DELPHI

TOOL DESIGN - MANUFACTURING DESIGN SPECIFICATIONS FOR TOOLING AND EQUIPMENT

SECTION C – DIMENSIONING & TOLERANCING

TABLE OF CONTENTS

TOOL DESIGN - MANUFACTURING DESIGN SPECIFICATIONS FOR TOOLING AND EQUIPMENT

SECTION C – DIMENSIONING & TOLERANCING

C.1 DIMENSIONS

1.1 Dimensions must be complete enough to describe the size, form, and location of each feature. **Coordinate dimensioning shall be used whenever possible.** The 0,0 start point should be in the lower left-hand corner of the dimensioned view. Exceptions would be starting from an internal hole or from a tooling ball. X,Y numbered hole charts should be provided for large plate type details with numerous holes.

1.1.1 Dimensions shall be shown in true profile views and refer to visible outlines rather than to hidden lines.

1.1.2 All dimension lines should be placed outside the outline of the detail view for clarity.

1.1.3 Hole locating dimensions and hole sizes are preferably shown in the circular view of the hole.

1.2 Dimensions shall be selected and arranged to avoid accumulation of tolerances, preclude more than one interpretation, and ensure satisfactory mating of parts.

1.2.1 Use interchangeability style dimensioning except in special instances.

1.2.2 If construction holes are used on mating details, reference dimensions must be given relating the holes to each other on the assembly drawing.

1.2.3 Show finish dimensions only. Process dimensions for rough machining, rough grinding, or prior to plating should not appear on the drawing.

1.3 Where practical, the finished part should be defined without specifying manufacturing methods. Thus only the diameter of a hole is given without indication as to whether it may be drilled, reamed, punched, or made by any other operation.

1.4 Counterbored holes are generally indicated by a note, giving the diameter and depth. Where the thickness of the remaining material has more significance, it may be dimensioned rather than the depth of the counterbore.

1.5 The preferred method of dimensioning Machine and Tool drawings is the decimal inch system.

1.5.1 This method is based on the use of two-place decimals; that is, two digits to the right of the decimal point. Three or four digits are used only where critical tolerances are required.

1.5.2 The exception to decimals is when calling for fractional thread and dowel sizes.

1.6 The secondary method of dimensioning is the millimeter system.

1.6.1 This method is based on the use of one-place decimals; that is, one digit to the right of the decimal point. Two or three digits are/used only where critical tolerances are required.

1.6.2 A value less than one millimeter will be shown with a zero to the left of the decimal point.

1.6.3 When all dimensions are in millimeters, the word “metric” must appear in 10.0mm high letters beside the title block.

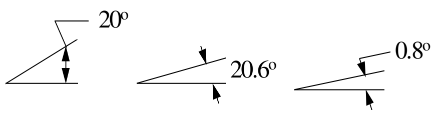
1.7 When dual dimensioning is requested or Required, the decimal-inch dimension is placed above the metric dimension. The metric dimension is designated with the suffix mm and enclosed within brackets.

Example: 1.00
[25.4 mm]

1.8 Angular dimensions must be expressed in degrees and decimals of a degree. (Do not use minutes and seconds.)

1.8.1 An angle less than one degree will be shown with a zero to the left of the decimal point.

Example:



C.2 DIMENSIONING RADII

2.1 The letter "R" should always follow the radius size.

2.2 If a spherical radius is being dimensioned, "Sphere R" should follow the size.

2.3 Where the center of a radius is outside the drawing or interferes with another view, the radius dimension line may be broken and foreshortened.

2.4 A curved line composed of two or more circular arcs should be dimensioned by noting the radii, locating their centers, and locating their tangency points.

2.5 Symmetrical outlines may be dimensioned on one side of the axis of symmetry only.

2.6 The recommended method for dimensioning a chamfer is to give a length and an angle. The dimension is the measurement along the length of the part, and not along the slope of the chamfer.

C.3 TOLERANCING

3.1 All dimensions required for the machining of a detail have a tolerance except those identified as reference, basic and gage. Tolerances, other than specified in the Title Block, may be expressed in one of the following ways:

3.1.1 As specific limits or tolerances shown directly on the drawing for a specific dimension.

3.1.2 In the form of a special note referring to specific dimensions.

3.2 When using the limit style dimensioning, the high limit (maximum material condition) should always be placed above the low limit (minimum material condition).

3.2.1 The high and low limits should contain the same number of decimal places.

3.3 When using the plus and minus tolerance method, the dimension of the specified size is given first and is followed by a plus and minus expression of tolerance.

3.4 Location tolerances for equally spaced holes are noted as non-accumulative.

3.5 Dowels shall be specified as either slip fit (SF) or press fit (PF).

3.6 Specify hole size with tolerance for bearings, bushings, etc. and shaft diameters, instead of indicating P.F. or S.F. Use tolerances from the recommended manufacturers catalog when available.

C.4 MICRO-FINISH TOLERANCE

4.1 General surface finish as now designated by the letter f are assumed to be 125 micro inch finish. As a high grade machine finish can be produced to 32 micro inch and also be ground at less expense, every finish should be reviewed for ease and cost to manufacture. The "when in doubt make smooth" is not cost effective and in general has little effect on the tolerances required.

4.2 The surface finishes are standardized by the American Standards Association B46.1-1955 This system has many surface characteristics but our initial specification will be confirmed to "ROUGHNESS HEIGHT".

C.5 GEOMETRIC TOLERANCING

5.1 Geometric tolerancing is a means of dimensioning and tolerancing a drawing with respect to the actual function and relationship of features which can be most economically produced.

Use of this tolerancing system is when:

5.2 Part features are critical to function.

5.3 Functional gaging techniques are desirable.

5.4 Datum references are desirable to insure consistency between manufacturing and gaging operations.

5.5 Standard tolerance is not already implied.