NOTICE WARNING CONCERNING COPYRIGHT RESTRICTIONS:
The copyright law of the United States (title 17, U.S. Code) governs the making of photocopies or other reproductions of copyrighted material. Any copying of this document without permission of its author may be prohibited by law.

A Tool for the Correct Appplication of Geometric Tolerances

by

Kurt A. Laughlin, Thomas R. Kurfess

EDRC 24-44-91

A TOOL FOR THE CORRECT APPLICATION OF GEOMETRIC TOLERANCES

by

Kurt A. Laughlin

Department of Mechanical Engineering

Carnegie Mellon University

Pittsburgh, PA 15213

Thomas R. Kurfess

Assistant Professor

Departments of Mechanical Engineering

and Engineering and Public Policy

Carnegie Mellon University

Pittsburgh, PA 15213

8 March 1991

This work has been supported by the Engineering Design Research Center, a NSF Engineering Research Center

University Libraries
Carnegie Mellon University
Pittsburgh PA 15213-3890

Abstract

The tolerance rules used by designers to specify the allowable variation of manufactured parts are often erroneosly applied. This paper describes a reformulation of the methodology standardized by ANSI Y14.5M-1982 that guides the user toward tolerance specifications that are complete and correct with regard to the standard. The reformulation can be automated for use with Computer-Aided Design and Drafting systems.

A Tool for the Correct Application of Geometric Tolerances

Kurt A. Laughlin

Department of Mechanical Engineering, Carnegie -Mellon University

Pittsburgh, PA 15213

Thomas R. Kurfess

Departments of Mechanical Engineering and Engineering and Public Policy

Carnegie -Mellon University

Pittsburgh, PA 15213

Introduction

The design of a mechanical object is not complete after the determination of its physical size and configuration. A perfect object, unvarying from the desired form does not exist in the realm of manufacturing. Even parts made on the same machines at the same time are not truly identical because of tool wear, variations in part set-up and fixturing, and small differences in the material properties from piece to piece. Although the designer cannot (and should not) specify every detail of the manufacturing process, there is a method by which they can manage these variations such that part performance is not affected. Dimensioning and tolerancing methods provide a system for the designer to control not only the variations in the size of a part but also in its form.

It has been said that the language of engineering is mechanical drawing. Here the ideas of the designer are transformed into a physical form that is understandable by all those involved in the manufacturing process. Modern standards and standardization agreements enable one to read drawings from many sources, even if they are in a different language. The standard governing Dimensioning and Tolerancing in the United States is ANSI Y14.5M-1982,

published and administered by the American Society of Mechanical Engineers. This document describes the principles and practices relating to controlling part variation. Unfortunately, no matter how explicit the text or numerous the examples, no standard can consider every possible situation. Nor can a standard be a substitute for training, experience or common sense. For a part to function properly the drawings used to fabricate it must be prepared properly.

Incorrect or incomplete instructions are likely to be misunderstood or ignored. In an attempt to aid the designer in using Y14.5M the its sections relating to Geometric Dimensioning and Tolerancing were distilled into a concise algorithm that includes queries for tolerance values, datum references, material condition, and specialized modifiers, and even discriminates between tolerances applied to features of size and those applied to surfaces. Converted to flowchart form, it can be used as a manual aid to check tolerancing callouts. It may be useful as a non-mandatory appendix to the Standard. Figure 1 is a generalized flowchart for ANSI Y14.5M-1982. Used as the logic for a computer program, the algorithm is being developed into a system that not only monitors specifications in a CAD environment but prompts the user for the information required. However, the program, much like the Standard, makes no claim that a "correct" callout, i.e. one conforming to the Standard, is appropriate or even functional in a given situation - for now these decisions must be based on experience.

Tolerances

Limit dimensioning (plus/minus or max/min) is probably the most familiar type of tolerancing. It establishes the limits for an object in one-dimension, such as a length or diameter. While seemingly adequate for defining part configuration they do not provide any control on the orientation of features, such as a boss or hole, with respect to the rest of the part. As the name states, limit dimensioning only defines boundaries for allowable part configurations - any variations within those limit boundaries are uncontrolled. Geometric tolerances allow the designer to specify a part's configuration more exactly and completely. The various types of geometric tolerances are named for the type of control they exert, such as flatness, parallelism, and position and the tolerance boundaries defined mimic the conditions described in their names: parallelism defines two planes parallel to a third, angularity two planes at a specified angle etc. The

geometric boundaries are generally smaller than the limit dimension boundaries and are related by the drawing callout to other part features called datums. Figure 2 shows a typical geometric tolerance. In this case the top surface of a block must be kept between two planes 0.030 units apart and parallel to the bottom (datum) surface. The position of the two planes is not fixed, but they must lie completely within the two limit dimension boundaries. Figure 3 illustrates this relationship in an exaggerated view.

For geometric tolerances to be understood, the information given in drawing callouts must be complete and correct. As a minimum, the type of tolerance must be shown and a tolerance value must be given. Some tolerance types require datums while others actually need two tolerance values. The omission of any of this information would make the tolerance meaningless. An additional conceptual concern is the selection of the tolerance type itself. It seems obvious that one would not control the circularity of a square bar, but tolerancing the concentricity of that bar may not appear to be incorrect.

Algorithm Development

This project attempts to supplement experience, at least in correctly supplying data, by providing a systematic method for specifying geometric tolerances. The algorithm is based on an analysis of the Dimensioning and Tolerancing standard, ANSI Y14.5M - 1982. It was found that the information required for virtually any tolerance application can be listed on one or more of nine data forms or menus. A tenth form is needed to collect datum information.

Through this generalization it is possible to design a compact flowchart to lead the designer through the specification process. The organization of the menus is shown in Figure 4. Figure 5 depicts the initial or main menu listing showing the applicable tolerance types. Some tolerance types apply only to surfaces or features of size. For this reason two menus exist, each listing the tolerance types applicable to the chosen feature type. Example tolerance type menus are given in Figure 6. These provide a convienent way to keep track of the information needed for the tolerance callout. In some cases the information required differs for a toleranced surface as opposed to a toleranced feature of size (see STRAIGHTNESS and GROUP 3). A check mark in the "SRF or "FOS" column indicates that a particular tolerance type is applicable to surfaces and

features of size, respectively. Note that these are actually two separate menus and that one uses only the version applicable to the case at hand.

Flowchart use is as follows: First, the user selects a feature to tolerance. They then determine whether the indicated feature is a feature of size or a surface. The main menu listing (showing all tolerance types applicable to the chosen feature type) is consulted. When the tolerance type is selected, the user chooses the proper menu based on surface/feature of size classification (if necessary) and supplies the information required. If datum referencing is required the last menu is used to apply the desired material condition modifiers.

Three examples are presented to illustrate the flowchart application.in some detail

Example 1: Flatness

In this simple example the user wishes to control the form of the upper surface of the part shown in Figure 7. The surface itself is located by the 2.1/1.9 dimension.

After determining that it is a surface and not a feature of size under consideration, the "SURFACE" menu is consulted (Figure 8). Choosing "FLATNESS" as the tolerance type gives the symbol shown. Moving down the chart, the flatness tolerance value is selected as .030, which is acceptable per the error-checking as it is smaller than the tolerance size limit positioning the surface (.030 < [.1 +.1 =] .2). Since the user does not wish to specify a unit length or total flatness value, the tolerance is now completely specified.

Example 2: Profile of a surface

Now consider an arbitrary surface defined by a mathematical or tabulated function of a dimension "x" as shown in Figure 9. A particular point on this surface needs to be located 2.64/2.14 above the surface labelled "A". The arbitrary surface, wherever it is located, can vary away from the ideal form shown, toward "A" only, by as much as .100. Assume that the entire surface must be controlled and not just linear elements or slices.

Since it is a surface to be toleranced the user again moves to the SURFACE menu and chooses "PROFILE - SURFACE" and its symbol. The tolerance value of .100 is given and the designer verifies that this is less than

the +/- .25 limit of size tolerance locating the surface. The design is concerned with the form of the surface relative to the bottom surface "A", so this is chosen as a datum. Datum A is not a feature of size so no material condition can be specified. The user would like to note that the surface many only vary inward so they use the "UNEQUAL BIAS" modifier which is indicated on the drawing by a short phantom line drawn parallel to the surface in the direction of allowable variation. The finished callout is shown in Fgure 10.

Example 3: Position of holes

In this instance the user would like to control the position of two holes on a circular disk of thin material as pictured in Figure 11. The hole centers must lie within .060 diameter circles located 180 degrees apart on a theoretical 2.00 diameter circle. This condition must be met when the holes are at their smallest allowable size. The theoretical circle is to have the same center as the disk. The angular position of the holes relative to the disk is unimportant. The user is not concerned with any tilt of the hole axes because of the thinness of the material.

Begining at the top of figure 12 the user refers to the main menu. Going along the "P branch, "POSITION" is selected from the "FOS" main menu. After picking a tolerance value the user then determines the material condition control to apply to the hole position tolerance. The symbol for maximum material condition is chosen to signify that the tolerane applies when the hole is at its minimum allowable size. The disk diameter, datum A, entered as the primary datum. Since secondary (angular position) or tertiary (tilt) datum control is not required, these datums need not be specified. Continuing on, the user notes that datum A is a feature of size. An additional step is then taken to assign a material condition modifier to datum A. At this point the tolerance callout is complete (Figure 13).

Software Implementation

In addition to the flowchart "hardware", the algoritim is used as the basis for a computerized tolerancing aid. This program is being designed to be used as an adjunct to NOODLES, a developmental solid modeler in use at Carnegie-Mellon University's Engineering Design Research Center. Here it will help graduate students and faculty unfamiliar with geometric tolerancing create

tolerance volume models for analysis by other systems such as assembly clearance evaluators..

Current work involves converting the logic behind the flowchart into a computer algorithm. Robust entry error checking and an automatic graphical menu system are planned. Later development will be directed toward interfacing with the NOODLES system and data structure. The NOODLES modeler generates solid shapes from zero (points), one (edges or lines), and two (faces or planes) dimensional linear primitive elements. The user can also create elements from the combination of other elements. Relationships such as connectivity between elements are stored for each model in the data structure. This highly versatile system provides easy access to the information the toierancing program needs to operate. For example, the PSET structure can be used to group one or more elements under a single label. These PSETs can then be labelled as datums and referenced by the datum name. Besides datums, this structure can also be used to group elements into a convenient physical organization: Since the NOODLES system uses only linear elements, curves are represented by many short line segments and curved surfaces by many small planes. These elements normally remain discrete but the PSET structure allows relevant surfaces to be linked and treated as one entity. Therefore, if a user wants to operate on a cylinder they need only create a PSET with all of the faces, vertices, and edges to avoid the tedium of specifying each element individually.

The important analysis operations of the toierancing program exploit the PSET structure. As stated previously, PSETs can be used to label datums. By counting the number of plane faces in a PSET the program can judge if it represents a (plane) surface or a more complex object. One face obviously indicates a surface, while two or more are taken as a feature of size. Through this simple process the program can flag surface/feature of size applicability and determine if datums require material condition modifiers.

After the tolerance callouts are determined, the system stores the toierancing information that the user entered through the menus. Other systems must take these data and generate tolerance boundaries around the original nominal shape. The information can also be used with more sophisticated systems to automatically create correct drafting callout blocks.

While other computerized design and drafting systems do not necessarily use the NOODLES data structures, essentially the same methods may exist. Normal drafting operations require similar automatic "groupings", such as moving circles or locating dimension lines. It is hoped that the program can draw on these procedures for the requisite information.

As one becomes familiar with the process, following a menu structure can be quite tiresome. At this point the menu system should be used as a program that operates in "background" and only makes itself known when an operator error occurs in tolerance specification. Possible detectable errors are: incomplete information, geometric tolerance larger than size tolerance, and composite feature tolerance larger than pattern locating tolerance. As research in other fields progresses, process and functional parameters may provide rules for future error checking.

Further Applications

This project is intended to give another tool to the mechanical designer in the field of computer aided design. The next step beyond computer implementation is a system to automatically generate tolerance volume boundaries based on the information processed by this program. The main thrust of research in this area is the determination of mathematical (that is to say, computer readable) models for tolerances. With a rigorous definition, a boundary volume can be generated for any shape under any constraint. Once the extreme (or statistically probable) shape variations are known other analyses such as fits, mold design, and volumetric design can be expanded to consider any "legal" shape, not just the nominal. Volume boundaries can also be used in inspection where three dimensional models of design tolerances and manufactured part shapes can be manipulated to simultaneously verify the compliance of interrelated features.

References

Dimensioning and Tolerancing, ANSI Y14.5M-1982, The American Society of Mechanical Engineers, New York, 1982

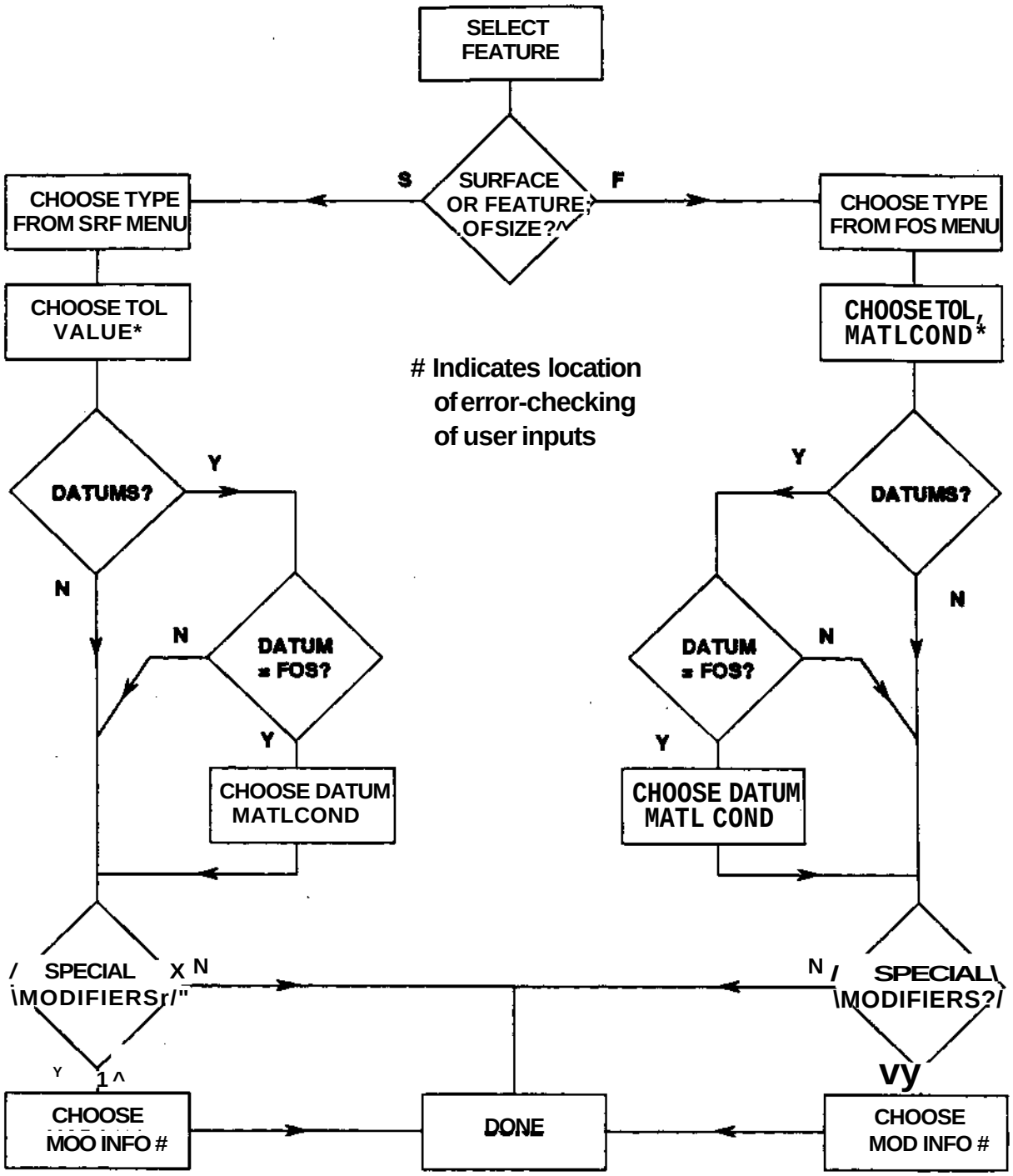

SELECT FEATURE
SURFACE OR FEATURE OF SIZE?
S
F
CHOOSE TYPE FROM SRF MENU
CHOOSE TYPE FROM FOS MENU
CHOOSE TOL VALUE*
CHOOSE TOL, MATLCOND*
# Indicates location of error-checking of user inputs
DATUMS?
Y
N
DATUM = FOS?
N
Y
CHOOSE DATUM MATLCOND
DATUMS?
Y
N
DATUM = FOS?
N
Y
CHOOSE DATUM MATL COND
SPECIAL MODIFIERS?
N
SPECIAL MODIFIERS?
N
Y
CHOOSE MOO INFO #
DONE
CHOOSE MOD INFO #

FIGURE1

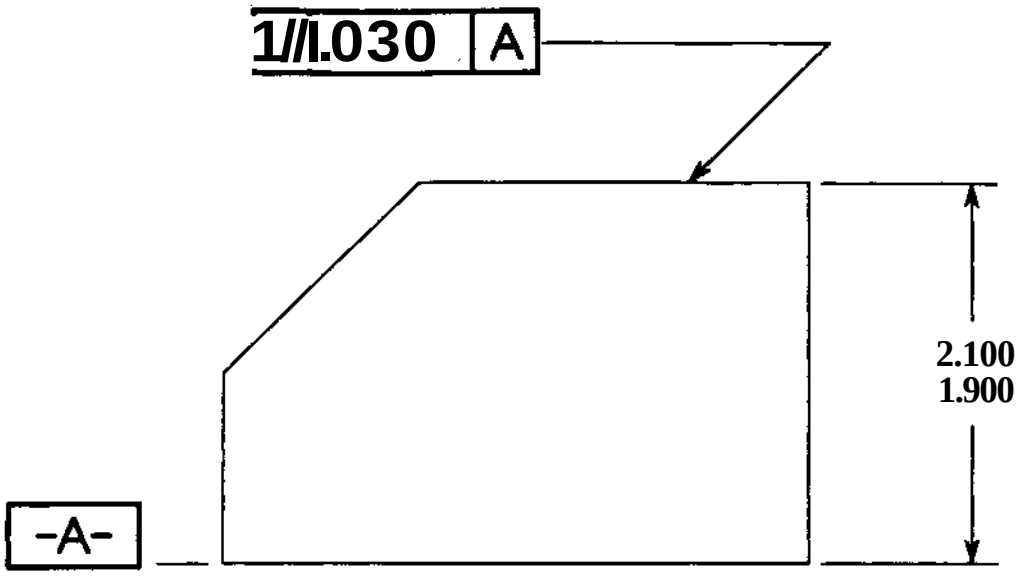


FIGURE 2

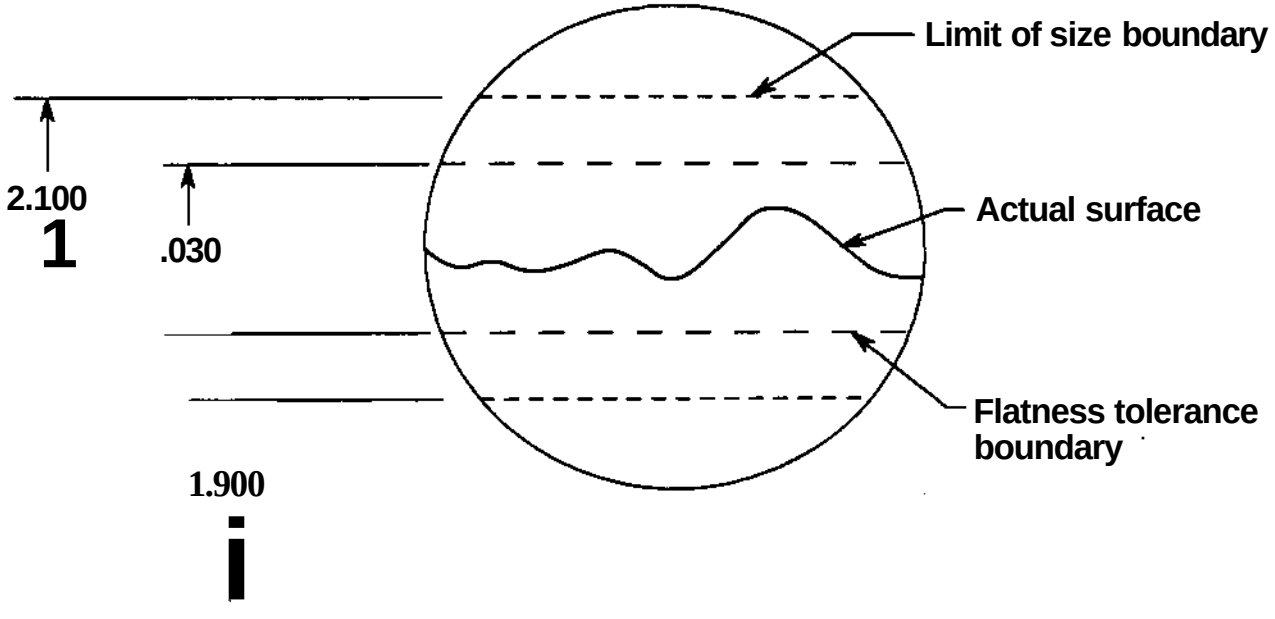


FIGURE 3

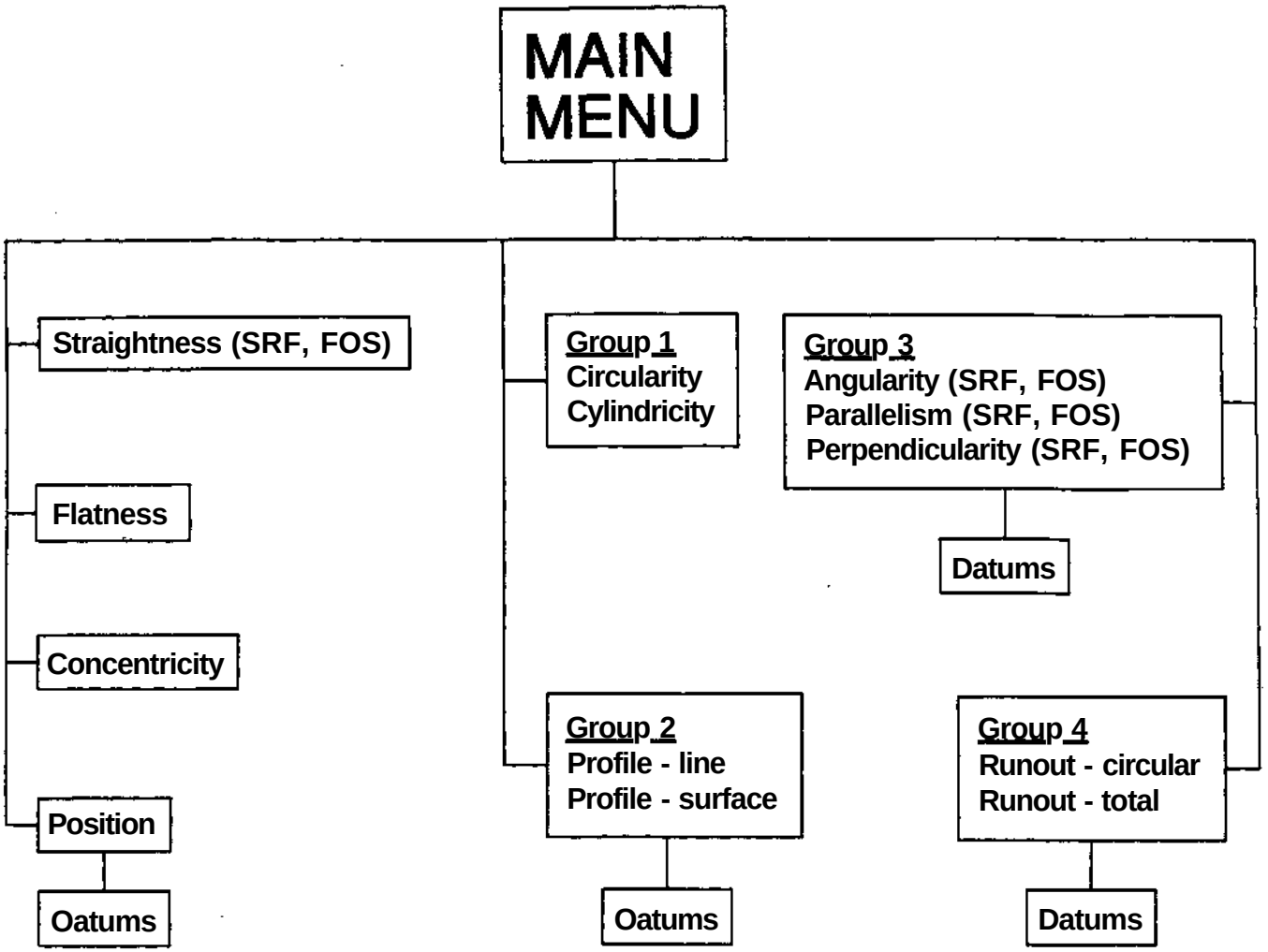


FIGURE 4

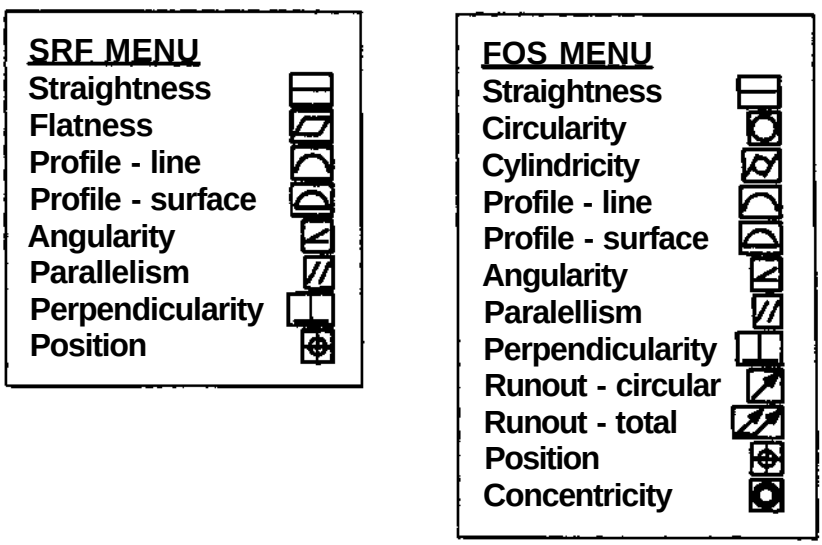


FIGURE 5

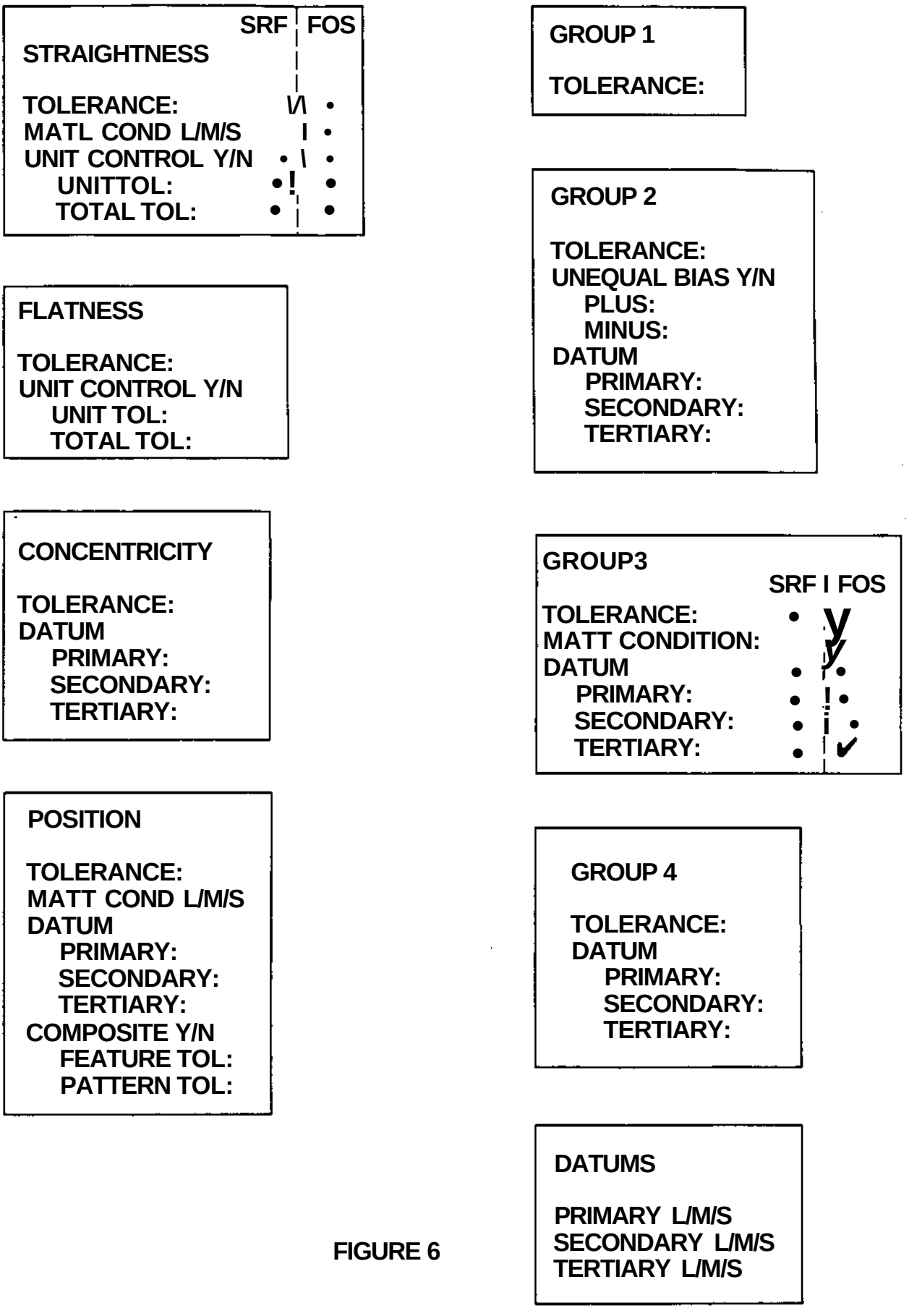

SRF
FOS
STRAIGHTNESS
TOLERANCE:
MATL COND L/M/S
UNIT CONTROL Y/N
UNITTOL:
TOTAL TOL:
FLATNESS
TOLERANCE:
UNIT CONTROL Y/N
UNIT TOL:
TOTAL TOL:
CONCENTRICITY
TOLERANCE:
DATUM
PRIMARY:
SECONDARY:
TERTIARY:
POSITION
TOLERANCE:
MATT COND L/M/S
DATUM
PRIMARY:
SECONDARY:
TERTIARY:
COMPOSITE Y/N
FEATURE TOL:
PATTERN TOL:
GROUP 1
TOLERANCE:
GROUP 2
TOLERANCE:
UNEQUAL BIAS Y/N
PLUS:
MINUS:
DATUM
PRIMARY:
SECONDARY:
TERTIARY:
GROUP3
SRF I FOS
TOLERANCE:
MATT CONDITION:
DATUM
PRIMARY:
SECONDARY:
TERTIARY:
GROUP 4
TOLERANCE:
DATUM
PRIMARY:
SECONDARY:
TERTIARY:
DATUMS
PRIMARY L/M/S
SECONDARY L/M/S
TERTIARY L/M/S

FIGURE 6

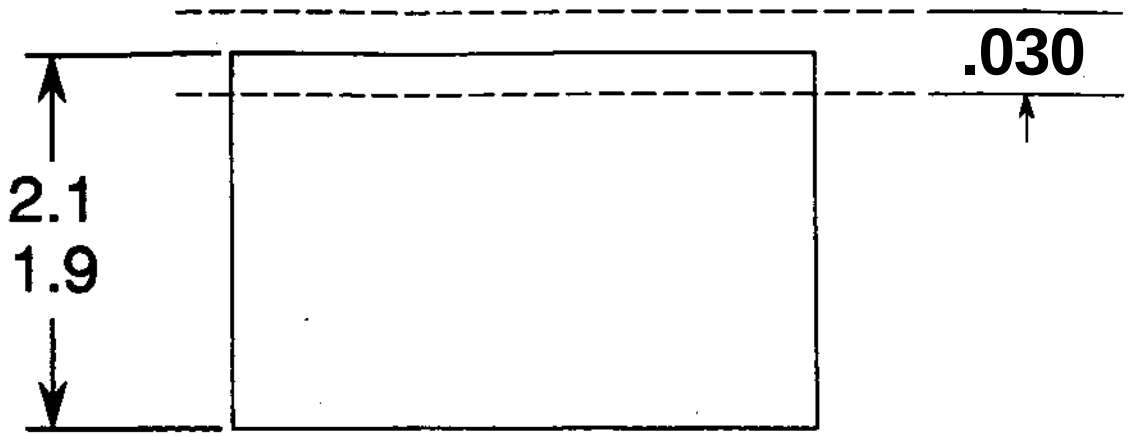


FIGURE 7

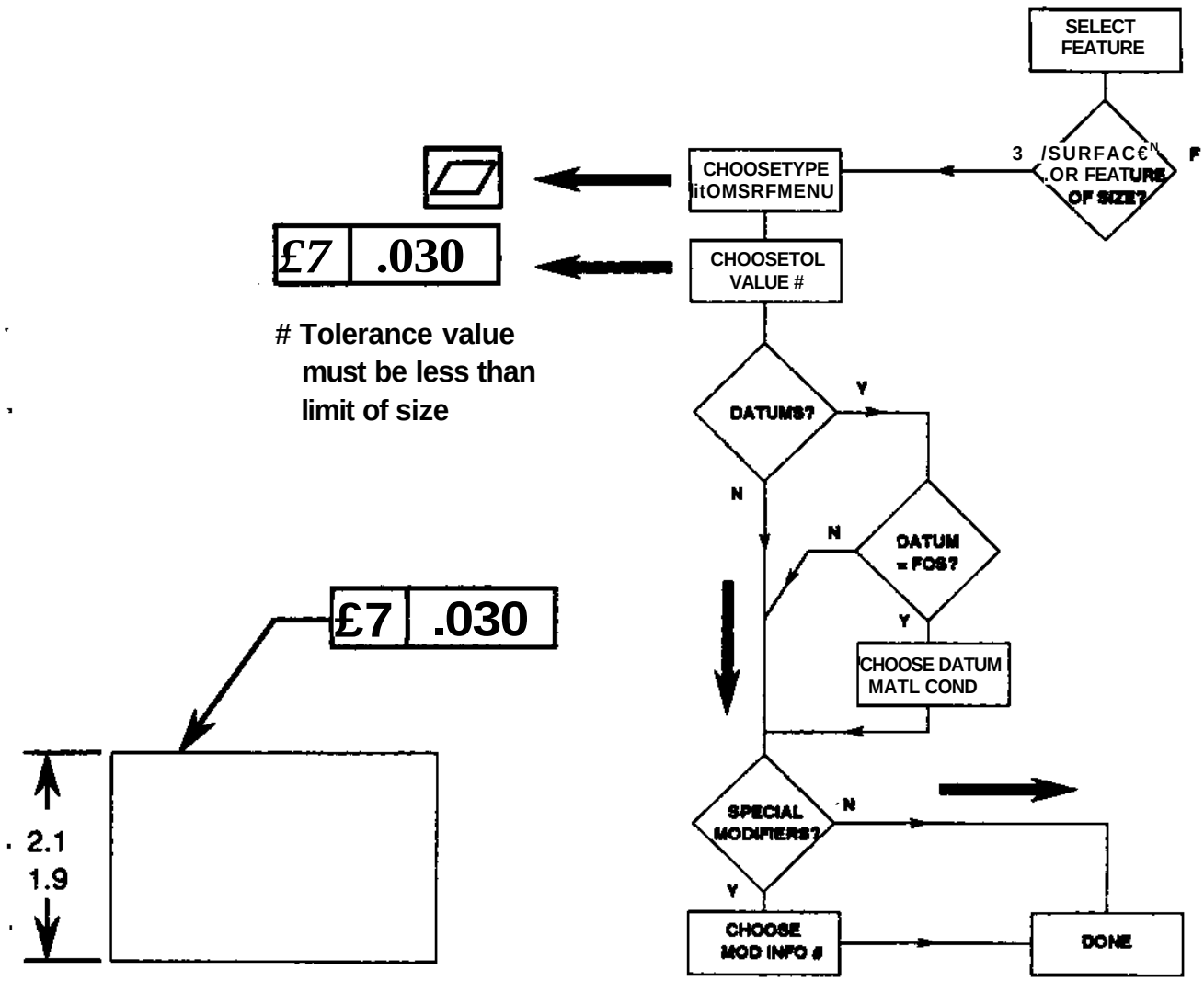


FIGURE 8

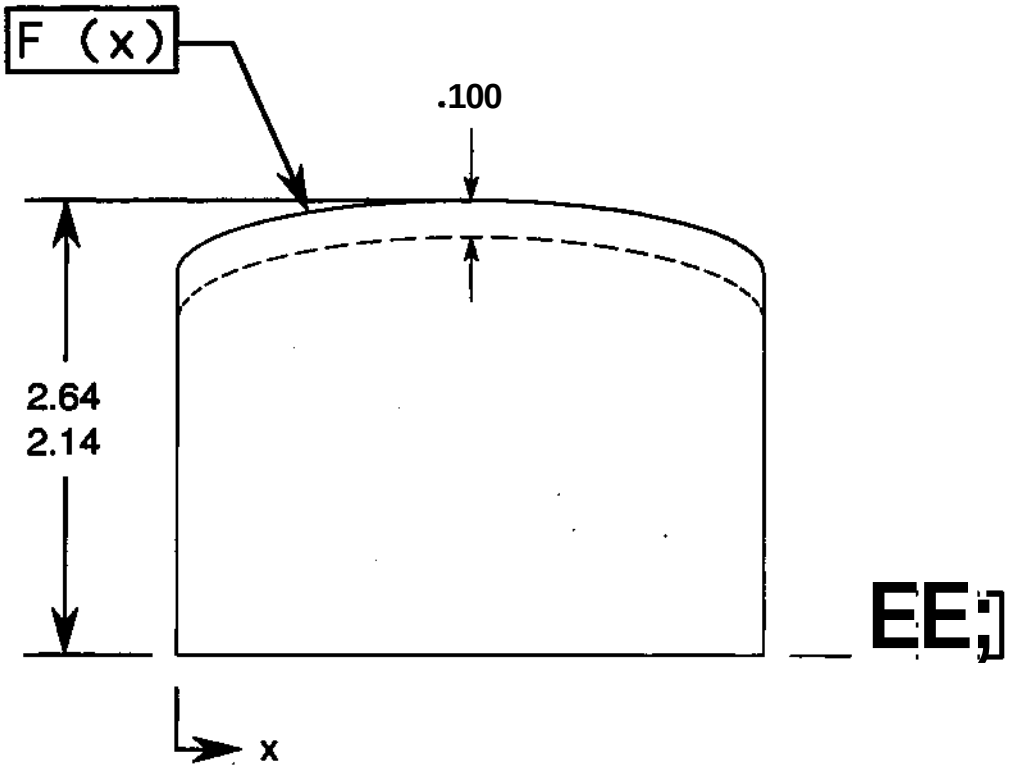
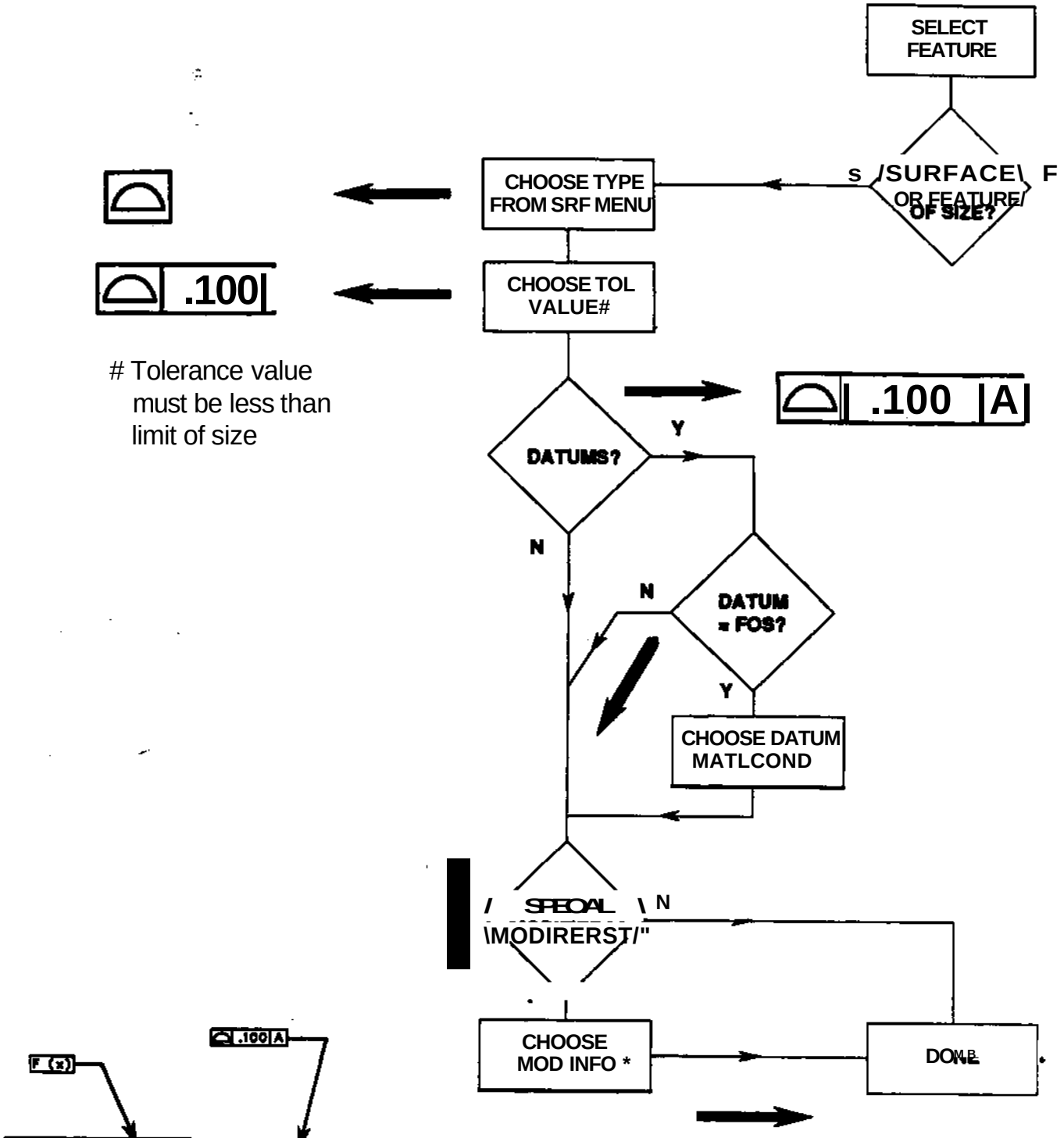


FIGURE 9


SELECT FEATURE
S /SURFACE\ F
OR FEATURE/
OF SIZE?
CHOOSE TYPE FROM SRF MENU
CHOOSE TOL VALUE#
.100
.100 A
Tolerance value must be less than limit of size
DATUMS?
Y
N
DATUM = POS?
N
Y
CHOOSE DATUM MATLCOND
/ SPECIAL \ N
\MODIFIERS/"
CHOOSE MOD INFO *
DONE
* Tolerance is biased. Direction is indicated by phantom line.

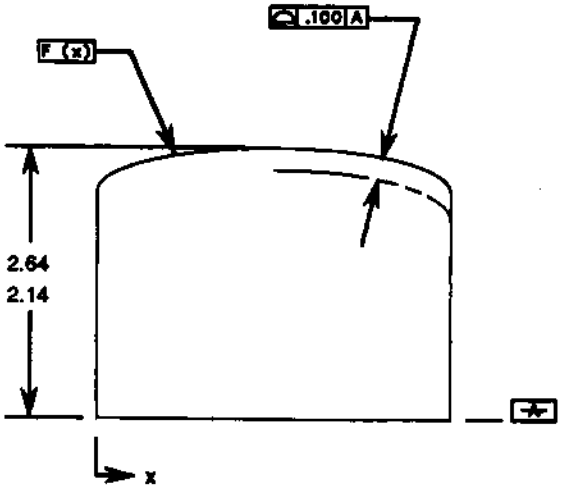

.100 A
F (x)
2.64
2.14
x

FIGURE 10

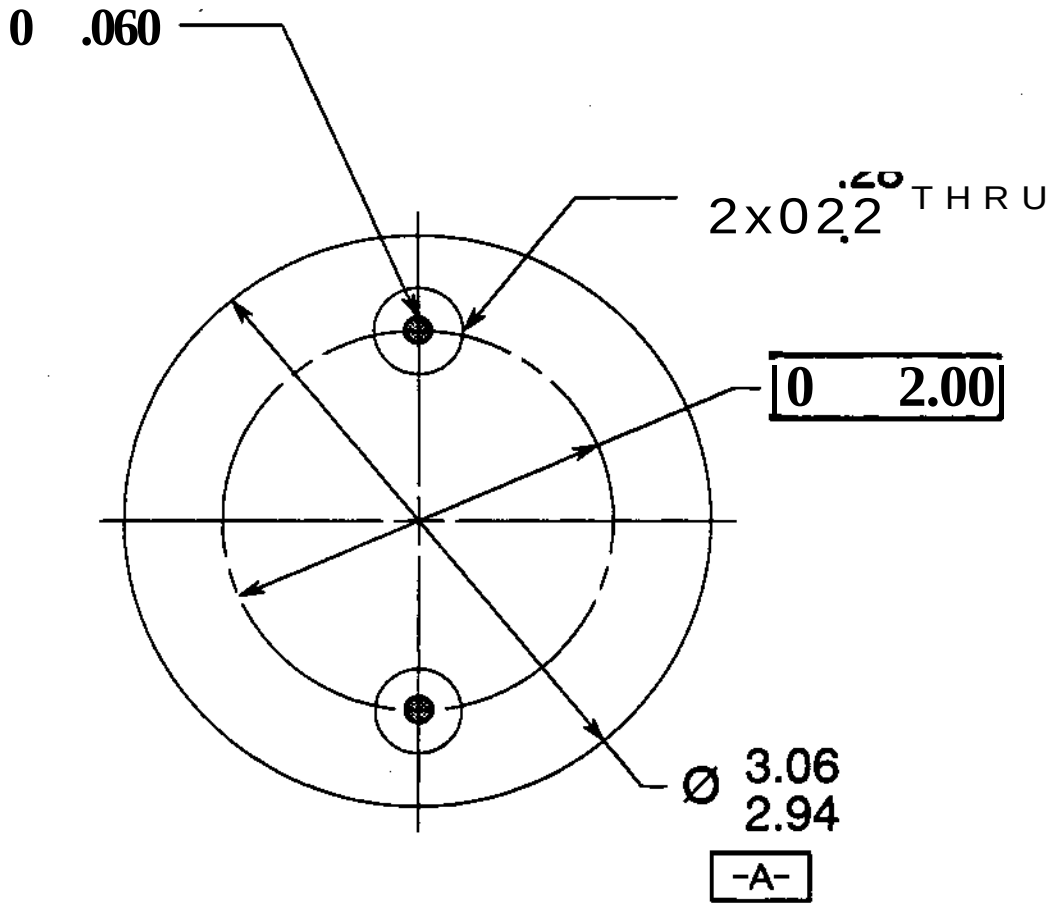


FIGURE 11

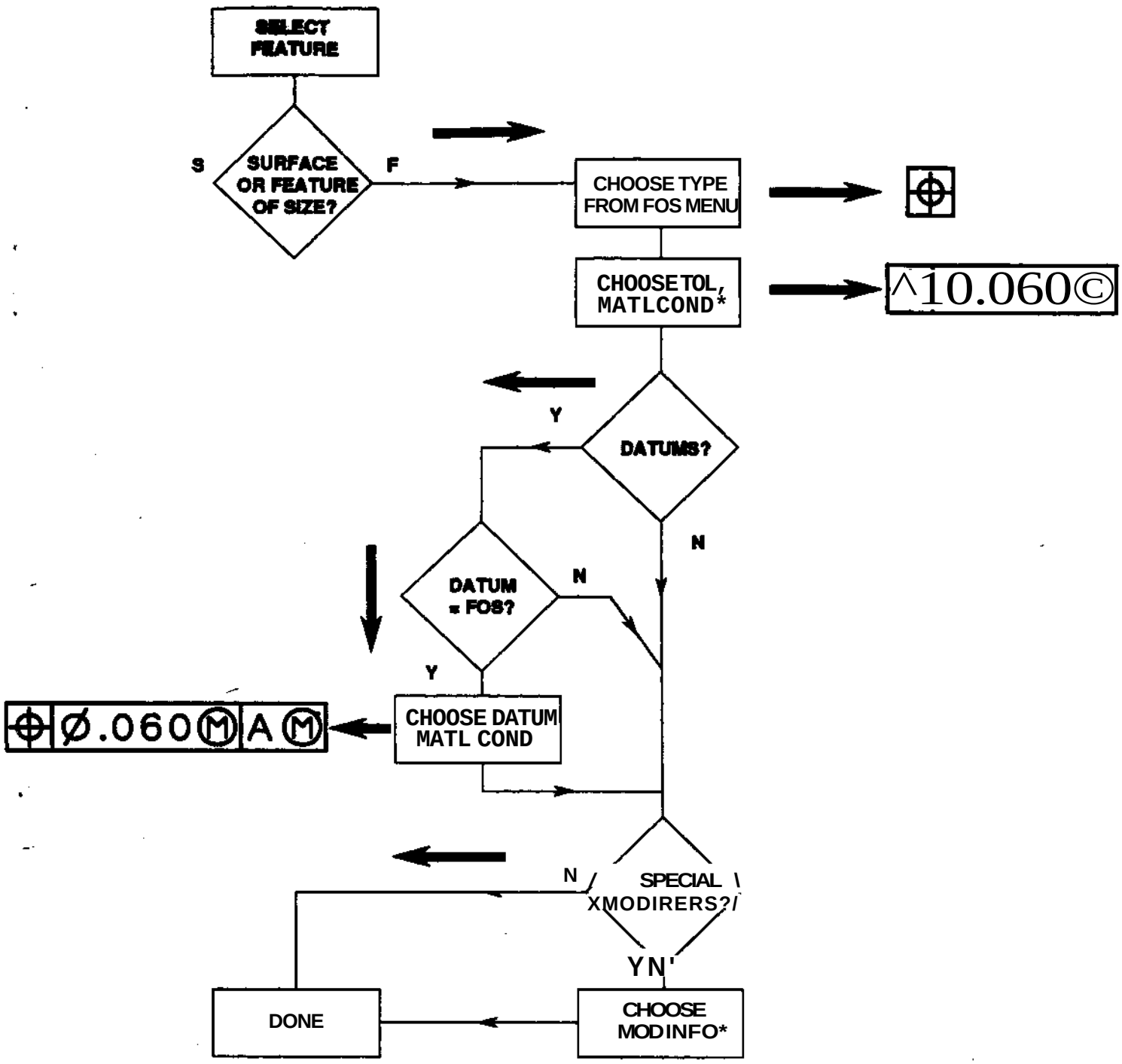

SELECT FEATURE
S
SURFACE OR FEATURE OF SIZE?
F
CHOOSE TYPE FROM FOS MENU
CHOOSE TOL, MATLCOND*
^10.060©
Y
DATUMS?
N
DATUM = FOS?
N
Y
CHOOSE DATUM MATL COND
Ø.060 M A M
N
SPECIAL XMODIRERS?
YN
CHOOSE MODINFO*
DONE

FIGURE 12

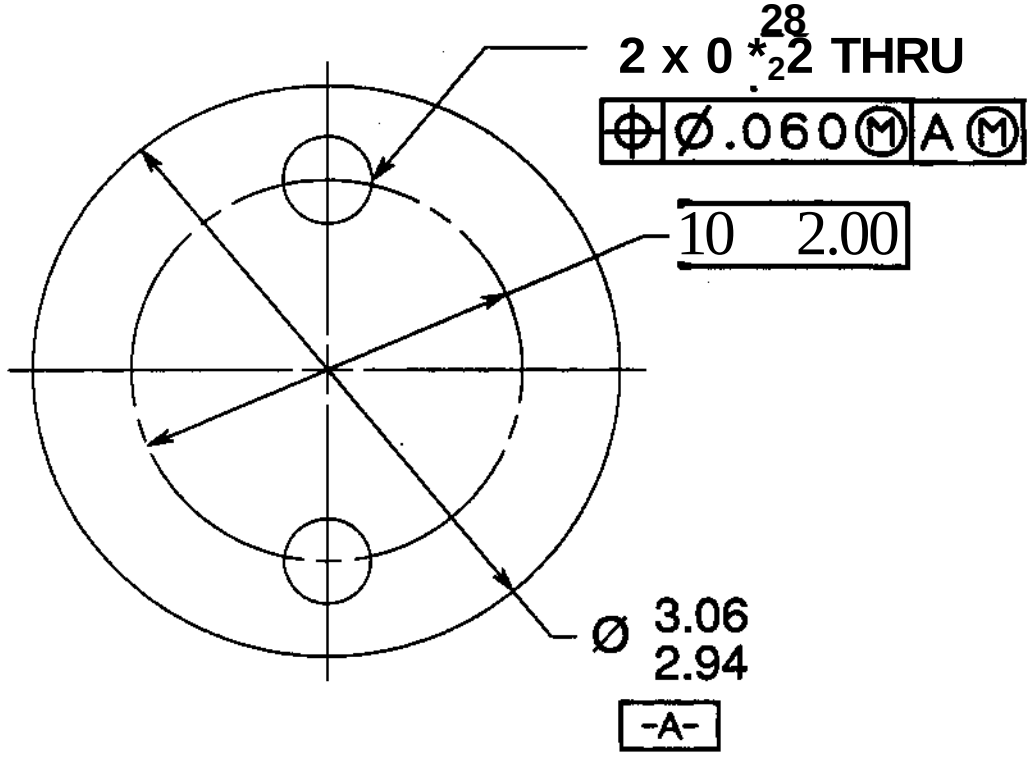


FIGURE 13