DEC STANDARD 114 SEC. 0 REV. A

TITLE: ENGINEERING DRAWING REQUIREMENTS - INDUSTRY STANDARDS ADOPTED BY DIGITAL ENGINEERING AND MANUFACTURING DOCUMENTATION ORGANIZATIONS

ABSTRACT: This standard lists the Industry Standards that are part of Digital's drawing requirements for engineering drawings.

FOR INTERNAL USE ONLY

DATE	ECO #	ORIGINATOR	APPROVED	REV
24-Sep-74	----	Richard Reilly	Eng. Comm.	Initial
7-Oct-82	ML001	Joe Kurta	Eng. Comm.	A

Document Identifier

Size	Code	Number	Rev
A	DS	EL00114-00-0	A

File Id: 13

digital

SECTION 0 - INDUSTRY STANDARDS ADOPTED BY DIGITAL ENGINEERING AND MANUFACTURING ORGANIZATIONS

TABLE OF CONTENTS/REVISION STATUS

1 INTRODUCTION

It is Digital's Engineering Support policy to adopt and use existing industry standards as the basis for engineering drawing practices, and to document any exceptions or variations from those industry standards in applicable Digital Standards.

1.1 PURPOSE

This section lists the industry standards adopted for use as Digital's drawing requirements for engineering drawings. It is provided to insure consistent application and implementation among the Digital engineering and manufacturing organizations that prepare and use engineering drawings.

1.2 SCOPE

This section identifies the specific version of each industry standard that is acceptable for use as Digital's engineering drawing practice. When a revised industry standard is released, it must be compared with the listed version and re-affirmed as an acceptable Digital engineering drawing practice.

This section does not list all industry standards that have been adopted for use by Digital; only those that pertain to engineering drawing practices. Industry standards that relate specifically to dimensioning and tolerancing practices are listed in Section 1.

DEC STD 187, Mechanical Fabrication Workmanship Standards, lists other industry standards, related to mechanical workmanship, that have been adopted for use by Digital.

1.3 RESPONSIBILITIES

Representatives of engineering and manufacturing organizations involved in design and documentation are responsible for reviewing subsequent revisions of the industry standards listed in this section to verify that each industry standard is still appropriate for use by Digital.

If, after reviewing a revised industry standard, substantial changes appear to render that industry standard inappropriate for Digital's use, the knowledgeable representative is responsible for initiating an update to this section.

1.3.1 Responsible Project Engineers

Project engineers are responsible for ensuring that the internal and external support organizations meet the requirements of this standard.

1.3.2 Standards and Methods Control

Standards and Methods Control is responsible for the content of this section and must review and approve all proposed changes. Submit all comments and proposed changes to:

Joe Kurta, Mgr.,Standards and Methods Control
ML04-4/E99, DTN: 223-8895

Standards and Methods Control is also responsible for the distribution of this standard and for making authorized changes. For additional copies, contact:

Digital Standards Administration
ML03-2/E56, DTN: 223-9475

1.4 REFERENCED DOCUMENTS

DEC STD 114, Section 1	Engineering Drawing Requirements - Dimensioning and Tolerancing On Engineering Drawings
DEC STD 187	Mechanical Fabrication Workmanship Standards
A-MN-ELENGRS-02-0	Drafting Manual, Volume 2

These documents are available from Digital Standards Administration, ML03-2/E56, DTN: 223-9475.

2 INDUSTRY STANDARDS

The industry standards adopted by Digital that pertain to engineering drawing practices are listed in Table 1. These standards are included as part of the Drafting Manual, Volume 2.

Table 1. Industry Standards Adopted By Digital For Engineering Drawing Requirements

Organization	Number	Title
ANSI	Y14.3-1975	Multi and Sectional View Drawings
ANSI	Y14.4-1957	Pictorial Drawings
ANSI	Y14.5-1973	Dimensioning and Tolerancing
ANSI	Y14.6-1978	Screw Thread Representation
ANSI	Y14.6aM-1981	Screw Thread Representation (Metric Supplement)
ANSI	Y14.7.1-1971	Spur, Helical, Double Helical, and Rack Gears
ANSI	Y14.7.2-1978	Bevel and Hypoid Gears
ANSI	Y14.9-1958	Forgings
ANSI/AWS	A2.4-79	Symbols for Welding and Non-Destructive Testing, Including Brazing
American Die Casting Institute		Product Standard for Die Casting
Aluminum Association of America		Standards for Aluminum, Sand, and Permanent Mold Castings

DEC STANDARD 114 SEC. 1 REV. A

ENGI-
NEERING
DRAWING
REQMNT.

TITLE: ENGINEERING DRAWING REQUIREMENTS - DIMENSIONING AND TOLERANCING FOR ENGINEERING DRAWINGS

ABSTRACT: This standard presents Digital's requirements for dimensioning and tolerancing engineering drawings. It provides guidelines for using inches and millimeters as units of measure, and for application of general tolerancing on engineering drawings.

FOR INTERNAL USE ONLY

DATE	ECO #	ORIGINATOR	APPROVED	REV
7-Oct-82	----	Joe Kurta	Eng. Comm.	A

Document Identifier

Size	Code	Number	Rev
A	DS	EL00114-01-0	A

File Id: 13

digital

SECTION 1 - DIMENSIONING AND TOLERANCING FOR ENGINEERING DRAWINGS

TABLE OF CONTENTS/REVISION STATUS

1 INTRODUCTION

1.1 PURPOSE

This standard presents Digital's requirements for representing dimensions and dimension tolerances on engineering drawings.

1.2 SCOPE

Dimensions on engineering drawings may be represented by either U.S. customary (inch) dimensions metric, or dual (inch and metric) dimensions.
This standard describes the correct method for specifying either type of dimension.

This standard also describes the U.S. customary (inch) and metric tolerance blocks used on pre-printed engineering drawing forms, and explains how to use them.

This standard does not specify when to use U.S. customary dimensions or metric dimensions; that decision is made by the mechanical engineer who is responsible for the engineering drawings for each product. The only general rule that can be applied is to use dual dimensioning on any engineering drawings that may be provided to customers.

This standard supersedes DEC STD 019, Decimal Dimensioning Standard and DEC STD 114, dated 24 September 1974, Metric Dimensioning on Engineering Drawings.

1.3 RESPONSIBILITIES

1.3.1 Standards and Methods Control

Standards and Methods Control is responsible for the content of this standard and must review and approve all proposed changes. Send all comments to Joe Kurta, Mgr., Standards and Methods Control, ML04-4/E99.

Standards and Methods Control is also responsible for the administration and distribution of this document, and for making authorized changes.

1.3.2 Responsible Product Engineer

The Product Engineer is responsible for determining if the design, manufacture, or maintenance of the product requires inch, metric, or dual dimensioning on its documentation. This individual is also responsible for ensuring that these requirements are implemented by internal and external support organizations.

1.4 REFERENCED STANDARDS

1.4.1 Digital Standards

DEC STD 013	Standard Engineering Drawing Formats and Forms
DEC STD 015	Abbreviations and Units Of Measurement
DEC STD 182	Engineering Documentation Acceptance Criteria

Copies of these standards may be obtained from:

Digital Standards Administration,
ML03-2/E56, DTN: 223-9475.

1.4.2 ANSI Standards

B46.1-1978	Surface Texture, Including Surface Roughness, Waviness and Lay
B87.1-1965	Decimal Inch
B94.6-1966*	Knurling
Y14.2M-1979	Line Conventions and Lettering
Y14.5-1973	Dimensioning and Tolerancing
Y14.6-1978	Screw Thread Representation
Y14.6AM-1981	Screw Thread Representation - Metric Supplement
Y14.7.1-1971	Gear Drawing Standards - Part 1 For Spur, Helical, Double Helical and Rack

Y14.7.2-1978 Gear and Spline Drawing Standards Part 2 - Bevel and Hypoid Gears

* 1981 revision available soon

Copies of these standards may be obtained from: American National Standards Institute, 1430 Broadway, New York, NY 10018.

1.4.3 Other Documents

Geometric Dimensioning and Tolerancing - A Working Guide by Lowell Foster.

Copies of the document may be obtained from: Addison-Wesley, Jacob-Way, Reading, MA 01867.

2 GENERAL REQUIREMENTS

Primarily, Digital has adopted ANSI Y14.5-1973 as its standard for dimensioning and tolerancing practice, supplemented by Geometric Dimensioning and Tolerancing - A Working Guide by Lowell Foster for Geometric dimensioning applications.

In certain areas of engineering documentation, Digital has other standards that supplement or replace those specified by ANSI Y14.5-1973. The standards specified by ANSI Y14.5-1973 and those available within Digital are listed for various documentation areas in Table 1-1.

Third angle projection shall be used throughout Digital; European first angle projection shall not be used.

3 UNITS OF MEASURE FOR DIMENSIONING AND TOLERANCING

3.1 U.S. CUSTOMARY (INCH) DIMENSIONING

For linear measurements on U.S. products, Digital uses U.S. Customary inches and decimal parts of an inch.

For inch measurements less than 1, omit zeroes left of the decimal point.

Dimensions and tolerances to four decimal places should be avoided if physical inspection is involved, because normal quality control practice requires gauges 10 times as accurate as the dimension to be measured, and gauges with 5-place accuracy are expensive and difficult to obtain.

Angular dimensions and tolerances shall be specified in degrees and, where necessary, in minutes and seconds. An angular dimension and both its tolerance limits shall use the same units of measure.

Table 1-1. Requirements Comparison, ANSI Y14.5-1973 References Versus Digital.

Subject	ANSI Y14.5	Digital
Drawing sheet size and format	Y14.1-1975	DEC STD 013, Section 1
Decimal inch dimensioning	B87.1-1965	DEC STD 114
Rules for rounding numbers	B87.1-1965	DEC STD 114
Line conventions	Y14.2-1973	Y14.2-1973
Lettering	Y14.2-1973	DEC STD 182
Knurling	B94.6-1966	B94.6-1966
Screw threads	Y14.6-1957	Y14.6-1957
Surface texture	B46.1-1962	B46.1-1962
Gears, splines, and serrations	Y14.7-1968	Y14.7-1968
Spur helical, double helical, and rack gear	Y14.7.1-1971	Y14.7.1-1971
Metric dimensioning	Y14.5-1973	This section and ANSI Y14.5-1973
Dual dimensioning	Y14.5-1973	This section and ANSI Y14.5-1973

3.2 METRIC DIMENSIONING

For metric linear dimensions, Digital uses millimeters and decimal parts of a millimeter.

The symbol Ø means diameter, and is to be used on all metric drawings. The term "dia" is not an acceptable abbreviation of diameter.

Until an acceptable metric fastener is selected, standard inch fasteners currently used for inch designs shall also be used on metric designs. Any deviations shall be reviewed by the mechanical engineer responsible for the particular product.

Angular dimensions and tolerances shall be specified in degrees and, where necessary, in minutes and seconds. The angular dimension and both its limits shall use the same units of measure.

3.3 DUAL DIMENSIONING

Dual dimensioning provides both inch and metric measurements on the same drawing. This method is used for products that are intended for both U.S. and non-U.S. markets. The preferred method of dual dimensioning is to draw a short horizontal line, and to place the metric measurement above the line and the inch measurement below the line.

For inch measurements less than 1, omit all zeroes left of the decimal point.

For other metric quantities (mass, force, pressure, stress, etc.), refer to DEC STD 015.

The symbol Ø means diameter, and is to be used on all metric drawings. The term "dia" is not an acceptable abbreviation of diameter.

Standard inch fasteners presently being used on inch designs shall also be used on dual-dimension designs. All deviations shall be reviewed by the mechanical engineer responsible for the particular product.

Angular dimensions and tolerances shall be specified in degrees and, where necessary, in minutes and seconds. An angular dimension and both its limits shall use the same units of measure.

4 DECIMAL DIMENSIONING

Decimal dimensioning shall be used for all new product drawings, and is generally preferred for all drawings. Decimal representation of the inch and millimeter both obey the following rules:

a. For decimal place sensitive dimensions and tolerances, use:

1-place decimal when the tolerance need not exceed $\pm$ 0.1,

2-place decimal when the tolerance need not exceed $\pm$ 0.02,

3-place decimal when the tolerance need not exceed $\pm$ 0.005.

Note

If a tighter tolerance is required, use 4-place decimal and specify that tolerance in the body of the drawing.

b. When a dimension and its tolerance are decimal place sensitive, the dimension and both limits shall all have the same number of digits to the right of the decimal point. For a dimension-sensitive tolerance, having an equal number of digits is not significant.

c. For unilateral tolerances (one limit zero), both limits shall be specified.

d. For a diameter, the least significant digit should be an even number if feasible, so that halving it does not change the implied tolerance level.

e. Tolerances used with dimension lines shall show the tolerances following the dimension, with the plus tolerance 1/2 line up and the minus tolerance 1/2 line down, so that the minus tolerance is directly under the plus tolerance, as shown in Figure 1-1.

f. Tolerances specified in text shall be shown all on one line, with the plus tolerance following the dimension and preceding the minus tolerance, as shown in Figure 1-2.

g. Limit dimensions used with dimension lines for external features shall show the maximum value listed above the minimum value; limit dimensions for internal features shall show the minimum value listed above the maximum value. To avoid legibility problems on microfilm, the upper and lower limits shall not be separated with a dimension line or separator bar. Figure 1-3 illustrates this configuration.

h. Limit dimensions specified in text shall be shown on a single line, separated by a dash. The larger shall be first for external features, and the smaller shall be first for internal features. Figure 1-4 illustrates this configuration.

i. Decimal points shall be uniform, dense, and large enough to be clearly visible, in accordance with the requirements of DEC STD 182.

j. Numbers shall not contain embedded commas (often used to set off thousands).

k. Nominal designations, such as thread size, wire gauge, and sheet metal thickness, shall remain as stated, and shall not be subjected to units conversion.

l. Surface finishes shall be specified in microinches on both inch and metric drawings. Preferred values are 1000, 500, 250, 125, 63, 32, and 16 microinches.

5 TOLERANCING

Digital uses two different tolerance methods, one of which is number of decimal places sensitive; the other is dimension sensitive. Which is used for a particular drawing depends on the application.

5.1 NUMBER OF DECIMAL PLACES SENSITIVE

This type of tolerance is concerned with the number of digits to the right of the decimal point. It is straightforward in application, but it is difficult to keep the tolerances meaningful when converting between inches and millimeters, or when making a major change in a dimension.

5.2 DIMENSION SENSITIVE

Dimension sensitive tolerance is well adapted to converting from inch dimensioning to metric dimensioning or the internationally required dual dimensioning.

Dimensioning sensitive tolerance is based on the SI (International Standards Organization) system of tolerances. The tolerances depend on the value of the associated dimension and the criterion of maintaining tolerance as the dimension increases. This method is preferred where constant design accuracy is desired.

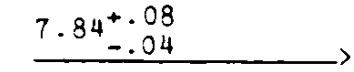

Figure 1-1. Tolerances on Drawing

7.84+.08-.04

Figure 1-2. Tolerances in Text

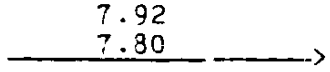

Figure 1-3. Limits on Drawing

7.92 - 7.80

Figure 1-4. Limits in Text

6 FRACTIONAL DIMENSIONING

6.1 NEW PARTS DOCUMENTATION

Fractions are not allowed on new parts documentation because of the difficulty in making accurate conversions between decimal, fractional, and metric units, and because fractional notation can cause legibility problems when the documentation is reproduced from microfilm.

Fractions may be used on drawings that do not pertain to hardware parts or products, such as plant engineering floor plans or other drawings that:

a. Will not be distributed or reproduced on microfilm.

b. Will not require dual dimensioning.

Commercial tools and standard hardware sizes that represent measurable dimension commonly expressed as fractions shall either be specified in the decimal equivalent of the tool and gauge, or as maximum/minimum limits that include the fractional dimension and allowable tolerances. Such fractions shall be converted to decimals by use of the method described under subhead 9.

Nominal sizes expressed as fractions, such as 1/2-inch iron pipe (which nowhere measures 0.5 inch), shall remain as fractions.

6.2 EXISTING PARTS DOCUMENTATION

If fractional dimensions are shown on current drawings of existing parts, they shall be converted to decimal notation when the drawings are revised by an ECO. Although exact conversion of the fractions to decimal may require 3 or 4 places, such precision may not be actually needed; the Design Engineer should reassess the actual requirements and specify the decimal precision accordingly.

7 TITLE BLOCKS

Dimensional tolerances for a drawing are determined by the design requirements for the particular part or product, and require the cooperation of Mechanical Engineering, Manufacturing Engineering, and Design Drafting. Tolerances may be specified for each individual dimension, or covered by one of the general tolerance notations.

General tolerances, which are made applicable unless otherwise specified, are selected to apply to the majority of dimensions on a drawing. The use of general tolerances saves drawing time by eliminating the need to indicate the tolerance of each dimension. General tolerances are never used where precision is required, but only when there is a need to establish a limit. Digital provides a 1-, 2-, and 3-place general tolerance format, and two dimension-sensitive general tolerance formats.

Dimensions shown without tolerances are controlled by the general tolerance notes in the title block of the drawing. The preprinted title blocks provide a selection of the general tolerances most commonly used by Digital, and require that the one being used be indicated. Figure 1-5 shows the standard inch title block; the standard metric title block is shown in Figure 1-6. The formats of both of these title blocks are controlled by DEC STD 013.

Formats created by computer-aided design (CAD) techniques are permitted to display only the general tolerances selected for use, provided the format is authorized by DEC STD 013.

Use of a standard Digital title block (Figure 1-5 or 1-6) requires that the designer select the general tolerance range that most appropriately fits the functional requirements of the part or assembly and the machine, method, or process used to fabricate or assemble the part without impairing its function or unnecessarily increasing its cost. The standard title blocks provide the following four tolerance options:

a. The first option is the use of the 1-, 2-, and 3-place decimal block (not available with the metric title block). When this option is selected, the other tolerance options should be crossed out, as shown in Figure 1-7.

b. The second and third options are choosing either of the dimension-sensitive tolerance ranges.

c. The fourth option is to use the blanks to specify any desired range. The specified range can be made dimension-sensitive by choosing tolerances to provide approximately constant accuracy, where the accuracy is the ratio of the tolerance to the dimension. If option 2, 3, or 4 is chosen, option 1 must be crossed out to make the drawing acceptable for release, as shown in Figure 1-8.

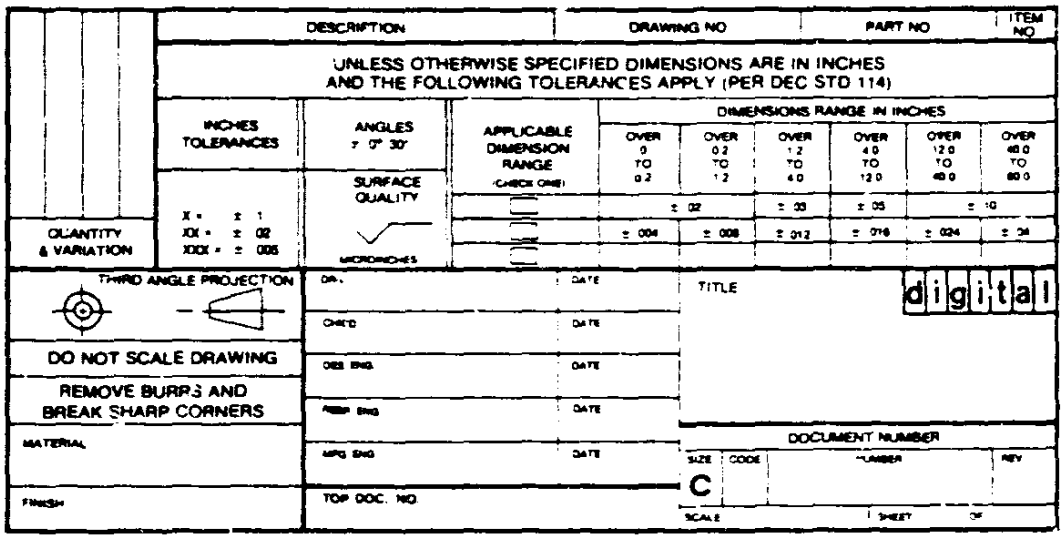

Figure 1-5. Standard Digital U.S. Customary (Inch) Title Block

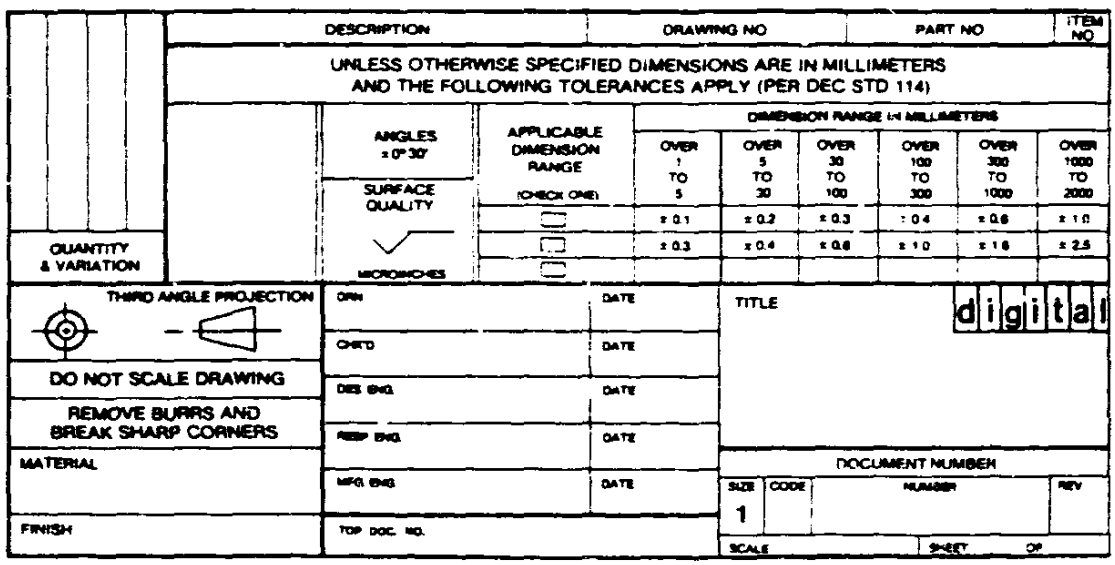

Figure 1-6. Standard Digital Metric Title Block

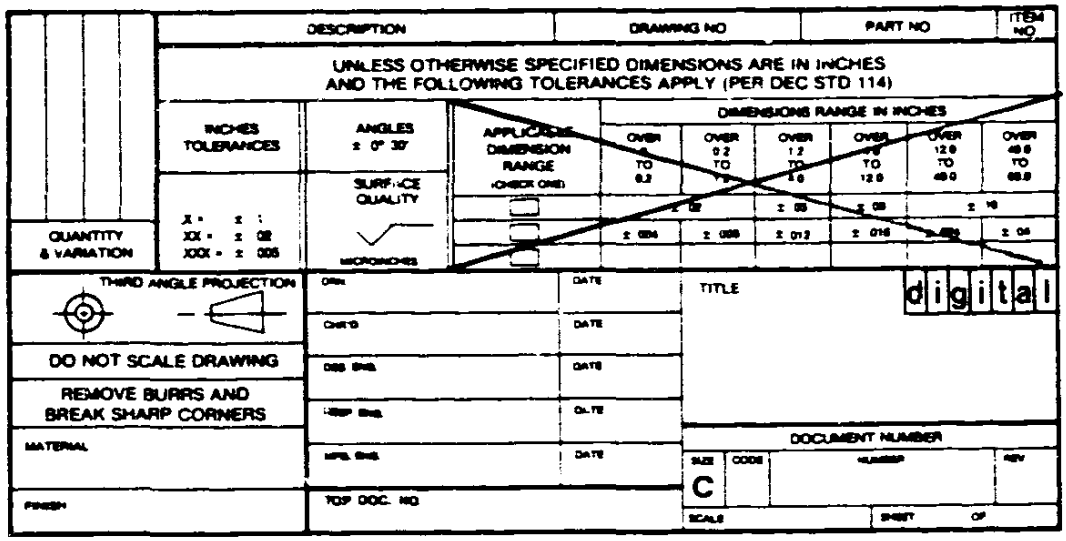

Figure 1-7. First Option Selected

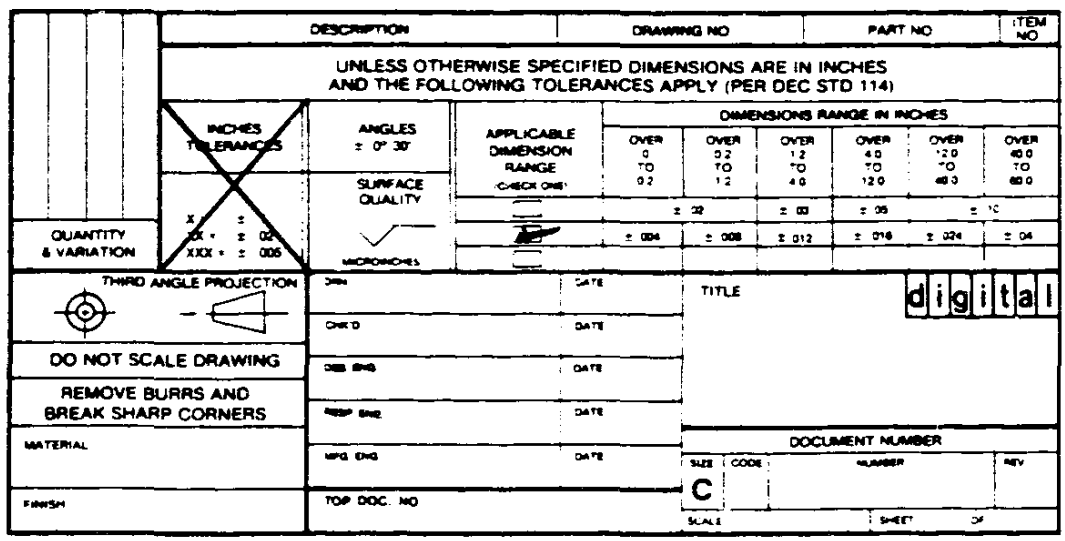

Figure 1-8. Third Option Selected

8 CONVERTING INCH DIMENSIONS TO METRIC

Conversion of inch dimensions into metric is done in one of two ways, depending on the requirements. Both methods start with precise conversion; the nearest value method then rounds to the nearest value at the same accuracy, while the tighter tolerance method rounds inside the original tolerance.

The following rules should be used to round off the nearest value:

a. Do not change the last digit retained (last digit) if the digit beyond it (next digit) is less than 5.

b. If the next digit is greater than 5, increase the last digit by 1.

c. If the next digit is 5, make the last digit even, adding 1 if it is odd.

To round to a tighter tolerance, truncate the upper limit and round up the lower limit. The following examples show both methods.

15.79 - 15.36 inches = 401.066 - 390.144 millimeters (exact)
= 401.07 - 390.14 millimeters (nearest)
= 401.06 - 390.15 millimeters (tighter)

9 CONVERTING FRACTIONS TO DECIMAL

When a drawing revision is required as part of an ECO to an existing part, all dimensions and tolerances should be checked for conformance to the requirements of this standard. Any conflicts should be considered by the responsible engineer, for inclusion in the ECO.

The following rules should be used to round off decimals that are too long:

a. Do not change the last digit retained (last digit) if the digit beyond it (next digit) is less than 5.

b. If the next digit is greater than 5, increase the last digit by 1.

c. If the next digit is 5, make the last digit even, adding 1 if it is odd. Table 1-2 shows the result of 2-, 3-, and 4-places for common fractions from 1/64 to 63/64.

Table 1-2. Decimal Equivalents of Common Fractions

4ths	8ths	16th	32nds	64ths	To 4 Places	To 3 Places	To 2 Places
				1/64	.0156	.016	.02
			1/32	-------	.0312	.031	.03
				3/64	.0469	.047	.05
		1/16	-------	-------	.0625	.062	.06
				5/64	.0781	.078	.08
			3/32	-------	.0938	.094	.09
				7/64	.1094	.109	.11
	1/8	-------	-------	-------	.1250	.125	.12
				9/64	.1406	.141	.14
			5/32	-------	.1562	.156	.16
				11/64	.1719	.172	.17
		3/16	-------	-------	.1875	.188	.19
				13/64	.2031	.203	.20
			7/32	-------	.2188	.219	.22
				15/64	.2344	.234	.23
1/4	-------	-------	-------	-------	.2500	.250	.25
				17/64	.2656	.266	.27
			9/32	-------	.2812	.281	.28
				19/64	.2969	.297	.30
		5/16	-------	-------	.3125	.312	.31
				21/64	.3281	.328	.33
			11/32	-------	.3438	.344	.34
				23/64	.3594	.359	.36
	3/8	-------	-------	-------	.3750	.375	.38
				25/64	.3906	.391	.39
			13/32	-------	.4062	.406	.41
				27/64	.4219	.422	.42
		7/16	-------	-------	.4375	.438	.44
				29/64	.4531	.453	.45
			15/32	-------	.4688	.469	.47
				31/64	.4844	.484	.48
1/2	-------	-------	-------	-------	.5000	.500	.50
				33/64	.5156	.516	.52
			17/32	-------	.5312	.531	.53
				35/64	.5469	.547	.55
		9/16	-------	-------	.5625	.562	.56
				37/64	.5781	.578	.58
			19/32	-------	.5938	.594	.59
				39/64	.6064	.609	.61
	5/8	-------	-------	-------	.6250	.625	.62
				41/64	.6406	.641	.64
			21/32	-------	.6562	.656	.66
				43/64	.6719	.672	.67

Table 1-2. Decimal Equivalents of Common Fractions (Cont'd)

4ths	8ths	16th	32nds	64ths	To 4 Places	To 3 Places	To 2 Places
		11/16 -	-------	-------	.6875	.688	.69
				45/64	.7031	.703	.70
			23/32 -	-------	.7188	.719	.72
				47/64	.7344	.734	.73
3/4 -	-------	-------	-------	-------	.7500	.750	.75
				49/64	.7656	.766	.77
			25/32 -	-------	.7812	.781	.78
				51/64	.7969	.797	.80
		13/16 -	-------	-------	.8125	.812	.81
				53/64	.8281	.828	.83
			27/32 -	-------	.8438	.844	.84
				55/64	.8594	.859	.86
	7/8 -	-------	-------	-------	.8750	.875	.88
				57/64	.8906	.891	.89
			29/32 -	-------	.9062	.906	.91
				59/64	.9219	.922	.92
		15/16 -	-------	-------	.9675	.938	.94
				61/64	.9531	.953	.95
			31/32 -	-------	.9688	.969	.97
				63/64	.9844	.984	.98
4/4 -	-------	-------	-------	-------	1.0000	1.000	1.00