1.0 General

The use of metric components is increasing rapidly in the United States. The automotive industries, Department of Defense (DoD), electronics industries, farming equipment industry, and most major export industries are converting to metric in order to meet foreign demands and compete in the world market. Since the U.S. is the only industrialized country in the world still using the inch/pound system, the need to utilize a universal measuring system is essential in facilitating international communication and trade.

This section provides instructions for drawings that are dimensioned in metric units. Soft metric is where the inch dimensions are converted to metric units and standard metric threads are used. This section is in accordance with ASME Y14.5-2009 where the interpretation of dimensioning and tolerancing is the same for the inch or the metric system. Dual dimensioning is where inch and mm units are stated on drawing. Dual dimensioning is no longer shown or recommended in the ASME Y14.5 dimensioning standard. Dual dimensioning has a problem where the number of inch verses metric millimeter (mm) decimal places may not give interchangeability or proper design functions. One way around this problem is to give the metric an extra decimal place to match the inch number of places. Typically CAD software automatically establishes the metric number of decimal places as one less than the inch number of decimal places. The solution is to set the CAD software to round off an extra decimal place and do not state plus-and-minus tolerance in the title block, except for angular tolerance. A general note may need to be added stating the tolerance for 2, 3, and 4 decimal places of metric dimensions and tolerance if the title block tolerance creates a conflict.

2.0 Definitions

2.1 Hard Metric – Hard metric drawings are designed with preferred metric standard sizes called the Renard series with no inch conversion and using the System International (SI) metric units.

2.2 Soft Metric – Soft metric conversion drawings maintain the original inch design but are converted to express the units of measurement in the SI metric language, including dimensioning and tolerancing in millimeters (mm). Soft conversion drawings are used when expensive tooling and production equipment cannot be immediately replaced or when the transition period to metric is limited. Soft metric may also include using metric fasteners.

2.3 Tolerances – Tolerance is the amount a dimension can vary. A tolerance may be found next to a dimension, in the title block, in a feature control frame, in general notes, on the drawing, or in a document (with a document number) stated on the drawing.

2.4 Gagemaker's Tolerance – Gagemaker's tolerance is typically 10 percent of the part tolerance. BASIC dimension SIZES stated on a drawing are given in a rectangle box, typically represented by holes and gage pins in fixtures. The Y14.5 dimensioning standard does not state the amount or percentage of gagemaker's tolerance and the tolerance is never given in the title block tolerance. The 10% gage tolerance includes manufacturing conditions such as temperature adjustment, Statistical Process Controls (SPC), etc.

2.5 Title Block Tolerance – When a three-place dimension on the face of a drawing has no stated tolerance next to the dimension, then the tolerance is generally found in the title block for three decimal places. If an implied 90 degree angle is shown on the drawing, such as for a rectangular part having eight corners, then the tolerance for all eight corners is the amount of tolerance stated in the title block for angles. For example, if the title block tolerance for angles is ±0°30′, then the implied 90 degree angles are 90°00′±0°30′. See Figure 1.

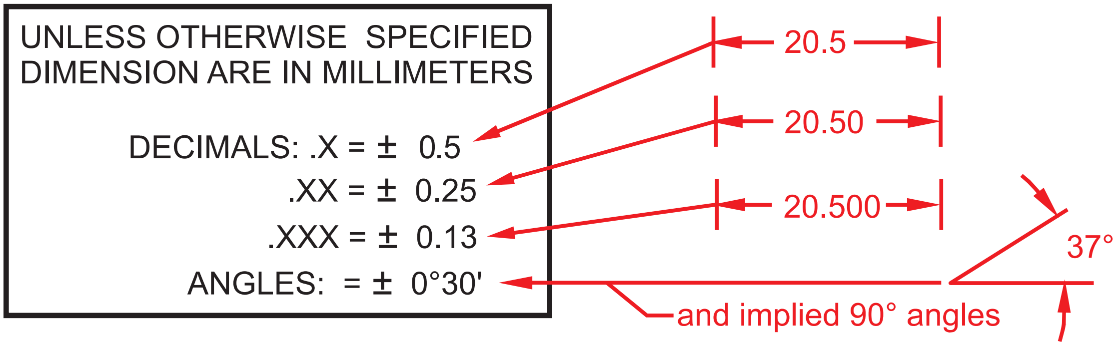

Figure 1

3.0 Standard Units

3.1 Metric Units – The units for linear dimensions shall be the millimeter (mm). Other units frequently used on drawings are:

"kilograms" for mass

"newtons" for force

"Celsius" for temperature

"kilopascals" for pressure

"megapascals" for stress

"kilowatts" for power

"newton-meters" for torque

3.2 Common Units to the Inch and Metric System – Some measurements can be stated so that the callout will satisfy the units of both systems; for example, .006 inch per inch, or 0.006 mm per mm can both be expressed simply as the ratio ".006:1" or in a note "TAPER .006:1". Angular dimensions are also specified the same in both systems; for example, 35°30′15″ or "35.609°".

3.3 Threads – Converting a .750-10 UNC thread to a M20 will provide a 20% stronger fastener, whereas a .500-13 UNC to a M12, will yield a 11% weaker fastener, where the materials are similar. Try to convert all threads to metric coarse threads, and avoid using metric fine threads in new designs. Use mm for diameters and lengths; use micrometers for plating thickness and surface roughness.

3.4 Other – If a standard part was originally in inches, it should continue to be defined in inches, if the part will still function as designed in a metric assembly.

3.5 General Notes - All drawings with dimensions in the metric system require two general notes:

ALL DIMENSIONS ARE IN MILLIMETERS.

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

3.6 Decimal Point - The decimal point must be large, and dark enough to clearly distinguish it when reduced size drawing copies are made. Do not use a comma for a decimal point on U.S. drawing. The CAD data file should be changed to black with no colors because copies may not be dark enough for some colors, such as blue.

3.7 Leading Zero – A leading zero is used or placed before a decimal point for metric dimensions or tolerances that are less than one millimeter; for example, 6.5±0.013.

3.8 Decimal Places – The decimal metric system may be shown as one-, two-, three-, or four-place decimals.

3.9 General Note Tolerance - If a general note states that all fillet radii are 0.08, then the general note must state a tolerance such as 0.08±0.02, or the tolerance will be found in the title block.

3.10 Rounding-off Decimal Values – Rounding-off is that condition where a lesser number of decimal places is justified for non critical design conditions.

3.10.1 Last Digit less than 5 - When the last digit is less than 5, there is no change to the last digit after rounding down.

0.1404 - 0.140 - 0.14 - 0.1

3.10.2 Last Digit greater than 5 - When the last digit is greater than 5, the last digit after rounding down is increased by one.

0.768 - 0.77 - 0.8

3.10.3 Last Digit is a 5 - When the last digit is 5, round down to the last even number.

0.0935 - 0.094

0.3125 - .312

3.11 Metric Decimal Examples – See Figure 2.

DIMENSIONAL TEXT METRIC mm	COMMENTS
25.00 - 25.45	Smaller number is first; dash; larger number is last.
25.45 25.00	Larger dimension number is above the smaller number.
$\boxed{25}$ *basic dim.* $\boxed{\oplus \mid \varnothing 0.13}$ *feature control frame*	The 25mm is a basic dimension with no decimal places. The tolerance for this 25mm is given in a feature control frame. The tolerance in the feature control frame can have a different number of decimal places than the basic dimension.
0.5 MIN or MAX	A metric dimension can be stated as minimum or maximum.
25.22 ±0.23	A metric dimension can have a ± tolerance with a zero in front.
$25^{+0.45}_{\;0}$	A unilateral metric tolerance has a single zero.
$25.15^{+0.30}_{-0.15}$	A bilateral tolerance must have the same number of decimal places.
25.22 ±0.23 [.993 ±.009]	On a metric drawing using dual dimensioning, the inch equivalent is shown in brackets.

Figure 2