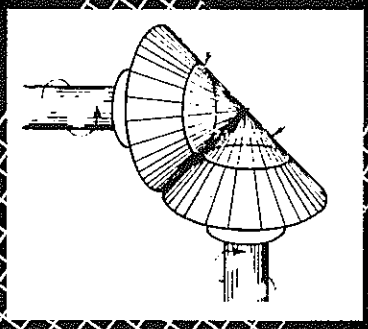
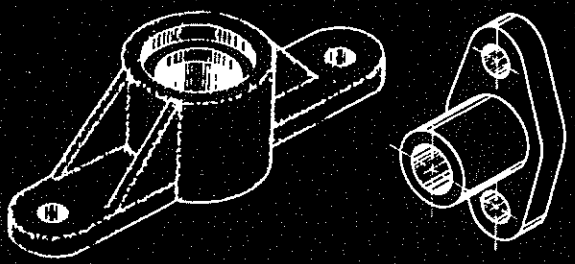
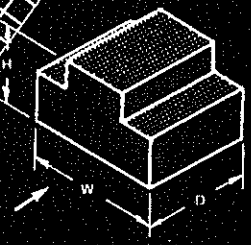
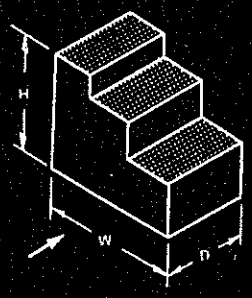
Interpreting Engineering Drawings

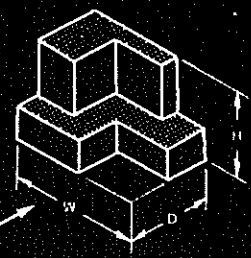


Fifth Edition

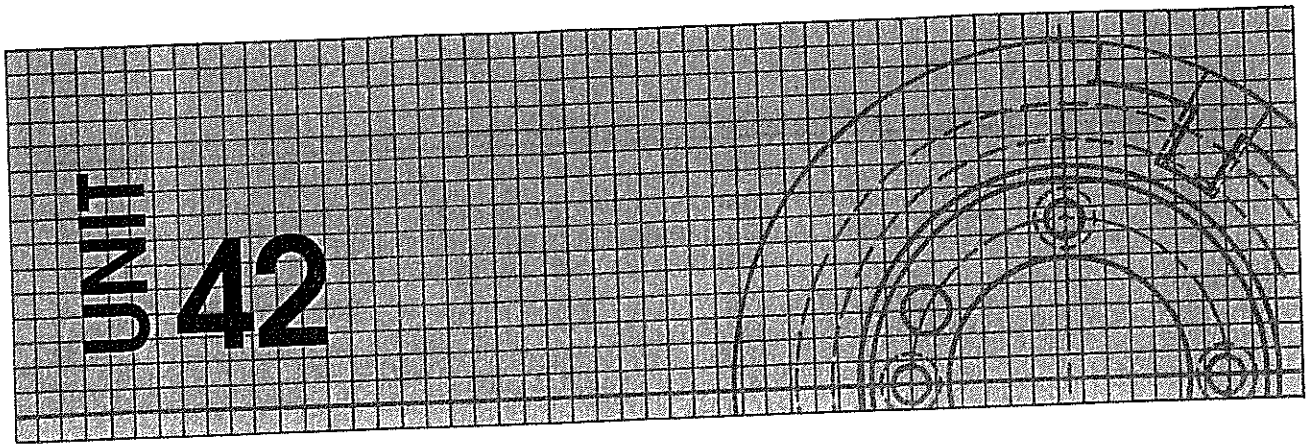








Cecil H. Jensen
Raymond D. Hines



MODERN ENGINEERING TOLERANCING

An engineering drawing of a manufactured part conveys information from the designer to the manufacturer and inspector. It must contain all information necessary for the part to be correctly manufactured. It must also enable the inspector to determine precisely whether the finished parts are acceptable.

Therefore, each drawing must convey three essential items of information: the material to be used, the size or dimensions of the part, and the shape or geometric characteristics of the part. The drawing must also indicate the permissible variation of size and form.

The actual size of a feature must be within the size limits specified on the drawing. Each measurement made at any cross section of the feature must not be greater than the maximum limit of size, nor smaller than the minimum limit of size, figure 42-1. Although each part is within the prescribed tolerance zones, the parts may not be usable because of their deviation from their form.

In order to meet functional requirements, it is often necessary to control errors of form including: squareness, roundness, and flatness, as well as deviation from true size. In the case of mating parts, such as holes and shafts, it is usually necessary to insure that they do not cross the boundary of perfect form at their maximum material size (the smallest hole or the largest shaft) because of being bent or otherwise deformed. This condition is shown in figure 42-2, where features are not permitted to cross the boundary of perfect form at the least material size (the largest hole or the smallest shaft).

The system *of geometric tolerancing* offers a precise interpretation of drawing requirements. Geometric tolerancing controls geometric characteristics of parts. These characteristics include: flatness, roundness, angularity, profile, and position. Other techniques, such as datum systems, datum targets, and projected tolerance zones were developed in order to facilitate this precise interpretation.

Geometric tolerances need not be used for every feature of a part. Generally, if each feature meets all dimensional tolerances, form variations will be adequately controlled by the accuracy of the manufacturing process and equipment used. A geometric tolerance is used when geometric errors must be limited more closely than might ordinarily be expected from the manufacturing process. A geometric tolerance is also used to state functional or interchangeability requirements.

GEOMETRIC TOLERANCING

A geometric tolerance is the maximum permissible variation of form, orientation, or location of a feature from that indicated or specified on a drawing. The tolerance value represents the width or diameter of the tolerance zone, within which the point, line, or surface of the feature should lie.

From this definition, it follows that a feature would be permitted to have any variation of form or take up any position within the specified geometric tolerance zone.

For example, a line controlled in a single plane by a straightness tolerance of .006 in. must be contained within tolerance zone .006 in. wide, figure 42-3.

Points, Lines, and Surfaces

The production and measurement of engineering parts deal, in most cases, with surfaces of objects. These surfaces may be flat, cylindrical, coni-

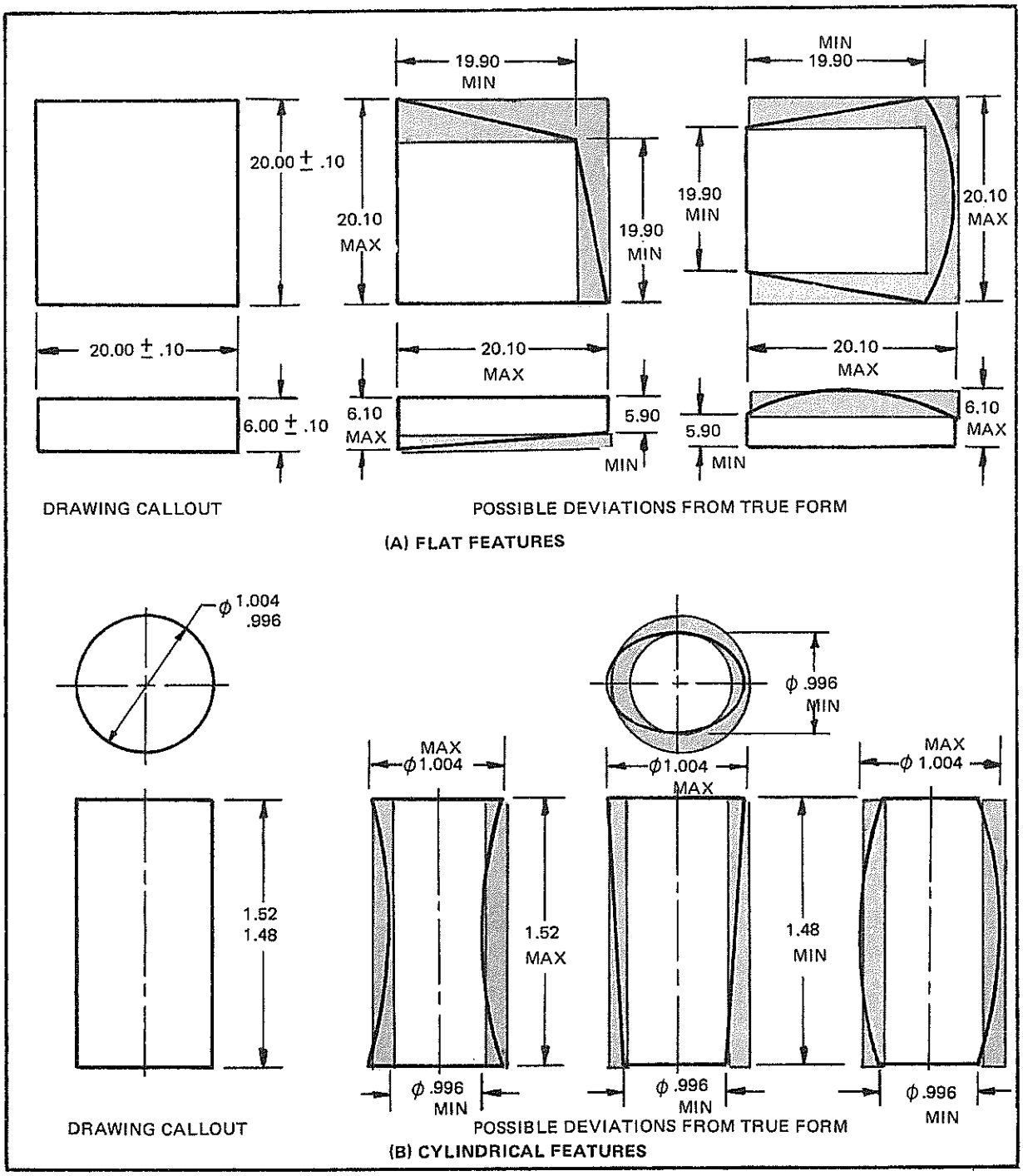


Figure 42-1 Deviations of shape permitted by toleranced dimensions

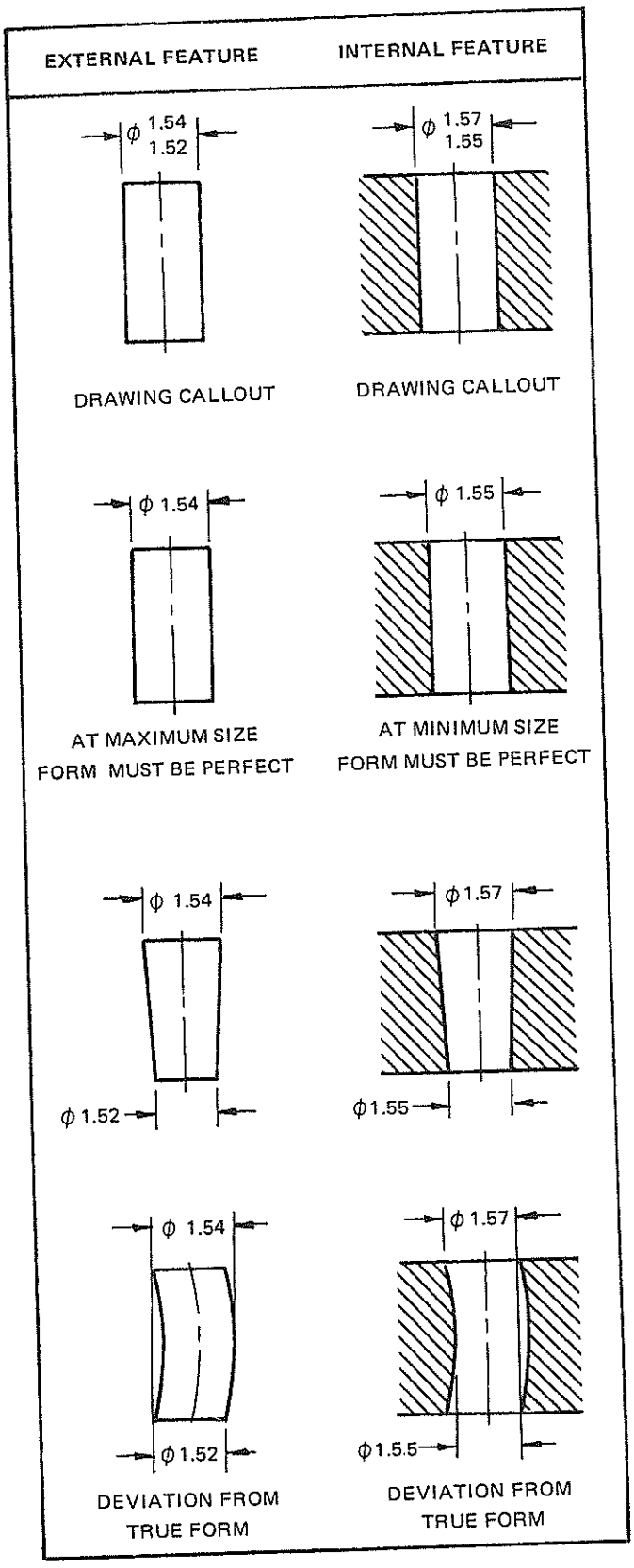


Figure 42-2 Examples of deviation of form when perfect form at the maximum material size is required

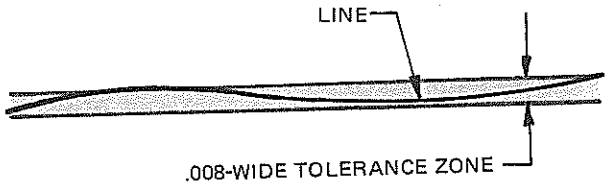


Figure 42-3 Tolerance zone for straightness of a line

cal, or spherical or have some more or less irregular shape or contour.

Measurement, however, usually has to take place at specific points. A line or surface evaluated dimensionally by making a series of measurements at various points along its length.

Surfaces are considered to be composed of a series of line elements running in two or more directions.

Points have position but no size, and therefore position is the only characteristic that requires control. Lines and surfaces have to be controlled for form, orientation, and location. Therefore geometric tolerances provide for control of these characteristics, figure 42-4. (Symbols will be introduced as required, but all are shown in the figure for reference purposes.)

FEATURE CONTROL FRAME

Some geometric tolerances have been used for many years in the form of notes such as PARALLEL WITH SURFACE "A" WITHIN .001" and STRAIGHT WITHIN .12". While such notes are now obsolete, the reader should be prepared to recognize them on older drawings.

The current method is to specify geometric tolerances by means of the feature control frame, figure 42-5.

A feature control frame consists of a rectangular frame divided into two or more compartments. The first compartment (starting from the left) contains the geometric characteristic. The second compartment contains the allowable tolerance. Where applicable, the tolerance is preceded by the diameter symbol and followed by a modifying symbol. Other compartments are added when datums must be specified.

FEATURE	TYPE OF TOLERANCE	CHARACTERISTIC	SYMBOL	SEE UNIT
INDIVIDUAL FEATURES	FORM	STRAIGHTNESS	—	42 & 43
		FLATNESS	▱	44
		CIRCULARITY (ROUNDNESS)	○	44
		CYLINDRICITY	⌭	44
INDIVIDUAL OR RELATED FEATURES	PROFILE	PROFILE OF A LINE	⌒	51
		PROFILE OF A SURFACE	⌓	51
RELATED FEATURES	ORIENTATION	ANGULARITY	∠	46 & 47
		PERPENDICULARITY	⊥	
		PARALLELISM	//	
	LOCATION	POSITION	⌖	49
		CONCENTRICITY	◎	
	RUNOUT	CIRCULAR RUNOUT	*↗	52
		TOTAL RUNOUT	*⌰	
SUPPLEMENTARY SYMBOLS		AT MAXIMUM MATERIAL CONDITION	Ⓜ	43 & 49
		REGARDLESS OF FEATURE SIZE	Ⓢ	
		AT LEAST MATERIAL CONDITION	Ⓛ	
		PROJECTED TOLERANCE ZONE	Ⓟ	
		BASIC DIMENSION	XX	45
		DATUM FEATURE	-A-	45
		DATUM TARGET	Ø.50 / A2	48

Figure 42-4 Geometric characteristic symbols

* MAY BE FILLED IN

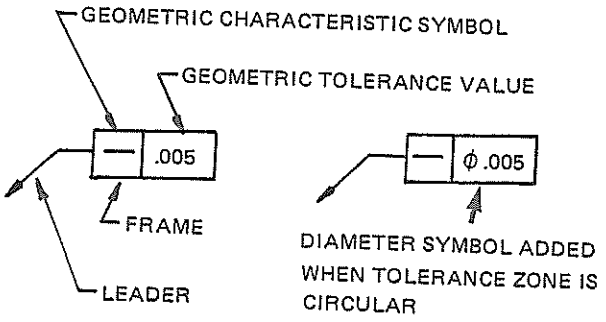
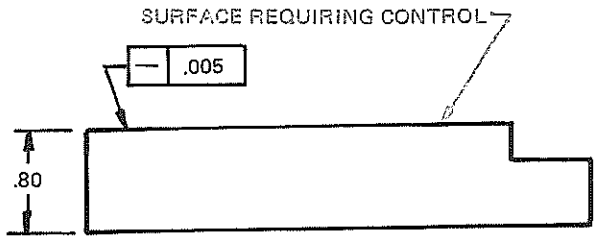
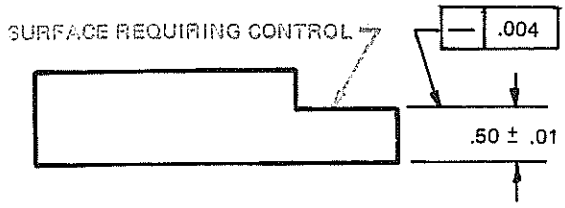


Figure 42-5 Feature control frame

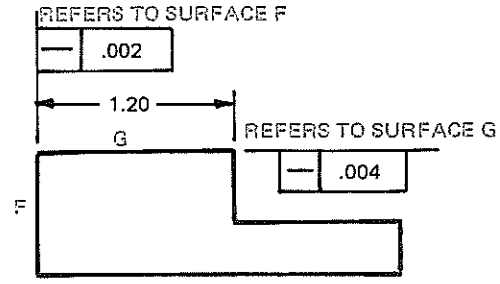
60



RUNNING A LEADER FROM THE FRAME TO THE FEATURE

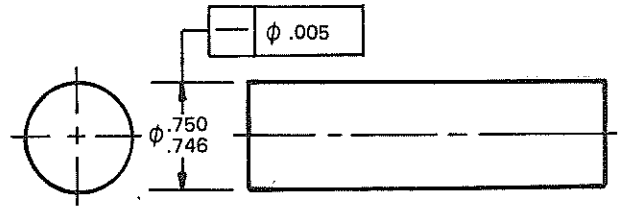


ATTACHED TO AN EXTENSION LINE USING A LEADER

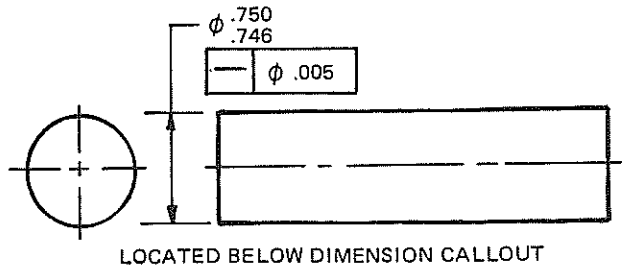


ATTACHED DIRECTLY TO AN EXTENSION LINE

(A) CONTROL OF SURFACE OR SURFACE ELEMENTS



ATTACHED TO THE DIMENSION LINE



LOCATED BELOW DIMENSION CALLOUT

(B) CONTROL OF FEATURE OF SIZE

Figure 42-6 Application of feature control frame

The feature control frame is related to the feature by one of the methods below.

Applicatin to Surfaces (figure 42-6A)

The arrowhead of the leader from the feature control frame should touch the surface of the feature or the extension line of the surface, but not in line with the dimension.

- Attaching a side or end of the frame to an extension line extending from a plane surface feature.

  The leader from the feature control frame should be directed at the featue in its characteristic profile. Thus, in figure 42-7, the straightness tolerance is directed to the side view, and the circularity tolerance to the end view. This may not always be possible; a tolerance connected to an alternative view, such as circularity tolerance connected to a side view, is acceptable. When it is more convenient, or when space is limited, the arrowhead may be directed to an extension line, but not in line with the dimension line.

Application to Features of Size (Figure 42-6B)

- Locating the frame below or attached to the leader directed callout or dimension pertaining to the feature. (See Unit 43.)
- Locating the frame below the size dimension to control the center line, axis, or center plane of the feature. (See Unit 43.)

When two or more feature control frames apply to the same feature, they are drawn together with a single leader and arrowhead, figure 42-8.

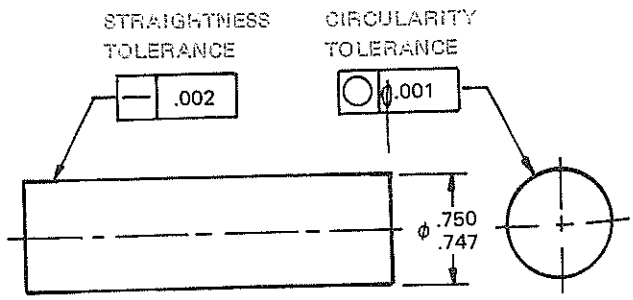


Figure 42-7 Preferred location of feature control symbol when referring to a surface

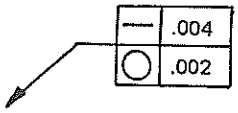


Figure 42-8 Combined feature control frames directed to one surface

FORM TOLERANCES

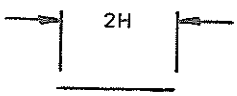
Form tolerances control straightness, flatness, circularity, and cylindricity. Orientation tolerances control angularity, parallelism, and perpendicularity.

Form tolerances are applicable to single (individual) features or elements of single features and, as such, do not require locating dimensions. The form tolerance must be less than the size tolerance.

Form and orientation tolerances critical to function and interchangeability are specified where the tolerances of size and location do not provide sufficient control. A tolerance of form or orientation may be specified where no tolerance of size is given, e.g., the control of flatness.

STRAIGHTNESS

Straightness is a condition in which the element of a surface or a center line is a straight line. The geometric characteristic symbol for straightness is a horizontal line, figure 42-9. A straightness tolerance specifies a tolerance zone within which the considered element of the surface or center line must lie. A straightness tolerance is applied to the view where the elements to be controlled are represented by a straight line.



H = LETTER HEIGHT

Figure 42-9 Straightness symbol

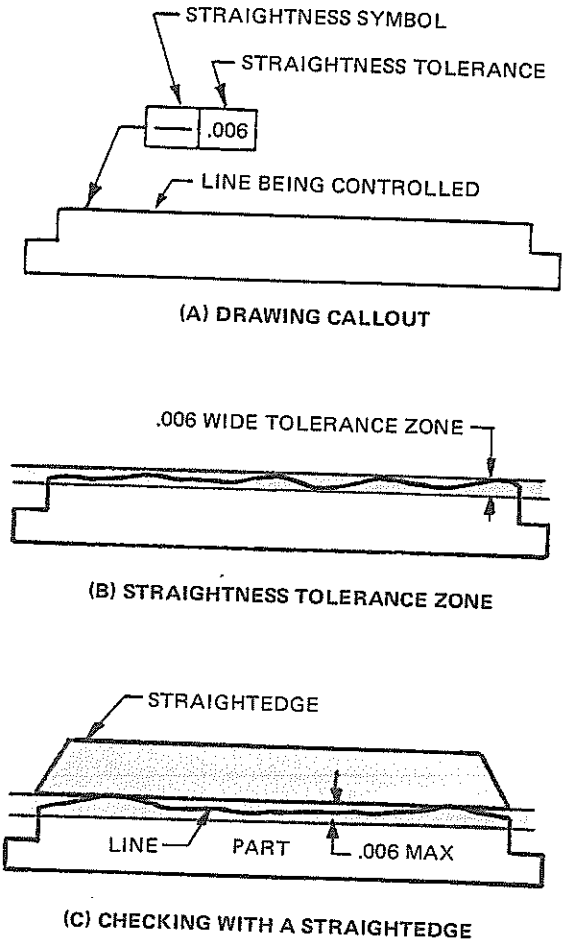


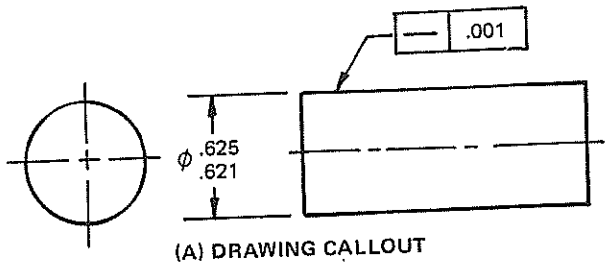
Figure 42-10 Straightness tolerance applied to a flat surface

STRAIGHTNESS CONTROLLING SURFACE ELEMENTS

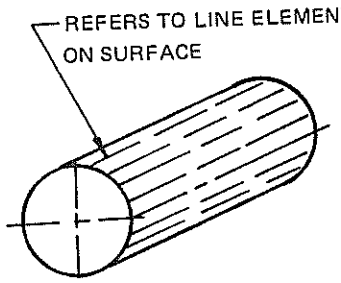
Straightness is fundamentally a characteristic of a line, such as the edge of a part or a line scribed on a surface. A straightness tolerance is specified on a drawing by means of a feature control frame, which is directed by a leader to the line requiring control, figure 42-10. It states in symbolic form that the line shall be straight within .006 in. This means that the line shall be contained within a tolerance zone consisting of the area between two parallel straight lines in the same plane, separated by the specified tolerance.

Theoretically, straightness could be measured by bringing a straightedge into contact with the line and determining that any space between the straightedge and the line does not exceed the specified tolerance.

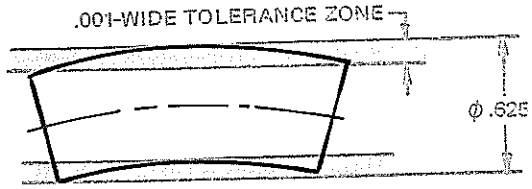
For cylindrical parts or curved surfaces which are straight in one direction, the feature control



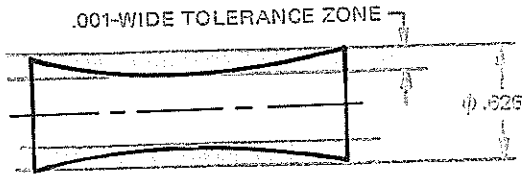
(A) DRAWING CALLOUT



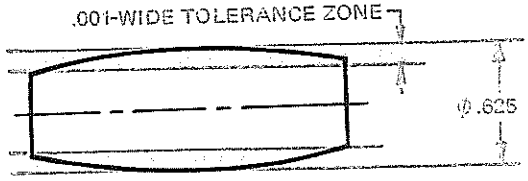
(B) REFERS TO LINE ELEMENTS ON SURFACE



Example 1 Bending error



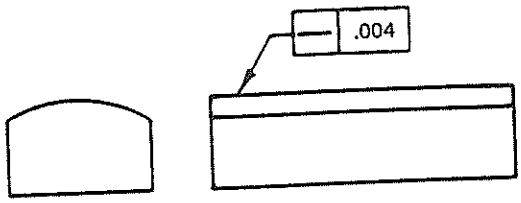
Example 2 Concave error



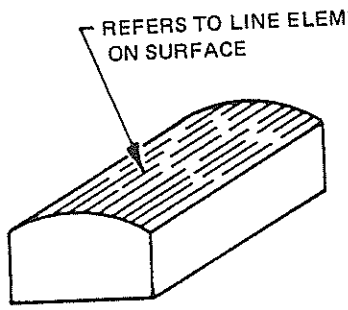
Example 3 Convex error

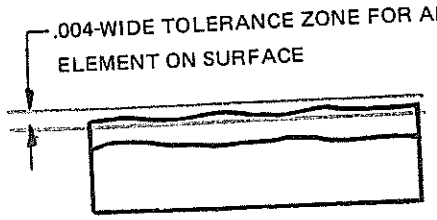
(C) POSSIBLE VARIATIONS OF FORM AT THE MAXIMUM MATERIAL SIZE

Figure 42-11 Specifying straightness of line elements of a cylindrical surface



(A) DRAWING CALLOUT





(B) INTERPRETATION

Figure 42-12 Straightness of surface line elements

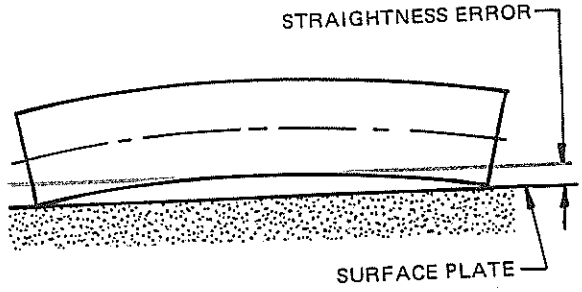


Figure 42-13 Measuring straightness of a cylindrical surface

frame should be directed to the side view, where line elements appear as a straight line, figures 42-11 and 42-12.

A straightness tolerance thus applied to the surface controls surface elements only. Therefore it would control bending or a wavy condition of the surface or a barrel-shaped part, but it would not necessarily control the straightness of the center line or the conicity of the cylinder.

Straightness of a cylindrical surface is interpreted to mean that each line element of the surface shall be contained within a tolerance zone consisting of the space between two parallel lines, separated by the width of the specified tolerance, when the part is rolled along one of the planes. Theoretically, this could be measured by rolling the part on a flat surface and measuring the space between the part

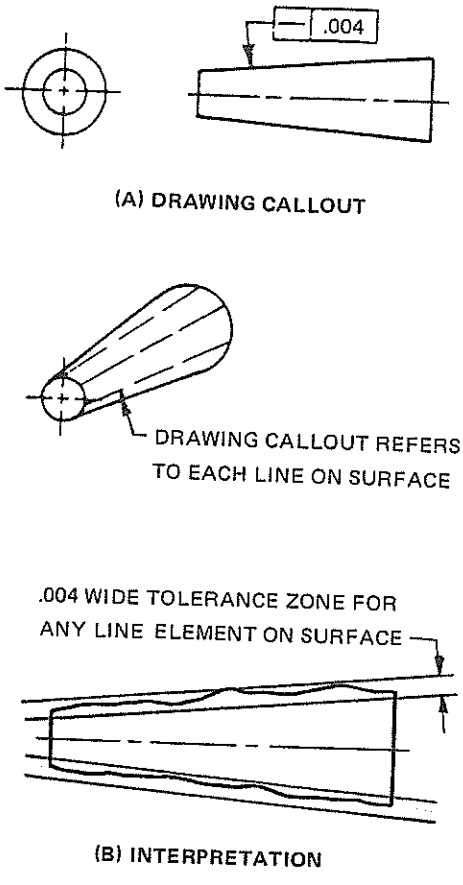


Figure 42-14 Straightness of line elements on a conical surface

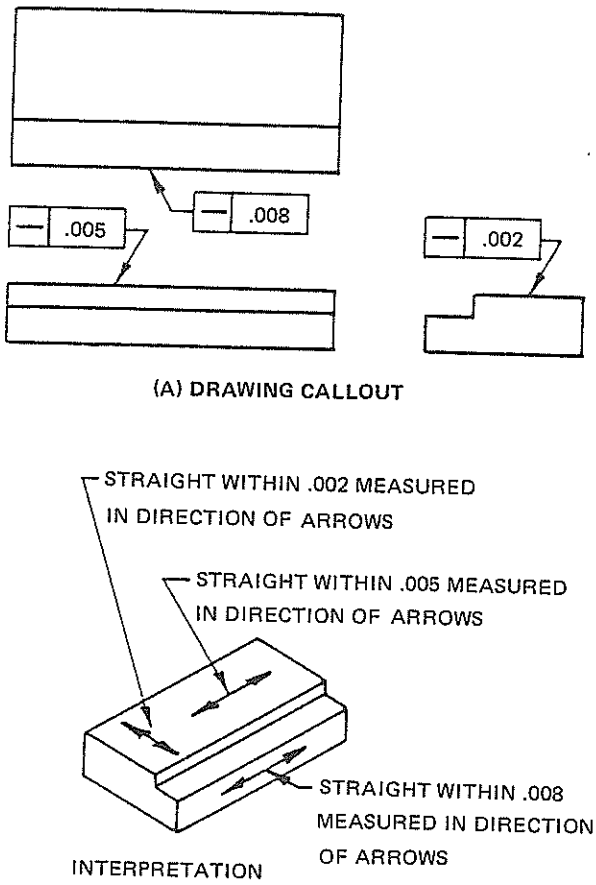


Figure 42-15 Straightness tolerances in several directions

and the plate to ensure that it did not exceed the specified tolerance, figure 42-13.

A straightness tolerance can be applied to a conical surface in the same manner as for a cylindrical surface, figure 42-14, and will ensure that the rate of taper is uniform. The actual rate of taper, or the taper angle, must be separately toleranced.

A straightness tolerance applied to a flat surface indicates straightness control in one direction only and must be directed to the line on the drawing representing the surface to be controlled and the direction in which control is required, figure 42-15. It is then interpreted to mean that each line element on the surface in the indicated direction shall lie within a tolerance zone.

Different straightness tolerances may be specified in two or more directions when required. However, if the same straightness tolerance is required in two coordinate directions on the same surface, a flatness tolerance rather than a straightness tolerance is used.

If it is not otherwise necessary to draw all three views, the straightness tolerances may all be shown on a single view by indicating the direction with short lines terminated by arrowheads, figure 42-15C.

REFERENCE

ANSI Y14.5M Dimensioning and Tolerancing