Update 79
Gary Whitmire

DRAFTING MANUAL
Dimensioning and Tolerancing Rules

Section 6.2.2
Page 1
June, 2009

1.0 General

This section is in accordance with ASME Y14.5-2009 Dimensioning and Tolerancing. Note that Rule #1 is the only rule that is numbered in the 2009 standard. All of the other rules fall under the category of "General Rules".

2.0 Rule Number #1

The Rule #1 principle was stated by William Taylor back in 1905. The idea was to use a ring gage over an external diameter or a gage pin into a hole to simulate the interchangeability of the mating part. Rule #1 is automatic in the United States and applies to all dimensions of size with a few exceptions. Rule #1 states: "Where 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." The actual size must be within the specified tolerance at any cross-section. As the feature of size departs from Maximum Material Condition (MMC) towards Least Material Condition (LMC) size, the feature can be out of perfect form as long as the MMC envelope is not exceeded. See Figure 1 for an illustrated definition. Maximum Material Condition (MMC) size is the smallest internal size, and the largest external size. Least Material Condition (LMC) is the largest internal size, and the smallest external size.
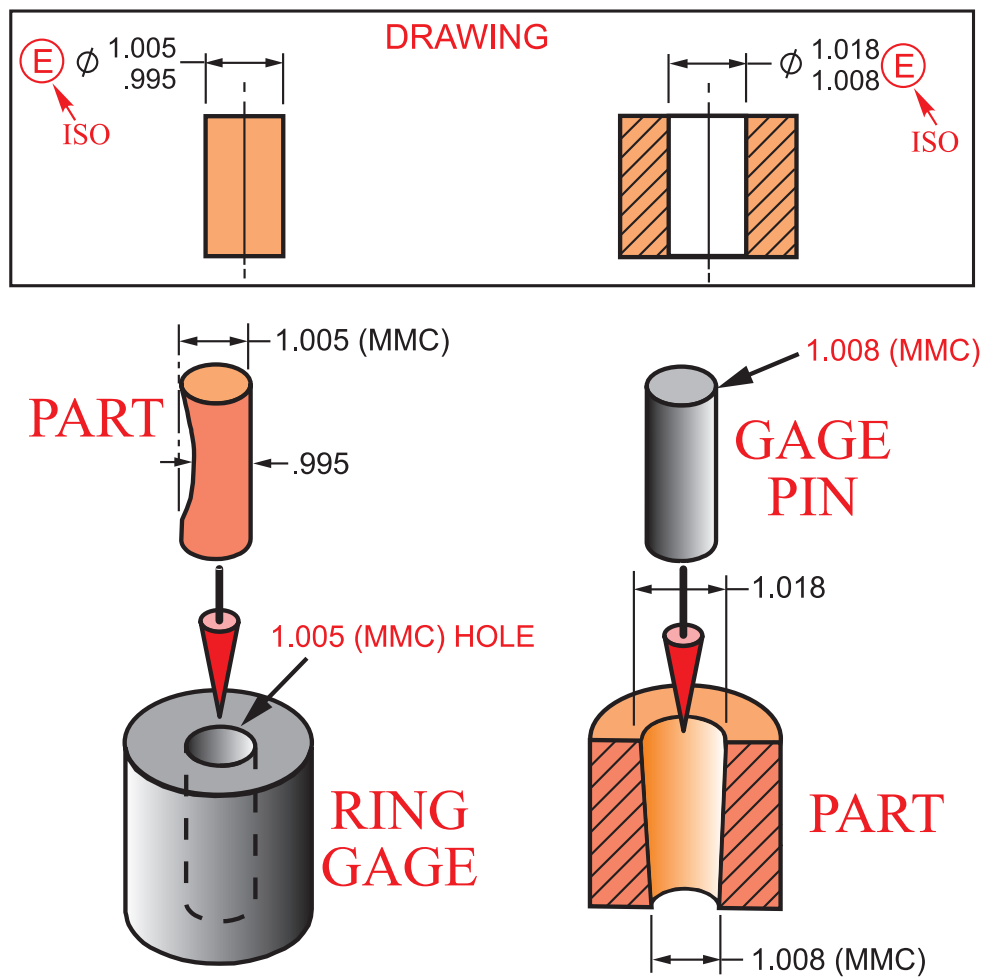

Figure 1

Update 79
Gary Whitmire

DRAFTING MANUAL
Dimensioning and Tolerancing Rules

Section 6.2.2
Page 2
June, 2009

2.1 Rule #1 International Symbol – The ISO calls Rule #1 "The Taylor Principle" and perfect form at MMC is NOT required. Where the Taylor Principle (Rule #1) is required for a feature of size, ISO uses the letter E in a circle (Envelope control) placed next to a dimension of size.

2.2 Perfect Form at LMC - Perfect form is not required at Least Material Condition (LMC) unless the letter L in a circle is specified in a feature control frame, then perfect form applies at LMC.

2.3 Features Related to Other Features – Rule #1 does not apply to the interrelationship of one feature to another feature. Figure 2 illustrates an acceptable part with respect to Rule #1. If there is a need to relate a feature to another feature, it must have a Geometric Dimensioning and Tolerance (GD&T) control such as Position, or Perpendicularity.
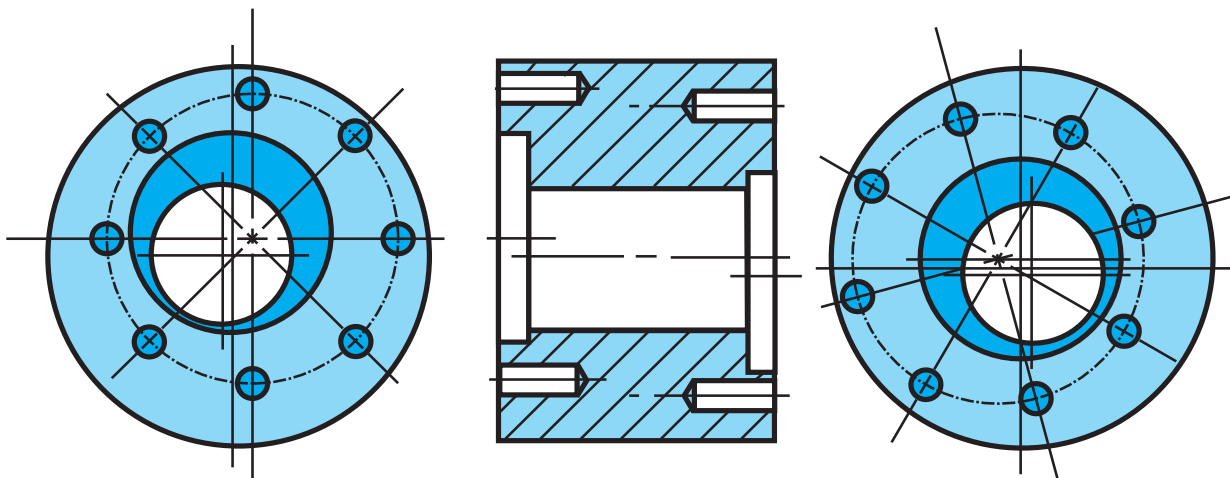

Figure 2

2.4 Removing Rule #1 – The Rule #1 requirement can be removed by adding the Independency symbol letter I in a circle placed next to a dimension or, adding a drawing note stating "PERFECT FORM AT MMC NOT REQD".

2.5 Non-rigid Features - Rule #1 is not applicable to non-rigid features or commercially milled shapes as shipped from the mill such as wire, tubing, sheets, etc. Once a part is produced from these milled shapes, Rule #1 is applicable.

2.6 Rule #1 Comments:

A. Perfect form is not physically possible; manufacturing should never produce parts near MMC. The ideal condition is for manufacturing to produce all features of size near the middle of size.

B. The full form (3D) GO gage to measure Rule #1 is at MMC size such as a pin to measure a hole and a ring gage to measure a pin, see Figure 3. The length of the MMC gage should be the length or depth of the feature being measured. Rule #1 can also be measured with a Coordinate Measurement Machine (CMM) that simulates a full form gage. The CMM is calculating an arithmetic mean based on the number of points of surface contact where the acceptable size must be towards the LMC size.

Update 79
Gary Whitmire

DRAFTING MANUAL

Dimensioning and Tolerancing Rules

Section 6.2.2
Page 3
June, 2009

C. LMC size of a feature is measured using a two-point measuring device, such as calipers, at all cross-sections.

D. Rule #1 is not understood by everyone. Some companies state in the general notes that perfect form is required at MMC for all features of size.

2.7 Why Rule #1:

A. Rule #1 automatically maintains interchangeability.

B. On welded components Rule #1 applies after welding because the weldment becomes one item.

C. Rule #1 automatically protects the corporation from bad parts if the MMC envelope is exceeded.

3.0 Rule Number 2

Rule #2 was stated in the 1994 dimensioning standard but it is now understood in the 2009 standard. Rule #2 stated that the letter S in a circle stood for Regardless of Feature Size (RFS) where the stated tolerance in the feature control frame would remain the same regardless of the feature's size. This will agree with the international (ISO) standard. See Figure 3.

Figure 3

4.0 General Rules

4.1 Dimensioning Standard and Date – A drawing need not be in accordance with the latest dimensioning and tolerancing standard. However, a general note is required on all drawings stating which version of the dimensioning standard is being used:

DIMENSIONING AND TOLERANCING IS IN ACCORDANCE WITH ASME Y14.5-2009.

4.2 Design Intent - Dimensioning and tolerancing shall clearly define engineering intent and shall conform to the following:

A. Dimensioning and tolerancing shall be complete so there is full understanding of the characteristics of each feature.

B. Each necessary dimension of an end product shall be shown. No more dimensions than those necessary for complete definition shall be given; keep reference dimensions to a minimum.

C. Dimensions shall be selected and arranged to suit the function and mating relationship of a part and shall not be subject to more than one interpretation.

Update 79
Gary Whitmire

DRAFTING MANUAL
Dimensioning and Tolerancing Rules

Section 6.2.2
Page 4
June, 2009

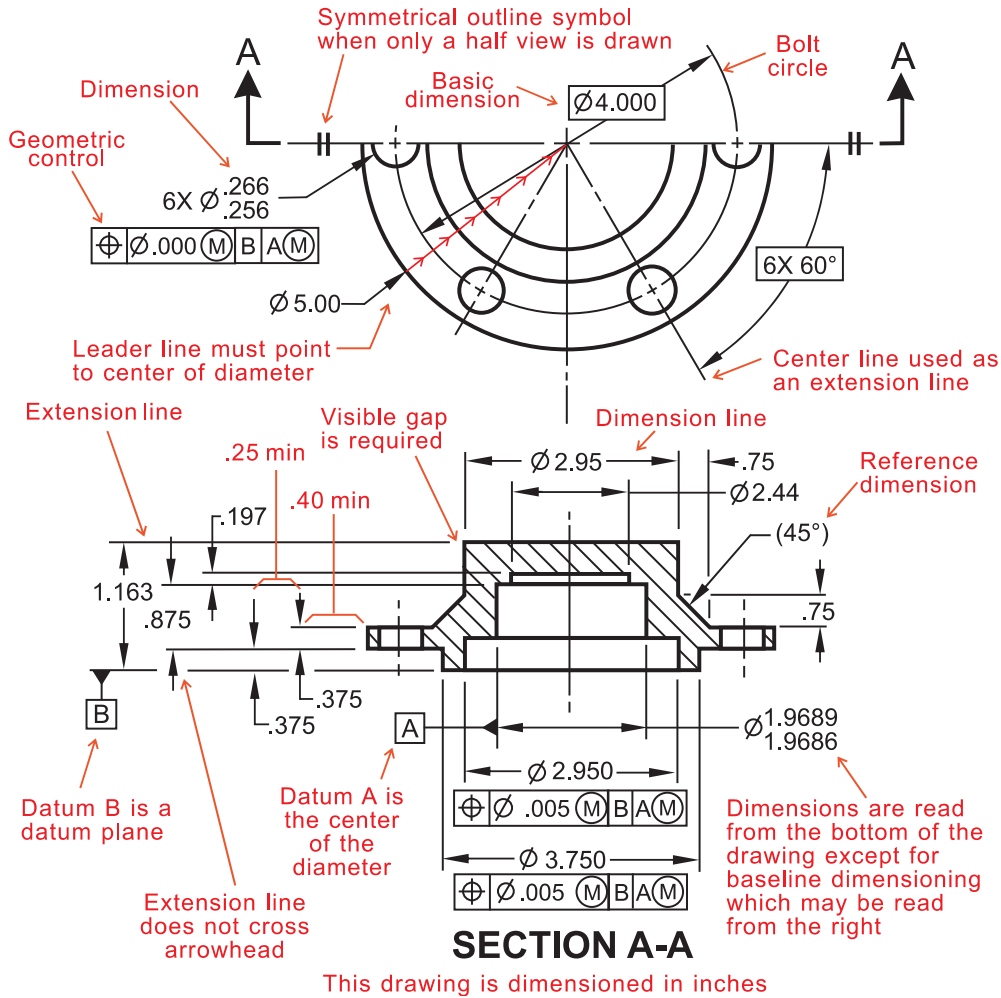**4.3 Illustrated Dimensioning Guidelines and Rules-** See Figure 4.

Figure 4

4.4 Feature Control Frame – A feature control frame which is divided into compartments shall contain a geometric characteristic symbol, geometric tolerance value, if applicable, modifiers, and datum reference letters. See Figure 5 for a few examples.
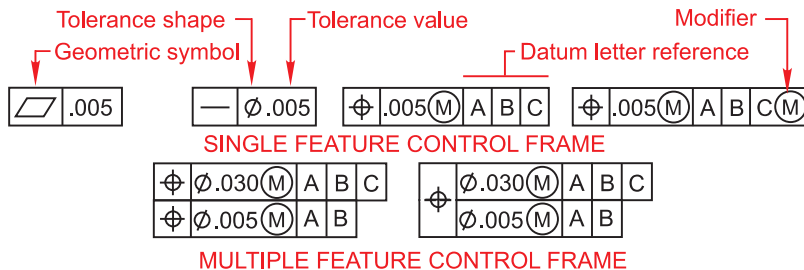

Figure 5

Update 79
Gary Whitmire

DRAFTING MANUAL

Dimensioning and Tolerancing Rules

Section 6.2.2
Page 5
June, 2009

4.5 Thread Pitch Cylinder – The GD&T tolerance stated in a feature control frame for a threaded feature is understood to apply to the axis of the pitch cylinder. The pitch cylinder axis is an imaginary axis passing through the center of the thread profile. Measuring the axis of the pitch cylinder for GD&T Perpendicularity or Position is very time consuming and requires special measuring equipment. It is preferred that the words MAJOR DIA or MINIOR DIA be specified under the feature control frame. In the Figure 6 example, the thread axis of the major diameter must fall within the .005 cylindrical tolerance zone for Position to datum "A". Datum "A" could be the axis of a shaft.
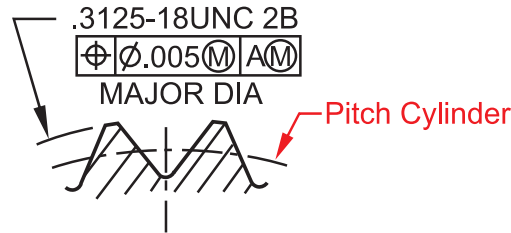
Figure 6

4.6 Gears and Splines – The GD&T tolerance stated in a feature control frame for a gear or a spline must state under the feature control frame "MAJOR DIA", "MINOR DIA", or "PITCH DIA"; this is an axis control. In the Figure 7 example, the major, minor, or pitch diameter axis must fall within the .005 cylindrical tolerance zone for position and, typically, perpendicularity to datum "A". Note that 95% of the time, the primary datum is controlling perpendicularity and in this case within the .005 tolerance zone. Datum "B" could be a hole in the center of the gear. Also note that if the perpendicularity is .004, then the position must be within .001; where the total equals .005.
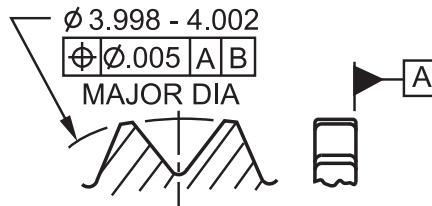
Figure 7

4.7 Features Drawn at 90 Degree Angles – A 90 degree angle is understood where lines or features are drawn at 90 degrees and no angle dimension is given. The tolerance for the understood 90 degree angles is found in the title block tolerance as shown in Figure 8.
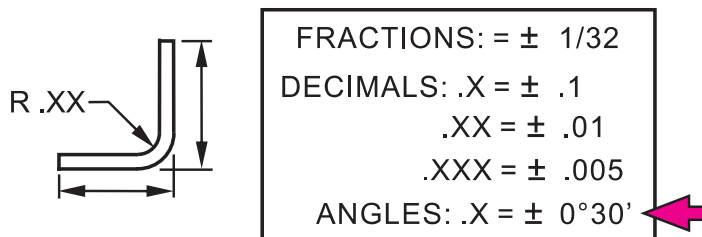
Figure 8

Update 79
Gary Whitmire

DRAFTING MANUAL
Dimensioning and Tolerancing Rules

Section 6.2.2
Page 6
June, 2009

4.8 BASIC 90 Degree Angles – Two, three, or four features drawn at an implied increment of 90 degrees need not have a angular dimension unless it is needed to improve the interpretation of the drawing. See Figure 9.
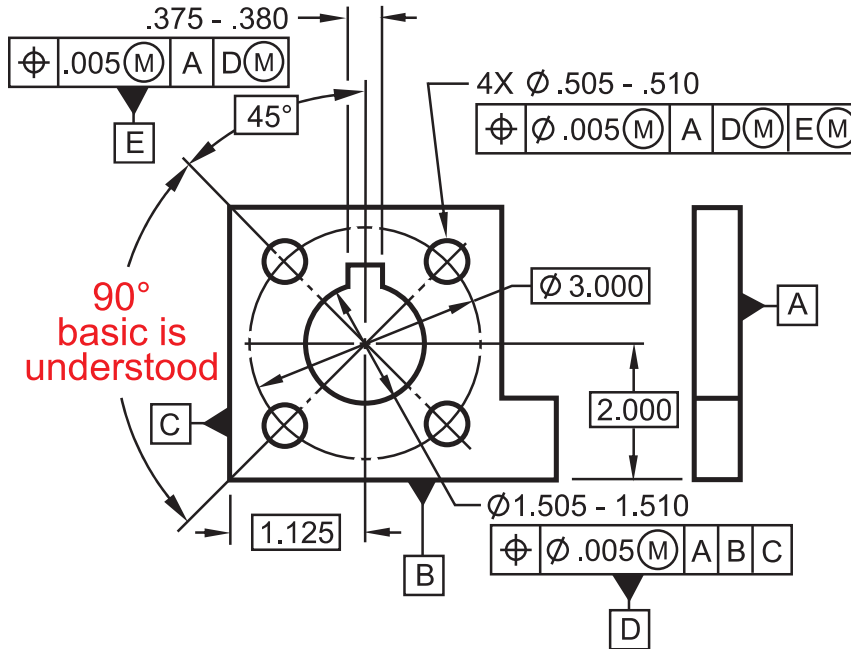

Figure 9

4.9 Specifying Manufacturing Methods - The drawing should define a part without specifying manufacturing methods. The words drill, ream, punch, or made by any other operation should not be specified. Where manufacturing, processing, quality assurance, or environmental information is essential to the definition of engineering requirements, it shall be specified on the drawing or in a document referenced on the drawing.

4.10 Gage or Code Numbers - Wires, cables, sheets, rods, and other materials manufactured to gage or code numbers shall be specified by linear dimensions indicating the diameter or thickness. The gage or code number may be shown in parentheses following the dimension with a tolerance so that receiving and inspection can accept the material.

4.11 Temperature - Unless otherwise specified, all dimensions are applicable at 20°C (68°F) per ANSI/ASME B89.6.2. Compensation may be made for measurements made at other temperatures.

4.12 Free-State Condition - Dimensions and tolerances apply in a free-state condition. This principle does not apply to non-rigid parts where a restraining note may be required to simulate the assembled condition or the use of a free-state symbol (letter F in a circle) where the free-state variation is the distortion of a part after removal of forces applied during manufacture such as weight and flexibility of the part and the release of internal stresses resulting from fabrication. This is when the part must meet the tolerance requirements while in the free-state.

Update 79
Gary Whitmire

DRAFTING MANUAL
Dimensioning and Tolerancing Rules

Section 6.2.2
Page 7
June, 2009

4.13 Tolerances – Each dimension shall have a tolerance, except for those dimensions specifically identified as reference, maximum, minimum, or stock (commercial stock size). The tolerance may be applied directly to the dimension, indicated by a general note, title block tolerance, Gage maker's tolerance, or within a feature control frame for BASIC dimensions.

   A. Unless otherwise specified, all tolerances apply for the full depth, length, and width of a feature.

   B. Tolerance values may be expressed in a CAD product definition data set per ASME Y14.41.

   C. The depth of a hole is understood to be from the outer surface of the part unless a dimension specifies otherwise. See Figure 10.
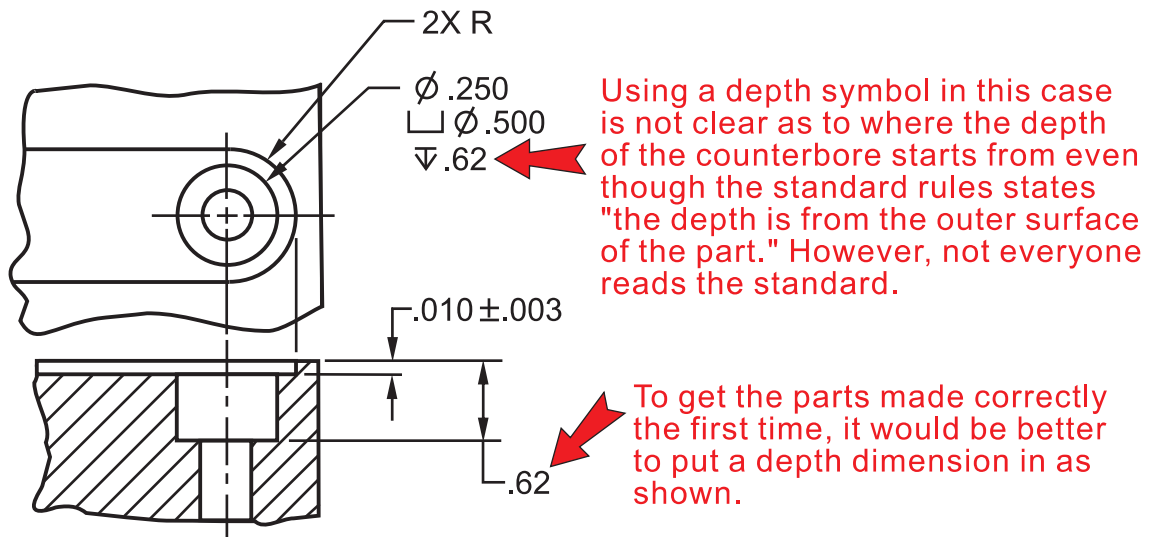

Using a depth symbol in this case is not clear as to where the depth of the counterbore starts from even though the standard rules states "the depth is from the outer surface of the part." However, not everyone reads the standard.

To get the parts made correctly the first time, it would be better to put a depth dimension in as shown.

Figure 10

   D. Each dimension shall have a tolerance, except for those dimensions specifically identified as reference, maximum, minimum, or commercial stock size.

4.14 Dimension Drawing Levels - Dimensions and tolerances apply only at the drawing level where they are specified. A dimension specified for a given feature on one level does not apply at a higher level.

4.15 Process Dimensions - It is permissible to identify as non-mandatory certain process dimensions that provide for finish allowance, shrink allowance, and other requirements, provided the final dimensions are given on the drawing.

   6.00 SHRINK ALLOWANCE NONMANDATORY (MFG DATA), FINAL LENGTH 5.80±.04

4.16 Un-dimensioned Drawings – Dimensions may not be needed on loft, printed wiring, templates, and master layouts prepared on stable material.

4.17 Coordinate System – Where the coordinate system is shown on the drawing, it shall be right-handed unless otherwise specified. Each axis shall be labeled and the positive direction shown.

Update 79
Gary Whitmire

DRAFTING MANUAL
Dimensioning and Tolerancing Rules

Section 6.2.2
Page 8
June, 2009

4.18 Zero BASIC – Zero BASIC is understood where axes, center lines, center planes, and surfaces are shown at right angles or parallel to each other where BASIC dimensions or geometric tolerances have been specified such as position or profile.

4.19 Dimensions Coming from Centerlines (not a rule in the standard) – Figure 11 illustrates the condition where dimensions are generated from centerlines with no knowledge as to which feature the centerlines represent on the part. Figure 11 is a poorly dimensioned part and needs a datum(s) or a general note stating the functional features such as:

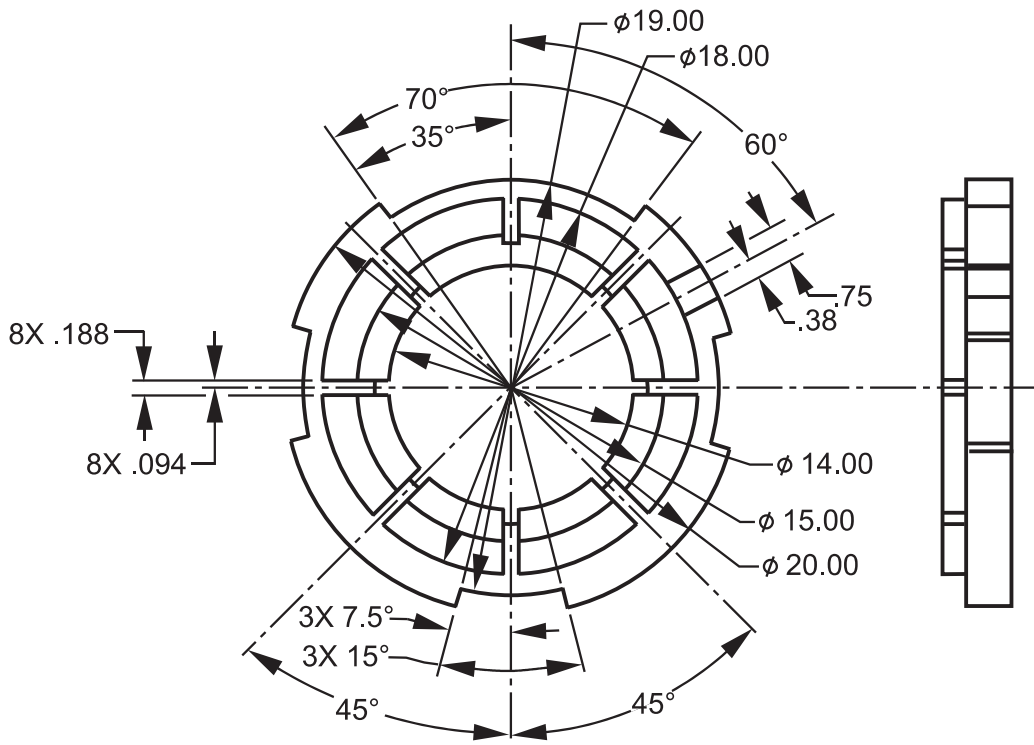ALL DIMENSIONS ARE COMING FROM DIAMETER XX.XX AND THE VERTICAL SLOT.

Figure 11