

JEDEC DESIGN STANDARD

DESIGN REQUIREMENTS FOR OUTLINES OF SOLID STATE AND RELATED PRODUCTS

JEDEC PUBLICATION 95

DESIGN GUIDE 4.2

General Requirements

**JEDEC
SOLID STATE TECHNOLOGY ASSOCIATION**

Date:
Item:

Issue:

2 GENERAL REQUIREMENTS

2.1 Geometric Dimensioning and Tolerancing

The JEDEC Solid State Product Engineering Council has mandated the use of Geometric Dimensioning and Tolerancing (GD&T) for all outlines. ANSI Y14.5M - 1982 "Dimensioning and Tolerancing" is the latest standard and is the basic reference for this Handbook and outlines in Publication 95.

GD&T uses the principle of worst case analysis in order to assure complete interchangeability and noninterference or interference (as desired) of mating parts.

Some concepts of GD&T are discussed below. The reader is referred to ANSI Y14.5M - 1982 for a complete treatment of this subject.

2.2 Definitions

Feature

A physical portion of a part, such as a surface, hole or slot.

Basic Dimension

A numerical value used to describe the theoretically exact size, profile, orientation or location of a feature or datum target. Permissible variations from the basic dimension are established by tolerances on associated dimensions, in notes or in feature control frames.

True Position

The theoretically exact location of a feature established by basic dimensions.

Datum

A theoretically exact point, axis, or plane established by tooling which is used in conjunction with a Datum Feature. The location or geometric characteristics of features of a part are established in relation to the datum.

Datum Feature

The physical portion of a part that, in conjunction with suitable tooling, establishes the Datum. Note, a centerline, by itself, cannot be a datum. It must be the centerline of a

JEDEC Standard No. 95-1**PAGE 2-2****2.2 Definitions (continued)**

physical feature such as a hole or boss etc. In these cases, it is the diameter of the hole or the width of the boss that is designated as the datum. The point of reference is the centerline of these physical features.

Maximum Material Condition (MMC)

A feature of size that contains the maximum amount of material; e.g., maximum shaft diameter or minimum hole diameter.

Least Material Condition (LMC)

A feature of size that contains the least amount of material; e.g., minimum shaft diameter or maximum hole diameter.

Locus Condition

The envelope established by the combined effects of tolerance of size or form and location. Note that ANSI Y14.5M-1982 defines virtual condition as the combined effect of Maximum Material Condition and geometric tolerance (position, orientation etc.). A definition for the combined effect of Least Material Condition and geometric tolerance is not given. Since the latter is an important characteristic that is used in analyzing the mating conditions between leads and lands, it is necessary to set up additional definitions to allow the analysis to proceed in a logical manner. To this end, the following two definitions have been formulated; Outer Locus (OL) and Inner Locus (IL).

The following formulas play an important part in analyzing mating and registration conditions in the interconnection industry. These formulas are written with positional tolerances. This is the most useful form for analyzing mating conditions. In the general form, these formulas can use geometric tolerances (perpendicularity, etc.) in place of the positional tolerances.

Outer Locus (OL)**Male Feature¹**

This is the worst case (largest) boundary or envelope of the outer surface of the feature as it moves around the tolerance zone of position.

$$OL = MMC + Pos. Tol.....(1)$$

¹ See appendix B, Section 1, for examples.

2.2 Definitions (continued)

Female Feature²

This is the worst case (largest) boundary or locus formed outside of the surface of the feature as it moves around the tolerance zone of position.

$$OL = LMC + \text{Pos. Tol.} \dots\dots\dots(2)$$

Inner Locus (IL)**Male Feature²**

This is the worst case (smallest) boundary or locus formed inside the surface of the feature as it moves around the tolerance zone of position.

$$IL = LMC - \text{Pos. Tol.} \dots\dots\dots(3)$$

Female Feature²

This is the worst case (smallest) boundary or envelope of the inner surface of the feature as it moves around the tolerance zone of position.

$$IL = MMC - \text{Pos. Tol.} \dots\dots\dots(4)$$

Profile Tolerancing³

This is a tolerance zone that lies along the true profile of the part. All elements of the profile must lie in this zone. Form as well as combinations of form, size and orientation can be controlled by this tolerance.

² See appendix B, Section 1, for examples

³ See appendix B, Section 3, for examples.

2.2 Definitions (continued)

Coplanarity⁴

This is the condition where an interrupted surface, or two or more surfaces, must have all the elements in one plane. The tolerance zone is established by two parallel planes between which all elements of the interrupted surface must lie. This is analogous to the flatness requirement for a continuous surface. For this Standard, use the method described in Appendix B, section 4.2.2.

2.3 Principles of Mating or Registration

The principles of mating or registration between two components of a system are based on worst case considerations. Two classes of components will be considered: Guided components such as DIPs whose leads enter a hole in a PC board and unguided components such as leadless chip carriers that mate with contacts in a socket. The designation of "unguided" does not imply a lack of positional control. The distinction between the two classes is necessitated by the different methods of analysis.

2.3.1 Guided Components.

2.3.1.1 Through Hole Mount

A typical application is the mounting of a DIP on a PC board with the leads engaging plated through holes. **Figure 2-1 view (a)** shows the tip, dimension "B" in relation to the diameter "D" of the plated through hole (plating is not indicated).

The body diameter of the lead is smaller than the plated through hole in accordance with good soldering practice. The small tip of the lead has to be held in proper relation to the hole in order to have the chamfer guide the lead into the hole. The Locus condition is used in the analysis since that condition combines size and positional tolerance effects. In normal applications, the Maximum Material Condition of the lead and hole is used as shown in Figure 2-1. However, formula 5 is also applicable when the least Material Condition modifier is used. This is so since the Inner Locus (IL) of "D" represents the smallest effective hole and the Outer Locus (OL) of "B" represents the largest effective lead tip.

⁴ See appendix B, Section 4, for examples.

2.3.1.1 Through Hole Mount (Continued)

Figure 2-1, View (b) shows the Locus condition of the two features. The positional tolerances shown in the equations is the sum of all variations in location from a datum that is common to both features. Unless a common datum or starting point is used, the analysis is not valid. Interference is indicated by negative clearance.

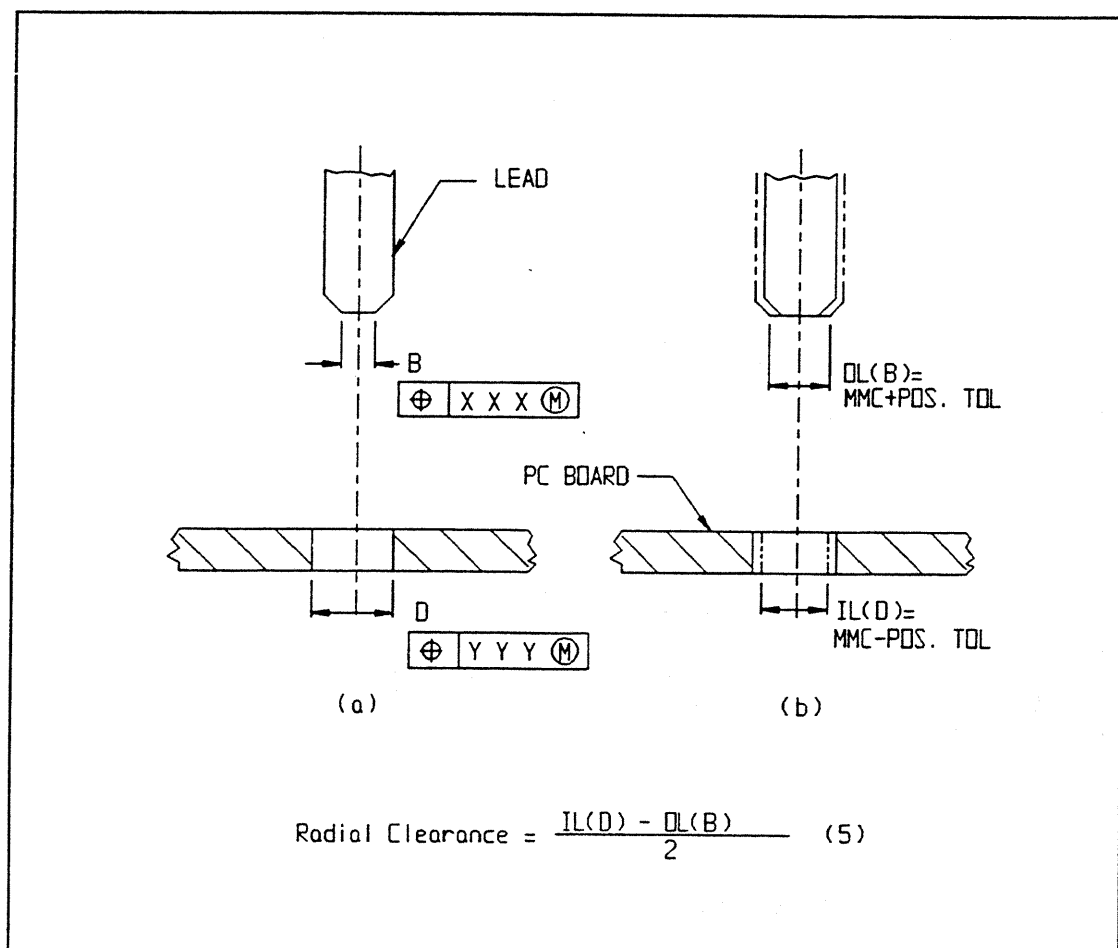


Figure 2-1
Radial Clearance - Printed Wiring Board

2.3.1.2 Socket or Connector Mount

Another typical application is socketing a DIP. **Figure 2-2, View (a)** shows the tip, dimension "B", of the lead in relation to the diameter, "D", of the lead-in of the socket cavity. **Figure 2-2, View (b)** shows the Locus condition of the two features. The comments of paragraph 2.3.1.1 with respect to positional tolerances and interference also apply to this application.

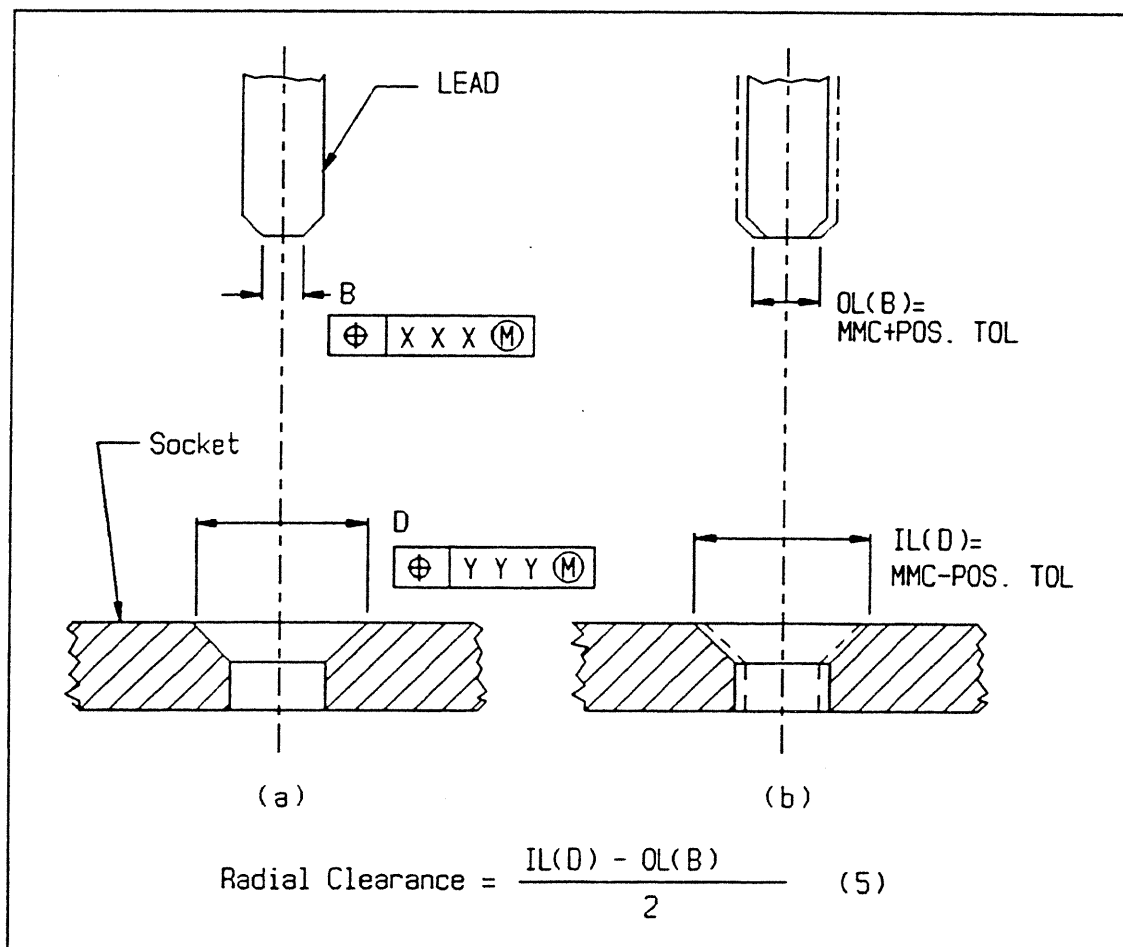


Figure 2-2
Radial Clearance - Socket