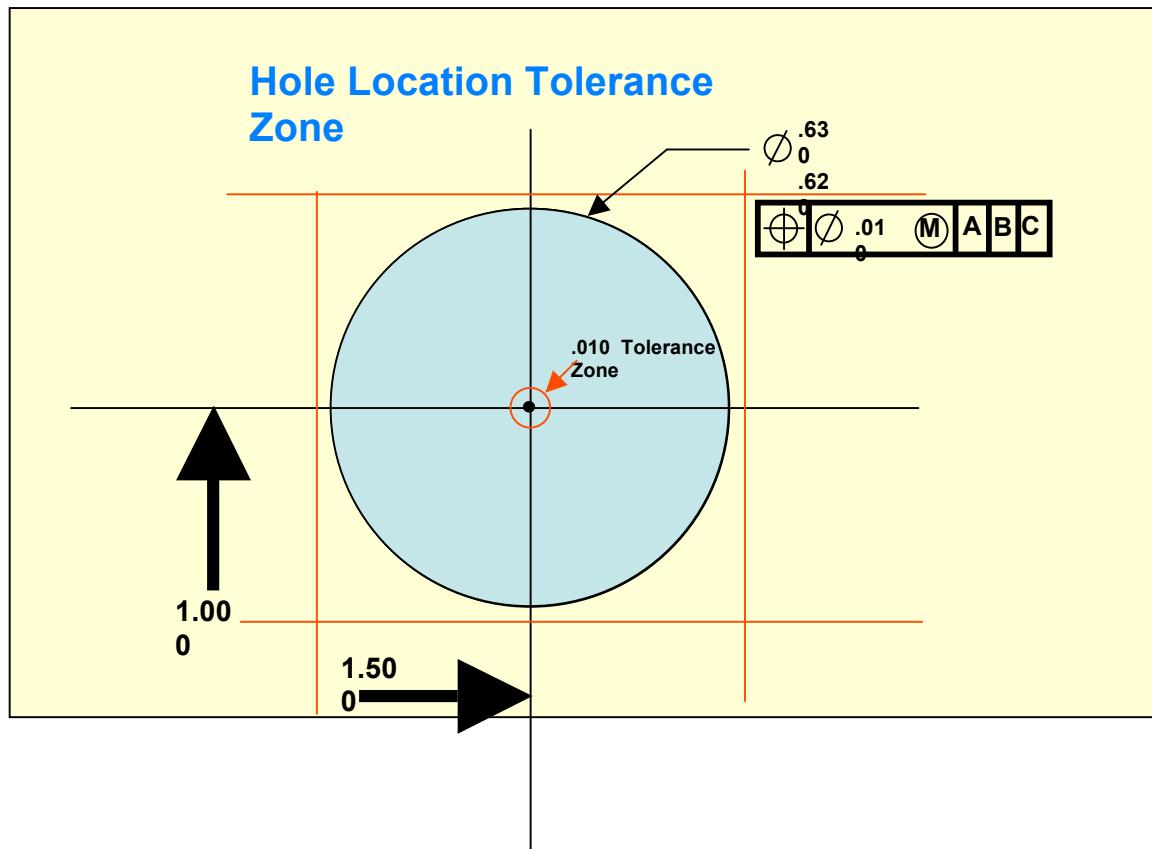


Figure 6-3. Hole-axis Tolerance zone

The MMC condition dictates a smaller position tolerance. If the hole is made to the Least Material Condition (LMC), resulting in a larger hole, then the hole location can be farther off and still align with the mating pin.

.010 when hole size is .620 (MMC)
.020 when hole size is .630 (LMC)

Geometric Tolerancing Applied to an Angle Block - 2D View

Geometrical tolerancing communicates the definition of a product in a very clear and concise three dimensional mathematical language. The drawing below shows a fully geometrically tolerated product.

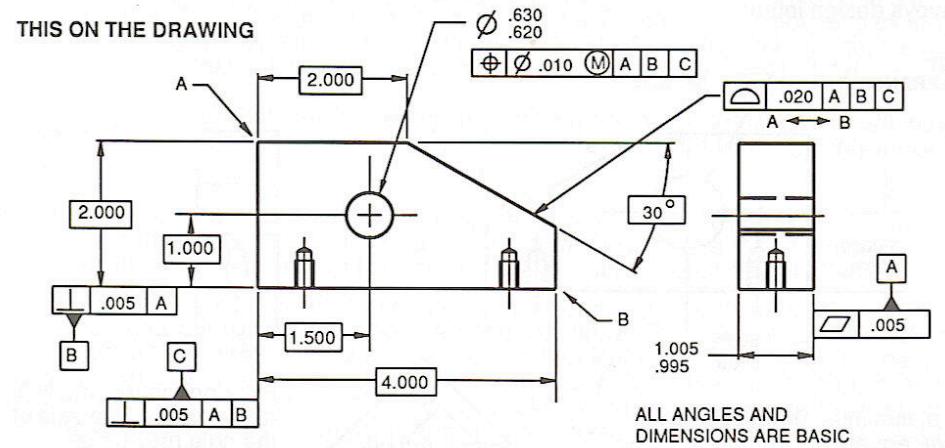


Figure 6-4. Fully toleranced design

The above drawing depicts the part as the designer intended it to be. In reality, **no part can ever be made perfect**. It will always be off by a few millionths of an inch. With that in mind, the drawing below illustrates how the GD&T instructions control the features of the part. The drawing is *greatly exaggerated* to show what would be undetectable by the naked eye.

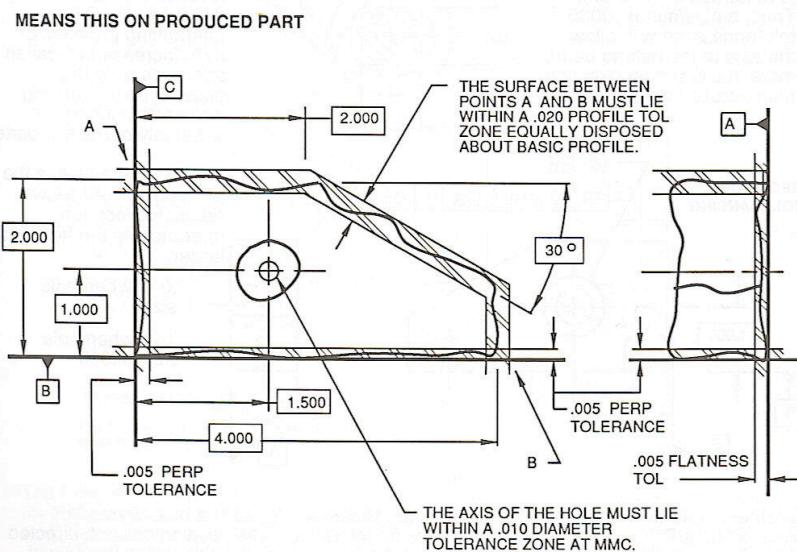


Figure 6-5. Actual produced part (greatly exaggerated).

As you can see, all surfaces must fall within the tolerance zone specified by the feature control frame.

Geometric Tolerancing -vs- Limit Tolerancing - What's The Difference?

This drawing is produced using limit tolerancing. There is no feature control frame, so the design relies on the limits established by the \pm dimensions, and the datums are all “implied”.

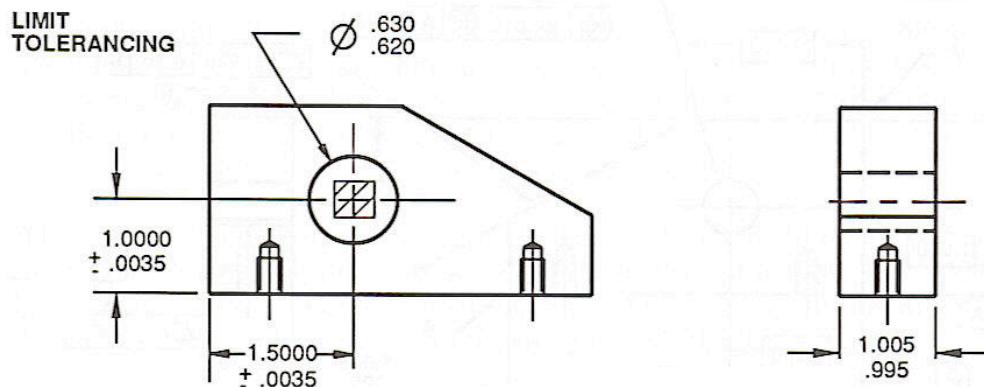


Figure 6-6. Limit tolerance effect

Notice that the position of the hole is implied as being oriented from the lower left hand corner. Because we are forced to use the plus/minus .0035 limit tolerance, the hole tolerance zone ends up looking like a square. A close look at the part reveals that the axis of the hole can be off farther in a diagonal direction than across the flat sides.

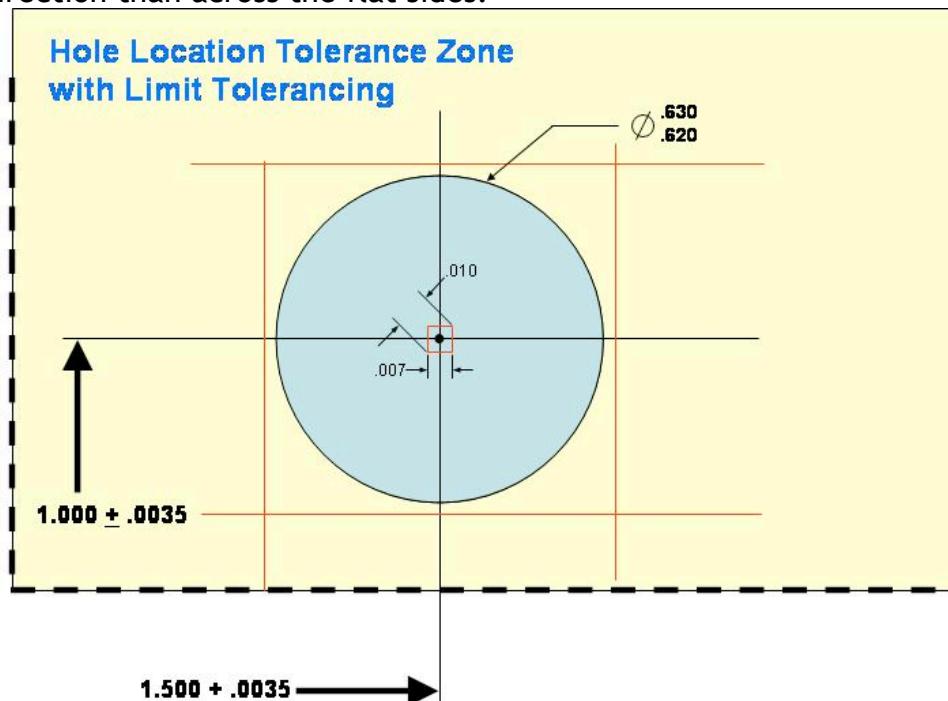


Figure 6-7. Square tolerance zone

Regardless of Feature Size - RFS

Modifier rule # 2 states that unless otherwise specified, all geometric tolerances are by default implied to be RFS - Regardless of Feature Size. Since all unspecified tolerances apply at RFS, there is no need for a RFS symbol. The drawing below illustrates how RFS affects the location tolerance of a feature.

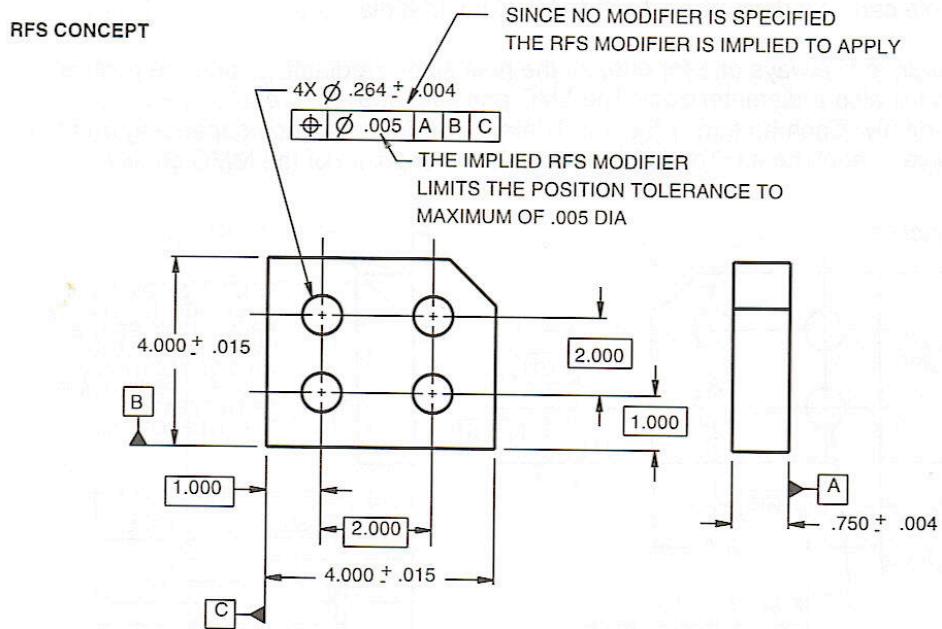


Figure 6-8. Effect of RFS modifier

What this means to the machinist is that no matter if the holes are machined at the upper limit of .268 or the lower limit of .260, their location is still restricted to the .005 position tolerance zone.

Summary

GD&T (geometric dimensioning and tolerancing) is an international design standard that uses a consistent approach and compact symbols to define and control the features of manufactured parts. GD&T is derived from the two separate standards of ASME Y14.5M and ISO 1101. Technically, GD&T is a drafting standard, but it helps inspectors improve their methods by emphasizing fit, form and function.

GD&T compares the physical, imperfect features of a part to its perfect, imaginary form specified in the design drawing. Through the use of standard geometric tolerances, GD&T controls flatness, straightness, circularity, cylindricity, and four form tolerances that independently control a feature. Other tolerances, such as location, runout, and orientation must be referenced to another datum. The profile tolerances can define a feature independently, but a related datum can further define the orientation and location.

A series of internationally recognized symbols are organized into a feature control frame. The control frame specifies the type of geometric tolerance, the material condition modifier, and any datums that relate to the feature.

Mass production of top quality automobiles would not be possible without the, accuracy and efficiency afforded by the guidelines of GD&T.