INTRODUCTION
1.0 GENERAL

1.1 Current dimensioning and tolerancing methods defined by the American Society of Mechanical Engineers (ASME) and approved by the American National Standards Institute (ANSI) can be utilized to reduce product cost. One reason cost can be reduced is that proper utilization of these methods will increase the clarity of the drawing requirements, which reduces ambiguity. It is much easier to achieve first-time quality when requirements are clearly stated. Another reason cost can be reduced is that all the functionally acceptable variation can be specified on the drawing. Permitting the maximum amount of variation has many manufacturing benefits that reduce part cost.

1.2 Current practices can achieve a clarity of requirements such that each dimension or tolerance can be shown so that only one meaning exists. Errors in manufacturing or inspection are reduced when only one meaning exists for specified dimensions and tolerances. There should not be any guessing about what must be produced.

1.3 Failure to use standardized dimensioning and tolerancing methods can result in ambiguous requirements, which may result in manufacturing, inspection, and engineering all having different opinions regarding what a dimension or tolerance means. The result can be lost time and scrap parts - all because of ambiguous requirements - not because of fabrication capabilities.

1.4 Maximizing allowable tolerances can reduce fabrication cost when increased tolerances permit less expensive processes to be used. Current dimensioning and tolerancing methods permit utilization of all functionally acceptable variation. Tolerance zone shapes are functional and therefore permit use of larger tolerances than do non-functional shapes. Functional shapes are typically related to the geometry of the controlled feature, such as a round position tolerance for a round hole.

1.5 Tolerance zones can be permitted to vary in relationship to the produced feature size. This maximizes allowable tolerance and ensures that all functionally good parts are acceptable.

1.6 Obsolete dimensioning and tolerancing techniques did not take advantage of the full functional tolerance and therefore unnecessarily restricted fabrication requirements.

2.0 COMPLETION LEVEL OF FIGURES

2.1 Figures contained in the dimensioning and tolerancing sections of this manual are complete to the extent necessary to illustrate the concept explained in the associated text. The figures are not complete technical drawings. These simplified figures make it easier to find the illustrated concept and avoid the confusion that can be caused by including features not yet explained.

2.2 Notes in the figures are considered part of the drawing when all characters are upper case. Explanations are shown using lower case letters.

3.0 BASIS FOR GUIDELINES

Each dimensioning and tolerancing section includes a statement in the first paragraph to indicate the standard in force at the time the section was written.

3.1 Guidelines defined in the dimensioning and tolerancing sections are based on the national standard noted in the first paragraph of each section. Some guidelines extend the principles of the national standard. Explanations of these principles are consistent with expected advances to be made as future editions of the standard are completed.

3.2 Explanations of extended principles are identified in the text by providing a caution regarding their usage. In some instances the caution indicates that a note should be added to the drawing for clarification of how the extended principle should be interpreted. If the standard is updated to include the extended principle, at that time it will no longer be necessary to use the notes on drawings completed subsequent to the new standard being issued. It should be recognized that any use of an extended principle introduces some risk of disagreement regarding interpretation.

3.3 Any extended principle shown in this manual is not a prediction of material to be contained in a future edition of the national standard. These extensions are only an attempt to fill voids currently in the standard and to advance capabilities. It is likely that many of the extensions will be considered for future editions of the standard, but it is also possible that different interpretations could be included when the standard is updated.

3.4 It is possible to avoid a conflict between drawings made today and future standards that may create new or changed guidelines. Each drawing should include a note that states the issue of the standard on which the drawing is based. Additional notes should be applied to each specification that is an extension of principles. The notes must explain how the extended principle is interpreted.

3.5 To indicate the applicable standard, a note such as the following is recommended:

> DIMENSIONING AND TOLERANCING IN
> ACCORDANCE WITH ASME Y14.5M-1994.

It is important to include the year of the standard in the note since requirements have advanced in each edition.

3.6 Some items in these dimensioning and tolerancing sections of the manual are not covered by the standard but are widely used practices. These items are identified in the text with a

*Supersedes issue of September 1995

caution. When these items are used, a note should be included on the drawing to indicate how to interpret each of them. As these practices are included in future editions of the standard, the notes may be omitted from subsequent drawings. References to the “national standard” or “the standard” contained in Section K6 of this manual are related to ASME Y14.5M-1994.

4.0 GAGING EXAMPLES

4.1 Explanations of some tolerance applications include gages for tolerance verification. The gage dimensions and example calculations omit gage design tolerances that would be required to fabricate gages. This permits a clear explanation of the tolerance interpretation for the workpiece.

5.0 UNITS OF MEASURE

5.1 Dimensioning and tolerancing fundamentals are the same regardless of the unit of measurement used on the drawing. Only the dimension value and the treatment of decimal values is impacted. Unit values and how they are shown are explained in the General Tolerances section.

5.2 Some figures are illustrated with metric units and others include inch units. The unit of measurement has no effect on the dimensioning and tolerancing concept.

SYMBOLS

1.0 GENERAL[1]

This section is based on ASME Y14.5M-1994. Symbols unique to ANSI Y14.5M-1982 are also described for those who are contractually obligated to comply with the older standard.

1.1 Symbols permit consistency in the way dimensions and tolerances are specified, and each symbol has a clearly defined meaning. Symbols take less time to apply on a drawing than would be required to state the same requirements with words. The symbols also require considerably less space. See Figure 1.

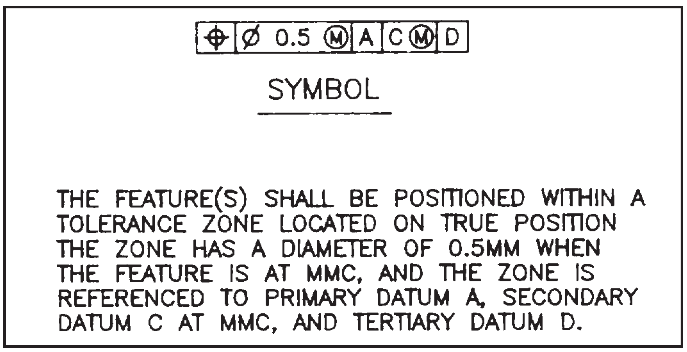


Figure 1. Equivalent Symbol and Note

1.2 The symbols are presented in two groups for easier use of this section as a reference. General dimensioning symbols are shown first. Some of these symbols are also used in tolerance specifications. The second set of symbols are used for tolerances.

1.3 Symbol sizes are shown in the figures as values proportional to the letter '**h**'. The letter '**h**' represents the predominant character height on a drawing. If a symbol dimension is shown as 1.5h, and the predominant character height on the drawing is to be 3mm, then the symbol dimension is 4.5mm (1.5 x 3mm).

1.4 Symbol proportions defined in the standard are recommendations. Some companies find it desirable to vary from the recommended proportions for improved microfilm reproduction capability. Symbol proportions within a company, and certainly within a single drawing, should be consistent.

1.5 Symbols are not generally used in text or notes lists. Abbreviations and symbol names are used in text or notes lists.

2.0 GENERAL SYMBOLS

General symbols are used with dimensions to clarify the requirement defined by a dimension value and to minimize the number of words or abbreviations placed on a drawing.

[1] Figures in this section were created using Auto CAD software, a product of Autodesk, Inc. and plotted on a Summagraphics (Houston Instrument Engineered) DMP 160 series plotter.

2.1 *Diameter* - A diameter symbol is placed in front of any dimension value that is a diameter. See Figure 2.

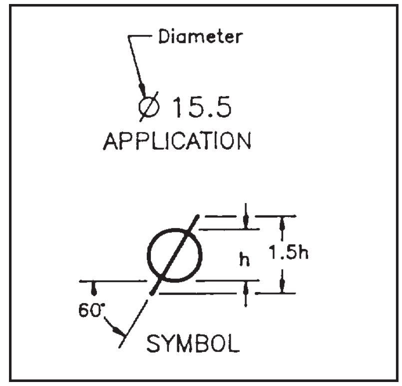


Figure 2. Diameter Symbol

2.2 *Counterbore Symbol* - A counterbore symbol combined with a diameter symbol is placed in front of a specified counterbore or spotface diameter. See Figure 3.

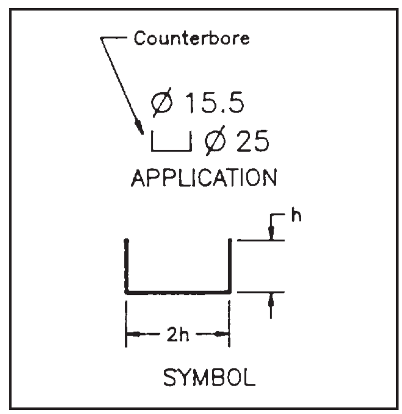


Figure 3. Counterbore Symbol

2.3 *Countersink* - The countersink symbol combined with a diameter symbol is placed in front of a specified countersink diameter. See Figure 4.

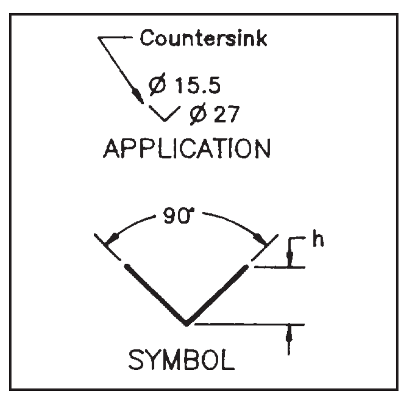


Figure 4. Countersink Symbol

2.4 *Depth* - A downward-pointing arrow is used for the depth symbol, and it is placed in front of the depth value in such applications as for counterbore and hole depths. See Figure 5.

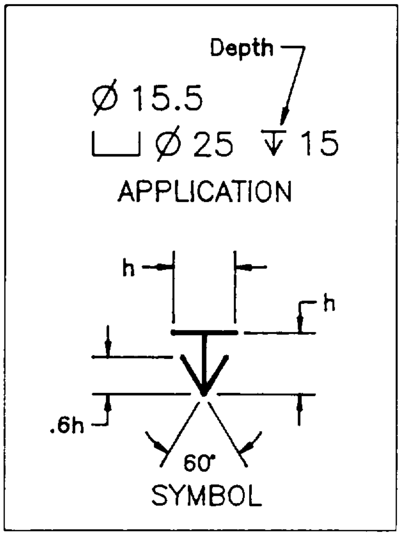


Figure 5. Depth Symbol

2.5 *Dimension Origin* - A circle used in place of one of the arrowheads on a dimension line indicates the origin for measurement. See Figure 6. This symbol is not used as a replacement for datums and datum references.

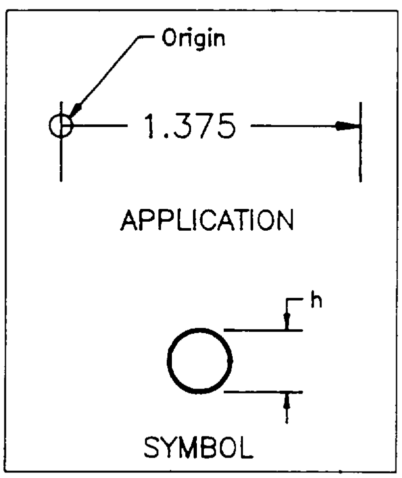


Figure 6. Dimension Origin

2.6 *Square* - A square is placed in front of dimensions for square features. See Figure 7. The square symbol is used in a manner similar to the diameter symbol. One dimension is adequate for a square shape when this symbol is used - eliminating the need for a second dimension.

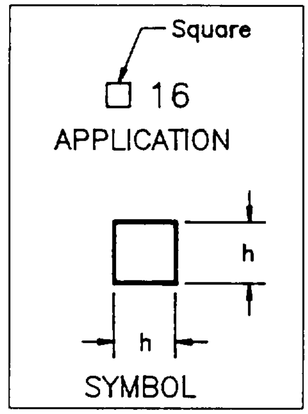


Figure 7. Square Symbol

2.7 *Reference* - Parentheses enclosing a dimension value indicates the dimension is a reference value. See Figure 8.

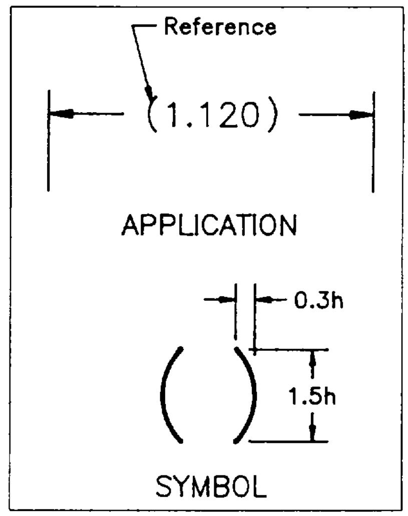


Figure 8. Reference Value

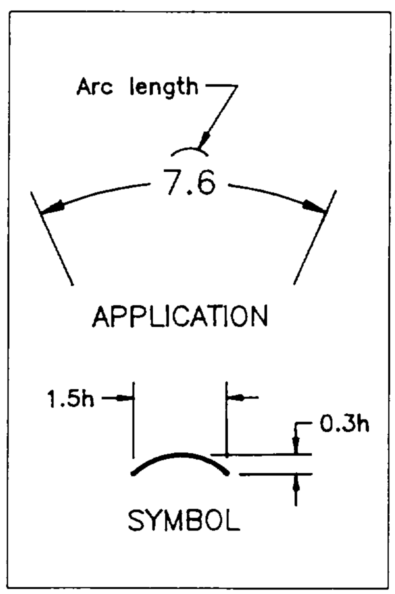


Figure 9. Arc Length

2.8 *Arc Length* - An arc drawn above a dimension value indicates the shown value is the distance measure along the defined arc. See Figure 9. It may be a distance measured along a curved surface or it may be a distance between two features such as holes in a curved part.

* Supersedes issue of September 1984

2.9 *Slope* - The slope symbol is placed at the left end of the slope specification. See Figure 10. The slope value is specified as the amount of change per unit of length.

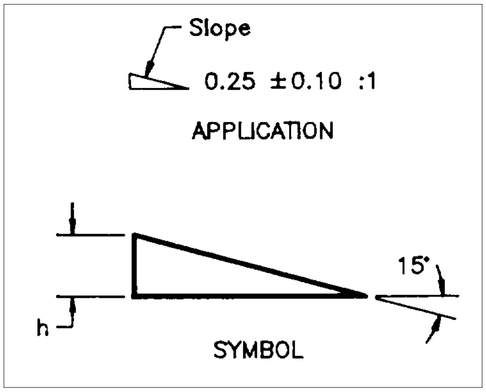


Figure 10. Slope Symbol

2.10 *Conical Taper* - A conical taper symbol is placed in front of a taper specification. See Figure 11. The specification indicates the amount of change in diameter over a unit length of the axis.

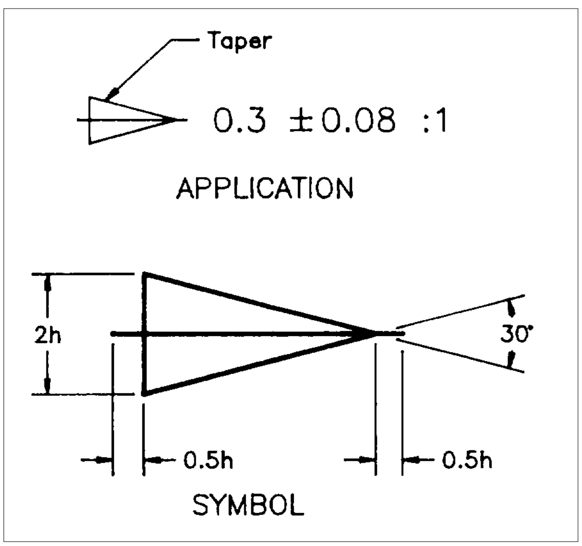


Figure 11. Conical Taper Symbol

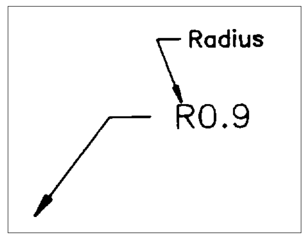
2.11 LETTERS - All letters used for symbols have a height equal to '**h**'.

2.11.1 *Letter X* - The letter **X** may be used to indicate a number of times or places a feature or item is repeated. If the **X** is to indicate the number of times or places, no space is shown between the number of occurrences and the letter **X.**

example:
3X .250-20UNC-3A

The letter **X** can also be used in place of the word **BY**. A space is placed on each side of the letter **X** for this usage.

example:
.125 X .750



2.11.2 *Radius* - The letter **R** is placed in front of any value that indicates a radius dimension. See Figure 12. Prior to the 1982 standard, the letter **R** was shown following the dimension value.

Figure 12. Radius Dimension

2.11.3 *Controlled Radius* - The abbreviation **CR** is defined by the 1994 standard to indicate a controlled radius dimension. CAUTION - This abbreviation is not defined in the 1982 standard.

2.11.4 *Spherical Radius* - The letters **SR** are placed in front of a dimension for a spherical radius. See Figure 13.

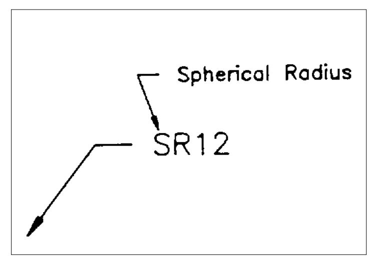
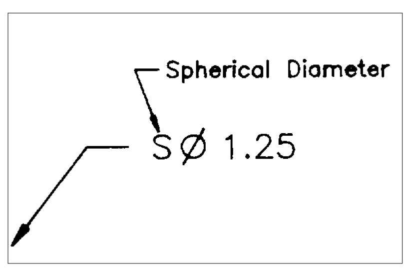


Figure 13. Spherical Radius



2.11.5 *Spherical Diameter* - A letter **S** precedes the diameter symbol, and both are placed in front of a spherical diameter dimension. See Figure 14.

Figure 14. Spherical Diameter

3.0 TOLERANCING SYMBOLS

Tolerancing symbols can be categorized as being applicable to datum identification, form, orientation, location, profile, runout, and modifiers.

3.1 DATUM IDENTIFICATION SYMBOLS - These symbols provide a means to identify datum features.

3.1.1 *1994 Datum Feature Symbol* - This symbol became the U.S. standard in 1994 and is used widely in foreign countries since it is defined by the International Standards Organization. See Figure 15.

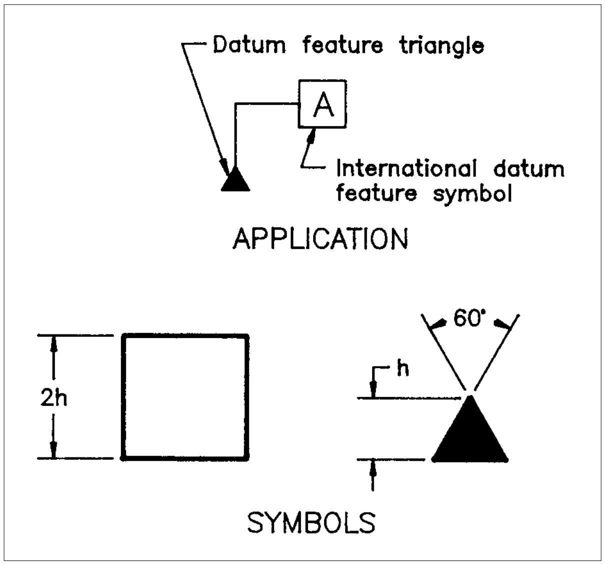


Figure 15. 1994 Datum Feature Symbol

3.1.1.1 The 1994 symbol always has a leader extending from it to the point of application. The leader includes a triangle at the end. It may be filled or left open. The triangle is sometimes referred to as a suction cup, but it is correctly identified as a *datum feature triangle*.

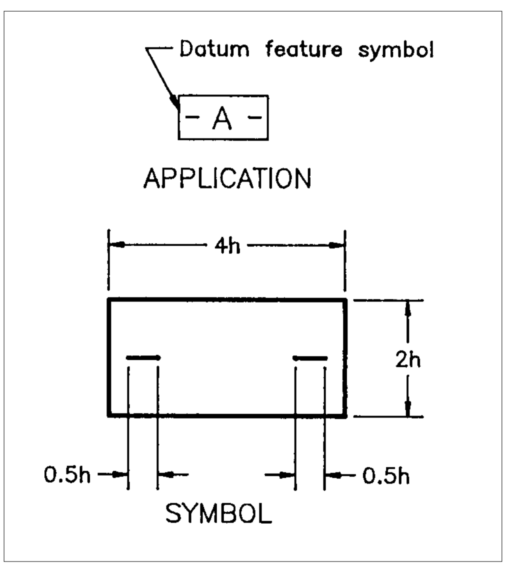


Figure 16. 1982 Datum Feature Symbol

3.1.1.2 *1982 Datum Feature Symbol* - The symbol shown in ANSI Y14.5M-1982 and earlier standards is a rectangle with two short dashes. See Figure 16. One dash is placed on each side of the letter used to identify the datum feature. Depending on the requirement, this symbol may be attached to an extension line, a dimension line, a leader, a feature control frame, or placed adjacent to a dimension value.

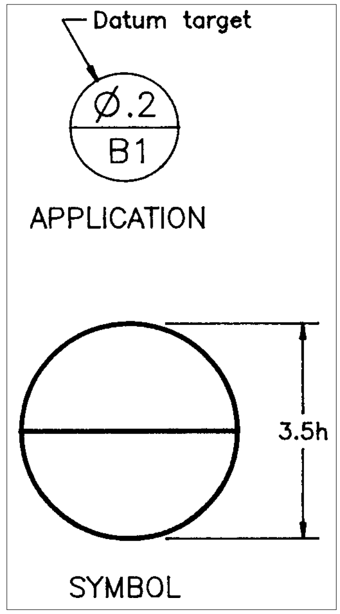


Figure 17. Datum Target Symbol

3.1.2 *Datum Target Symbol* - The datum target is a circle with a horizontal line across the middle. See Figure 17. The datum identification goes in the bottom half of the symbol, and, in the case of a datum area, the size can be shown in the top half.

3.1.3 *Target Point Symbol* - A datum target point is shown by drawing an X at the location of the point. See Figure 18. This same symbol is also used to indicate the end view of a datum target line.

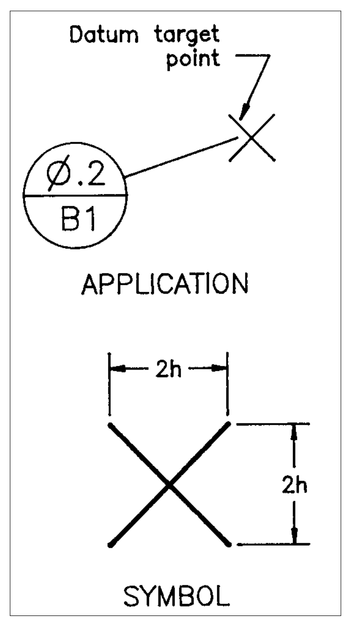


Figure 18. Datum Target Point Symbol

3.2 FORM - Form tolerance symbols include straightness, flatness, circularity, and cylindricity. Although profile can be used to control form, profile tolerance symbols are contained in a separate category of tolerance.

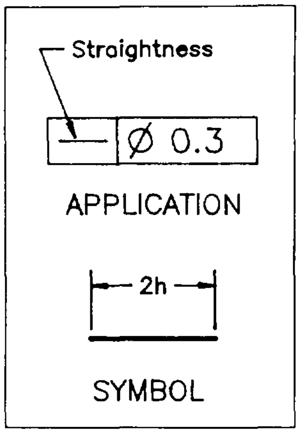


Figure 19. Straightness Symbol

3.2.1 *Straightness* - A straight line is used to indicate a straightness requirement. See Figure 19. It is only applied in a feature control frame, and may be used to control straightness of surface elements. It may also be used to control the straightness of an axis or center plane.

3.2.2 *Flatness* - The flatness symbol appears as an oblique view of a square surface. See Figure 20. This symbol is used in feature control frames and is only used to control the form variations on flat features.

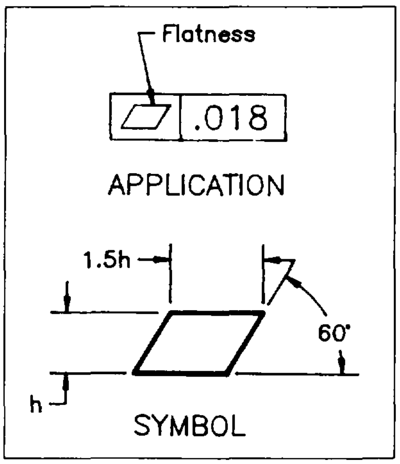


Figure 20. Flatness Symbol

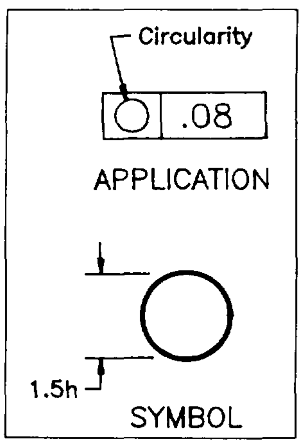


Figure 21. Circularity Symbol

3.2.3 *Circularity* - Circularity is indicated by a circle. See Figure 21. It controls the amount of form error permitted on the surface of a circular feature at individual cross sections.

3.2.4 *Cylindricity* - This symbol is a circle with two parallel lines drawn tangent to the circle. See Figure 22. It is used to control the surface errors on a cylindrical feature. It simultaneously controls circularity and parallelism of the elements on the cylinder.

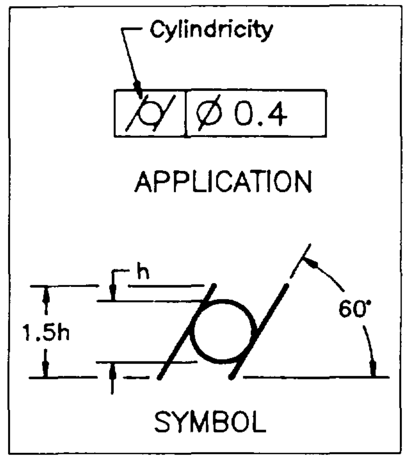


Figure 22. Cylindricity Symbol

3.3 ORIENTATION - Orientation tolerance symbols include parallelism, perpendicularity, and angularity.

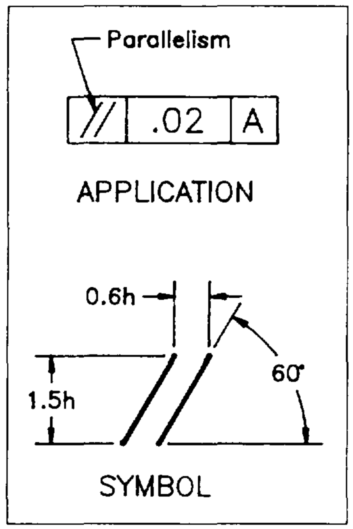


Figure 23. Parallelism

3.3.1 *Parallelism* - Parallelism is indicated by two parallel straight lines. See Figure 23.

3.3.2 *Perpendicularity* - Perpendicularity is indicated by two perpendicular lines. See Figure 24.

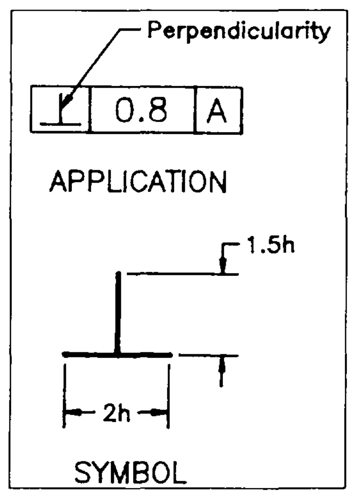


Figure 24. Perpendicularity

* Supersedes issue of September 1984

3.3.3 *Angularity* - Angularity is indicated by two lines forming a 30° angle. See Figure 25.

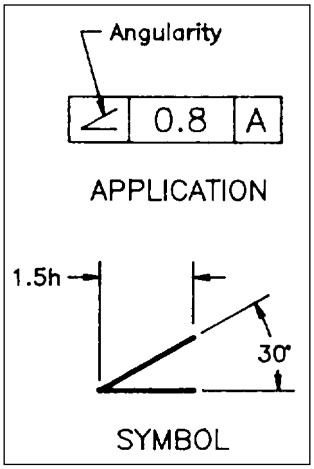


Figure 25. Angularity Symbol

3.4 LOCATION - Position tolerances are the most common location tolerance. Concentricity and symmetry are also types of location tolerances. Profile tolerances can be used to control location, but these are covered separately.

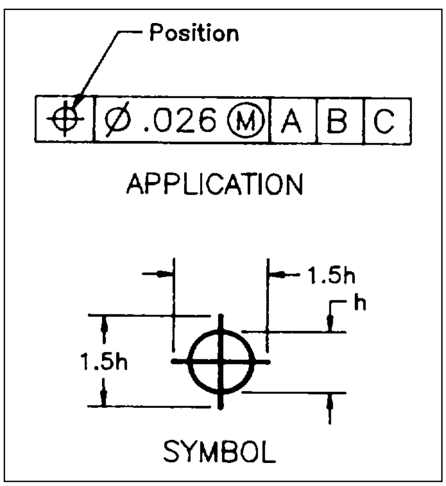


Figure 26. Position Tolerance Symbol

3.4.1 *Position* - The position tolerance symbol is a circle with a horizontal and vertical line drawn through the circle. See Figure 26.

3.4.2 *Concentricity* - Concentricity is indicated by two concentric circles. See Figure 27. The need for this symbol has been challenged, but it is still used to specify concentricity. Care should be taken to avoid the use of this symbol if the needed control can be achieved by using either position or runout tolerances.

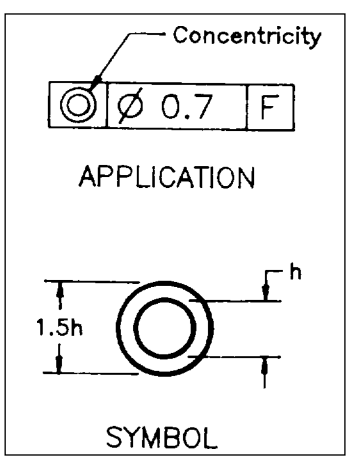


Figure 27. Concentricity Symbol

3.4.3 *Symmetry* - A special symmetry symbol existed prior to the 1982 standard. See Figure 28. This symbol became obsolete in the 1982 standard. Under the 1982 standard, any symmetry requirement is specified using a position tolerance. The symbol has been reinstated by the 1994 standard.

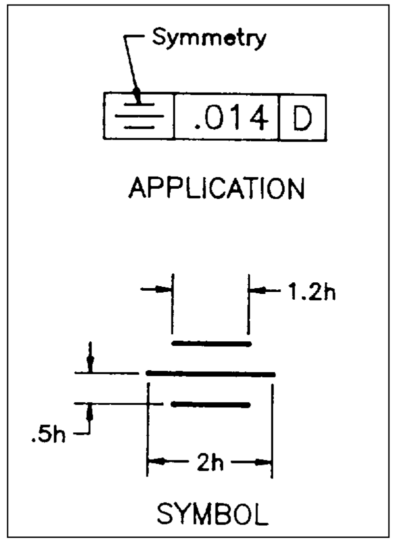


Figure 28. Symmetry Symbol

3.5 PROFILE - Line profile and surface profile are two distinctly different tolerance types, and each has a specific symbol. Both tolerance types may be used for multiple levels of control, which are described in detail in the section on profile tolerances.

3.5.1 *Line Profile* - Line profile is indicated by an arc. See Figure 29. The arc is open at the bottom. It is easy to remember the line profile symbol since it is drawn with a single curved line.

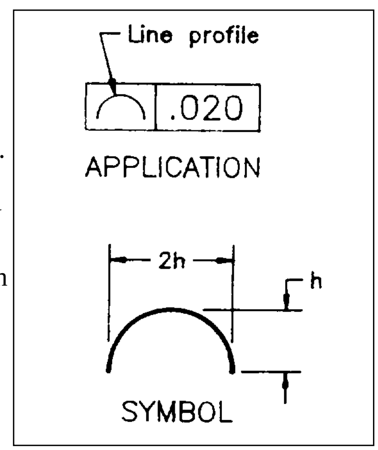


Figure 29. Line Profile

3.5.2 *Surface Profile* - The surface profile symbol is similar to the line profile symbol, except the surface profile has a straight line drawn across the bottom. See Figure 30. Although the surface profile symbol is closed, it is not filled.

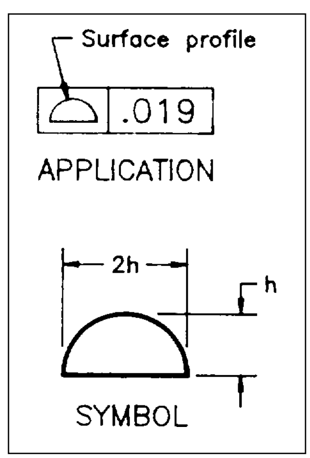


Figure 30. Surface Profile Symbol

*Supersedes issue of August 1993

3.6 RUNOUT - Two types of runout can be specified to achieve the necessary surface control of circular elements or surfaces relative to an axis of rotation. Either circular runout or total runout may be controlled.

3.6.1 *Circular Runout* - A single arrow pointing in an upward direction indicates circular runout. See Figure 31. The arrow may be filled or unfilled.

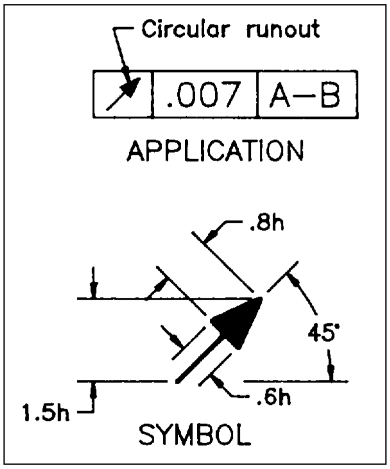


Figure 31. Circular Runout Symbol

3.6.2 *Total Runout* - Two parallel arrows pointing upward and connected by a horizontal line at the bottom of the arrows indicates a total runout requirement. See Figure 32. The arrows may be filled or unfilled.

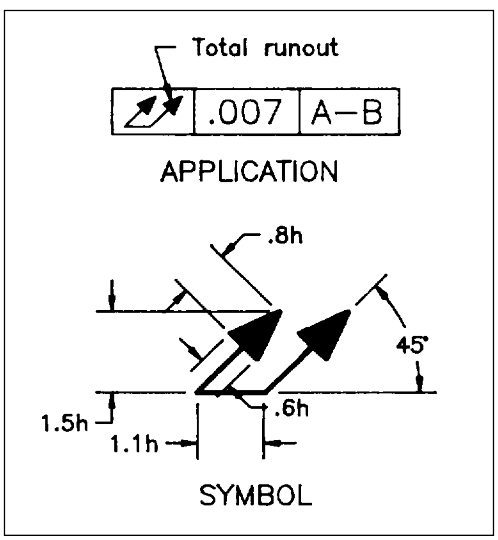


Figure 32. Total Runout Symbol

3.7 MODIFIERS - Modifiers are used to provide information about how the specified tolerances apply to the controlled feature. As an example, if a tolerance applies when a feature is at its maximum material condition, then the maximum material condition modifier is shown with the tolerance specification. Some modifiers may be used with dimensions. The specific application of each modifier is explained in the sections on tolerance application.

3.7.1 *Maximum Material Condition* - A letter **M** within a circle is the maximum material condition modifier. See Figure 33. The equivalent abbreviation for use in notes is MMC.

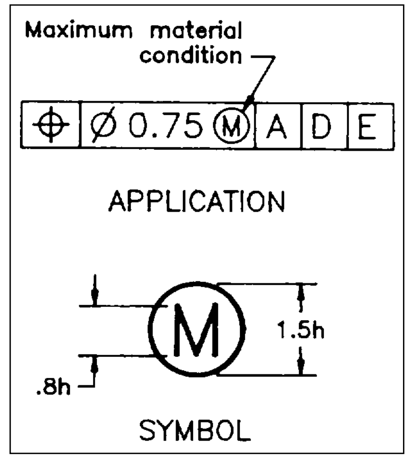


Figure 33. Maximum Material Condition Symbol

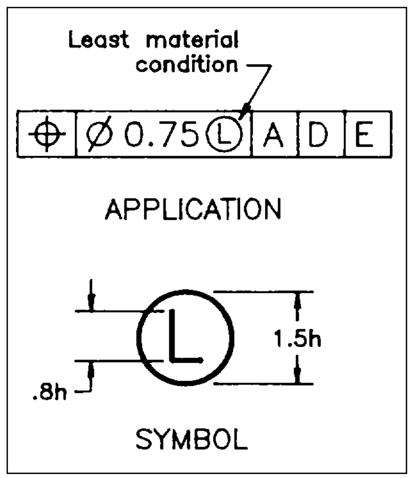


Figure 34. Least Material Condition Symbol

3.7.2 *Least Material Condition* - A letter **L** within a circle is the least material condition modifier. See Figure 34. The equivalent abbreviation for use in notes is LMC.

3.7.3 *Regardless of Feature Size* - A letter **S** within a circle indicates regardless of feature size. See Figure 35. The symbol is optional—no longer required. The equivalent abbreviation for use in notes is RFS. In compliance with ASME Y14.5M-1994, the RFS condition is assumed to apply to all tolerances unless specified otherwise.

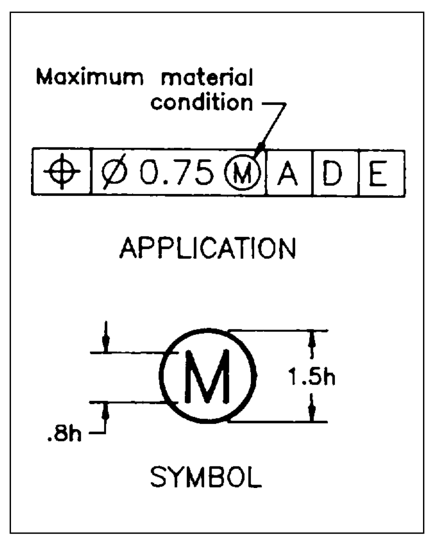


Figure 35. Maximum Material Condition Symbol

3.7.4 *Projected Tolerance* - A tolerance zone that projects beyond the surface of the part is a projected tolerance zone and is indicated by a letter **P** inside a circle. See Figure 36.

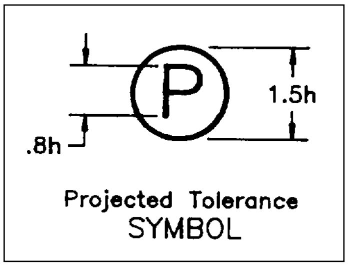


Figure 36. Projected Tolerance Symbol

3.7.5 *Statistical Value* - A tolerance value calculated on the basis of statistical analysis should be identified. For this purpose, ASME Y14.5-1994 introduced the letters **ST** enclosed within what looks like an elongated hexagon. See Figure 37. CAUTION - This symbol is not in the 1982 or earlier standards.

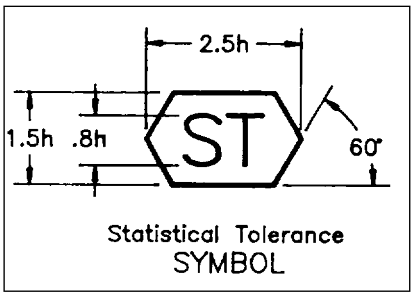


Figure 37. Statistical Value Symbol

3.7.6 *Tangent Plane* - A tolerance zone applicable to a tangent plane rather than the feature surface requires special notation. A tangent plane application is indicated by the letter **T** placed inside a circle. See Figure 38. CAUTION - This symbol is not in the 1982 or earlier standards.

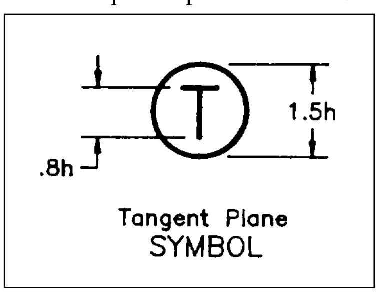


Figure 38. Tangent Plane Symbol

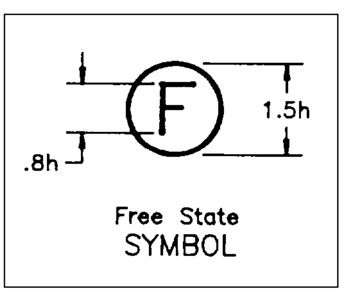


Figure 39. Free State Symbol

3.7.7 *Free State* - A letter **F** inside a circle indicates that a dimension or tolerance is applicable in the free state. See Figure 39. CAUTION - This symbol is not in the 1982 or earlier standards.

3.8 BETWEEN - A double-ended arrow placed between two letters indicates that a control extends between two labeled points on a feature. See Figure 40. CAUTION - The double-ended arrow is not in the 1982 or earlier standards.

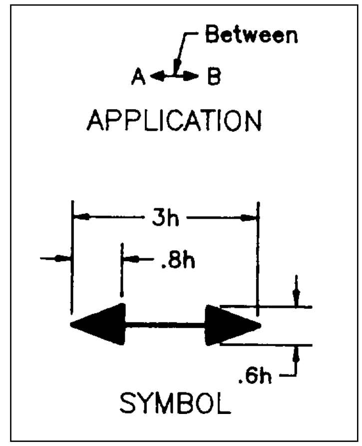


Figure 40. Between Symbol

4.0 FEATURE CONTROL FRAMES

Geometric tolerances are specified with feature control frames. See Figure 41. The feature control frames are always created and read from left to right. The tolerance characteristic is first, followed by the tolerance and any applicable datum references. Whether or not a diameter symbol or modifier is included in the tolerance specification depends on the application and the desired control.

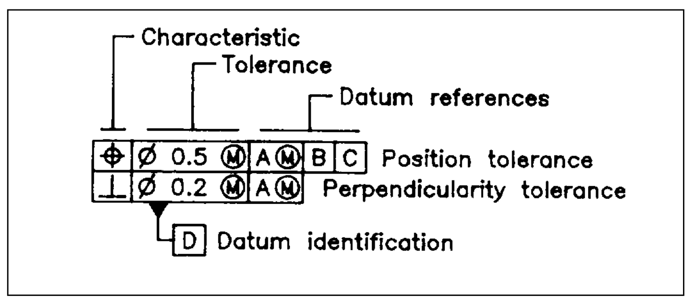


Figure 41. Geometric Tolerances are Specified wtih Feature Control Frames

4.0.1 *Feature Control Frame Lines* - A feature control frame may be made of one line containing a single specification, or multiple lines may be used to indicate multiple controls on the same feature or features. The controlled item may be identified with a datum feature symbol applied to the feature control frame.