SAE International™

SURFACE VEHICLE RECOMMENDED PRACTICE

SAE J2370 REV. DEC2003

Issued 2001-05
Revised 2003-12

Superseding J2370 JUN2003

Geometric Dimensions and Tolerancing for Curved Hose

Foreword—The design engineer assigning tolerances for size, length and geometry must consider function, mating relationships, virtual condition, datums, and economy. Owing to the construction of rubber hoses (synthetic elastomeric material) and being a "non-rigid" product that can be deformed by shipping conditions, measurement forces, and in many cases it's own weight, care must be taken in applying GD&T procedures.

American Society of Mechanical Engineers, ASME, Dimensioning and Tolerancing Standard Y14.5M is in wide use today in a variety of industries to prevent ambiguous interpretation of design intent between designers, customers, suppliers,, manufacturers, and quality engineers. ASME Y14.5M Dimensioning and Tolerancing, standardizes definitions and methods for expressing a designer's intent for a work piece in the form of an engineering drawing. It may be most cost effective for customer and supplier, to agree upon a frequency of gaging using this document, along with high volume cost effective gaging methods currently used in industry

Due to variation in hose wall thickness, material, shape, length, diameter, etc.; no tolerances for true position outer boundary (see Section 3) will be included with this document. The true position tolerance for the outer boundary shall be determined by customer and supplier and should be based on feasibility analysis conducted prior to final release of design record.

1. Scope—To provide the curved hose industry and their customers with a recommended practice for applying GD&T procedures to curved hoses and to provide generic curved hose drawings that represent the application of GD&T to typical curved hose parts. Dimensioning and Tolerancing will be in accordance with ASME Y14.5M.

2. References

2.1 Applicable Publications—The following publications form a part of this specification to the extent specified herein. Unless otherwise indicated, the latest issue of the publications shall apply.

2.1.1 SAE PUBLICATION—Available from SAE, 400 Commonwealth Drive, Warrendale, PA 15096-0001.

SAE J20—Coolant System Hoses

2.1.2 ASME PUBLICATION—Available from ASME, 345 East 47 Street, New York, NY 10017-2330.

ASME Y14.5M 1994—Dimensioning and Tolerancing

3. Definitions

3.1 Geometric Dimensioning and Tolerancing (GD&T) is:

a. The engineering product definition methods that geometrically describe design intent and provide the documentation base for the design of the quality control and production systems.
b. A technique of communication between customer engineers and supplier engineers that promotes a uniform interpretation of the acceptance requirements for a component in terms of its dimensions and tolerances.

GD&T provides the method for dimensioning and tolerancing in a language that helps to eliminate confusing and inconsistent notes. It replaces them with symbols that refer to a common code, ASME Y14.5M: which is the American National Standard. This code describes the dimensions and tolerances of the component with reference to the relationship of its features to each other and their functional interfaces with mating parts, assemblies, etc.

3.2 Reference to Standards Documents—When this document applies, there shall be a note on the drawing or in a document referenced on the drawing, which refers to ASME Y14.5M, SAE J20, and SAE J2370.

3.3 Reference to Gauging—This document and the ASME Y14.5M are not intended as gauging standards. Any reference to gauging is included for explanatory purposes only.

3.4 Figures—The figures in this document are illustrations intended only as an aid to the user in understanding the application of GD&T to curved hose. Any numerical values shown are for illustrative purposes only.

3.5 Theoretical Design Attitude—The theoretical design attitude is: The position of the hose when in the final installed attitude unless otherwise agreed upon by customer and supplier.

3.6 Boundary—The outside contour of the hose is controlled by a true position outer boundary tolerance zone. The tolerance zone follows the basic centerline of the hose, and its zone is equal to the hose outside diameters' MMC size plus the positional tolerance. The outside surface of the hose must reside within this boundary. To invoke this concept, the term BOUNDARY is placed beneath the feature control frame (see Figures 1 to 3).

4. Drawing Views

4.1 Projection—The drawing must specify either third (3rd) or first (1st) angle projection including the appropriate projection symbol.

4.2 Minimum Views—A minimum of two orthographic views is recommended on three dimensional hose shapes.

4.3 End View—A true end view of the hose, dimensioning any radially located features (orientation marks, stripes, etc.) to the next adjacent body length. This view is required for any radially located feature that is used as a tertiary datum.

4.4 Identification—Views are to be identified such as: Top or Plan, and Bottom or Front.

5. Dimensions

5.1 Dimensions, Angles—All dimensions are to be in metric. Angles are to be in decimal degrees.

5.2 Origin—Dimensions may originate from any hose end.

5.3 Basic Dimensions—The contour of the hose must be defined with basic dimensions.

5.3.1 COORDINATE BOX—Dimensions may be shown in a Coordinate Table or Chart based on the "right hand rule" (see Figure 4). All coordinate dimensions in a Table or Chart must be noted that: "All chart dimensions are basic".

5.3.2 XYZ DIMENSIONS

a. If charted, the XYZ location dimensions as installed are required.
b. Dimensions may be shown as ordinate dimensions on the hose, and should be indicated as basic.

6. Datums General—Features of the hose must be identified as datum features for the purpose of establishing geometric relationships imposed by a feature control frame. Datums are theoretically exact points, axes, and planes. These elements exist within a framework of three mutually perpendicular intersecting planes known as the datum reference frame. In the datum reference frame, from left to right, the first datum is the primary, the second datum is the secondary and the third datum is the tertiary. Sufficient datum features, those most important to the design of a part are chosen to position the part in the datum reference frame. This reference frame exists in theory only and not on the part. Therefore, it is necessary to establish a method of simulating the theoretical reference frame from the actual features of the part. Due to the nature of a rubber hose (synthetic elastomeric materials) and being a non-rigid material subject to free state variation, and because its surface is curved or free-form, the datum planes may be described in one of two methods (referred to as methods 1 or 2) described as follows.

6.1 Method 1—Because of the requirement to inspect the contour of the hose in its theoretical design attitude, and because it's surface is curved or free-form, the datum planes will generally be located completely offset from the actual datum features (see Figures 2 and 3). The three mutually perpendicular planes will remain parallel and normal to the horizon, and will reside in the checking fixture (see Figure 5). When multiple datum features of equal importance are used to establish a single datum, the appropriate datum reference letters and associated modifiers, separated by a dash, are entered into one compartment of the feature control frame.

6.1.1 PRIMARY DATUM—The primary datum feature is the Internal Diameter(s) (I.D.) of the hose. Because both ends must be used, and due to their equal importance, a multiple datum is created by using multiple datum features located on the I.D. of the ends. Datum targets are used to identify only the functional portion to be used, since controlling the entire surface is not required.

6.1.2 SECONDARY DATUM—The secondary datum is a multiple datum feature using the end(s) of the hose and is always RFS (regardless of feature size).

6.1.3 TERTIARY DATUM—A tertiary datum is required and will be a specified physical feature on the hose. This datum may be multiple datum feature(s) located on the ends (see datums C and F in Figures 1 and 3) or a datum target(s) (see datum E in Figure 2). It should be kept in mind that when an alignment mark is used as a datum, there may not be a "surface of the part to physically contact the processing equipment or mating assembly surface." Therefore, the act of aligning the mark may affect the value of measurements (on a fixture) or assembly. Never the less, the use of an alignment mark may be desirable because of other considerations. This feature must be dimensioned and toleranced for width and length plus any other dimensions and tolerances necessary to define its shape or location.

6.1.4 DATUMS SIMULATED, (FOR INSPECTION PURPOSES)—Datum feature simulators will be used to fixture the hose in its basic theoretical design attitude, using the specified datum features identified in the feature control frame. The primary datum feature(s) will be affixed to gage plugs with diameters equal to the maximum inside diameter to obtain a least material condition (LMC). The secondary datum feature(s), the hose end(s) will be positioned against datum feature simulators (surfaces) using a minimum of one point contact. The tertiary datum feature(s) will be aligned to an index mark(s) on the fixture; or if datum targets are designated, positioned against datum target features simulators. Other than locating on the specified datum features and other datum target features (support points) as designated on the drawing, the hose will be considered to be in it's "free-state."

6.2 Method 2

6.2.1 The part may also be described by establishing a datum reference frame at one end of the hose and providing additional support or restraint as necessary (see Figures 2 and 3).

6.2.2 PRIMARY DATUM—The primary datum feature is the internal diameter (I.D.) of one of the hose ends. The datum is the axis of the internal diameter over a prescribed length. The axis is theoretically the intersection of two mutually perpendicular planes, which are the origin of measurement in two (of three) directions.

6.2.3 SECONDARY DATUM—The secondary datum feature is the end surface of the hose. The datum is a plane which is perpendicular to the primary datum axis (and planes) and is the origin of measurement in the third direction.

6.2.4 TERTIARY DATUM—The tertiary datum feature may be an alignment mark or datum target which can be used when orientation is essential. Since the three datum planes and directions of measurement have already been established, the tertiary datum merely orients the part in space relative to them.

6.2.5 Since a hose is flexible, supplemental support or restraint may be required beyond those identified in the datum reference frame. These features may be covered in a note on the drawing and must be used for inspection as indicated. By placing the datum and additional support features in a holding fixture in relative location and orientation, according to the car coordinate information, the hose can be inspected as installed (see Figures 2 and 3).

7. Drawing Notes and Symbols

7.1 True Position Symbol—The hose contour is specified by using a True Position Symbol with a cylindrical tolerance zone at Maximum Material Condition (MMC) for the feature tolerance. Datums, if applicable are specified as Regardless of Feature Size (RFS).

7.2 Support Points—All vehicle hose support points, or restrictions (brackets etc.) are to be identified and located with appropriate geometric tolerancing dimensions and symbols.

7.3 Boundary—The outside contour of the hose is controlled by a true position outer boundary tolerance zone. The tolerance zone follows the basic centerline of the hose, and its zone is equal to the hose outside diameters' MMC size plus the positional tolerance. The outside surface of the hose must reside within this boundary. To invoke this concept, the term BOUNDARY is placed beneath the feature control frame (see Figures 1 to 3).

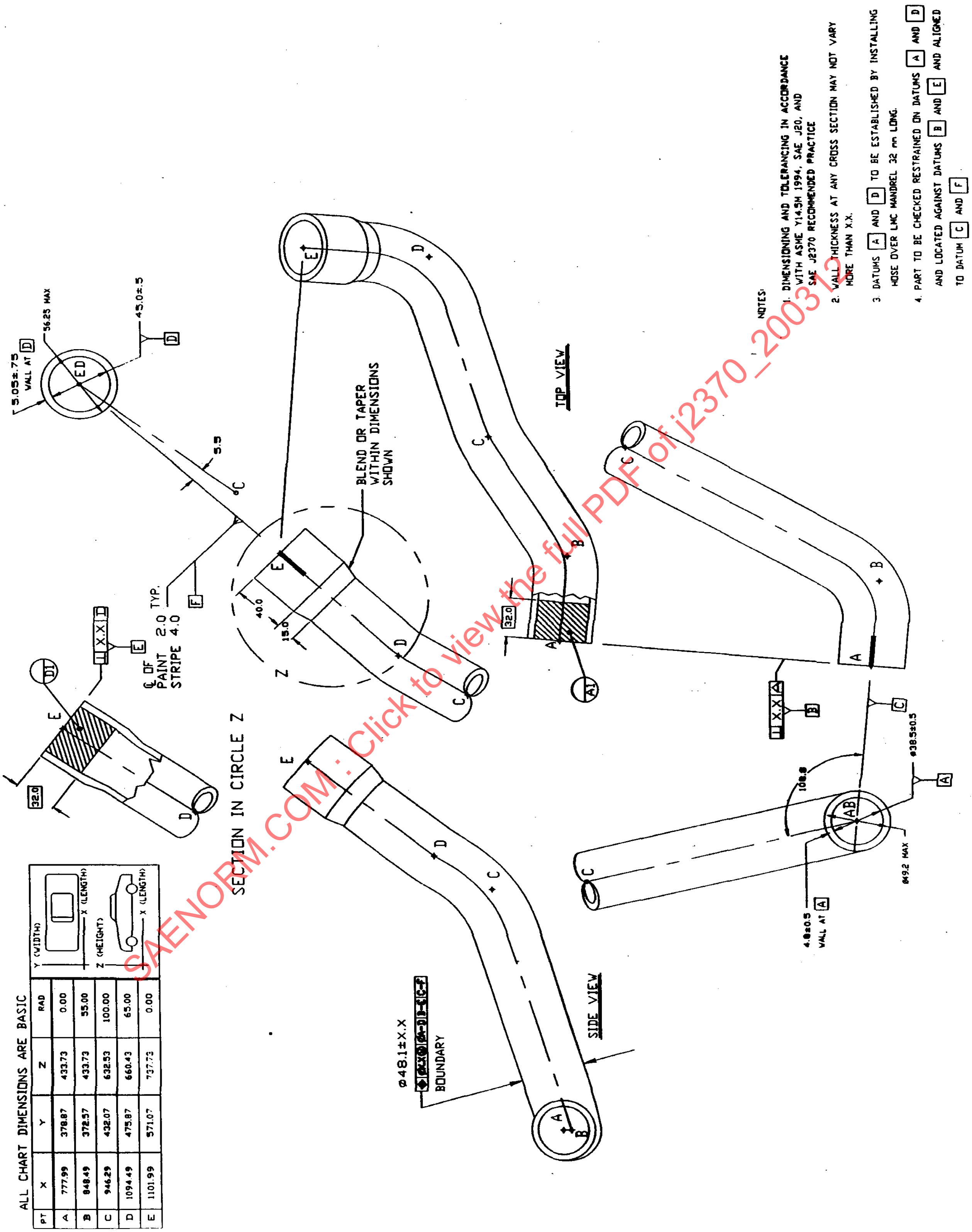
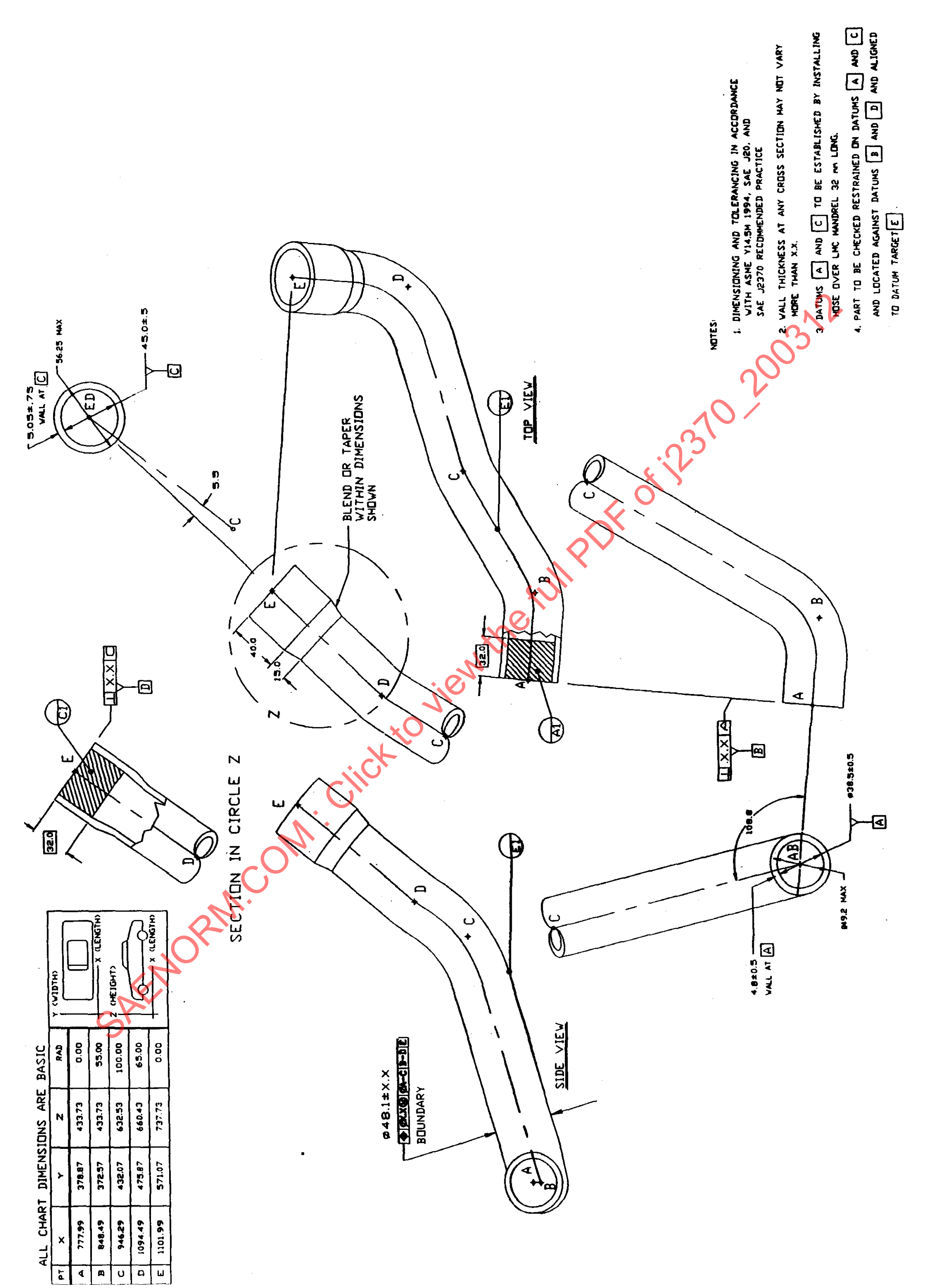
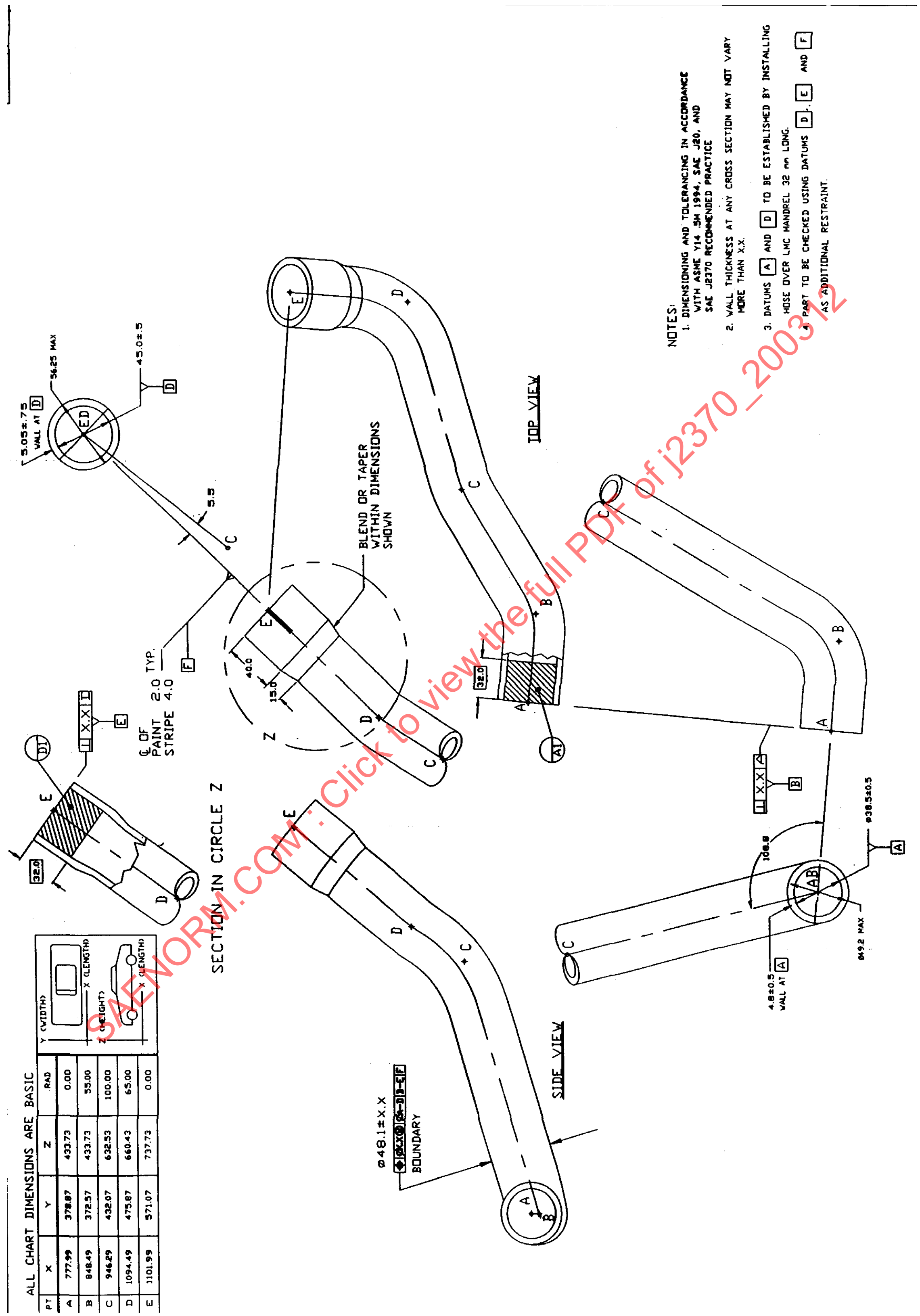


FIGURE 1—DRAWING WITH MULTIPLE DATUM FEATURES



PT	X	Y	Z	RAD
A	777.99	378.87	433.73	0.00
B	848.49	372.57	433.73	55.00
C	946.29	432.07	632.53	100.00
D	1094.49	475.87	660.43	65.00
E	1101.99	571.07	737.73	0.00

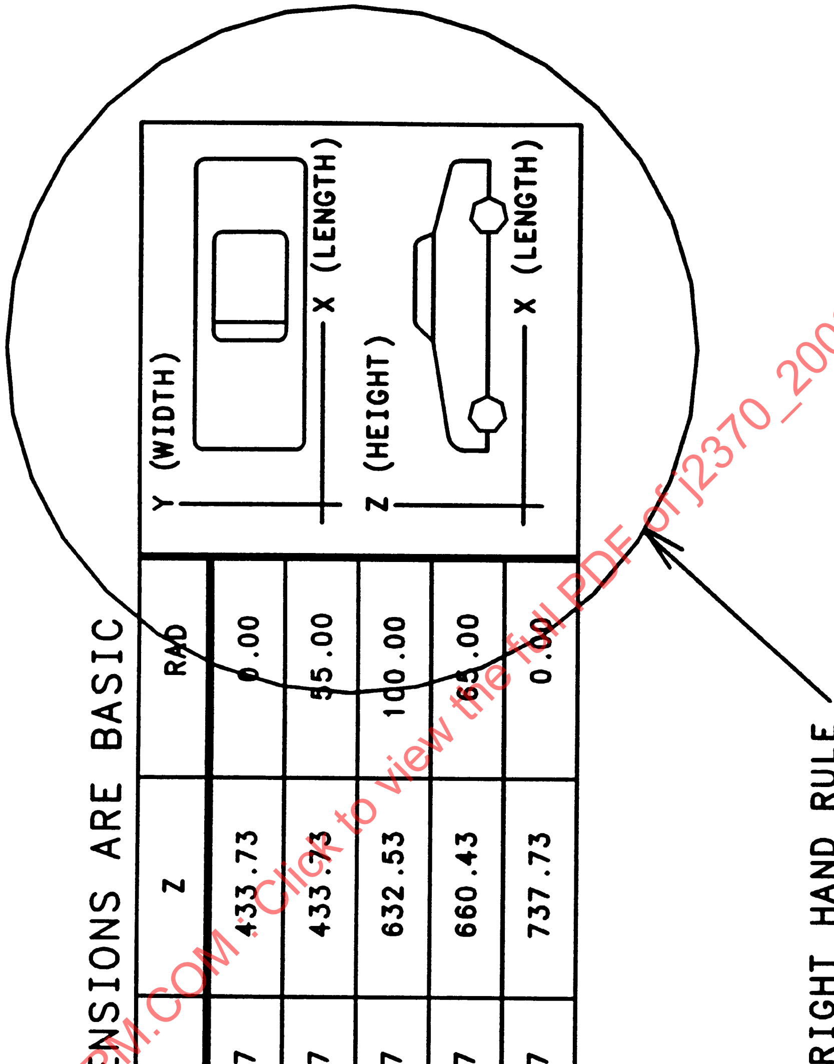
FIGURE 2—DRAWING WITH DATUM TARGETS



PT	X	Y	Z	RAD
A	777.99	378.87	433.73	0.00
B	848.49	372.57	433.73	55.00
C	946.29	432.07	632.53	100.00
D	1094.49	475.87	660.43	65.00
E	1101.99	571.07	737.73	0.00

FIGURE 3—DRAWING WITH DATUM FEATURES AND

DATUM PLANES OFFSET FROM FEATURES



ALL CHART DIMENSIONS ARE BASIC

PT	X	Y	Z	RAD
A	777.99	378.87	433.73	0.00
B	848.49	372.57	433.73	55.00
C	946.29	432.07	632.53	100.00
D	1094.49	475.87	660.43	65.00
E	1101.99	571.07	737.73	0.00

FIGURE 4—RIGHT HAND RULE AND REFERENCE COORDINATE TABLE