1 of 1

The submitted manuscript has been authored by a contractor of the U. S. Government under contract No. W-31-109-ENG-38. Accordingly, the U. S. Government retains a nonexclusive, royalty-free license to publish or reproduce the published form of this contribution, or allow others to do so, for U. S. Government purposes.

A GRAPHIC VIEW OF GEOMETRIC DIMENSIONING AND TOLERANCING

Ralph A. C. Swanson

ABSTRACT

This paper points out a weakness frequently encountered in communication between Engineering Documents and Manufacturing Practices. In addition this paper gives an overview of a symbolic language designed to provide important communication between the Engineering Document and the Manufacturing process for every part and assembly. It will define the application of the characters used. Unlike the English language where the same word can have several meanings, the characters used in this symbolic language have only one.

INTRODUCTION

It is widely held that Geometric Dimensioning and Tolerancing (G.D. & T.), if done properly, leads to a more efficient production process and to higher quality engineered products than would normally be obtained using the older non-coordinate methods. For instance, G.D. & T. is used by Department of Defense contractors and in the automotive industry. Unfortunately, it is not well understood by some users, and it is not as effective as it could be.

G.D. & T. may be likened to learning a new language. Each individual would need one or two weeks of intensive, graded training initially with follow-up refresher and/or advanced topic sessions. Everybody involved in engineering, design, manufacturing and QA and QC has to be trained and productivity, during the learning period will be affected.

DISCUSSION

It is most important that G.D. & T. is perfectly understood by both Engineering and Manufacturing Personnel so that there are no errors in communication. It must be understood

that the symbolic language does not cause parts or assemblies to be held to tighter tolerances. Its main purpose is to clarify and make engineering documents more useful in many ways. All engineering documents for parts and assemblies contain dimensions and notes to describe the features. At times dimensions alone do not describe the features completely. Dimensions do not tell you a feature is to be flat or straight. They do not tell you that features are parallel or perpendicular. They do not tell how round is round. Notes which try to describe these qualities may be lengthy and confusing. Dimensions alone do not tell you if the high or low side of the feature is required. Dimensions do not describe exact locations or highlight datums. On a two dimensional document dimensions can't describe a three dimensional feature accurately. If a note says two holes are to be straight, is one hole as straight as the other. How are features described on two different parts to make sure they match? What controls features to make sure they are correct for their full depth? How do you control an irregular profile? How is its exact location known? To describe these conditions, a new language is needed, namely, a symbolic language that describes each feature exactly as specified and leaves no question of the feature's location or condition.

The key is understanding what information is required on the document and how to interpret this information. This paper will point out the advantages of using this symbolic language. The object of "A GRAPHIC VIEW OF GEOMETRIC DIMENSIONING AND TOLERANCING" is to make it clear that this symbolic language is applicable both for Engineering and Manufacturing. This paper will describe the issues and language without diagrams or pictures. I will present "A GRAPHIC VIEW OF THIS SYMBOLIC LANGUAGE" which points out the

MASTER

relationships between features contained on parts. It will show how this symbolic language can be an international language and the how, when, why and where it is applied and define each characteristic.

CONSIDERATIONS

Manufacturing and inspection should be interested in how a part is made and how it will operate or its purpose in relationship to other parts. This information can be communicated by symbolic characteristics that relate to function. The function or how a part will operate or the purpose of the part and its relationship to other parts can be determined by the Symbolic Characteristic attached to the part feature.

The way a part is to work, such as revolving, sliding, and swinging will determine its function and shape.

The purpose of a part, such as being fixed, holding other parts, etc., also has certain characteristics for its function and shape.

The relationship of a part, such as guiding, controlling other parts and locating parts have certain other characteristics for determining its function and shape.

There are three categories for function characteristics:

- Operation of parts uses characteristics that describe action such as cylindricity, parallelism, position, profile of a surface, circular runout and total runout. These are for parts that revolve, move together, fit together, press on one another and act as defined by the characteristic attached to the feature.
- Purpose of parts use characteristics that describe stationary objects such as flatness, straightness, parallelism, circularity and profile of a line. The parts are fixed or connected together or fit other parts.
- Relationship of parts use characteristics that compare one part to another part for perpendicularity, parallelism, angularity, position and total runout. The parts depend on each other per the characteristic attached to the features.

Some characteristics fall into more than one of the three categories, but the definition does not change, only the function of the feature with parts and assemblies. Additional characteristics add information to satisfy function and shape of part, such as datums, maximum material condition, regardless of feature size, least material condition and projected tolerance zone. These give more information to clarify location, size and depth of features and are added to features along with other characteristics to give exact definition.

The information is set by the Engineering Design and Manufacturing Techniques. Inspection Systems also must be considered. The categories of function characteristics make manufacturing aware of what is required and gives inspection an insight into important features. It works as a production tool by keeping consistent information and relating it to the complete unit. Besides, this allows Engineering to update parts and keep the integrity of the assembly.

The Characteristics and their Application are defined on the following pages. Each characteristic stands by itself, but must be used in conjunction with information so that every single feature is completely defined.

An example of this is the documentation for an engine crankshaft and crankcase. The crankshaft has characteristics for features that control operation and its relationship to other feature parts and assemblies. The crankcase has characteristics that control purpose and its relationship to other features, parts and assemblies. The parts are related by similar characteristics of specific features, such as the shaft controlled by Total Runout and Position. The crankcase bores also contain these characteristics fixing their relationship.

COMPLICATIONS

Without the symbolic characteristics each feature on each part and assembly would stand alone, unrelated to the other features on parts. This can cause manufacturing and inspection to look at individual features which may seem correct, but in fact may not match related features on other parts and assemblies. In other words a feature may be cylindrical within tolerance, while a feature on a related part may have been inspected for total

runout so that the parts may not fit together properly. The location of a feature can cause problems if one part has features located from a different feature of its mating part. This can be rectified only by making each assembly unique. Records of this occurrence seldom reflect the features on either part. Therefore, each assembly that varies from the official document cannot have replacement parts. Many firms face this situation over and over again. In addition to the parts being inspected differently, manufacturing of parts was done on different types of machines. This happened because the Symbolic Characteristics controlling features was not used. If a requirement was for a feature to be Straight within the tolerance and the feature was controlled by Profile of a Line, the feature may have been far from straight.

Notes on Engineering Documents can be complicated and confusing. Here are some examples from actual documents:

"The front and rear must be perpendicular with the bottom and sides within twenty five to forty seconds of a degree and flat from end to end within plus or minus thirty thousandths. The top should also be the same." Misinterpretation of this note can be costly from the manufacturing and inspection point of view. Another actual note read "This diameter of four and one half inches plus or minus two tenths should be round and square with bore within thirty thousandths". No one could figure out exactly how round the shaft was to be. They made the shaft better than was required, which was more costly.

These are only two examples, but over my thirty five years of experience I have encountered thousands of examples of unclear documentation. The consequence is that the parts are more costly than necessary or do not meet minimum specifications. Dealing with foreign companies can make matters worse because unclear language can produce unacceptable products.

The need for a symbolic language came about long before language was a factor. Still many large Engineering and Manufacturing Organizations have not used it or have not taken the time to understand it. They seem to have somehow worked around the symbolic language approach, no matter the cost. This seems hard to believe especially when their competitors use symbolic language for their products on a national and

international front. How long they will avoid the symbolic language will be measured by the consumer.

COMMUNICATION

Exactly what is this symbolic language? The symbolic language is called "GEOMETRIC DIMENSIONING AND TOLERANCING" (G.D. & T.). This is found in the "American National Standards Engineering Drawings and Related Documentation Practices" ANSI Y 14.2 m1982 Dimensioning and Tolerancing.

G.D. & T. is a means of specifying on an Engineering document the design requirements with respect to the actual function and relationship of part features. G.D. & T. is both an Engineering design language, functional production and inspection technique language. G.D. & T. is required when datum reference is desirable to insure consistency between manufacturing and inspection operations. G.D. & T. should be used when functional gauging techniques are desirable. G.D. & T. is used to eliminate rejected parts and assembly problems.

When should G.D. & T. be used? G.D. & T. should be used when the tolerance of size and location do not provide the required control for features critical to function or interchangeability.

The effectiveness of G.D. & T. depends a great deal on understanding and application of datums. A datum is a point, line, plane or axis which is assumed to be theoretically exact for the purpose of computation or reference and from which the location of features of a part or assembly may be established or related. Datums should be selected on a basis of the functional requirements of part or assembly. The datum is signified by a dash which is followed by a letter and another dash inside a rectangle. The datum may be attached to a feature or dimension or may have an arrow pointing to the feature. The Primary datum has three point minimum contact with the feature. Secondary datum has a two point contact with a feature. The tertiary datum has one point contact with the feature.

Important Benefits of G.D. & T.: G.D. & T. provides nominal dimensions for numerically controlled machines and insures interchangeability

and reduces inspection time by legalizing the use of functional gauges. G.D. & T. simplifies accurate tolerance calculations through the rule total tolerance equals clearance. G.D. & T. insures consistent interpretation and facilitates both domestic and international communication. It provides an additional fifty seven percent larger tolerance zone when the condition of position is used. G.D. & T. can be applied to existing documents. It contains only sixteen characters.

A benefit not often recognized is when G.D. & T. is not required. This makes the user aware there is no need for features to be controlled. The following are some: Single Place Dimensions or Two Place used on a document means very loose tolerances. Four and Five Place Dimensions means very tight tolerances. Reference Dimensions, Unspecified Angles, Fabricated Parts, General Hardware, Gaskets and Seals and Shims or Spacers with no critical size do not require G.D. & T.

The Symbolic Language has some terms which must be defined. Symbols which express language are characters. The requirements for a part to meet specifications are the conditions. The shape of part and condition stated on the document is the control. Specific components of a part, e.g., straight, flat, round or curved are features. Any deviation in feature is an element. Dimensional statement of how far an actual surface or feature is permitted to vary from the desired location implied on the document establishes the form. Dimensional statement of how far an actual feature is permitted to vary from the perfect location implied by the document as related to a datum, datums or other features is its location. Dimensional statement of how far an actual surface or feature is permitted to vary from the desired form implied by the document during full three hundred and sixty degree rotation of part on datum axis is the runout.

Classification and requirements of conditions. The following are considered form conditions: The straightness condition is required when surface is to be controlled in one direction. The flatness condition is required when surface control is in more than one direction. The cylindricity condition is required when control is required over full length of curve. The circularity condition is required when control is required for cross section of curve.

Profile conditions: The profile of a line condition is required when control is required for outline of curved cross section. The profile of a surface condition is required when control for the outline of curve is required for the full length.

Orientation conditions: The angularity, perpendicularity, and parallelism conditions are required to control two related features.

The following are considered runout conditions: (Rotating Parts)

The circular runout condition is required when the cross section is to be controlled from axis of datum. The total runout condition is required when Position, Roundness, Angularity and Profile are to be controlled to axis of datum for entire curved surface.

CONTROLLING

Explanation of conditions that require control: Unless otherwise specified, the tolerance of a part or size control of an individual feature control is the quality of a single surface as well as its size. No element of part feature shall extend outside the perfect or true form which is implied in the document. Each tolerance of form or location and datum reference for screw threads applied to pitch diameter, unless major or minor diameter is noted. When a pattern of holes is dimensioned relative to the location of another hole, this hole must be identified as a datum and the hole pattern is located with respect to it.

The requirements for the Shape of the part to meet specifications requires a special way to define the part features and specify the correct characteristics.

The following are Graphic Visualizations of these characteristics. The characteristics have only one meaning and one symbol.

Flatness which is used to describe a three dimensional flat surface. The Symbol for flatness is a polygon.

To demonstrate the meaning of flatness, let us suppose a cup slides off a table top. Is the table top flat? How is flatness measured? The condition for flatness is when all elements are in a single plane or all elements on a surface in a plane

are within the part tolerance. The surface is measured along a perpendicular line from a controlled surface. Flatness is a form condition which considers an individual feature. The control characteristic is applied to the feature, along with the control tolerance. No datum is required. In the example, the floor the table sat on is not the controlled surface. In fact, the table top may have been measured as an individual feature where all elements on the top were within tolerance from the other elements on the table top. There is no comparison from one FLAT feature to any other feature.

Straightness is used to describe a straight two dimensional line. The symbol for straightness is a short horizontal line. The condition for straightness is when all elements are on a single line or the elements on the line are within the tolerance zone. The straightness tolerance must be less than feature size. The line is measured perpendicular to a controlled feature. In the example this may be the side of the doorway. The control symbol is applied to feature or feature dimension, along with the control tolerance. No datum is required for straightness which is a form condition and considers an individual feature. There is no comparison from one straight feature to any other feature. Accordingly, flatness and straightness have nothing to do with parallelism. We just assume they do.

Circularity is used to describe a two dimensional round cross section. The symbol for circularity is a circle. To demonstrate the meaning of circularity, we know that hardware and parts that hold things together are generally round. It would be difficult to tighten an oval bolt into an oval threaded hole. How are circular parts controlled to make sure they are round and to make two round parts fit together? The condition for circularity or roundness is when all the elements on a single three dimensional curved surface are equidistance or within the desired tolerance from the axis when measured on a plane which is perpendicular to a plane which intersects the common axis. The perpendicular plane revolves on the axis so that measurement can be made along the complete cross section. Circularity is a form condition which considers an individual feature. The control symbol is applied to a feature, along with control tolerance. No datum is required. There is no comparison from one round cross section to any other feature.

Cylindricity is used to describe a round three dimensional object. The symbol for cylindricity is a circle with two tangent lines, one on each side of the circle that are at a forty five degree angle. The condition for cylindricity is when all elements on the circular object for the entire length of the surface are simultaneously equidistant or within tolerance from the axis while revolving. The axis must be established and cylindricity tolerance must be within part size tolerance. Cylindricity controls circularity, straightness and parallelism. The control symbol is applied to the feature along with control tolerance. No datum is required since cylindricity is a form condition which considers an individual feature. There is no comparison of features controlled by cylindricity.

Circularity and cylindricity both deal with controlling round or curved features. Circularity control is used for stationary round cross sectional features. Cylindricity controls a round or curved feature for circularity, straightness and parallelism simultaneously along the full length as it revolves.

The characteristics, flatness, straightness, circularity and cylindricity all consider the condition of a feature as a deviation of elements on the feature.

The following characteristics are Profile Condition which consider both an individual and related feature. They may consider the condition of a feature as the deviation of elements on the feature or may consider another feature which is related. A datum may be needed.

Profile of a line is used to describe the path of a line in one orthographic view. The symbol for profile of a line is the upper half of a circle. The condition for a profile of a line is when all elements are on the profile in a single path in one orthographic view or within the tolerance limits measured along the profile. Profile of a line is a condition which considers both individual and/or related features. No datum is required when an individual feature is under consideration. A datum is required when related features are considered. The control Characteristic is applied to the feature. The control of profile of a line may consider other features.

Profile of a surface is used to describe the profile of a three dimensional surface for its entire length. The symbol for profile of a surface is the upper half of a circle with a horizontal line connecting

the start and end of curve. The condition for a profile of a surface is when all elements are on a single three dimensional surface for the entire length, width and circumference or when the elements are within the profile tolerance limits measured along the complete profile of the surface. Profile of a surface is a condition which considers both individual and related features.

The following Characteristics are Orientation condition which consider the relationship between two or more features. A datum may be required.

Perpendicularity is used to describe the position of two or more features in regards to their squareness. The symbol for perpendicularity is a short horizontal line bisected by a vertical line on top of the horizontal line. The condition for perpendicularity is when all elements on a feature or surface are ninety degrees from the controlled feature or within the tolerance zone. The measurement is made from the control feature or datum over the height and width of the feature to be controlled. The control tolerance must be less than the feature size tolerance. Perpendicularity is an Orientation condition which considers related features.

Angularity is used to describe the position of two or more features with respect to an inclined plane at a specific angle to a datum or axis. The symbol for angularity is a short horizontal line with another short line at a thirty degree angle from an axis at the left end of the horizontal line. The condition for angularity is when all elements on the feature or surface over the height and width of the feature are within the control tolerance. The control tolerance must be less than feature size tolerance. Angularity is an orientation condition which consider related features.

Parallelism is used to describe two or more features that are equidistant from one another. The symbol for parallelism is two lines which do not intersect and are on a forty five degree angle slanted to the right. The condition of parallelism is when all elements on the feature or surface are within the same distance or within the tolerance, when measured at three points ninety degrees from the controlled feature or surface. Parallelism is an orientation condition which considers related features. A datum may be necessary.

All characteristics described to this point have

their features defined in a stationary state, even if the actual function is not stationary. Gears, for instance, can be controlled in this manner. In other words the features are controlled when the part is fixed or not moving. The gauges and fixtures can be set to measure features. In production this means consistency.

The following characteristics for Runout are described as the features of a part rotating a full three hundred and sixty degrees. Circular runout is used to describe the relationship of the profile of one cross section to another while the features are revolving a full three hundred and sixty degrees. The symbol for circular runout is a short line at a forty five degree angle with an arrow on the end of the line pointing up.

To demonstrate the meaning of circular runout, a varying diameter shaft running on different size bearings must run true. The smaller diameter feature on the shaft must be on the same axis as the larger diameter feature. The shaft bearing clearances on the small and large shaft diameter must be equal. The condition for circular runout is when composite circular elements on circular cross sections vary from roundness. The deviation from desired form within the tolerance is measured during full revolution on the datum axis at the designated cross sections. The control symbol is applied to the feature, along with control tolerance. Circular runout is a runout condition which requires a datum.

Total runout is used to describe the relationship of the profile of one circular three dimensional feature for Circularity, Cylindricity, Straightness, Angularity and Parallelism to another circular feature while making a full three hundred and sixty degree revolution. The symbol for total runout is two short lines at forty five degree angles slanted to the right with a horizontal line connecting the lines at bottom and with arrows on the ends of the angled lines which point up.

The condition for total runout is when composite circular elements on a surface effect Circularity, Cylindricity, Straightness, Angularity and Parallelism. The deviation form within the tolerances is measured during a full three hundred and sixty degree revolution on the datum axis along the full length of the feature surface. The control symbol along with control tolerance is applied to the feature. Total runout is a runout

condition which requires a datum.

The one function characteristic and classification of condition not covered is for the position of features. This Fourth Category for a function characteristic is exact location. The classification and requirement of this is the position condition. Position considers the location of stationary features, point, line or plane and related features which are to be exact in respect to theoretical perfect location.

The condition of position is when the location of features are exact or within tolerance with respect to a theoretical perfect location, generally at the center of the feature. An initial feature or datum is selected to locate other features. Position controls the angle of a feature. The control tolerance and characteristic are applied to the feature. A datum may be required.

The following characteristics do not define a feature by themselves. They are added to the control characteristic to more exactly define the features.

Maximum material condition is an added characteristic. Its characteristic symbol is an "M" inside a circle. Maximum material condition means that any parts or features that fit together have maximum clearance. The feature to be fit to another feature would be on the smaller side of the tolerance. The feature that the part would fit into would be on the larger side of tolerance stated, giving maximum clearance. The characteristic follows the control tolerance and is placed in the Feature Control Symbol which is a rectangular box containing characteristics, tolerance and datum.

Least material condition is an added characteristic. Its characteristic symbol is an "L" in a circle. Least material condition means that any parts or features that fit together have a minimum clearance. The feature to be fit into another feature would be on the larger side of the tolerance. The feature that the part would fit into would be on the smaller side of the tolerance, giving the least clearance. The characteristic follows the control tolerance and is placed in the Feature Control Symbol which is a rectangular box containing characteristics, tolerance and datum.

Regardless of feature size is an added characteristic. Its characteristic symbol is an "S" in

a circle. Regardless of feature size means that whatever the state of the feature is should be considered. The characteristic follows the control tolerance and is placed in the Feature Control Symbol which is a rectangular box containing characteristics, tolerance and datum.

Projected tolerance zone is an added characteristic. Its characteristic symbol is a "P" inside a circle. Projected tolerance zone is used with position and means that the depth of a feature is held to the position control tolerance for the stated depth. This should be used when multiple parts must line up or the feature is deep and critical within the tolerance for the projected tolerance zone. The tolerance zone is always the width of the control tolerance for Flat, Straight, Parallel, Perpendicular and Angular features. For Profile or Circular features it is concentric curves to match feature.

Maximum material condition, least material condition, and regardless of feature size apply to flat, straight, perpendicular, parallel and angular features and to features controlled by position. They do not apply to features controlled by circularity, cylindricity nor features controlled by profile or runout.

CONCLUSION

G.D. & T. is a formalism for showing dimensions and tolerances relative to datum references on a drawing in an absolutely unambiguous manner so that nothing is left to interpretation by the user (shops, inspection, etc.). It is particularly useful when interchangeability is important (e.g., parts replacement in a hot cell, modest or large scale mass production) and where close contact between the various users is not practical (e.g., large projects, long duration projects where turnover of personnel is expected). A major advantage is that the need for trial fit up and rework is virtually eliminated.

This paper points out the need for a symbolic language and describes problems which arise when its not used. "GEOMETRIC DIMENSIONING AND TOLERANCING" should appear on many mechanical engineering documents. "A GRAPHIC VIEW OF GEOMETRIC DIMENSIONING AND TOLERANCING" only touches on the subject, but its obvious that it is important to clarify complex documents. The paper shows that if applied

correctly, it is fool-proof. Its pointed out what this means to manufacturing and inspection operations. Understanding and applying "GEOMETRIC DIMENSIONING AND TOLERANCING" are required to improve our products and will be rewarding if used in the national and international markets.

DISCLAIMER

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

G.D. & T. AND IT'S RELATIONSHIP

	LATHE	PLANE	MILL	BOREING	DRILL **	PUNCH **	SAW & BURN		SYMBOLS
STRAIGHT	X	X	X	X	X	X			
FLAT		X	X						
PERPENDICULAR	X	X	X	X	X	X			
ANGULAR	X	X	X	X	X				
PARALLEL	X	X	X	X	X				
CIRCULARITY	X		X	X	X				
CYLINDRICITY	X		X	X	X	X			
RUNOUT	X		X	X	X				
POSITION	X		X	X	X	X			
PROFILE	X		X						

CATEGORIES			CLASSIFICATIONS			
OPERATION(ACTION)	PURPOSE(STATIONARY)	RELATIONSHIP(COMPARE)	FORM	PROFILE	ORIENTATION	RUNOUT
	X		X			
	X		X			
		X			X	
	X	X			X	
X	X	X			X	
X	X		X			
X			X			
X		X				X
X		X				
		X		X		

X= G.D. & T. SYMBOLS USED.

**= NO G.D. & T. FOR CLEARANCE HOLES.

OTHER SYMBOLS

MAXIMUM MATERIAL CONDITION	
LEAST MATERIAL CONDITION	
REGARDLESS OF FEATURE SIZE	
PROJECTED TOLERANCE ZONE	
DATUM TARGET	
DIAMETERS (CYLINDRICAL) TOLERANCE ZONE	
BASIC, EXACT DIMENSION	
DATUM IDENTIFICATION SYMBOL	
FEATURE CONTROL SYMBOL (FRAME)	

RELATIONSHIP TO MICROFINISHES.

ALL G.D. & T.

8	HONE SURFACE -G.D. & T. REQUIRED
16	SPECIAL BORES -G.D. & T. REQUIRED
32	BORES-BEARING SURFACE -G.D. & T. REQUIRED
65	SPECIAL & BRG. SURFACE -G.D. & T. REQUIRED
125	GENERAL SURFACE -G.D. & T. REQUIRED
250	G.D. & T. MAY NOT BE REQUIRED
500	G.D. & T. MAY NOT BE REQUIRED

DATE

FILMED

7/6/94

END