ASSOCIATION CONNECTING
ELECTRONICS INDUSTRIES

IPC-2615

Printed Board Dimensions and Tolerances

IPC-2615

July 2000

A standard developed by IPC

2215 Sanders Road, Northbrook, IL 60062-6135
Tel. 847.509.9700 Fax 847.509.9798
www.ipc.org

The Principles of Standardization

In May 1995 the IPC's Technical Activities Executive Committee adopted Principles of Standardization as a guiding principle of IPC's standardization efforts.

Standards Should:
- Show relationship to Design for Manufacturability (DFM) and Design for the Environment (DFE)
- Minimize time to market
- Contain simple (simplified) language
- Just include spec information
- Focus on end product performance
- Include a feedback system on use and problems for future improvement

Standards Should Not:
- Inhibit innovation
- Increase time-to-market
- Keep people out
- Increase cycle time
- Tell you how to make something
- Contain anything that cannot be defended with data

Notice

IPC Standards and Publications are designed to serve the public interest through eliminating misunderstandings between manufacturers and purchasers, facilitating interchangeability and improvement of products, and assisting the purchaser in selecting and obtaining with minimum delay the proper product for his particular need. Existence of such Standards and Publications shall not in any respect preclude any member or nonmember of IPC from manufacturing or selling products not conforming to such Standards and Publication, nor shall the existence of such Standards and Publications preclude their voluntary use by those other than IPC members, whether the standard is to be used either domestically or internationally.

Recommended Standards and Publications are adopted by IPC without regard to whether their adoption may involve patents on articles, materials, or processes. By such action, IPC does not assume any liability to any patent owner, nor do they assume any obligation whatever to parties adopting the Recommended Standard or Publication. Users are also wholly responsible for protecting themselves against all claims of liabilities for patent infringement.

IPC Position Statement on Specification Revision Change

It is the position of IPC's Technical Activities Executive Committee (TAEC) that the use and implementation of IPC publications is voluntary and is part of a relationship entered into by customer and supplier. When an IPC standard/guideline is updated and a new revision is published, it is the opinion of the TAEC that the use of the new revision as part of an existing relationship is not automatic unless required by the contract. The TAEC recommends the use of the lastest revision.

Adopted October 6. 1998

Why is there a charge for this standard?

Your purchase of this document contributes to the ongoing development of new and updated industry standards. Standards allow manufacturers, customers, and suppliers to understand one another better. Standards allow manufacturers greater efficiencies when they can set up their processes to meet industry standards, allowing them to offer their customers lower costs.

IPC spends hundreds of thousands of dollars annually to support IPC's volunteers in the standards development process. There are many rounds of drafts sent out for review and the committees spend hundreds of hours in review and development. IPC's staff attends and participates in committee activities, typesets and circulates document drafts, and follows all necessary procedures to qualify for ANSI approval.

IPC's membership dues have been kept low in order to allow as many companies as possible to participate. Therefore, the standards revenue is necessary to complement dues revenue. The price schedule offers a 50% discount to IPC members. If your company buys IPC standards, why not take advantage of this and the many other benefits of IPC membership as well? For more information on membership in IPC, please visit www.ipc.org or call 847/790-5372.

Thank you for your continued support.

ASSOCIATION CONNECTING ELECTRONICS INDUSTRIES

IPC-2615

Printed Board Dimensions and Tolerances

Developed by the Dimensioning and Tolerancing Task Group (1-10a) of the Printed Board Design Committee (1-10)

Users of this standard are encouraged to participate in the development of future revisions.

Contact:

IPC
2215 Sanders Road
Northbrook, Illinois
60062-6135
Tel 847 509.9700
Fax 847 509.9798

Acknowledgment

Any Standard involving a complex technology draws material from a vast number of sources. While the principal members of the Dimensioning and Tolerancing Task Group (1-10a) of the Printed Board Design Committee (1-10) are shown below, it is not possible to include all of those who assisted in the evolution of this standard. To each of them, the members of the IPC extend their gratitude.

Printed Board Design Committee

Chairman
Rick Hartley
Applied Innovation

Dimensioning and Tolerancing Task Group

Chairman
John A. Sabo. C.I.D.
Rockwell Automation/
Allen Bradley

Technical Liaisons of the IPC Board of Directors

Stan Plzak
Pensar Corp.

Peter Bigelow
Beaver Brook
Circuits Inc.

Dimensioning and Tolerancing Task Group

Leon Cohen

C. Don Dupriest, Lockheed Martin Corporation

Will J Edwards, Lucent Technologies Inc.

Joe Fjelstad, Pacific Consultants LLC

Pierre Gadoua, Bae Systems Canada, Inc.

John H. Morton, C.I.D., Lockheed Martin Corporation

Deepak K. Pai, C.I.D., General Dynamics Information Sys. Inc.

John A. Sabo, C.I.D., Rockwell Automation/Allen-Bradley

Margaret Terpening, Boeing Phantom Works

Wally Younger, Nelco Technology, Inc.

Table of Contents

Figures

Tables

This Page Intentionally Left Blank

Printed Board Dimensions and Tolerances

1 PURPOSE

The purpose of this Standard is to establish acceptable principals and practices for dimensioning and tolerancing used to define end-product requirements for printed boards and printed board assemblies.

1.1 Scope This Standard covers dimensioning and tolerancing of electronic packaging as it relates to printed boards and the assembly of printed boards. The concepts defined in this Standard are derived from ASME Y14.5M-1994. Printed boards have such wide applications that there may be times where this standard does not address a specific case. In those cases, the user is referred to ASME Y14.5M 1994 for use of additional dimensioning and tolerancing concepts.

1.2 General This Standard covers dimensioning, tolerancing, and related practices for use on printed board drawings and in related documents. Uniform practices for stating and interpreting these requirements are established herein.

1.2.1 Units The International System of Units (Sl) is featured in this Standard.

1.2.2 Reference to This Standard Where drawings are based on this Standard, this fact shall be noted on the drawings or in a document referenced on the drawings. References to this Standard shall state ‘‘IPC-2615 or per IPC-2615.’’

1.2.3 Figures The figures in this Standard are intended only as illustrations to aid the user in understanding the principles and methods of dimensioning and tolerancing described in the text. The absence of a figure illustrating the desired application is neither reason to assume inapplicability nor basis for drawing rejection. In some instances figures show added detail for emphasis, in other instances figures are incomplete by intent. Numerical values of dimensions and tolerances are illustrative only.

1.2.4 Notes Notes herein in capital letters are intended to appear on finished drawings. Notes in lower case letters are explanatory only and are not intended to appear on drawings.

1.2.5 Reference to Gauging This document is not intended as a gauging standard. Any reference to gauging is included for explanatory purposes only.

1.3 References

1.3.1 IPC Specifications[1]

IPC-T-50 Terms and Definitions

IPC-D-310 Guidelines for Phototool and Artwork Generation

IPC-D-325 Documentation for Printed Boards and Printed Board Assemblies

IPC-D-330 Design Guide for Printed Boards and Printed Board Assemblies

IPC-2220 Design Standard Series for Printed Boards

IPC-6010 Performance Specification Series for Printed Boards

1.3.2 ANSI Standards[2] When the following American National Standards referred to in this Standard are superseded by a revision approved by the American National Standards Institute, Inc., the latest revision shall apply.

ANSI Y14.1-1980, Drawing Sheet Size and Format
ANSI Y14.2M-1979, Line Conventions and Lettering
ASME Y14.5M-1994, Geometric Dimensioning and Tolerancing
ANSI Z210.1-1976, Metric Practice

2 TERMS AND DEFINITIONS

The definition of terms shall be in accordance with IPC-T-50 and the following.

2.1 Actual Size The measured size.

2.2 Basic Dimension A numerical value used to describe the theoretically exact size, profile, orientation, or location of a feature or datum target. It is the basis from which permissible variations are established by tolerances on other dimensions, in notes, or in feature control frames (see 3.4.1).

2.3 Bilateral Tolerance A tolerance in which variation is permitted in both directions from the specified dimension.

2.4 Cumulative Tolerances The summation of all tolerances permitted between functionally related features:

1. IPC, 2215 Sanders Road, Northbrook, IL 60062
2. ANSI, 655 15th Street N.W., Suite 300, Washington, DC 20005-5794

a) ***Chain Dimensioning*** The maximum variation between two features is equal to the sum of the tolerances on the intermediate distances; this results in the greatest tolerance accumulation.

b) ***Base Line Dimensioning*** The maximum variation between two features is equal to the sum of the tolerances on the two dimensions from their origin to the features; this results in a reduction of tolerance accumulation.

c) ***Basic Dimensioning*** The maximum variation between two features is controlled by the positional tolerance on the two features; this results in zero tolerance accumulation.

2.5 Datum A theoretically exact point, axis, or plane derived from the true geometric counterpart of a specified datum feature. A datum is the origin from which the location or geometric characteristics of features of a printed board are established.

2.6 Datum Feature An actual feature of a printed board that is used to establish a datum.

2.7 Datum Axis The theoretical axis derived from the true geometric counterpart of a specified feature (i.e., tooling hole, fiducial) as established by the extremities of contacting points of the actual datum feature.

2.8 Datum Target A specified point or area on a printed board used to establish a datum.

2.9 Dependent of Size The concept that permits tolerances of form or position to vary in proportion to, and dependent on, the size of the feature.

2.10 Dimension A numerical value expressed in appropriate units of measure and indicated on a drawing and in other documents along with lines, symbols and notes to define the size, location or geometric characteristic of a printed board or printed board feature.

2.11 End Product (End Item) An end product is the individual printed board or assembly in its final or completed state.

2.12 Fabrication Allowance An amount added to a printed board feature, e.g., the diameter of a land, which is an accumulation of manufacturing variation. The fabrication allowance is intended to assure that manufacturing variation does not allow certain performance characteristics to be exceeded, such as minimum annular ring.

2.13 Feature The general term applied to a physical portion of a printed board or printed board assembly, such as a surface, hole, slot, or surface mount land.

2.14 Feature of Size One cylindrical surface or a set of two plane parallel surfaces, each of which is associated with a size dimension.

2.15 Fiducial A printed board artwork feature (or features) that is created in the same process as the printed board conductive pattern and that provides a common measurable point for component mounting with respect to a land pattern or land patterns.

2.16 Geometric Tolerance The general term applied to the category of tolerances used to control form, profile, orientation, and location.

2.17 Limits of Size The specified maximum and minimum sizes.

2.18 Least Material Condition (LMC) The condition in which a feature of size contains the least amount of material within the stated limits of size - for example, maximum hole diameter, or minimum printed board size.

2.19 Maximum Material Condition (MMC) The condition in which a feature of size contains the maximum amount of material within the stated limits of size; for example, minimum hole diameter, maximum printed board size.

2.20 Positional Tolerance The amount that a feature is permitted to vary from the location of true position.

2.21 Reference Dimension A dimension, usually without tolerance, used for information purposes only. It is considered auxiliary information and does not govern production or inspection operations. A reference dimension is a repeat of a dimension or is derived from other values shown on the drawing or on related drawings.

2.22 Regardless of Feature Size (RFS) The term used to indicate that a geometric tolerance or datum reference applies at any increment of size of the feature within its size tolerance. Regardless of feature size permits no additional positional, form, or orientation tolerance other than that stated in the applicable feature control frame. RFS can only be applied to features of size.

2.23 Simulated Datum A point, axis, or plane established by processing or inspection equipment.

2.24 Tolerance The total amount by which a specific dimension is permitted to vary. The tolerance is the difference between the maximum and minimum limits.

2.25 Tolerance, Statistical A tolerance that is based on statistical models, usually combining a variety of specific tolerances i.e., Root Mean Square (RMS) value.

2.26 Toleranced Dimension A dimension with a directly applied tolerance as opposed to a basic dimension that specifies the exact size or location of a feature.

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

2.28 Undimensioned Drawing An undimensioned drawing depicts to a precise scale on environmentally stable material a template or a pattern for which dimensioned detail drawings would be impractical.

2.29 Unilateral Tolerance A tolerance in which variation is permitted in one direction from the specified dimension.

2.30 Virtual Condition The boundary generated by the collective effects of the specified MMC or LMC limit of size of a feature and any applicable geometric tolerances.

3 GEOMETRIC CHARACTERS AND SYMBOLS

3.1 General This Section establishes the symbols for specifying geometric characteristics and other dimensional requirements typically used on PCB engineering drawings. Symbols should be of sufficient clarity to meet the legibility and reproducibility requirements of ANSI Y14.2M. For symbols not described in the following, refer to ASME Y14.5M-1994.

3.2 Use of Notes to Supplement Symbols Situations may arise where the desired geometric requirement cannot be completely conveyed by symbology. In such cases, a note may be used to describe the requirement, either separately or supplementing a geometric tolerance. See 6.3.5.2.

3.3 Symbol Construction Information related to the construction, form, and proportion of individual symbols described herein is contained in ASME Y14.5M-1994 Appendix C.

3.3.1 Geometric Characteristic Symbols The symbols denoting central geometric characteristics are shown in Table 3-1. Table 3-2 shows other symbols used in special applications.

3.3.2 Datum Feature Symbol The datum feature symbol consists of a capital letter enclosed in a square frame with a leader line extending from the frame to the concerned feature, terminating with a triangle (see Figure 3-1). The

CHARACTERISTIC	SYMBOL	TYPE OF TOLERANCE	USES
POSITION	⌖	LOCATION	HOLE AND LAND LOCATION
PROFILE OF SURFACE	⌓	PROFILE	BOARD EDGES
FLATNESS	⏥	FORM	BOW & TWIST

IPC-2615-t3-01

Table 3-1 General Geometric Characteristic Symbols

CHARACTERISTIC	SYMBOL	TYPE OF TOLERANCE	USES
STRAIGHTNESS	—	FORM	BONDING OF HEATSINK
CIRCULARITY	○	FORM	ROUND PRINTED BOARD, TIGHT FITTING CASE
ANGULARITY	∠	ORIENTATION	SPECIAL SLOT OF FEATURE CONTROL
PERPENDICULARITY	⊥	ORIENTATION	HOLE TO THICK BOARD RELATIONSHIP
PARALLELISM	//	ORIENTATION	EDGES TO TIGHT FITTING CASE
CONCENTRICITY	◎	LOCATION	METAL CORE BOARD HOLES, TEST FEATURE

IPC-2615-t3-02

Table 3-2 Special Application Symbols

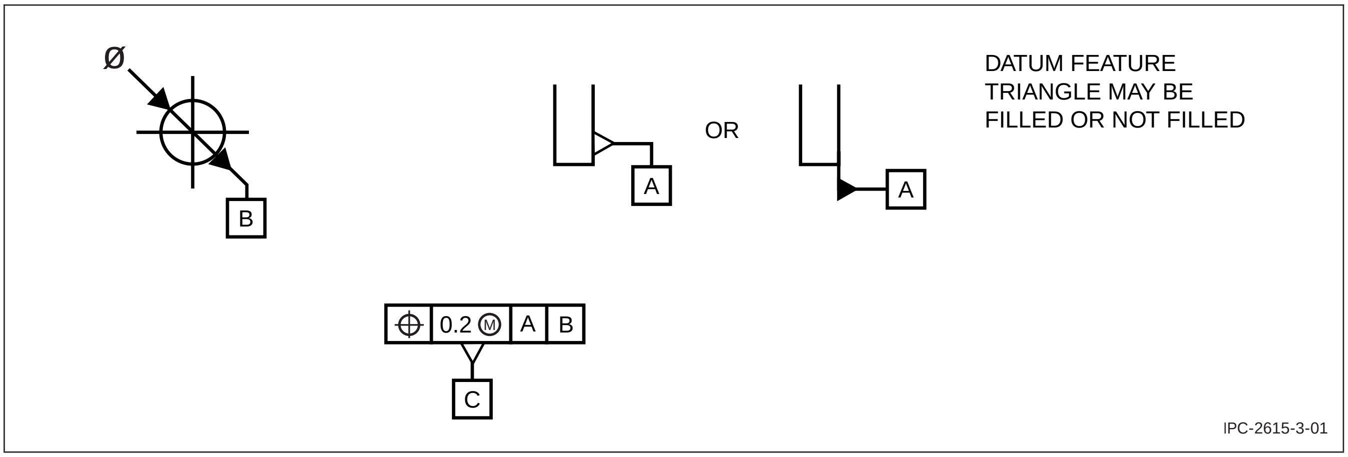


Figure 3-1 Datum Feature Symbol

triangle may be filled or not filled. The datum feature symbol is applied to the concerned feature surface outline, extension line, dimension line, or feature control frame as shown in Figure 3-2. The symbol frame is related to the datum feature by one of the methods prescribed in 3.5.

3.3.2.1 Letters of the Alphabet Letters of the alphabet (except I, O, and Q) are used as datum identifying letters. Each datum feature requiring identification shall be assigned a different letter.

3.3.2.2 Repeated Datum Feature Symbols Where the same datum feature symbol is repeated to identify that same feature in other locations on a drawing, it need not be identified as a reference.

3.3.3 Basic Dimension Symbol A basic dimension states only half of the requirement; a tolerance must be associated with the feature to complete the requirement.

The symbolic means of indicating a basic dimension is shown in Figure 3-3.

3.3.4 Material Condition Symbols The symbols used to indicate ''at maximum material condition,'' ''regardless of feature size,'' and ''at least material condition'' are shown in Table 3-3. The use of these symbols in local and general notes is prohibited. When no material modifier is used, RFS is assumed.

The use of MMC permits greater possible tolerance as the part feature deviates from its maximum material condition. It also assures interchangeability and permits functional gauging techniques. The maximum material condition principle is normally valid only when both of the following conditions exist.

Two or more features are interrelated with respect to location or orientation (e.g., a hole and an edge or surface, two holes, etc.). At least one of the related features is a feature of size. The feature (or features) to which the MMC principle is to apply must be a feature of size (e.g., a hole, slot, conductor, etc.) with an axis or center plane.

Least material condition applies when special design requirements will not accommodate MMC or when RFS is too strict. It can be used to maintain critical center locations of features for which the positional tolerance can be increased as the size of the feature departs from its least material condition. The amount of increase in positional tolerance permissible is equal to the feature size departure from LMC.

3.3.5 Diameter and Radius Symbols The symbols used to indicate diameter and radius are shown in Table 3-3. These symbols precede the value of a dimension or tolerance given as a diameter or radius, as applicable.

3.3.6 Reference Symbol A reference dimension (or reference data) is identified by enclosing the dimension (or data) within parentheses. See Table 3-3.

3.4 Geometric Tolerance Symbols Geometric characteristic symbols, the tolerance value, and datum reference letters, where applicable, are combined in a feature control frame to express a geometric tolerance.

3.4.1 Feature Control Frame A geometric tolerance for an individual feature is specified by means of a feature control frame divided into compartments containing the geometric characteristic symbol followed by the tolerance (see Figure 3-4). Where applicable, the tolerance is preceded by the diameter symbol and followed by a material condition symbol.

3.4.2 Feature Control Frame Incorporating Datum References Where a geometric tolerance is related to a datum, this relationship is indicated by entering the datum reference letter in a compartment following the tolerance

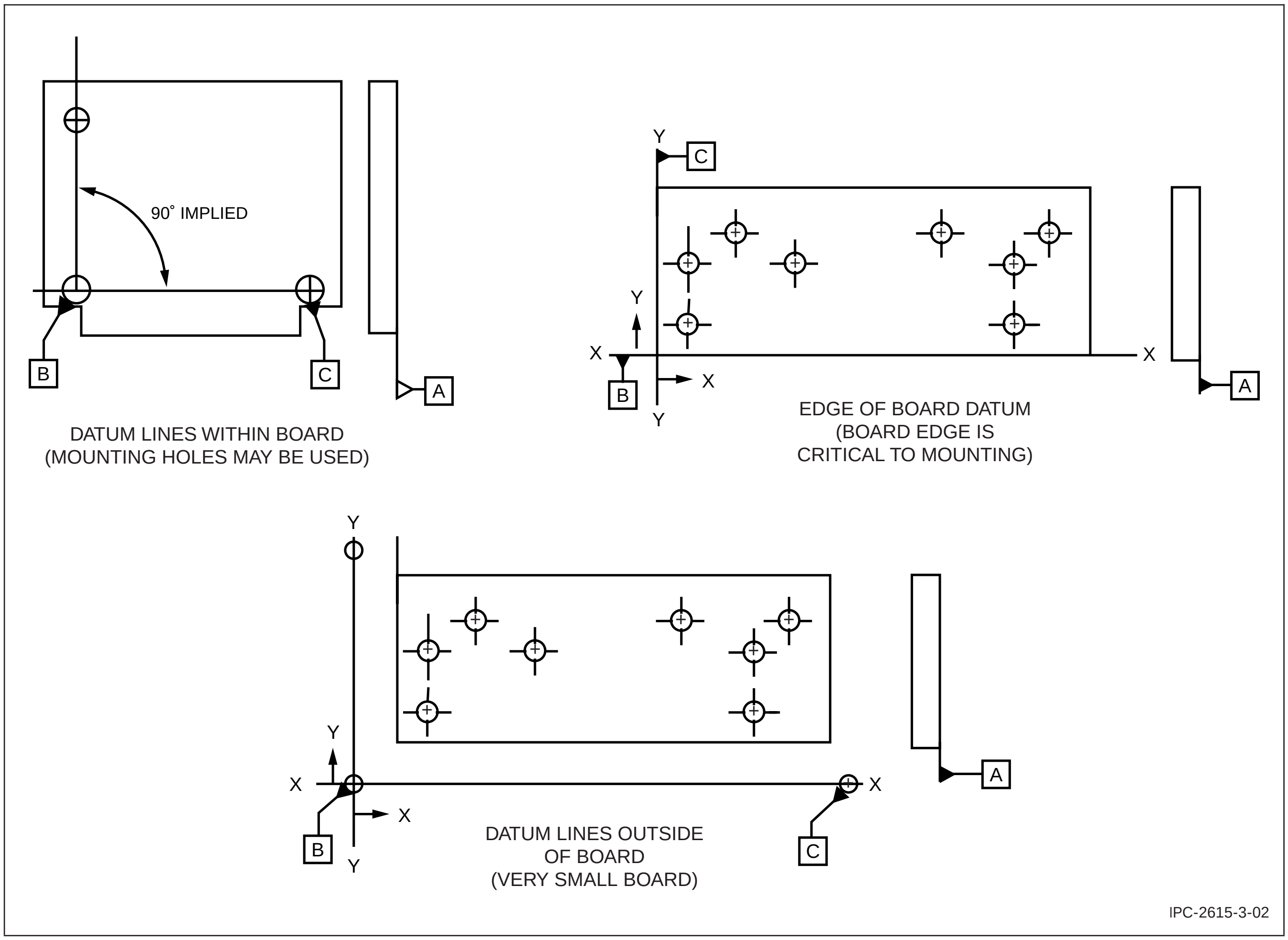


Figure 3-2 Examples of Datum Identification

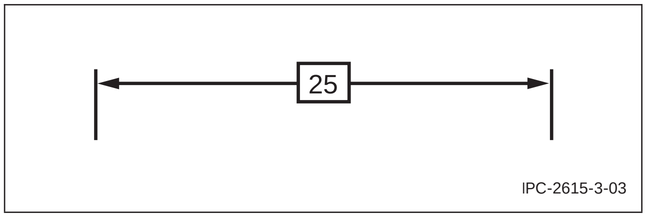


Figure 3-3 Basic Dimension Symbol

(see Figure 3-5). Where applicable, the datum reference letter is followed by a material condition symbol.

3.4.2.1 Multiple Datum Requirements Where more than one datum is required, the datum reference letters (followed by a material condition symbol, where applicable) are entered in separate compartments in the desired order of precedence, from left to right (see Figure 3-6). Datum reference letters need not be in alphabetical order in the feature control frame.

3.4.2.2 Composite Feature Control Frame A composite feature control frame is used where more than one tolerance is specified for the same geometric characteristic of a feature or features having different datum requirements. The composite frame contains a single entry of the geometric characteristic symbol followed by each tolerance and datum requirement, one above the other (see Figure 3-7 and 6.4.1).

3.4.2.3 Common Profile Tolerance Symbol The symbol used to indicate that a profile tolerance applies to surfaces all around the printed board is a circle located at the junction of the leader from the feature control frame (see Figure 3-8).

3.4.3 Combined Feature Control Frame and Datum Feature Symbol Where a feature or pattern of features controlled by a geometric tolerance also serves as a datum feature, the feature control frame and datum feature symbol are combined (see Figure 3-9).

3.4.3.1 Control Frame and Datum Feature Symbol Combinations Wherever a feature control frame and datum feature symbol are combined, datums referenced in the feature control frame are not considered part of the datum feature symbol. In the positional tolerance example, Figure

TERM	SYMBOL
MAXIMUM MATERIAL CONDITION	Ⓜ
LEAST MATERIAL CONDITION	Ⓛ
DIAMETER	⌀
RADIUS	R
REFERENCE	( )
ARC LENGTH	⌒
STATISTICAL TOLERANCE	⟨ST⟩
BETWEEN	↔
REGARDLESS OF FEATURE SIZE	Ⓢ

IPC-2615-t3-03

Table 3-3 Modifying Symbols

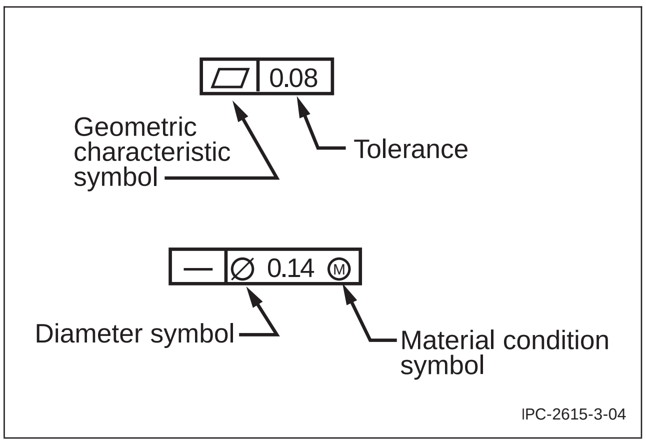


Figure 3-4 Feature Control Frame

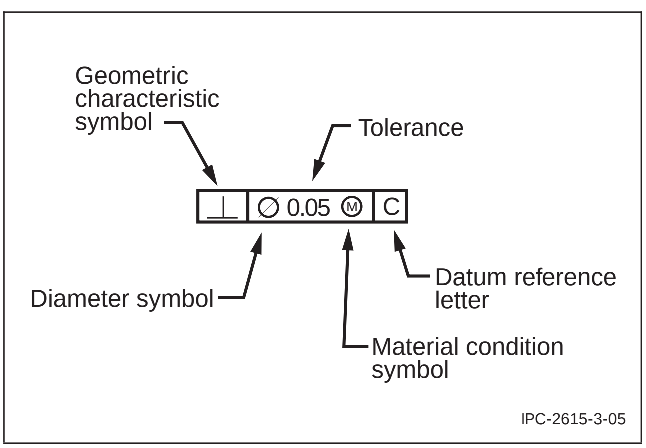


Figure 3-5 Feature Control Frame Incorporating Datum Report

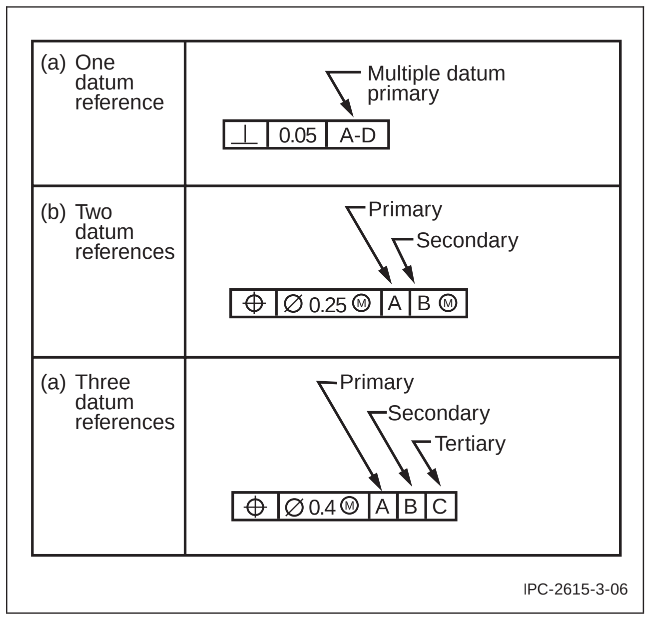


Figure 3-6 Order of Precedence of Datum Reference

3-9, a feature is controlled for position in relation to datums A and B, and identified as datum C. Whenever datum C is referenced elsewhere on the drawing, the reference applies to datum C not to datums A and B.

3.5 Feature Control Frame Placement The feature control frame is related to the considered feature by one of the following methods as depicted in Figure 3-10.

a) locating the feature control frame below or attached to a leader-directed callout or dimension pertaining to the feature;

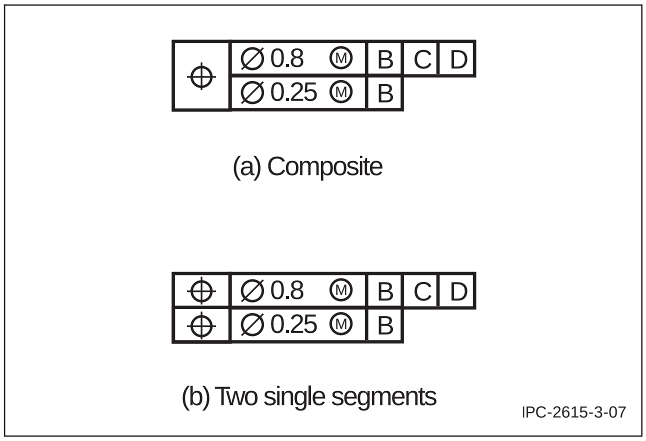


Figure 3-7 Multiple Feature Control Frames

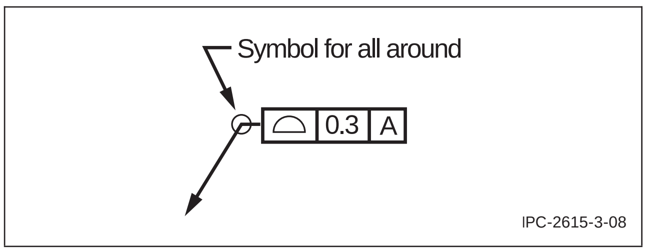


Figure 3-8 Symbol for All Around

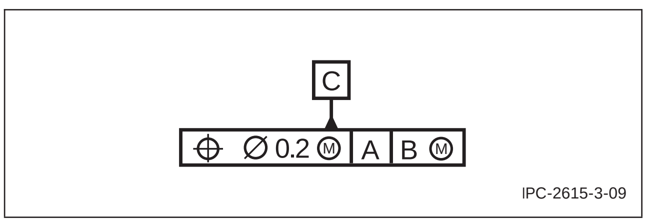


Figure 3-9 Combined Feature Control Frame and Datum Feature Symbol

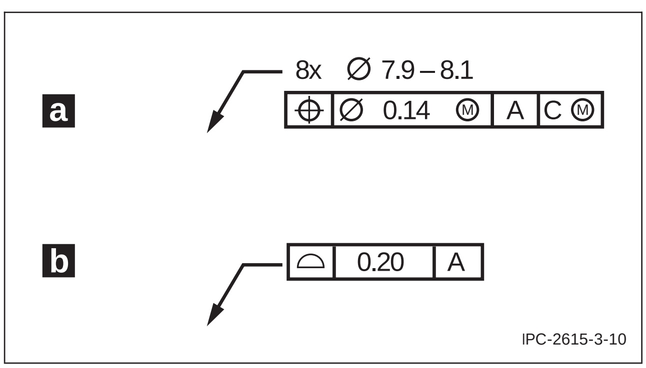


Figure 3-10 Feature Control Frame Placement

b) running a leader from the frame to the feature;

c) attaching a side or an end of the frame to an extension line from the feature, provided it is a plane surface;

d) attaching a side or an end of the frame to an extension of the dimension line pertaining to a feature of size.

4 GENERAL RULES

The following rules apply to the use of material condition modifiers.

4.1 Maximum Material Condition Principle (MMC) Effect of MMC. Where a geometric tolerance is applied on an MMC basis, the specified tolerance is interdependent on the size of the considered feature. The tolerance is limited to the specified value if the feature is produced at its MMC limit of size. Where the actual size of the feature has departed from MMC, an increase in the tolerance is allowed equal to the amount of such departure. The total permissible variation in the specific geometric characteristic is maximum when the feature is at LMC (see Figure 4-1). Referencing a datum feature on an MMC basis means the datum is the axis or center plane of the feature at the MMC limit. Where the actual size of the datum feature has departed from MMC, a deviation is allowed between its axis or center plane and the axis or center plane of the datum (see Table 4-1).

4.2 Regardless of Feature Size Where a geometric tolerance is applied on an RFS basis, the specified tolerance is independent of the size of the considered feature. The tolerance is limited to the specified value regardless of the actual size of the feature. Likewise, referencing a datum feature on an RFS basis means that a centering about its axis or center plane is necessary, regardless of the actual size of the feature (see Table 4-2).

4.3 Least Material Condition Principle Where a positional tolerance is applied on an LMC basis, the specified tolerance is interdependent on the size of the considered feature. The tolerance is limited to the specified value if the feature is produced at its LMC limit of size. Where the actual size of the feature has departed from LMC, an increase in the tolerance is allowed equal to the amount of such departure. The total permissible variation in position is maximum when the feature is at MMC. Likewise, referencing a datum feature on an LMC basis means the datum is the axis or center plane of the feature at the LMC limit. Where the actual size of the datum feature has departed from LMC, a deviation is allowed between its axis or center plane and the axis or center plane of the datum (see Table 4-3).

4.4 Limits of Size Unless otherwise specified, the limits of size of a feature prescribe the extent within which variations of geometric form, as well as size, are allowed. This control applies solely to individual features of size as defined in 2.14.

4.4.1 Individual Feature of Size (Rule #1) Where only a tolerance of size is specified, the limits of size of an individual feature prescribe the extent to which variations in its geometric form, as well as size, are allowed.

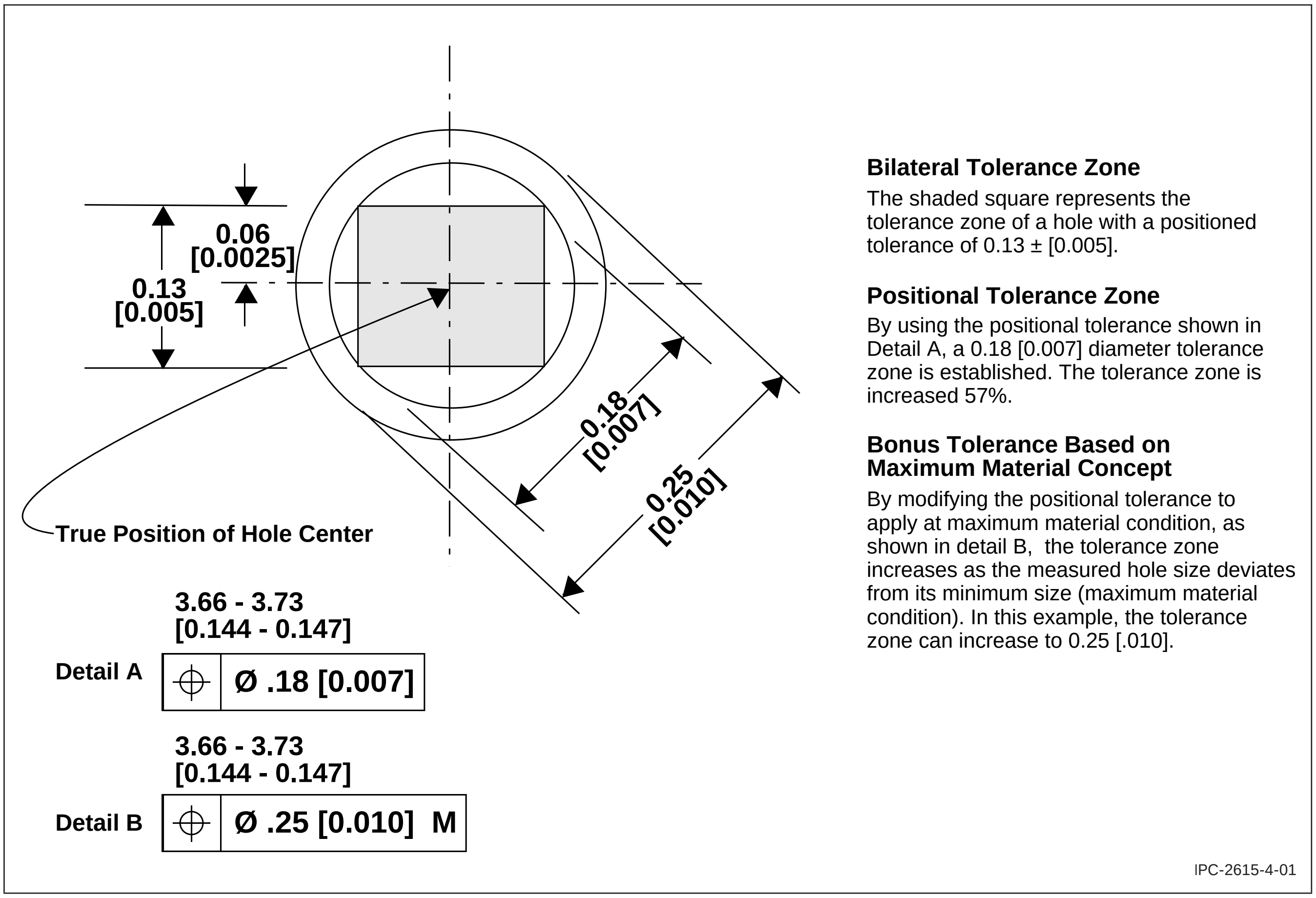


Figure 4-1 Positional Tolerancing at MMC

Ø .78 / .85 — ⌖ | Ø .01 Ⓜ

	size	tolerance
LMC	.85	.07
	.84	.06
	.83	.05
	.82	.04
	.80	.03
	.79	.02
MMC	.78	.01

IPC-2615-t4-01

Table 4-1 Maximum Material Condition Range

Ø .78 / .85 — ⌖ | Ø .01 Ⓢ

	size	tolerance
LMC	.85	.01
	.84	.01
	.83	.01
	.82	.01
	.80	.01
	.79	.01
MMC	.78	.01

IPC-2615-t4-02

Table 4-2 Regardless of Feature Size Range

4.4.1.1 Variations of Size The actual size of an individual feature at any cross section shall be within the specified tolerance of size.

4.4.1.2 Variations of Form (Envelope Principle) The form of an individual feature is controlled by its limits of size to the extent prescribed in the following paragraphs and illustrated in Figure 4-2.

a) The surface or surfaces of a feature shall not extend beyond a boundary (envelope) of perfect form at MMC. This boundary is the true geometric form

⌀ .78
.85

⊕ | ⌀ .01 Ⓛ

	size	tolerance
LMC	.85	.01
	.84	.02
	.83	.03
	.82	.04
	.80	.05
	.79	.06
MMC	.78	.07

IPC-2615-t4-03

Table 4-3 Least Material Condition Range

represented by the drawing. No variation in form is permitted if the feature is produced at its MMC limit of size.

b) Where the actual size of a feature has departed from MMC toward LMC, a variation in form is allowed equal to the amount of such departure.

c) There is no requirement for a boundary of perfect form at LMC. Thus, a feature produced at its LMC limit of size is permitted to vary from true form to the maximum variation allowed by the boundary of perfect form at MMC.

4.4.2 Relationship Between Individual Features The limits of size do not control the orientation or location relationship between individual features. Features shown perpendicular, coaxial, or symmetrical to each other must be controlled for location or orientation to avoid incomplete drawing requirements.

4.5 Applicability of MMC, RFS, and LMC Applicability of RFS, MMC, and LMC is limited to features subject to variations in size. They may be datum features or other features whose axes or center planes are controlled by geometric tolerances. In such cases, the following practices apply.

a) Tolerances of Position (Rule #2) RFS, MMC or LMC must be specified on the drawing with respect to the individual tolerance, datum reference, or both, as applicable.*

* Retained for explanation purposes on older drawings. Per ASME Y14.5M-1994, RFS is assumed unless otherwise indicated

b) All Other Geometric Tolerances (Rule #3). RFS applies, with respect to the individual tolerance, datum reference, or both, where no modifying symbol is specified. MMC must be specified on the drawing where it is required.

5 DATUM REFERENCING

5.1 General This section establishes the principle of datum referencing used to relate features of a printed board to an appropriate datum or datum reference frame. It contains the criteria for selecting, designating, and using features of a printed board as the basis for dimensional definition. A datum indicates the origin of a dimensional relationship between a toleranced feature and a designated feature or features on a printed board. The designated feature serves as a datum feature, whereas its true geometric counterpart establishes the datum.

5.1.1 Application Because a datum is theoretical, measurements cannot be made from a true geometric counterpart. A datum is assumed to exist in and be simulated by the associated processing equipment. For example, machine tables and surface plates, though not true planes, are of such quality that they are used to simulate the datums from which measurements are taken and dimensions verified. The flat surfaces of manufactured printed

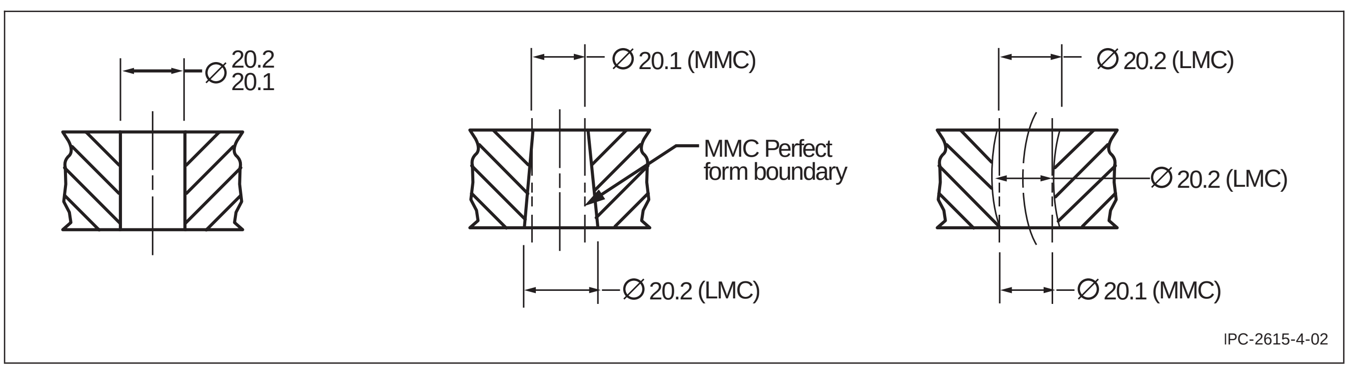


Figure 4-2 Variations of Form Allowed By Size Tolerance

board are seen to have irregularities due to both the circuitry and the variation in the laminate surface. Contact is made with a datum plane at a number of surface extremities or high points.

5.1.2 Datum Reference Frame Sufficient datum features, those most important to the design of a printed board, are chosen to position the printed board in relation to a set of three mutually perpendicular planes, jointly called a datum reference frame. This reference frame exists in theory only and not on the printed board. Therefore, it is necessary to establish a method for simulating the theoretical reference frame from the actual features of the printed board. This simulation is accomplished by positioning the printed board on appropriate datum features to adequately relate the printed board to the reference frame and to restrict motion of the printed board in relation to it (see Figure 5-1).

5.1.2.1 Plane Simulation Relationship These planes are simulated in a mutually perpendicular relationship to provide direction as well as the origin for related dimensions and measurements. Thus, when the printed board is positioned on the datum reference frame (by physical contact between each datum feature and its counterpart in the associated processing equipment), dimensions related to the datum reference frame by a feature control frame or note are thereby mutually perpendicular. This theoretical reference frame constitutes the three plane dimensioning system used for datum referencing.

5.1.2.2 Datum Options In the case of a printed board profile or a secondary datum feature of size position, a single datum reference frame will suffice. In others, additional datum reference frames may be necessary where physical separation or the functional relationship of features require that datum reference frames be applied at specific locations on the printed board. In such cases, each feature control frame must contain the datum feature references that are applicable. Any difference in the order of precedence or in the material condition of any datums referenced in multiple feature control frames requires different datum simulation methods and, consequently, establishes a different datum reference frame. See 5.3.9.

5.2 Datum Features A datum feature is selected on the basis of its geometric relationship to the toleranced feature and the requirements of the design. To ensure proper printed board interface and assembly, corresponding features of mating parts are also selected as a datum feature where practical. Datum features must be readily discernible on the printed board. A datum feature should be accessible on the printed board. For printed boards, the primary datum features will normally be the plane of the board, and typically the mounting (nonplated-through) holes used as secondary and tertiary datum features (see Figure 5-2). However, there may be occasions when the edges of the printed board, slots, or fiducials may serve as secondary and tertiary datum features (see Figures 5-3, 5-4, and 5-5, respectively). Using fiducials as datum features means that the profile is controlled in relation to the circuit pattern as opposed to the hole pattern, as is normally the case.

5.2.1 Datum Feature Symbols Datum features are identified on the drawing by means of symbols (see 3.3.2). The datum feature symbol is applied to the concerned feature surface outline, extension line, dimension line or feature control frame.

5.2.2 Datum Feature Control Measurements made from a datum plane do not take into account any variations of

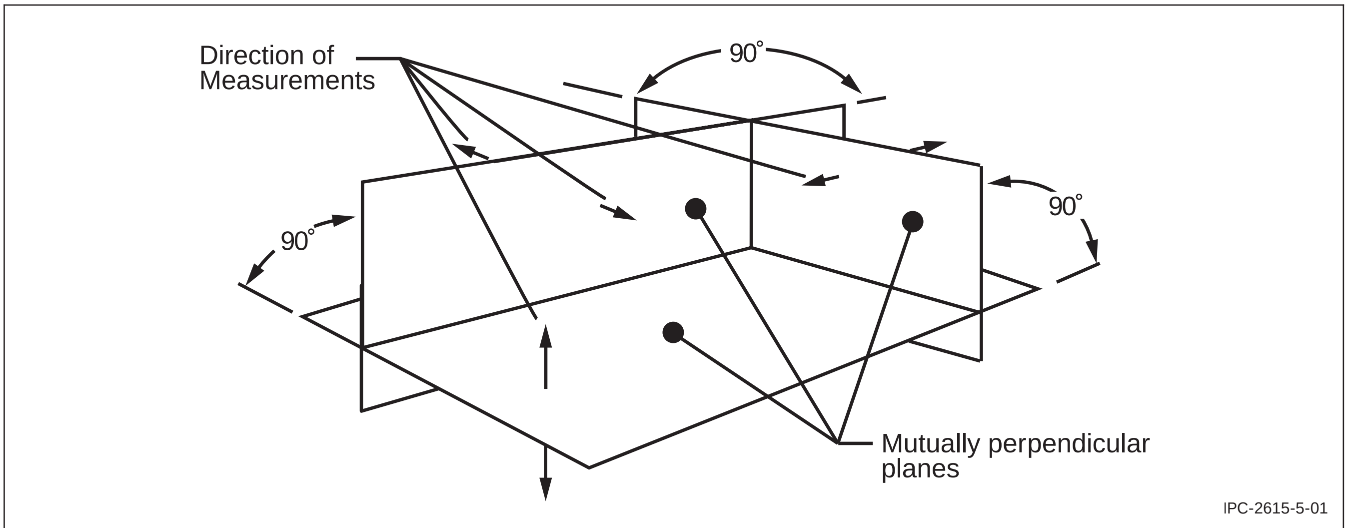


Figure 5-1 Datum Reference Frame

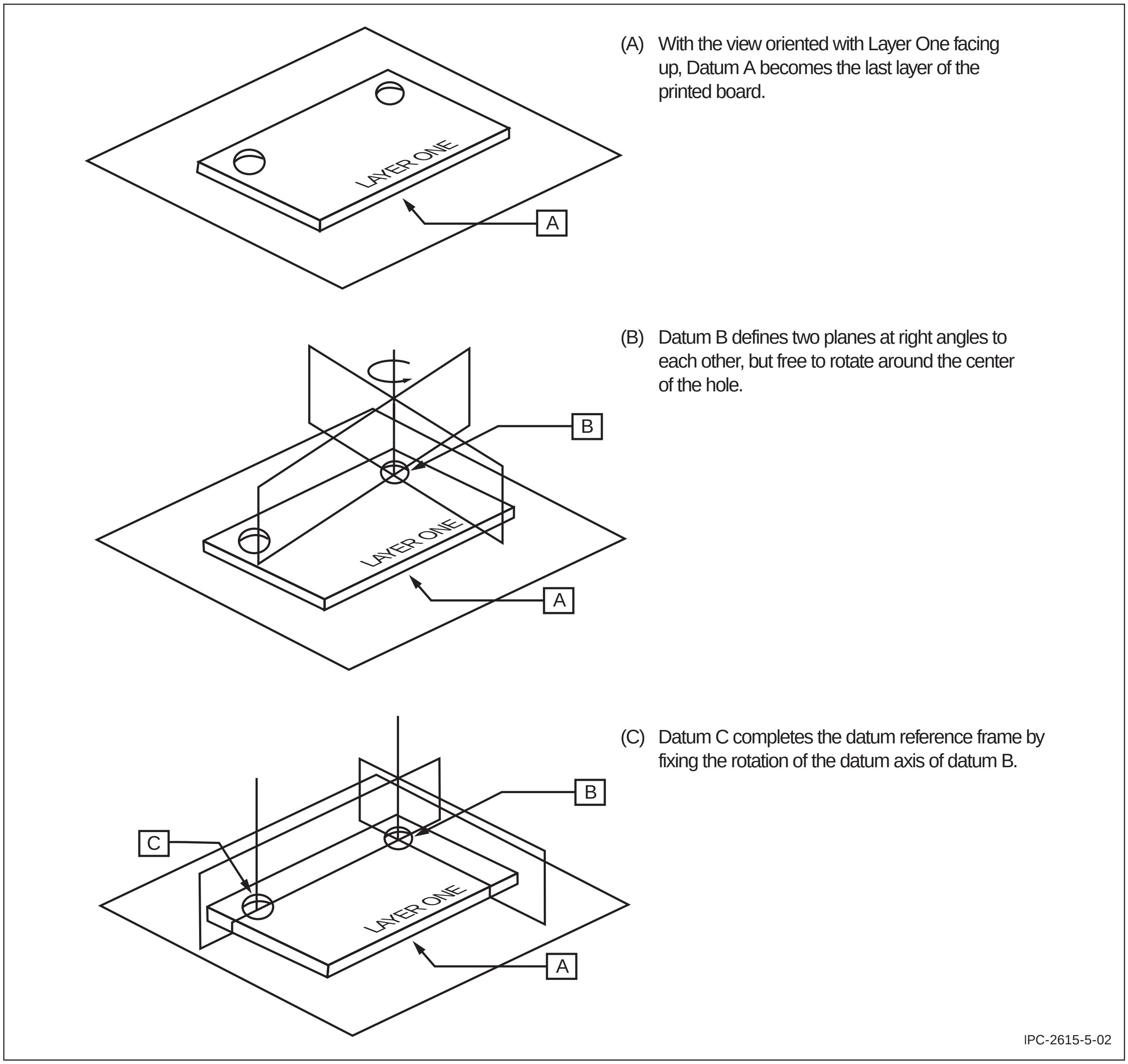


Figure 5-2 Datum Reference Frame to Printed Board Relationships

the datum surface from the datum plane. This can happen with a warped printed board lying on a surface plate. Consideration shall be given to the desired accuracy of datum features relative to design requirements and the degree of control necessary for the toleranced features related to them. If not sufficiently accurate, datum features may need to be controlled by specifying appropriate geometric tolerances. Where control of the entire feature becomes impractical, use of datum targets may be considered. Datum targets are used for assemblies only; not for unassembled printed wiring boards. See 5.4.

5.2.3 Specifying Datums in Order of Precedence To properly position a printed board on the datum reference frame, datums must be specified in an order of precedence in the feature control frame. Figure 5-6 illustrates a printed board where the datum features follow the preferred convention for noncircular printed boards. The desired order of precedence is indicated by entering the appropriate datum reference letters, from left to right, in the feature control frame. In Figure 5-6, the datum features are identified as surface A, and features (nonplated-through holes) B, and C. These features are most important to the design and function of the printed board. Features A, B, and C are the primary, secondary, and tertiary datum features, respectively, since they appear in that order in the feature control frame. Typically, the printed board is oriented with the component side or the designated Layer 1 facing up. This orientation

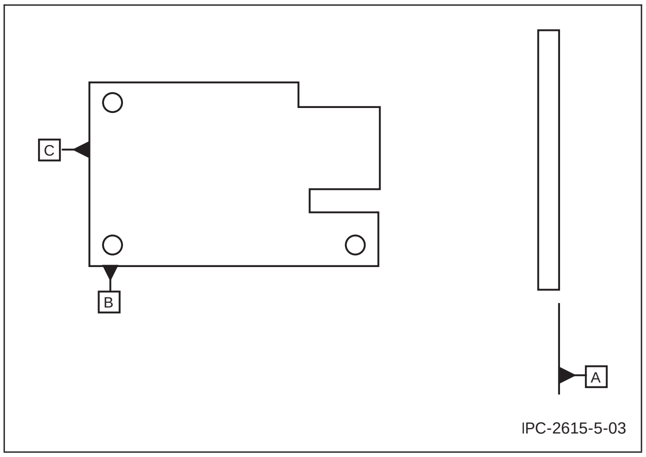


Figure 5-3 Datum Reference Using Printed Board Edges

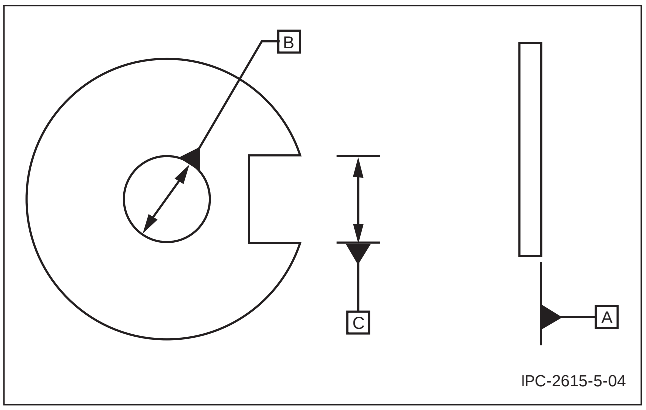


Figure 5-4 Hole and Slot Establishing Secondary and Tertiary Datums

establishes the opposite side of the printed board as the primary datum feature. The other two datum planes are established by either two holes, etched features or edges of the printed board. Coordinate zero for measurement should originate at the secondary datum feature. All datum features should either be located on grid or establish grid criterion. All datum features shall be located within the board profile or on the board profile itself.

5.2.3.1 Positioning Part on Datum Reference Frame Figure 5-2 illustrated the typical sequence for positioning the printed board shown in Figure 5-6 on a datum reference frame as simulated by the processing equipment. The primary datum feature relates the printed board to the datum reference frame by bringing a minimum of three points on the surface into contact with the first datum plane (see Figure 5-2a). The printed board is further related to the frame by bringing at least two points of the secondary datum feature into contact with the second datum plane as simulated by a pin in a drill press (see Figure 5-2b and Figure 5-7). The relationship is completed by bringing at least one point of the tertiary datum feature into contact with the third datum plane (see Figure 5-2c). As measurements are made from datum planes, positioning of the printed board on a datum reference frame in this manner ensures a common basis for measurements.

5.3 Establishing Datums The following paragraphs define the criteria for establishing datums from datum features.

5.3.1 Primary Datum Feature The primary datum feature for a printed board will usually be the plane surface of the board and may be either the first or last layer of the printed board. Most often, it will be the surface opposite Layer 1, but there may be cases where Layer 1 is an appropriate choice for the primary datum. The primary datum is simulated by a plane contacting the high points of that surface (see Figure 5-8). If irregularities on the surface of a primary or secondary datum feature are such that the printed board is unstable (that is, it wobbles) when brought into contact with the corresponding surface of a fixture, the printed board may be adjusted to an optimum position, if necessary, to simulate the datum. Unless otherwise specified on the drawing, it is understood that the contact points could be circuitry or the base laminate, depending on the flatness of the printed board.

5.3.2 Secondary and Tertiary Datum Features Not Subject to Size Variations The sequence for establishing the datum reference frame is illustrated in Figure 5-3 when the edges of a printed board are specified as secondary and tertiary datum features. The primary datum feature relates the part to the datum reference frame by bringing a minimum of three points on the surface into contact with the first datum plane. The secondary datum is established by bringing at least two points of the secondary datum feature into contact with the second datum plane. The relationship is completed by bringing at least one point of the tertiary datum into contact with the third datum plane. Measurements on the printed board are made from the datum planes. This method of establishing a datum reference frame is recommended when the edges are the most critical locating features of the printed board.

5.3.3 Secondary and Tertiary Datum Features Subject to Size Variations Datum features such as diameters and widths (e.g., slots) are subject to variations in size as well as form. Because variations are allowed by the size dimension, it becomes necessary to determine whether RFS or MMC applies in each case (see 4.1, 4.2, and 4.3). For a tolerance of position, the datum reference letter is always followed by the appropriate modifying symbol in the feature control frame. For all geometric tolerances, RFS is implied unless otherwise specified.

5.3.4 Specifying Datum Features RFS Where a datum feature of size is applied on an RFS basis, the datum is established by physical contact between the feature surface

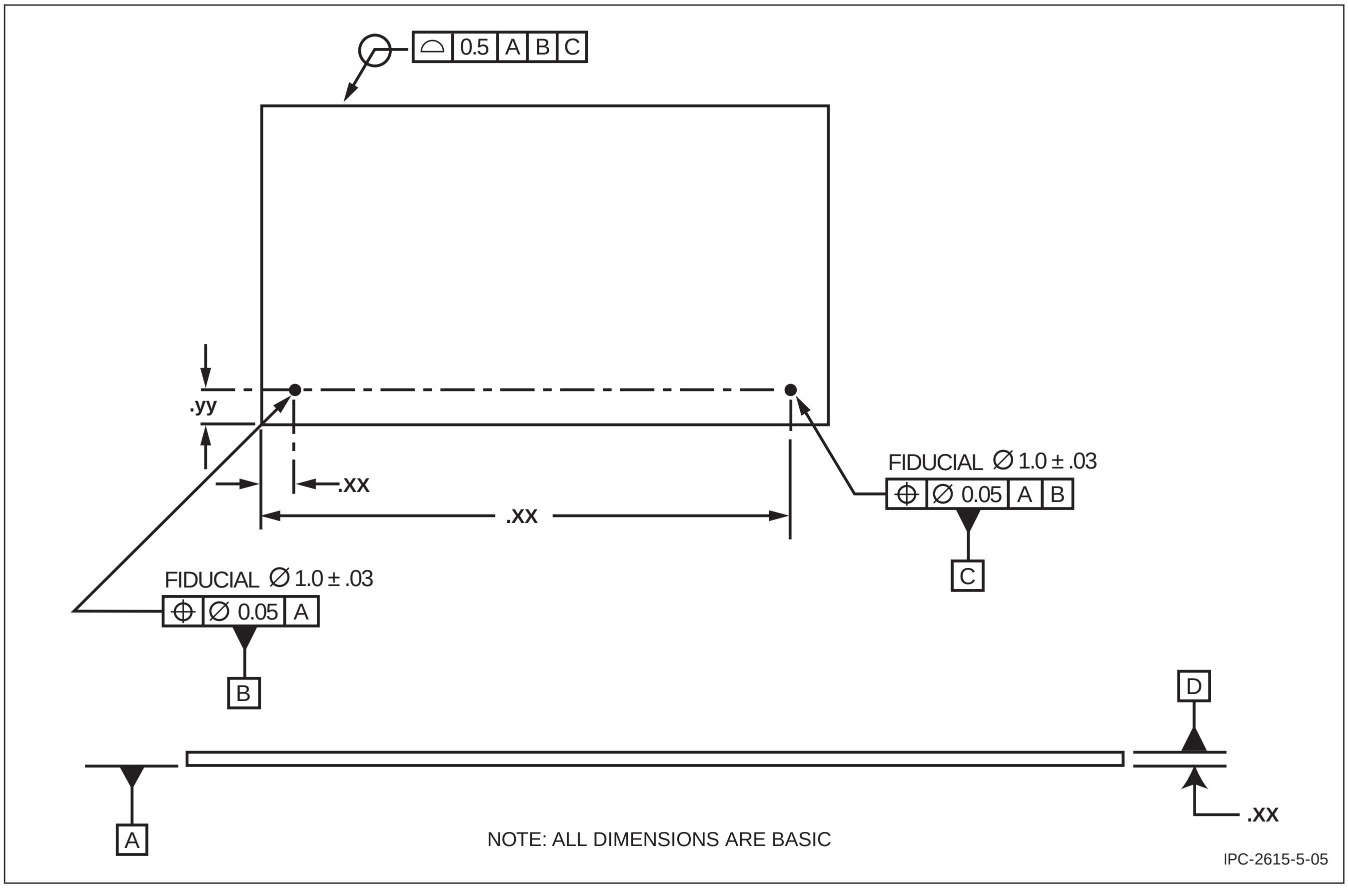


Figure 5-5 Additional Datum Example

or surfaces and surfaces of the processing or inspection equipment. Machine elements which are variable in size (such as tooling pins, surface plates, or guide rails) are used to simulate a true geometric counterpart of the feature and to establish the datum.

a) *Secondary Datum Feature – Diameter* For an external feature, the secondary datum (axis or center plane) is established by the axis of the smallest circumscribed, perfect cylinder which contacts the profile surface of the printed board. For an internal feature (a hole), the datum is the axis of the largest inscribed, perfect cylinder which contacts the hole surface. The contacting cylinder must be oriented perpendicular to the primary datum. Datum B in Figure 5-7 illustrates this principle.

b) *Secondary Datum Feature – Width* For an external feature, the secondary datum (axis or center plane) is established by the center plane between two parallel planes which, at minimum separation, contact the corresponding surfaces of the feature. For an internal feature (a slot), the datum is the center plane between two parallel planes which at maximum separation contact the corresponding surfaces of the slot. The contacting parallel planes must be oriented perpendicular to the primary datum.

5.3.5 Specifying Datum Features at MMC Where a datum feature of size is applied on an MMC basis, machine and gauging elements in the processing equipment, which remain constant in size, may be used to simulate a true geometric counterpart of the feature and to establish the datum. In each case, the size of the simulated datum is determined by the specified MMC limit of size of the datum feature or its virtual condition, where applicable.

5.3.5.1 Datum Feature Size Control Where a datum feature of size is controlled by a specified tolerance of form (profile), the size of the simulated datum is the MMC limit of size.

5.3.5.2 Secondary or Tertiary Datum Features of Size Where secondary or tertiary datum features of size in the same datum reference frame are controlled by a specified tolerance of location or orientation with respect to each other, the size of the simulated datum is the virtual condition of the datum feature (see Figure 5-9). This example illustrates both secondary and tertiary datum features specified at MMC but simulated at virtual condition.

5.3.6 Cylindrical Datum Features Cylindrical datum features can be holes or the profile of a circular shaped printed board. A cylindrical datum feature is always associated with two theoretical planes intersecting at right

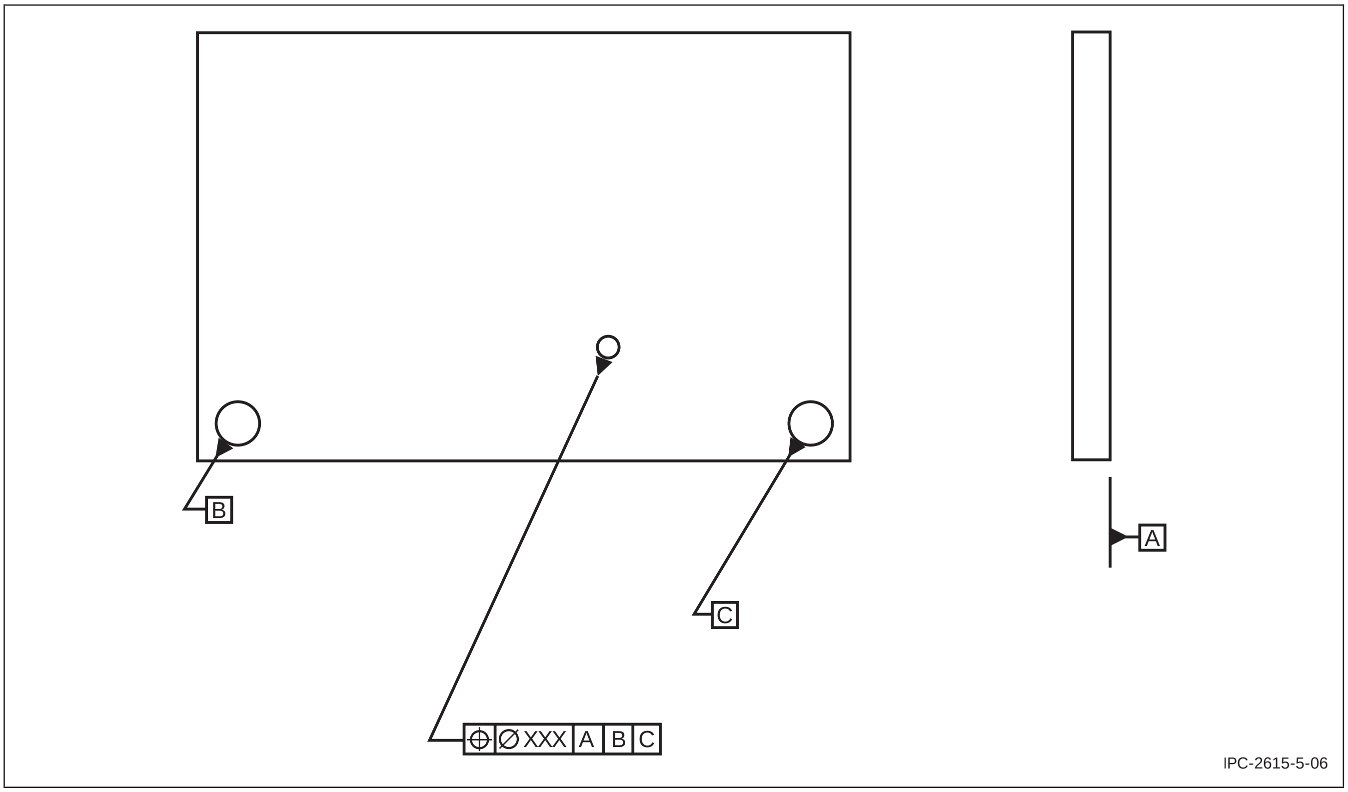


Figure 5-6 Datum Feature Identification and Reference

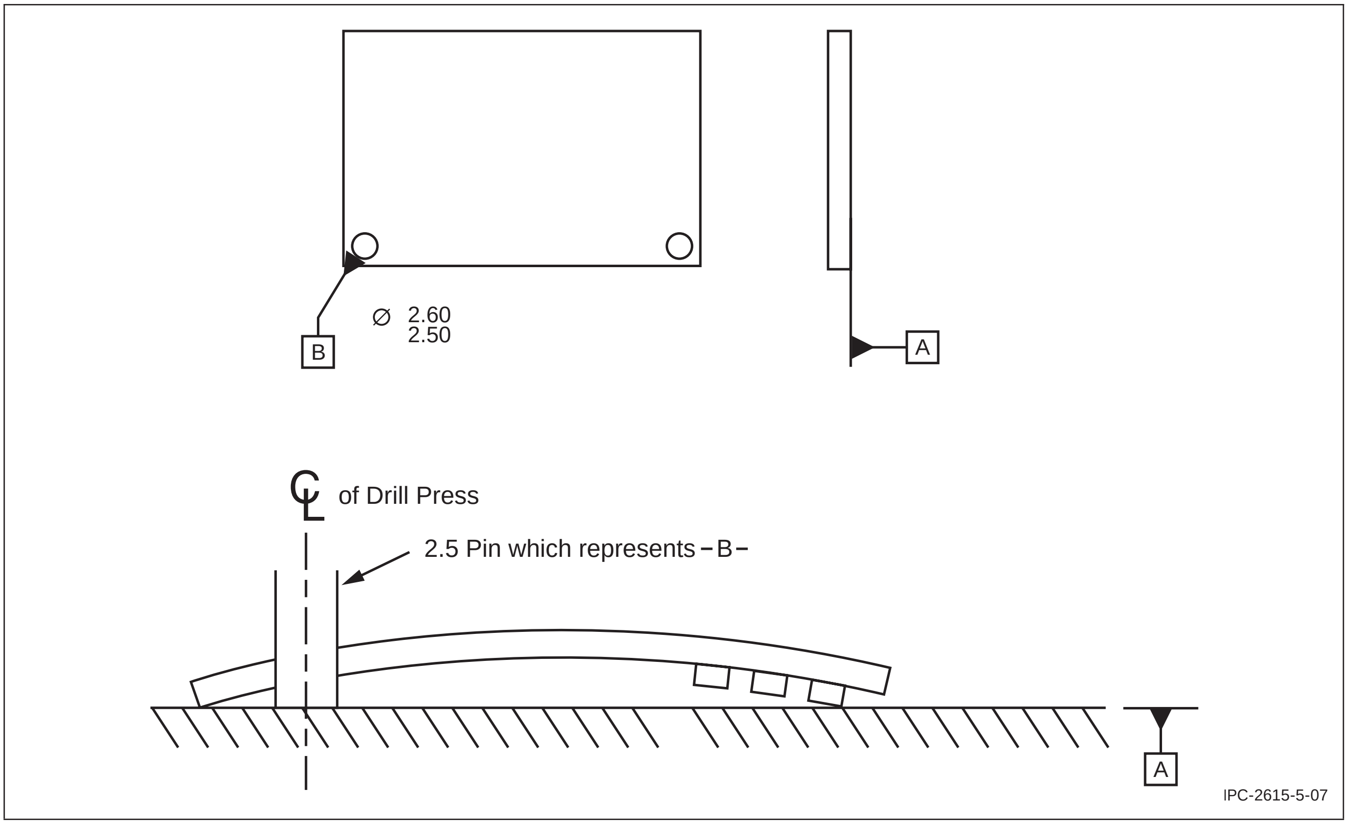


Figure 5-7 Secondary Datum Established By Internal Feature

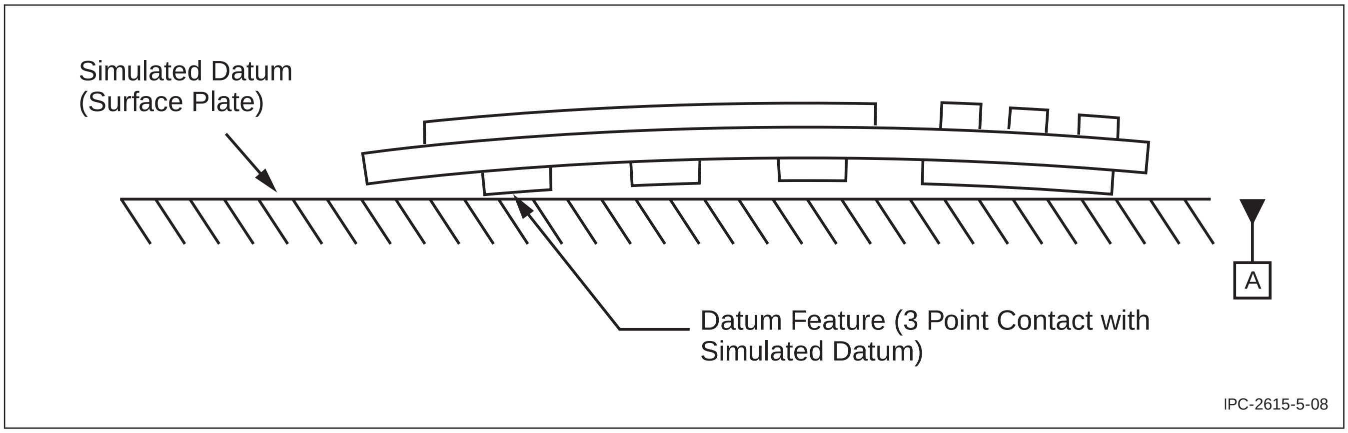


Figure 5-8 Datum Feature and Simulated Datum

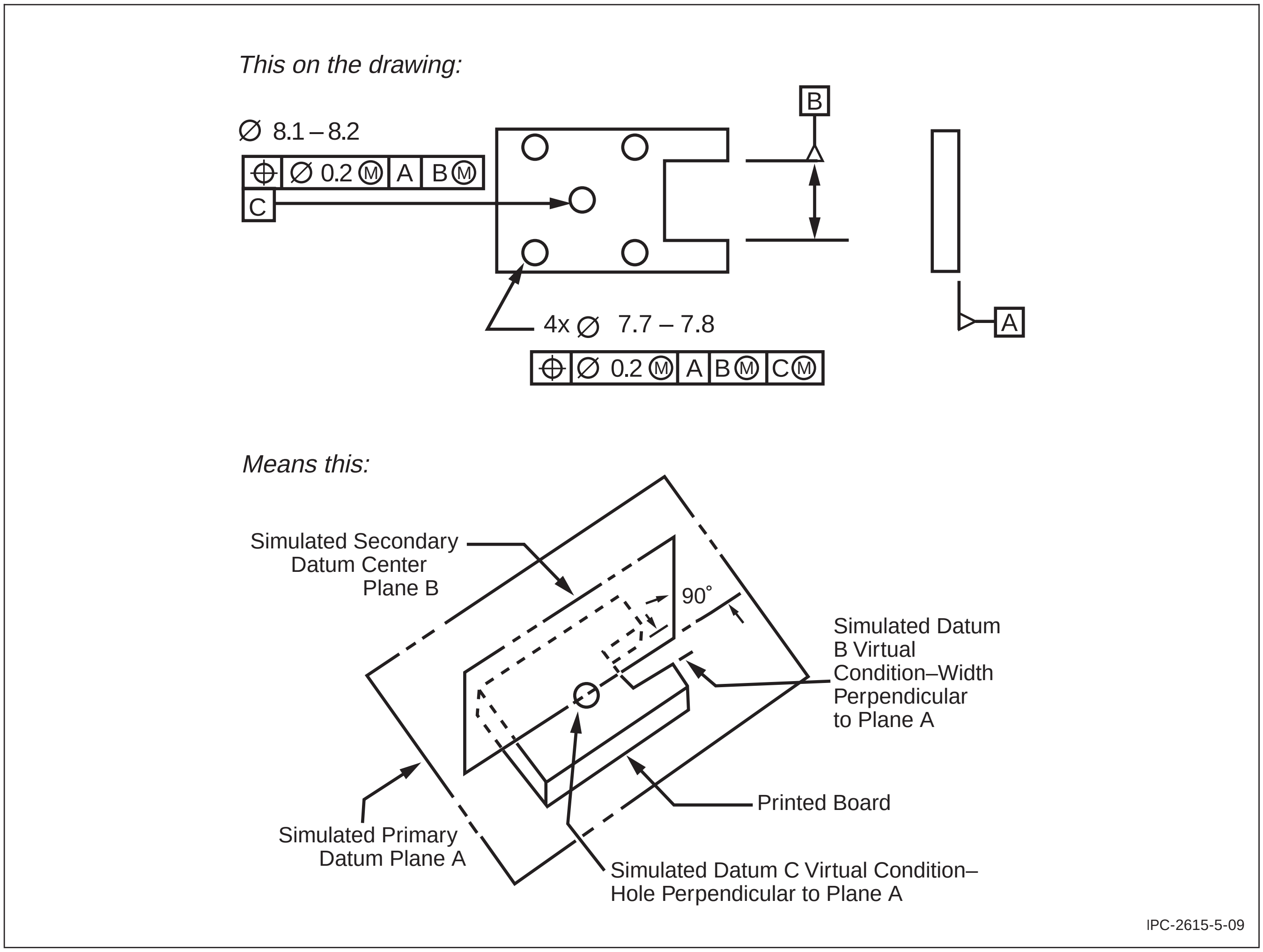


Figure 5-9 Virtual Condition of Datum Feature

angles on the datum axis. These planes indicate the direction of measurements made from the datum axis. The datum established by a cylindrical surface is the axis of a true cylinder simulated by the processing equipment (see 5.3.4 and 5.3.5). Since most printed boards use tooling holes for mounting or positioning, the use of cylindrical datum features, that is, holes, for the secondary and tertiary datum is the common method for establishing the datum reference frame.

5.3.6.1 Cylindrical External Datum Features Figure 5-10 illustrates a printed board having an external cylindrical datum feature. Primary datum feature K relates the printed board to the first datum plane. Since secondary datum feature M is cylindrical, it is associated with two theoretical planes – the second and third in a three-plane relationship. These two theoretical planes are represented on a drawing by centerlines crossing at right angles, as in Figure 5-10, part (a). The intersection of these planes coincides with the datum axis (see Figure 5-10(b)). Once established, the datum axis becomes the origin for related dimensions while the two planes (X and Y) indicate the direction of measurements. In this example, angular orientation of planes X and Y is immaterial, as rotation of the pattern of holes about the datum axis has no effect on the function of the printed board. In such cases, only two datum features are referenced in the feature control frame:

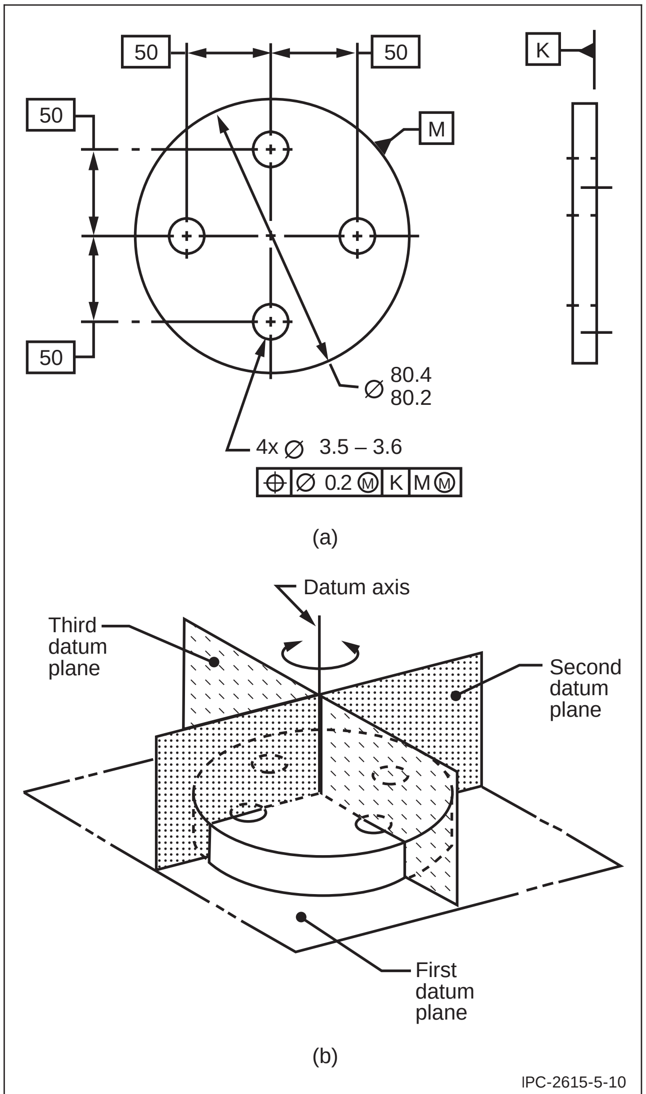


Figure 5-10 Part With Cylindrical Datum Features (a) primary datum feature K, which establishes a datum plane; and (b) secondary datum feature M, which establishes a datum axis.

5.3.6.2 Cylindrical Internal Datum Features Figure 5-11 illustrates a hole used as an internal cylindrical datum feature. It is associated with two theoretical planes – the second and third in a three-plane relationship. The intersection of these planes coincides with the datum axis. Once established, the datum axis becomes the origin for related dimensions while the two planes (X and Y) indicate the direction of measurements.

5.3.7 Angular Orientation When using a circular datum feature to establish the secondary datum plane, a tertiary datum feature must be specified to establish angular orientation. To establish angular orientation of two planes about the datum axis, a third or tertiary datum feature is referenced in the feature control frame. The feature can be a hole, or a slot (see Figures 5-4 and 5-9). In this example, angular orientation of the two planes intersecting through B is established by the center plane of slot C, the tertiary datum feature. Figure 5-12 illustrates the development of the theoretical datum reference frame for the positional tolerance applied in Figure 5-4.

5.3.8 Pattern of Features to Establish a Secondary Datum Multiple features of size, such as a pattern of holes at MMC, may be used as a group to establish a datum when part function dictates (see Figure 5-13). In this case, individual datum axes are established at the true position of each hole. These are the axes of true cylinders which simulate the virtual condition of the holes. When the printed board is mounted on the primary datum surface and rotated about the centroid of the secondary pattern of holes, a central datum axis (axis of rotation) is generated for establishing the datum reference frame.

5.3.9 Multiple Datum Reference Frames More than one datum reference frame may be established for certain printed boards, depending upon functional requirements. In Figure 5-14, datums A, B, and C constitute one datum reference frame, while datums A, D, and E constitute a second datum reference frame.

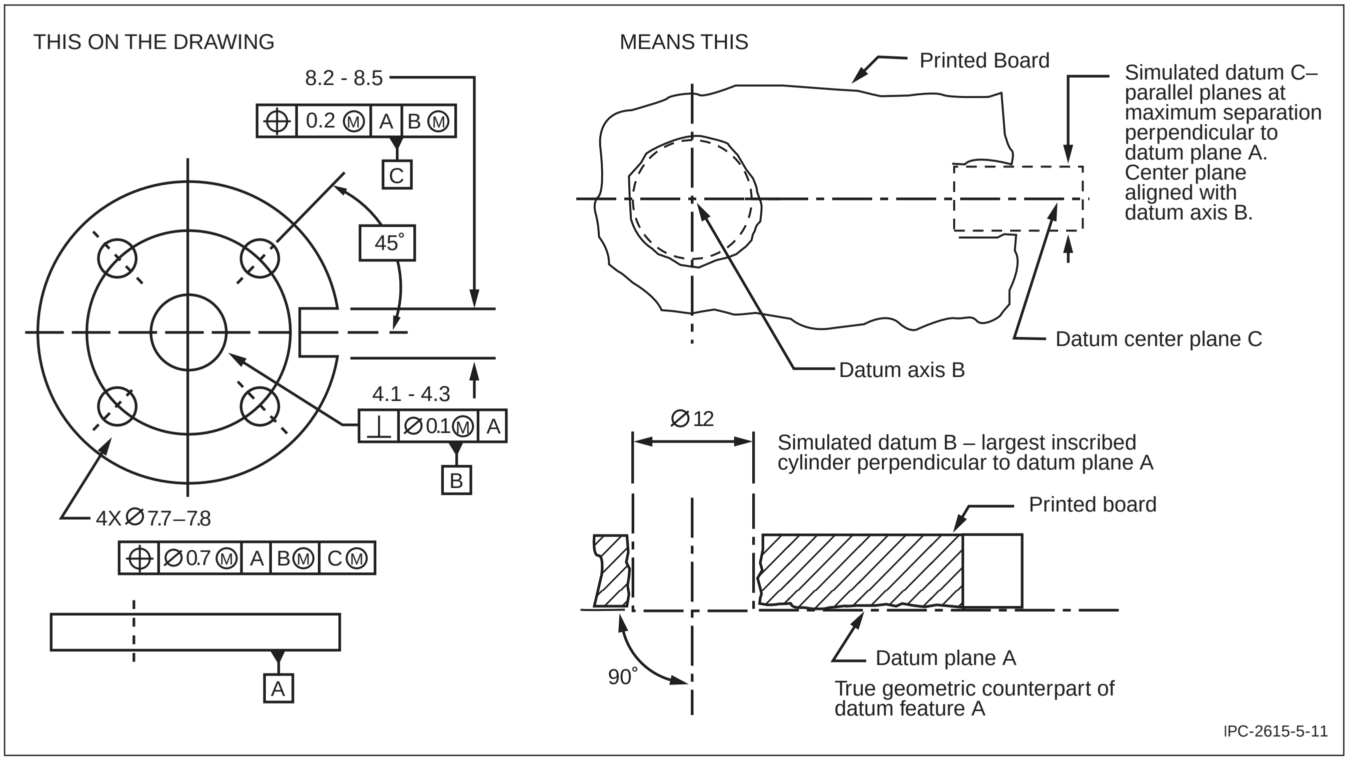


Figure 5-11 Cylindrical Internal Datum Features

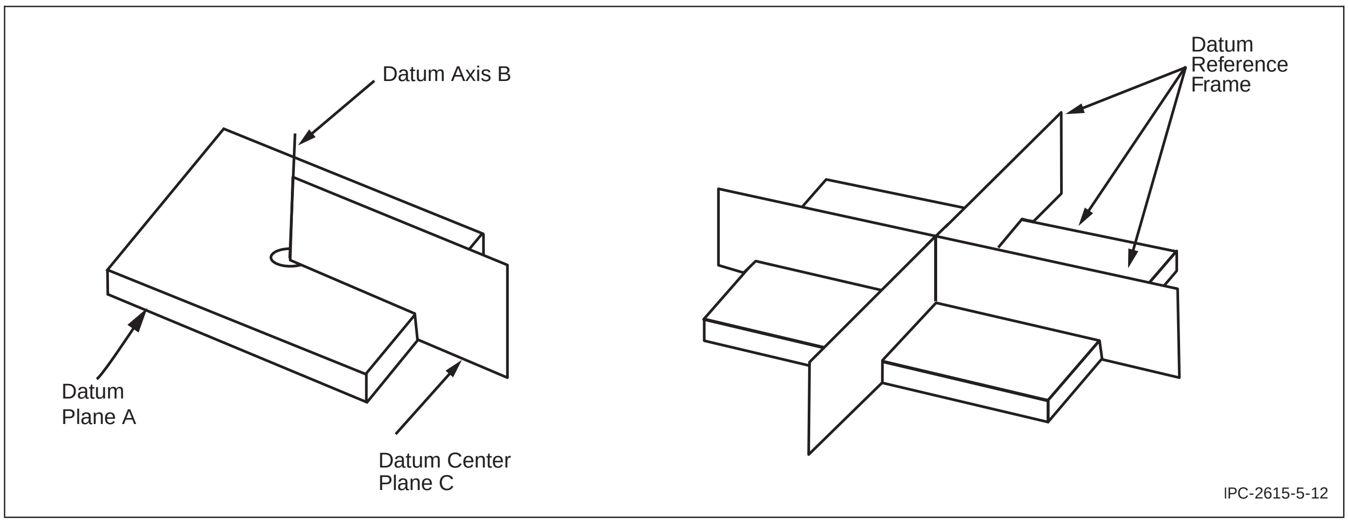


Figure 5-12 Development of a Datum Reference Frame

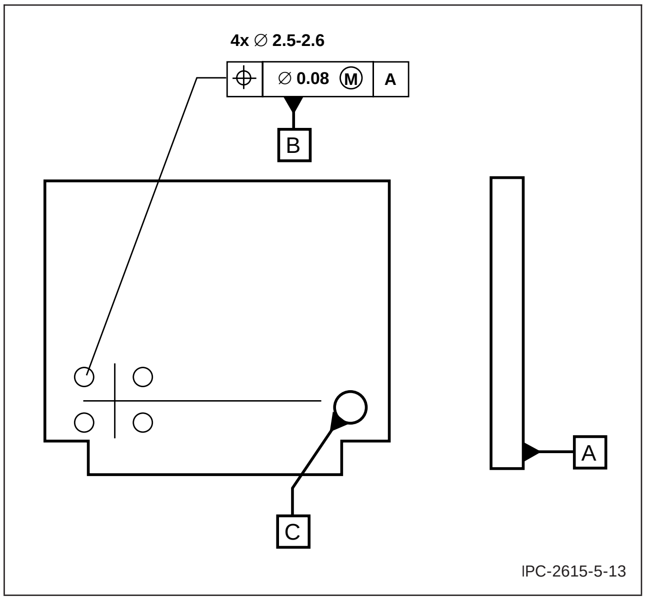


Figure 5-13 Pattern of Feature to Establish Secondary Datum

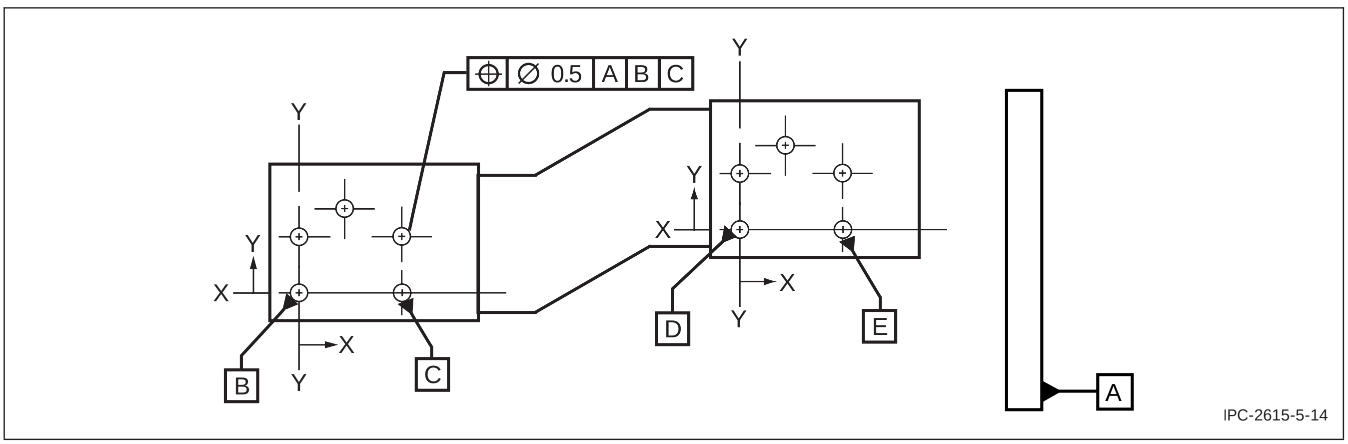


Figure 5-14 Multiple Datum Reference Conditions (Rigid/Flex) Examples

5.3.9.1 Referencing Datums in Feature Control Frames Only the required number of datums should be referenced in a feature control frame when specifying geometric tolerances. An understanding of the geometric control provided by these tolerances is necessary to effectively determine the number of datum references required for a given application. Additionally, functional requirements of the design should be the basis for selecting the related features to be referenced as datums. Figure 5-15 illustrates a printed board where three geometric tolerances are specified, each having the required number of datum references. Although common datum identifying letters appear in each frame, each combination is a different and independent requirement.

5.4 Datum Targets Datum targets designate specific points, lines, or areas of contact on a printed board that are used in establishing a datum reference frame. Because of inherent irregularities, the entire surface of some features cannot be effectively used to establish a datum. Examples are nonplanar or uneven surfaces produced by molding and thin section surfaces subject to bowing, warping, or other inherent or induced distortions. The use of datum targets can be beneficial when the bottom layer of an assembled printed board is specified as a primary datum; the existence of solder-side components and leads from the through-hole components elevating the printed board to an uneven plane, it is difficult to determine the datum plane for inspection purposes. Datum targets simplify locating the datum plane. See 5.4.1 through 5.4.2 for additional information.

5.4.1 Datum Target Symbols Points, lines, and areas on datum features are designated on the drawing by means of a datum target symbol (see Figure 5-16). The symbol is placed outside the printed board outline with a radial

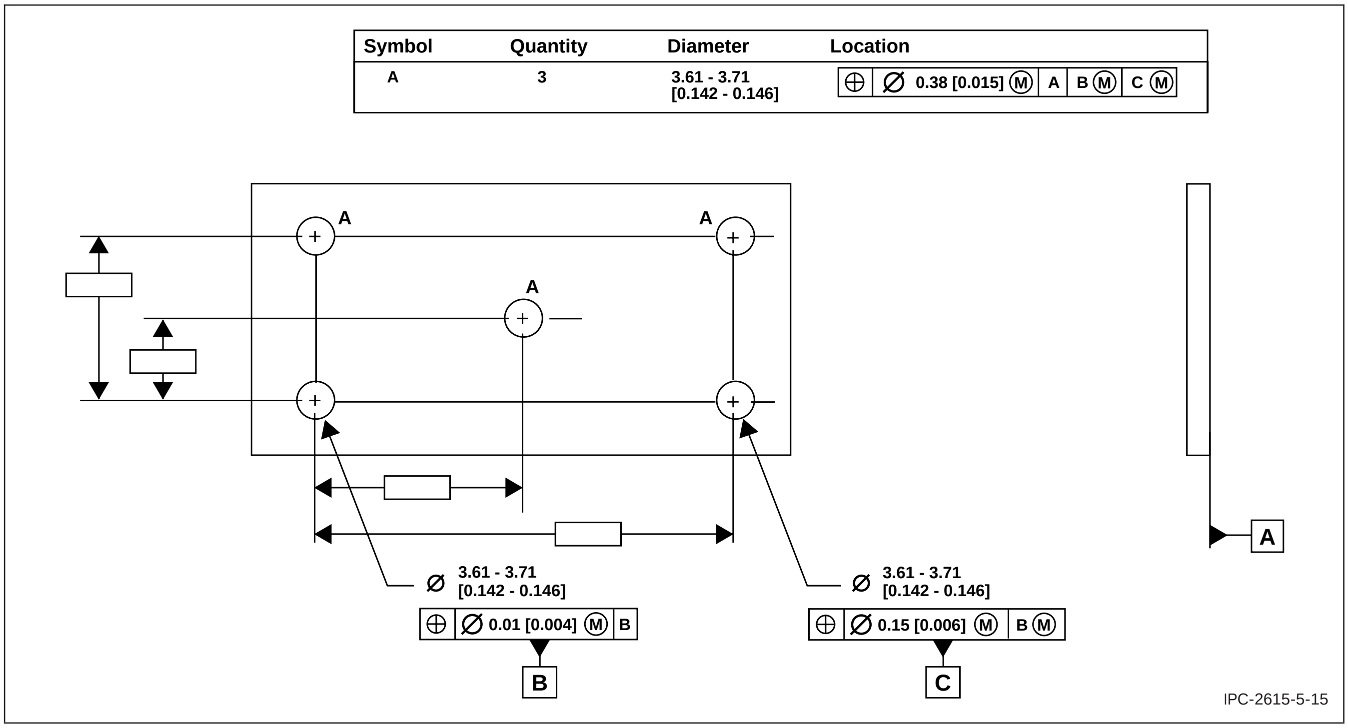


Figure 5-15 Referencing Datums in Feature Control Frames

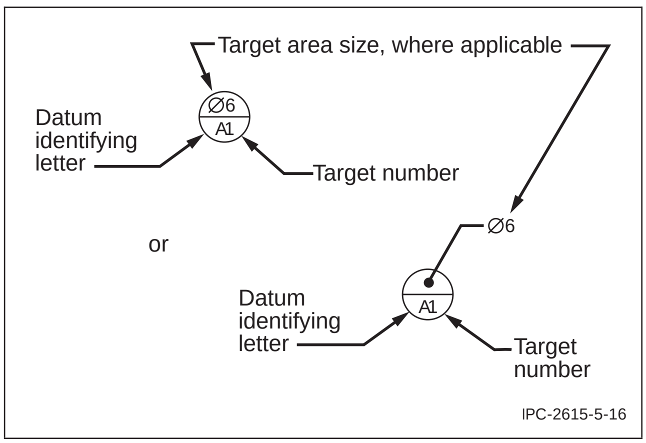


Figure 5-16 Datum Target Symbol

(leader) line directed to the target. The use of a solid radial (leader) line indicates that the datum target is on the near (visible) surface. The use of a dashed radial (leader) line indicates that the datum target is on the far (hidden) surface. The datum feature itself is identified in the usual manner with a datum feature symbol.

5.4.1.1 Datum Target Points A datum target point is indicated by the symbol ‘‘X,’’ which is dimensionally located on a direct view of the surface. Where there is no direct view, the point location is dimensioned on two adjacent views (see Figure 5-17).

5.4.2 Datum Target Dimensions The location and size, where applicable, of datum targets are defined with either basic or toleranced dimensions. If defined with basic dimensions, tooling or gaging tolerances are assumed to apply. Figure 5-18 illustrates a printed board where datum target points are located by means of basic dimensions. In this example, three mutually perpendicular planes areestablished by three target points on the primary datum feature, two on the secondary, and one on the tertiary.

5.4.3 Datum Planes A primary datum plane is established by at least three target points or areas not on a straight line (see Figure 5-19). Secondary and tertiary datum planes are usually established by two and one target points or areas, respectively.

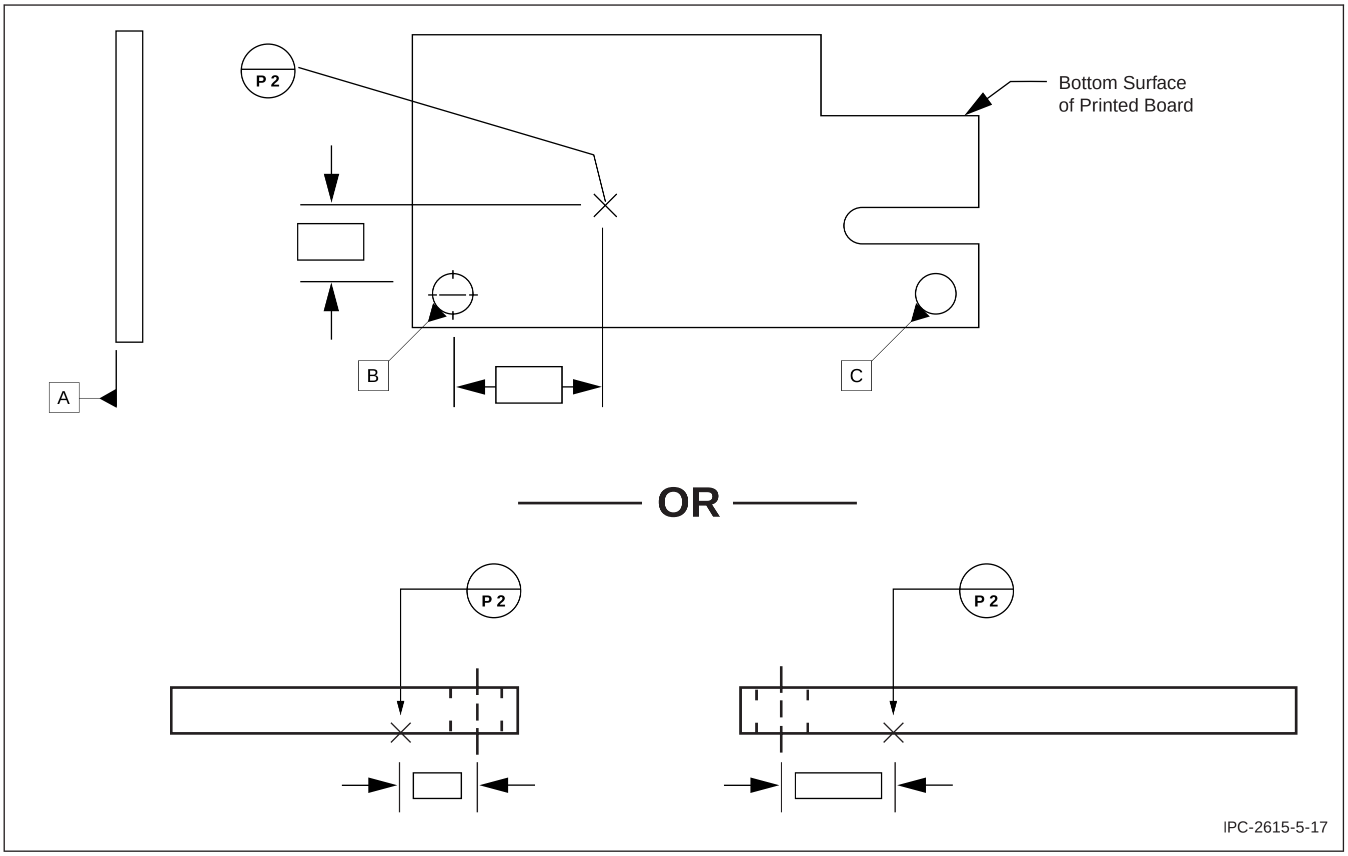


Figure 5-17 Datum Target Point

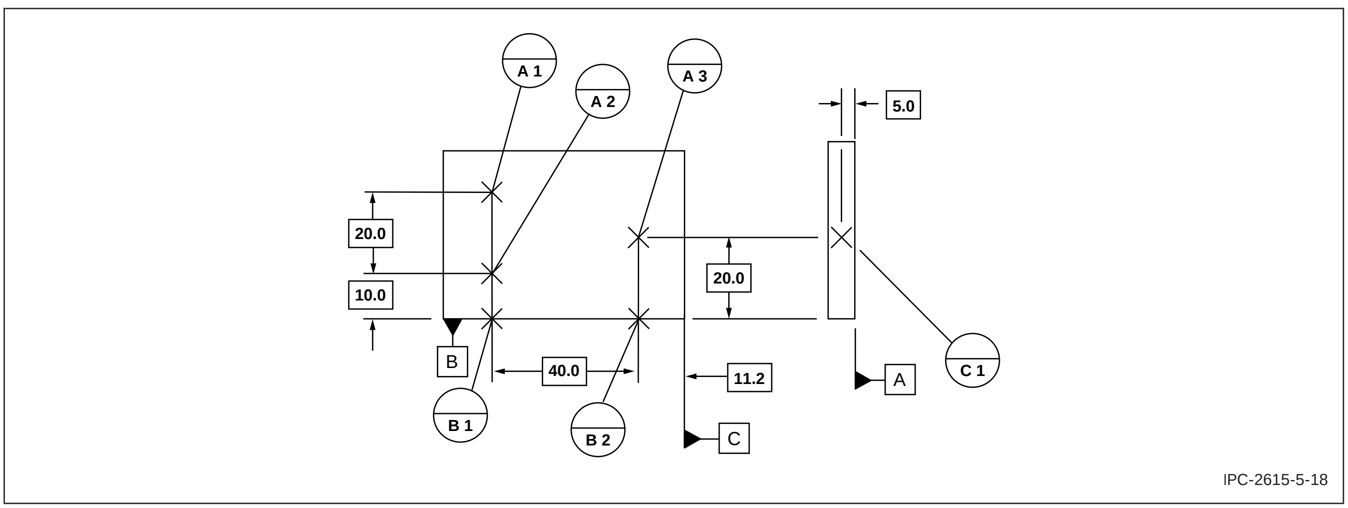


Figure 5-18 Dimensioning Datum Targets

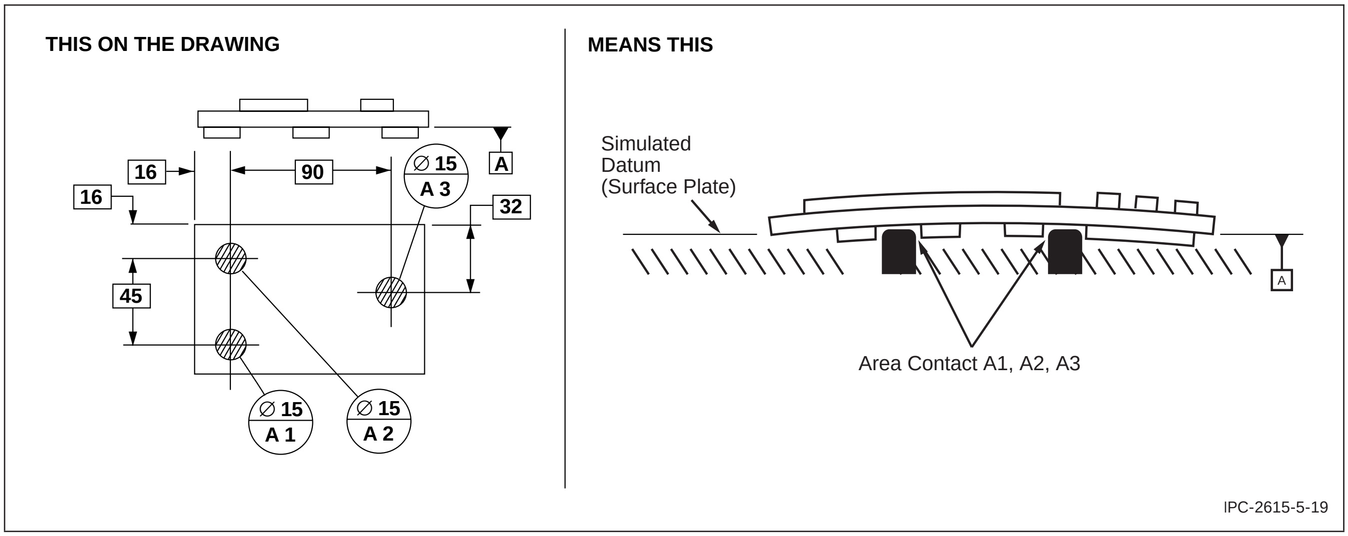


Figure 5-19 Primary Datum Plane Established

6 TOLERANCES OF LOCATION

6.1 General

This Section establishes the principles of tolerances of location used to control the following relationships:

a) center distance between such features as holes, slots, and tabs;

b) location of features as a group from datum features, such as plane;

c) features with center distances equally disposed about a datum axis or plane.

6.2 Positional Tolerancing A positional tolerance defines a zone within which the center, axis, or center plane of a feature of size is permitted to vary from true (theoretically exact) position. Basic dimensions establish the true position from specified datum features and between interrelated features. A positional tolerance is indicated by the position symbol, a tolerance, and appropriate datum references placed in a feature control frame.

6.2.1 Feature Locations Given by Basic Dimensions The location of each feature (hole, slot, etc.) is given by basic dimensions. Many drawings are based on a schedule of general tolerances, usually provided near the drawing title block. See ANSI Y14.1. Dimensions locating true position must be excluded from the general tolerance in one of the following ways:

a) applying the basic dimension symbol to each of the basic dimensions [see Figure 6-1, parts (a) and (b)];

b) specifying on the drawing (or in a document referenced on the drawing) the general note: UNTOLERANCED DIMENSIONS LOCATING TRUE POSITION ARE BASIC [see Figure 6-1), part (c)]

6.2.2 Feature Control Frame A feature control frame is added to the note used to specify the size and number of features (see Figures 6-2 and 6-3). These figures show different types of feature pattern dimensioning.

6.2.3 Establish Datums for Dimensions Locating True Positions It is necessary to identify features on printed board to establish datums for dimensions locating true positions. On printed boards, this is generally accomplished by using nonplated-through holes as in Figures 3-2 and 5-2. The intended datum features are identified with datum feature symbols and the applicable datum references are included in the feature control frame. For information on specifying datums in an order of precedence, see 5.2.3. see Figure 6-4

6.2.4 Application to Base Line and Chain Dimensioning True position dimensioning can be applied as base line dimensioning or as chain dimensioning. For positional tolerancing, unlike plus and minus tolerancing as shown in Figure 6-5, basic dimensions are used to establish the true position of features. Assuming identical positional tolerances are specified, the resultant tolerance between any two holes will be the same for chain dimensioning as for base line dimensioning. This also applies to angular dimensions, whether base line or chain type.

6.3 Fundamental Explanation of Positional Tolerancing

6.3.1 Material Condition Basis Positional tolerancing is applied on an MMC, RFS, or LMC basis. The appropriate symbol follows the specified tolerance and applicable datum reference in the feature control frame (see 4.5).

6.3.2 MMC as Related to Positional Tolerancing The positional tolerance and maximum material condition of mating features are considered in relation to each other.

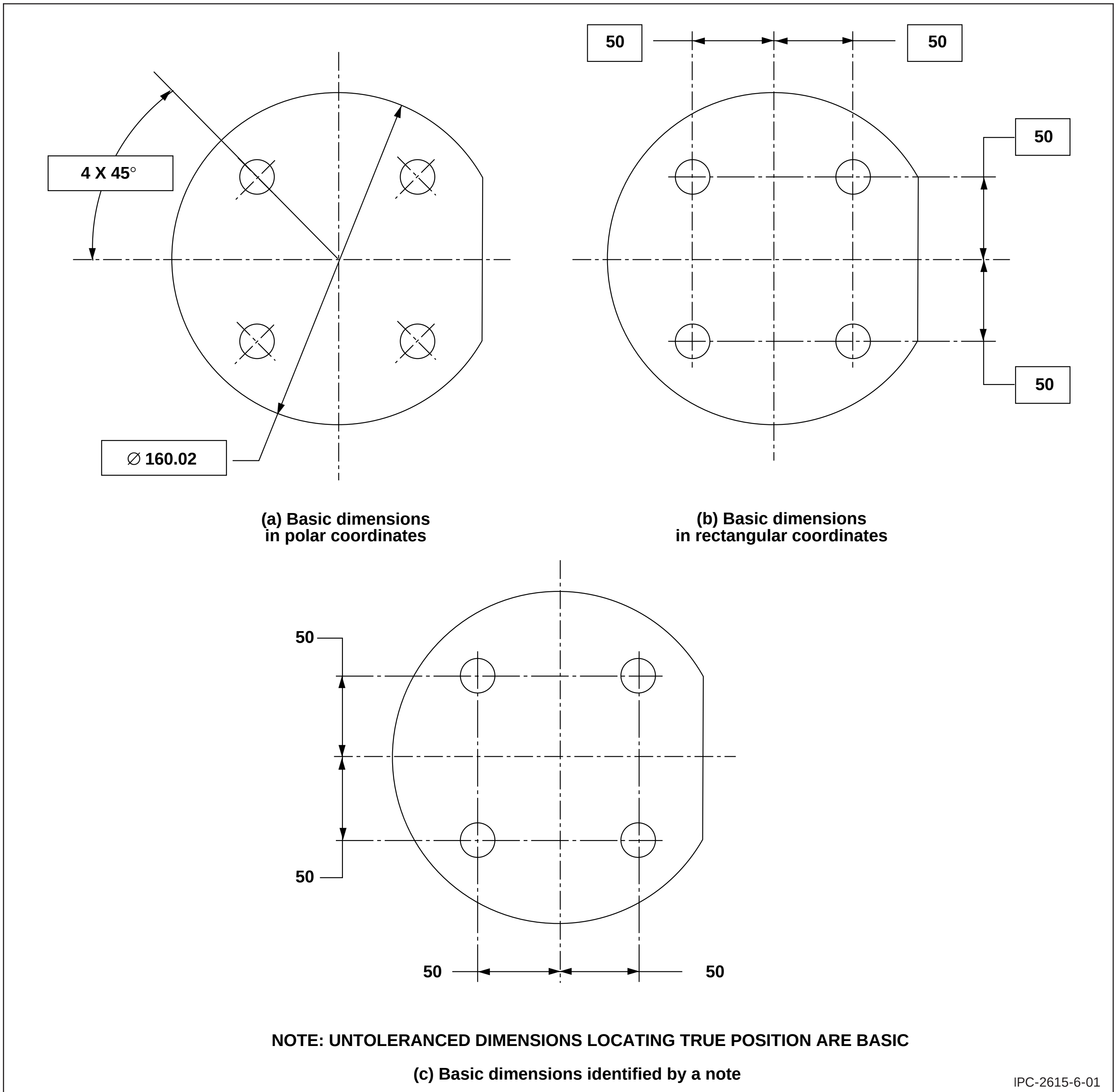


Figure 6-1 Identifying Basic Dimensions

MMC by itself means a feature of a finished product contains the maximum amount of material permitted by the toleranced size dimension for that feature. Thus, for holes, slots, and other internal features, maximum material is the condition where these features are at their minimum allowable sizes. For lands, surface mount features, tabs, and other external features, maximum material is the condition where these features are at their maximum allowable sizes.

6.3.2.1 Positional Tolerance Applied at MMC A positional tolerance applied at MMC may be explained in either of the following ways. (a) In Terms of the Surface of a Hole. While maintaining the specified size limits of the hole, no element of the hole surface shall be inside a theoretical boundary located at true position (see Figure 6-6). (b) In Terms of the Axis of a Hole. Where a hole is at MMC (minimum diameter), its axis must fall within a cylindrical tolerance zone whose axis is located at true position. The diameter of this zone is equal to the positional tolerance as shown in Figure 6-7, parts (a) and (b). This tolerance zone also defines the limits of variation in the attitude of the axis of the hole in relation to the datum surface (see Figure 6-7 (c)). It is only when the feature is at MMC that the specified positional tolerance applies.

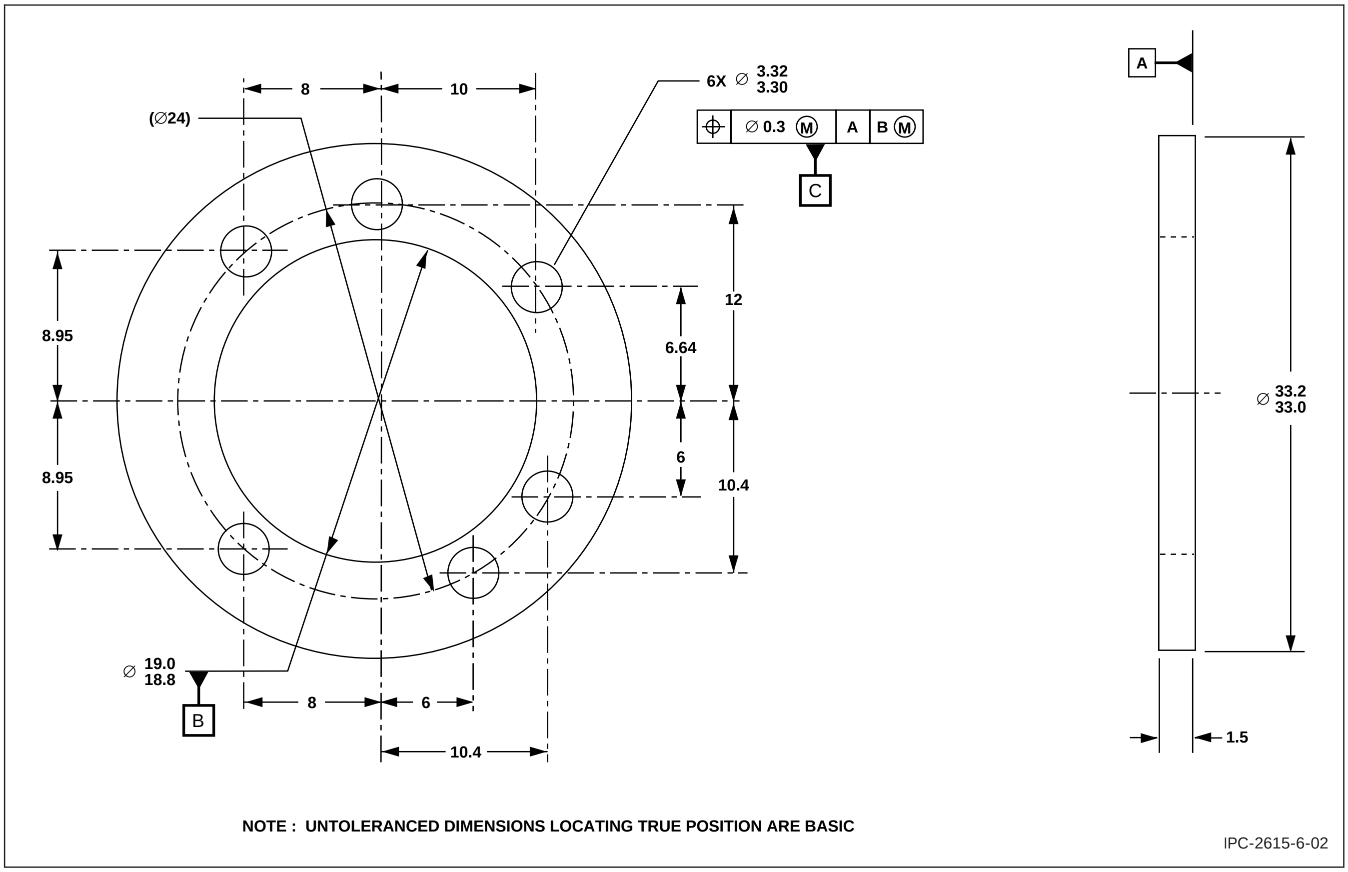


Figure 6-2 Positional Tolerances With Datum Reference

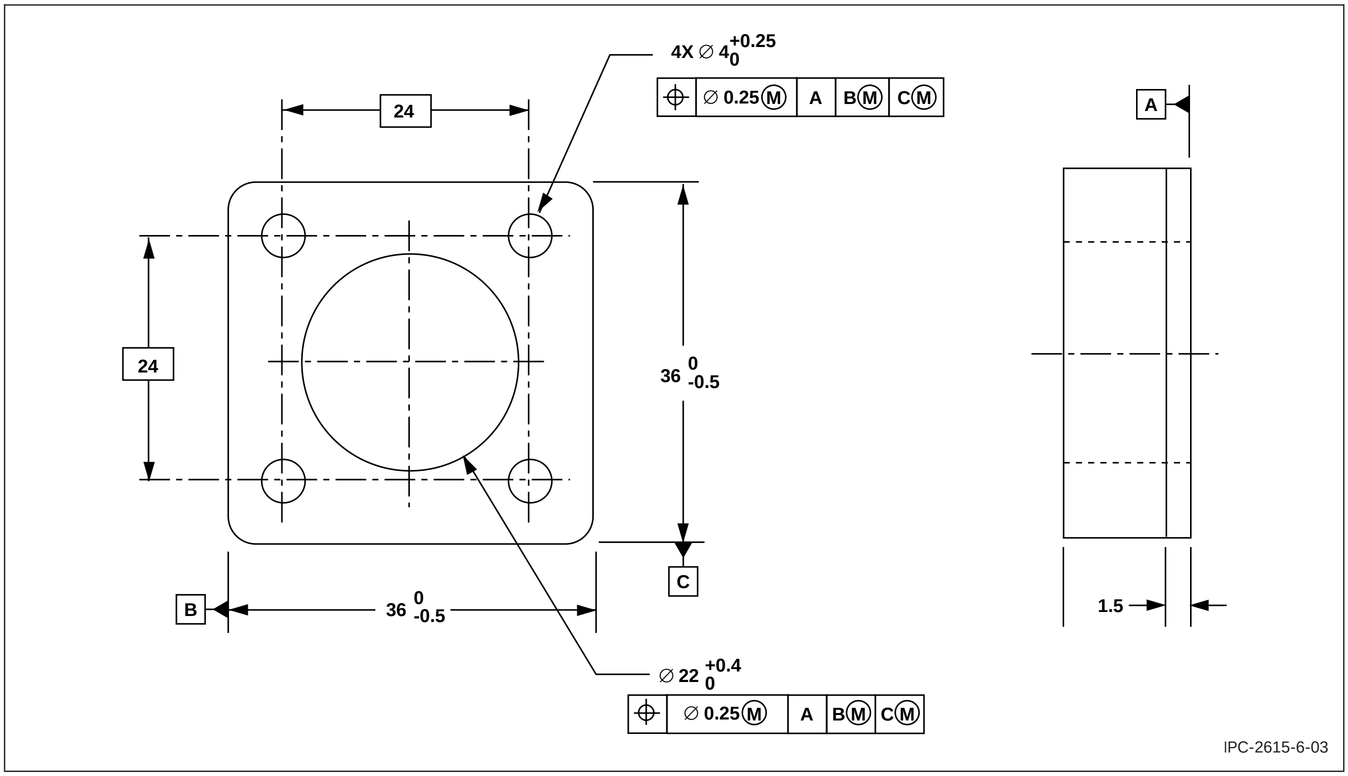


Figure 6-3 Positional Tolerancing

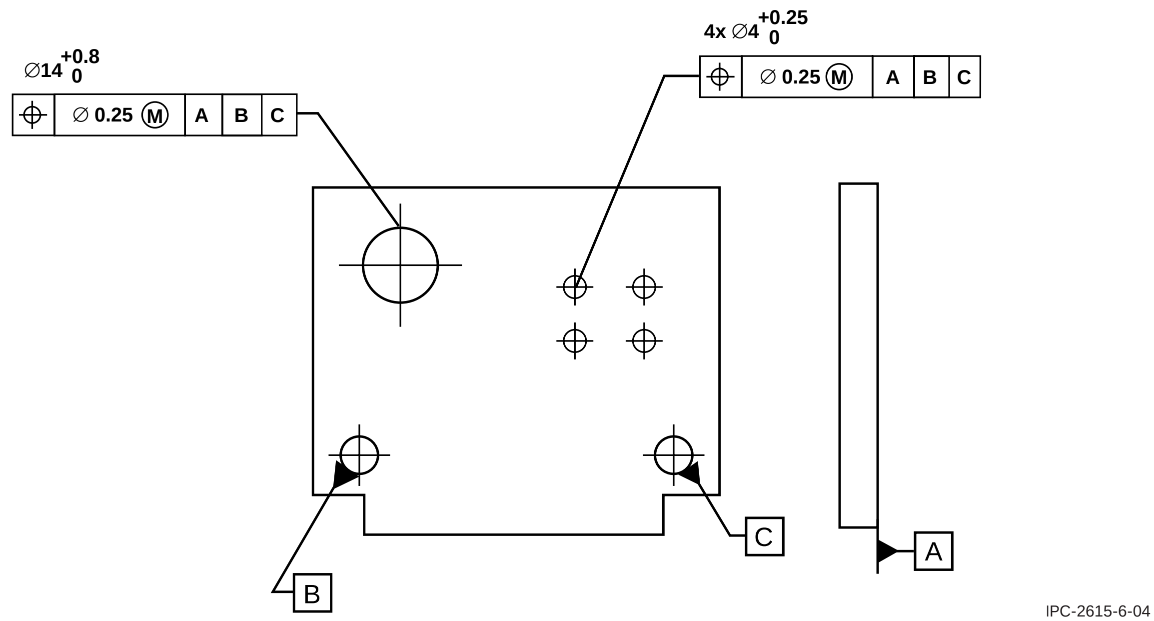


Figure 6-4 Establishing Datums for True Position Location

Where the actual size of the feature is larger than MMC, additional positional tolerance results (see Figure 6-8). This increase of positional tolerance is equal to the difference between the specified maximum material limit of size (MMC) and the actual size of the feature.

6.3.2.2 RFS as Related to Positional Tolerancing In certain cases, the design or function of a part may require the positional tolerance or datum reference, or both, to be maintained regardless of actual feature sizes. RFS, where applied to the positional tolerance, requires the axis of each feature to be located within the specified positional tolerance regardless of the size of the feature. This requirement imposes a closer control of the features involved and introduces complexities in verification.

6.3.2.3 Positional Location Example In Figure 6-10, the six holes may vary in size from 6.30 to 6.40 diameter. Each hole must be located within the specified positional tolerance regardless of the size of that hole. A hole at LMC (6.40 diameter) is as accurately located as a hole at MMC (6.30 diameter). This positional control is more restrictive than the MMC principle.

6.3.2.4 Functional Requirements Variations The functional requirements of some designs may require that RFS be applied to both the hole pattern and datum feature. That is, it may be necessary to require the axis of an actual datum feature (such as datum diameter B in Figure 6-10 to be the datum axis for the holes in the pattern regardless of the datum feature's size. The RFS application does not permit any shift between the axis of the datum feature and the pattern of features, as a group, where the datum feature departs from MMC. This may become necessary in applications such as rigid mounting features on a printed board where mating chassis features have a tight (or interference) fit in order to function properly. It typically is not necessary for component mounting holes because sufficient tolerances are in place to allow a clearance fit between component leads and holes.

6.3.3 LMC as Related to Positional Tolerancing Where positional tolerancing at LMC is specified, the stated positional tolerance applies when the feature contains the least amount of material permitted by its toleranced size dimension. Thus for holes, slots, and other internal features, least material condition is the condition where these features are at their maximum allowable sizes. For profile features or other external features, LMC is the minimum allowable size. Specification of LMC further requires perfect form at LMC. Perfect form at MMC is not required. Where the feature departs from its LMC size, an increase in positional tolerance is allowed, which is equal to the amount of such departure.

Specifying LMC is limited to positional tolerancing applications where MMC does not provide the desired control and RFS is too restrictive (see Figure 6-11). LMC is used to maintain a desired relationship between the surface of a feature and its true position at tolerance extremes. Considerations critical to the design are usually involved (see Figure 6-12).

6.3.4 Multiple Patterns of Features Located by Basic Dimensions Relative to Common Datums Where two or more patterns of features are located by basic dimensions

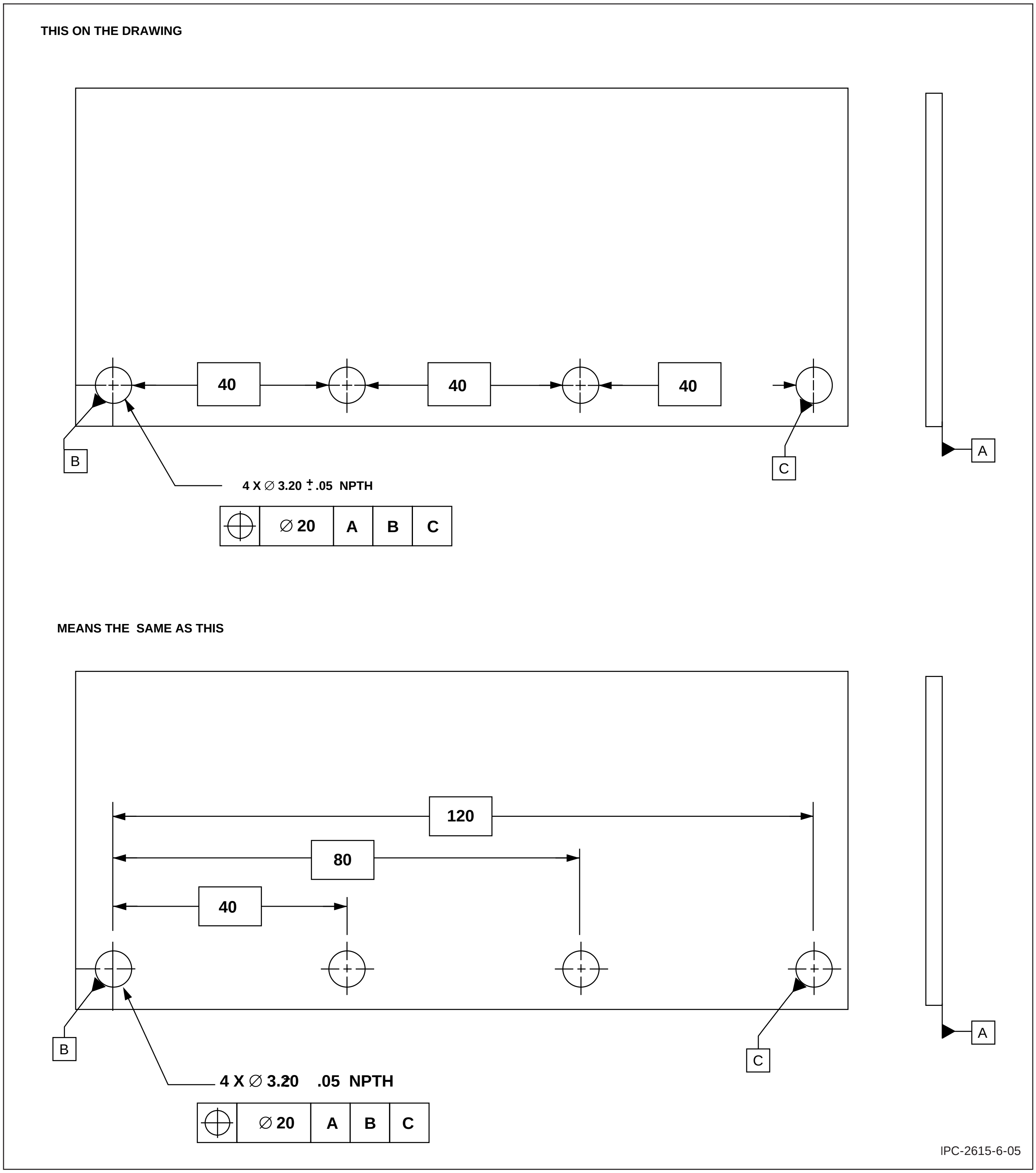


Figure 6-5 Basic Dimensioning Using Chain or Baseline Format

relative to common datum features referenced in the same order of precedence, the following apply.

6.3.4.1 Multiple Patterns of Features Example One In Figure 6-13, each pattern of features is located relative to common datum features not subject to size tolerances. Since all locating dimensions are basic and all measurements are from a common datum reference frame, verification of positional tolerance requirements for the printed board can be collectively accomplished in a single setup or gage as illustrated by Figure 6-14. The actual centers of all holes must lie on or within their respective tolerance zones

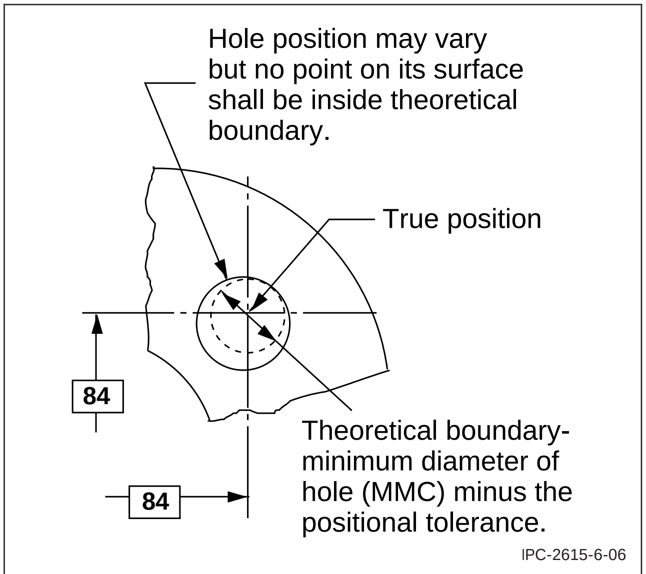


Figure 6-6 Boundary for Surface of Hole at MMC

when measured from datums A, B, and C. **Note:** The explanation given in Figure 6-14 still applies where independent verification of pattern locations becomes necessary due to size or complexity of the printed board.

6.3.4.2 Multiple Patterns of Features Example Two Multiple patterns of features, located by basic dimensions from common datum features that are subject to size tolerances, are also considered a single composite pattern if their respective feature control frames contain the same datums in the same order of precedence with the same modifying symbols. If such interrelationship is not required between one pattern and any other pattern or patterns. a notation such as SEP REQT is placed beneath each applicable feature control frame (see Figure 6-15).

This allows each feature pattern, as a group, to shift independently of each other relative to the axis of the datum feature and denotes an independent relationship between the patterns.

6.4 Feature Pattern Location Where design requirements permit the location of a pattern of features as a group to vary within a larger tolerance than the positional tolerance assigned to each feature in the pattern, composite positional tolerancing is used. For example, this would apply to a set of mounting holes to allow the pattern to float while maintaining a tighter tolerance center to center among the pattern.

6.4.1 Composite Positional Tolerancing Composite application of positional tolerancing provides for location of feature patterns as well as the interrelation of features within these patterns. Requirements are annotated by the use of a composite feature control frame. See 3.4.2.2. Each complete horizontal entry in the feature control frame of Figure 6-16 constitutes a separate requirement. The position symbol is entered once and is applicable to both horizontal entries. The upper entry is referred to as the pattern-locating control. It specifies the larger positional tolerance for the location of the pattern of features as a group. Applicable datums are specified in a desired order of precedence. The lower entry is referred to as the feature-relating control. It specifies the smaller positional tolerance for each feature within the pattern (feature-to-feature relationship) and repeats the primary datum (same as upper part of frame).

6.4.1.1 Pattern of Features Located from Specified Datums Each pattern of features is located from specified datums by basic dimensions. Figure 6-17 provides the explanation and interpretation of the illustration in Figure 6-16. The lower entry, in addition to providing interrelationship control of the features in each pattern, controls the extent of attitude variation (perpendicularity in the case of Figure 6-16) of each feature axis in relation to the plane established by datum A. As can be seen from the sectional view of the tolerance zones in Figure 6-17, the axes of both the large and small zones are parallel. The axes of the 6-12 holes may vary obliquely (out of perpendicular) only within the confines of the respective smaller positional tolerance zones. The axes of the holes must lie within the larger tolerance zones and also within the smaller tolerance zones. In certain instances, a portion of the smaller zones may fall beyond the peripheries of the larger tolerance zones. However, this portion of the smaller tolerance zone is not usable because the axis of the feature must not violate the larger tolerance zone. **Note:** The zones in Figure 6-17 are shown as they exist at MMC of the features depicted in Figure 6-16. The large zones would increase in size by the amount the features depart from MMC, as would the smaller zones; the two zones are not cumulative.

6.5 Bi-directional Positional Tolerancing of Features Where it is desired to specify a greater tolerance in one direction than another, bi-directional tolerancing may be applied. Bi-directional positional tolerancing results in a noncircular tolerance zone for locating round holes; therefore, the diameter symbol is omitted from the feature control frame in these applications (see Figure 6-18).

6.6 Position of Non-Circular Features The basic principles of true position dimensioning and positional tolerancing for circular features, such as holes, apply also to noncircular features, such as open end slots, tabs, and elongated holes. For such features of size, a positional tolerance is used to locate the center plane established by parallel surfaces of the feature. The tolerance value represents a distance between two parallel planes. The diameter symbol is omitted from the feature control frame (see Figures 6- 19 and 6-20).

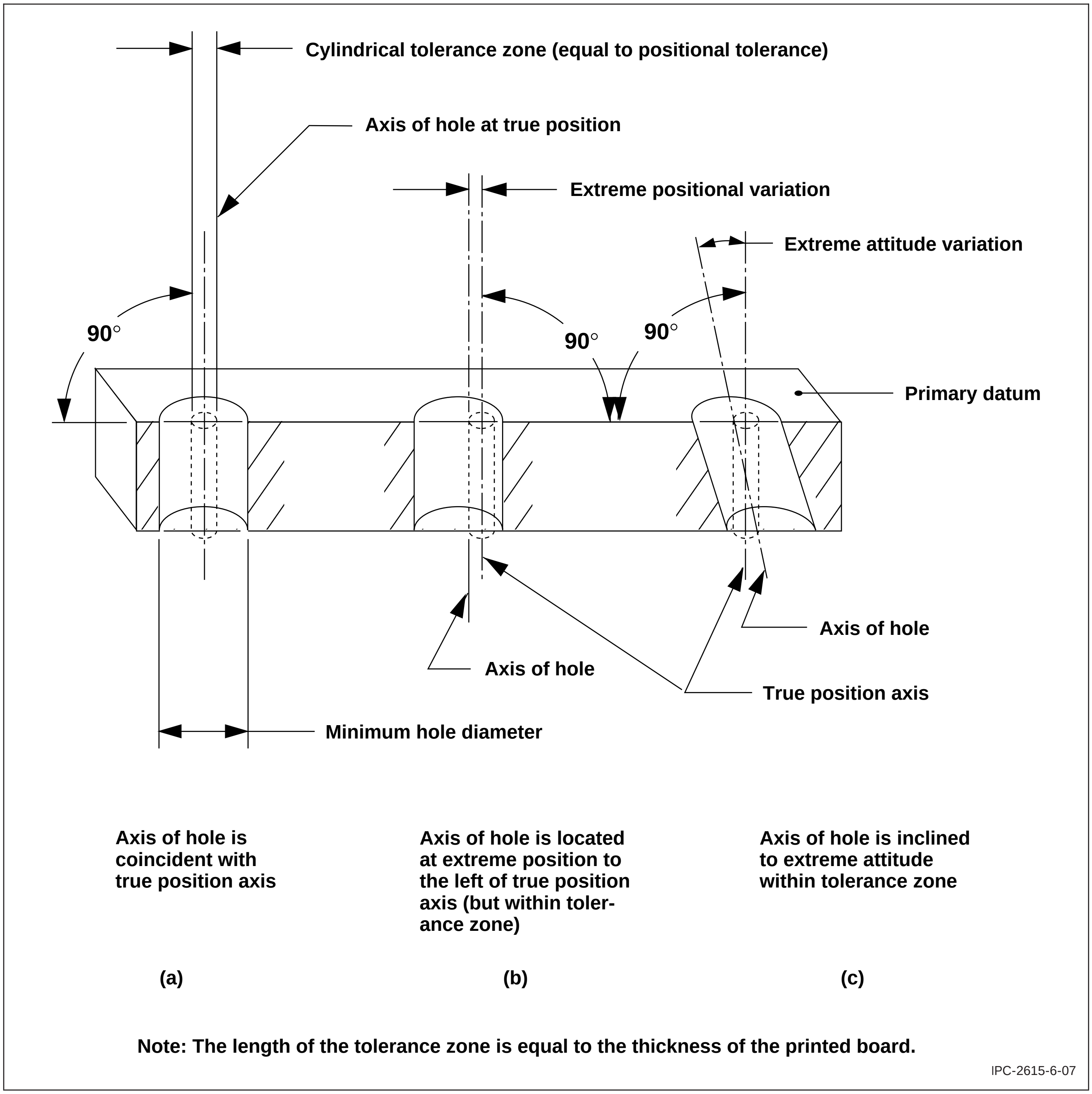


Figure 6-7 Hole Axes in Relation to Positional Tolerance Zones

6.6.1 Non-circular Features at MMC Where a positional tolerance of a noncircular feature - for example, a slot - applies at MMC, the following apply.

In Terms of the Surfaces of a Slot. While maintaining the specified width limits of the slot, no element of its side surfaces shall be inside a theoretical boundary defined by two parallel planes equally disposed about true position and separated by a distance equal to that shown for W in Figure 6-21.

6.7 Undimensioned Drawings (Artwork) An undimensioned drawing, as referenced in IPC-D-325 and applied to printed board artwork actually is dimensioned in digital database without tolerances. Positional tolerances for features such as circular lands for plated-through holes are defined by minimum annular ring and registration requirements in combination with the hole position tolerance. The term ‘‘undimensioned’’ comes from the standpoint of a pattern that is used as supplied, without reference to dimensions, except for location.

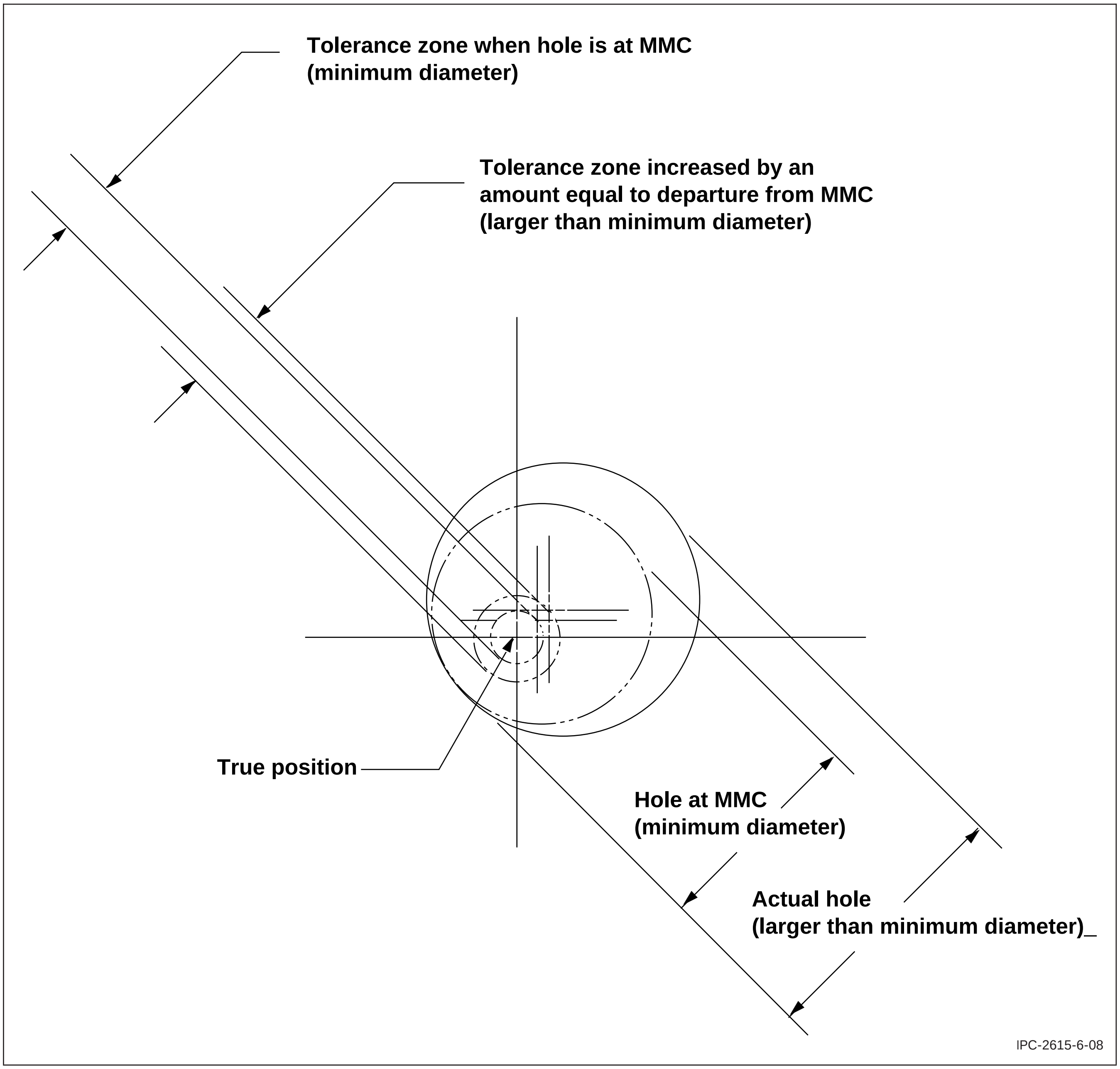


Figure 6-8 Increase in Positional Tolerance Where Hole is Not at MMC

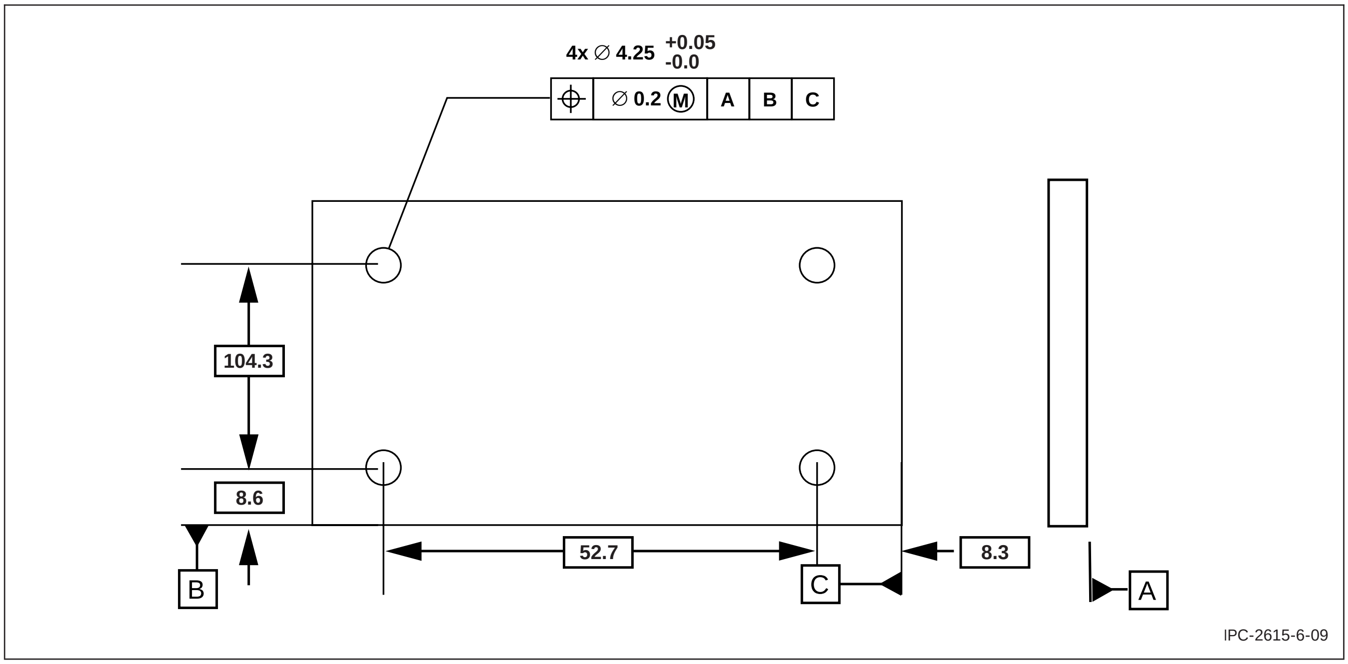


Figure 6-9 Conventional Positional Tolerancing at MMC

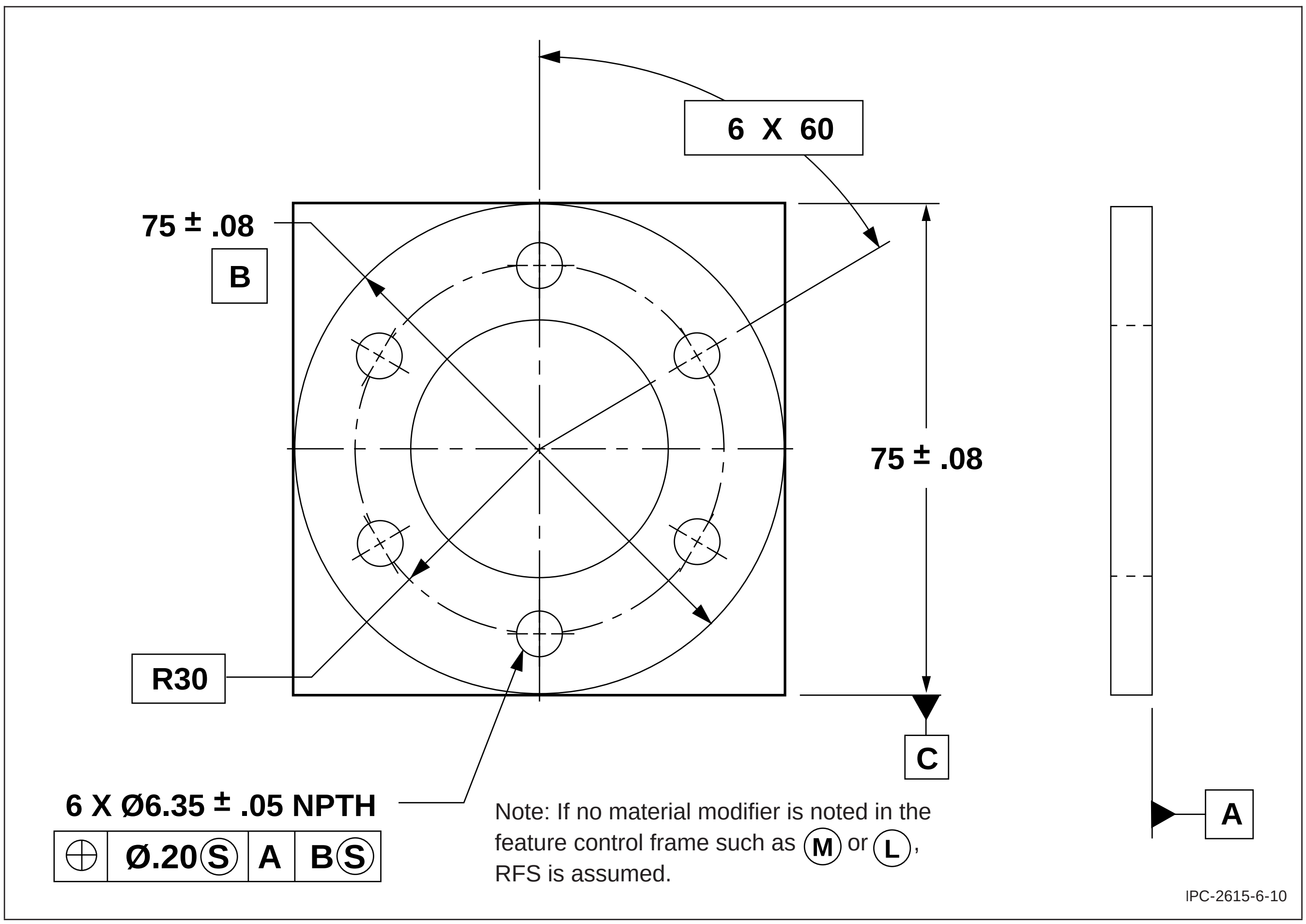


Figure 6-10 Regardless of Feature Size Applied to A Feature and A Datum

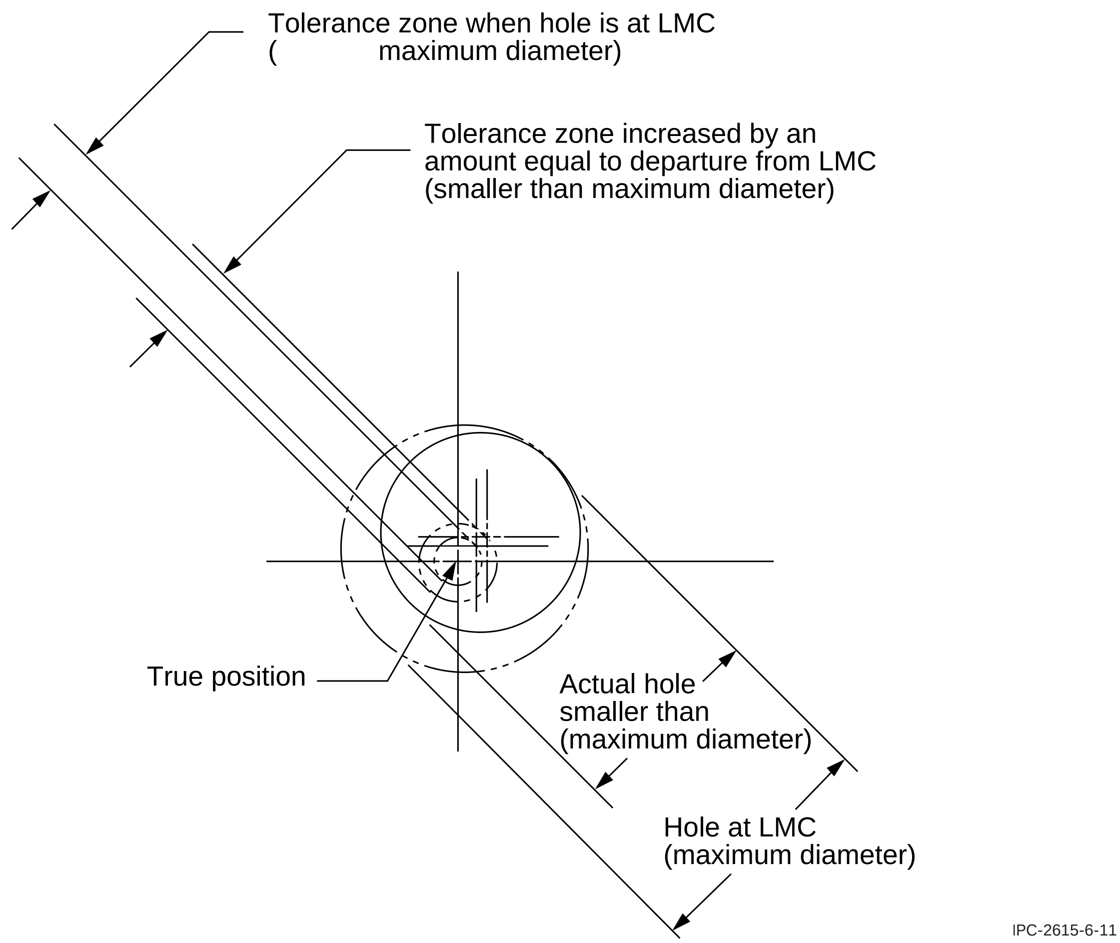
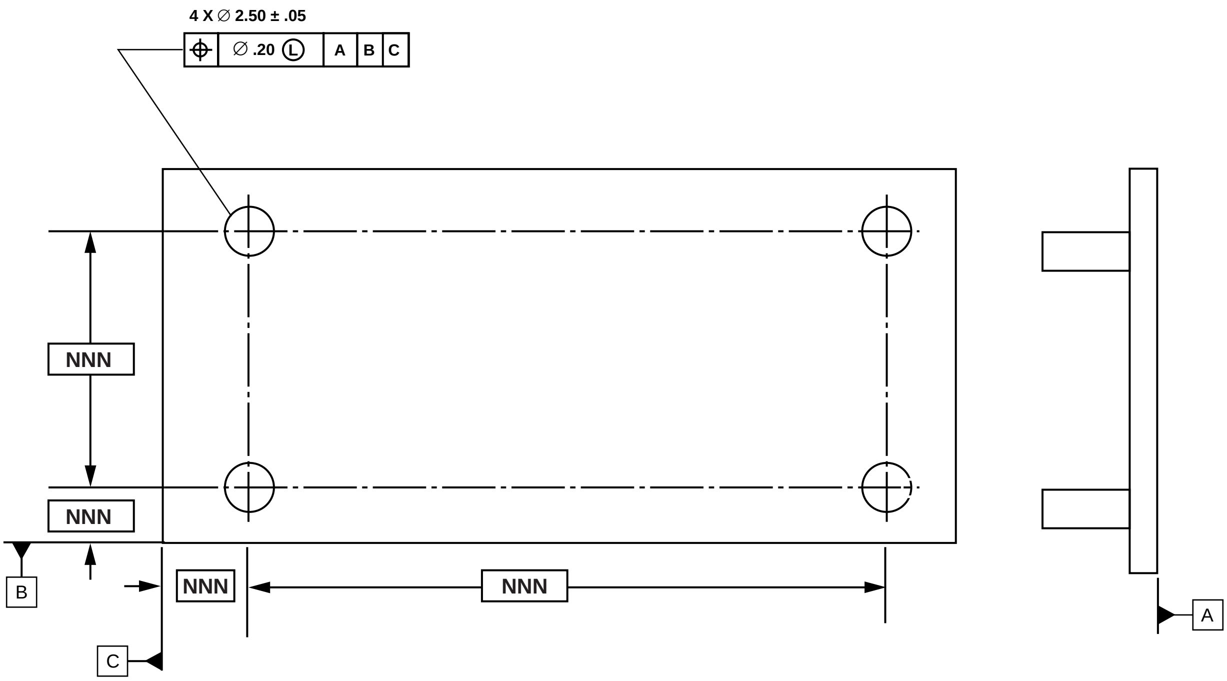


Figure 6-11 Increase in Positional Tolerance Where Hole is not at LMC

THIS ON THE DRAWING



MEANS:

AT 2.55∅, OR LMC, THERE IS A .20 DIAMETER TOLERANCE ZONE FOR POSITION
AT 2.50∅, THERE IS A .25 DIAMETER TOLERANCE ZONE FOR POSITION
AT 2.45∅, OR MMC, THERE IS A .30 DIAMETER TOLERANCE ZONE FOR POSITION

(BONUS TOLERANCE IS ADDED AS THE PIN GETS SMALLER WHEN LMC IS APPLIED)

IPC-2615-6-12

Figure 6-12 LMC Applied to A Pattern of Mounting Pins

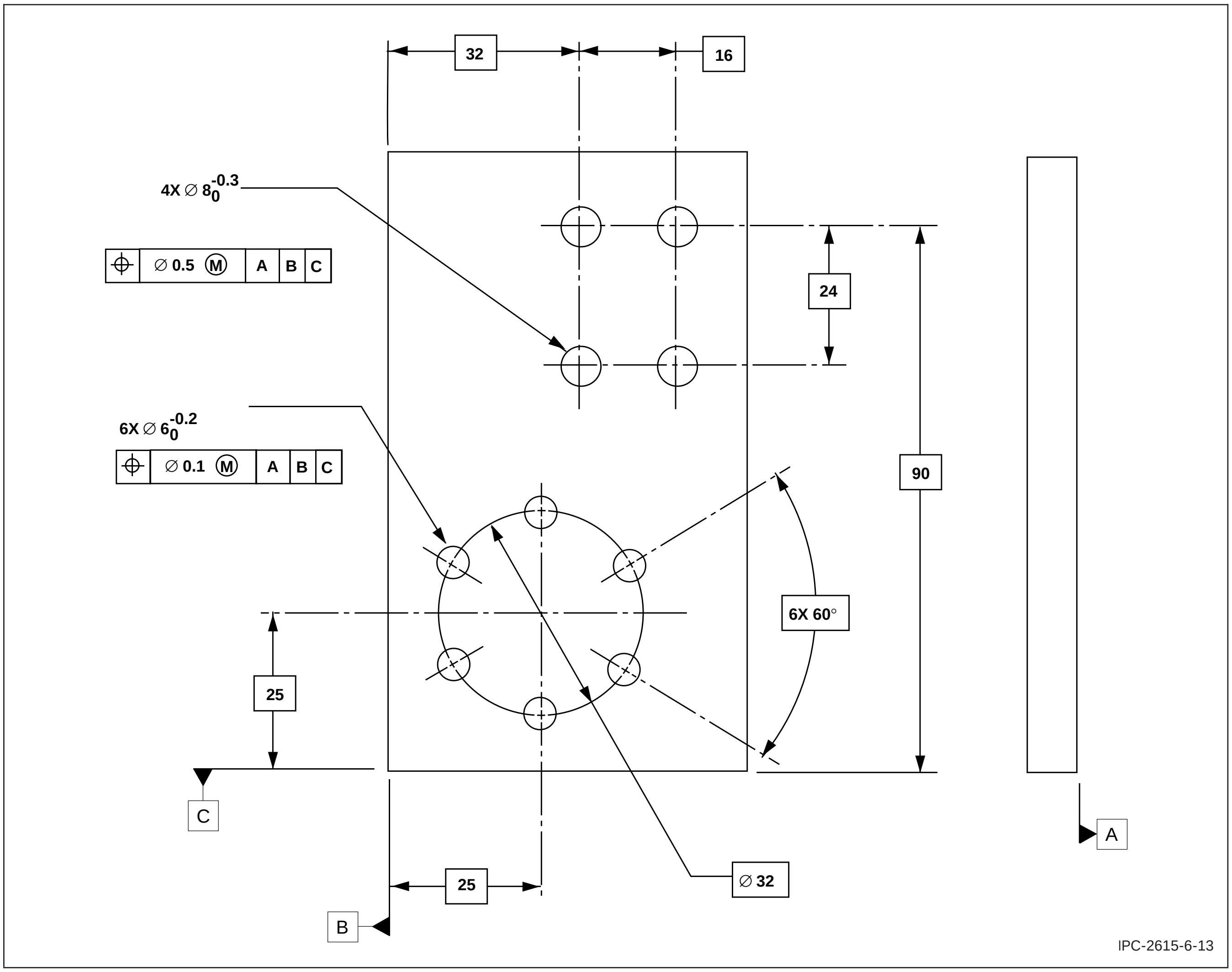


Figure 6-13 Multiple Patterns of Features

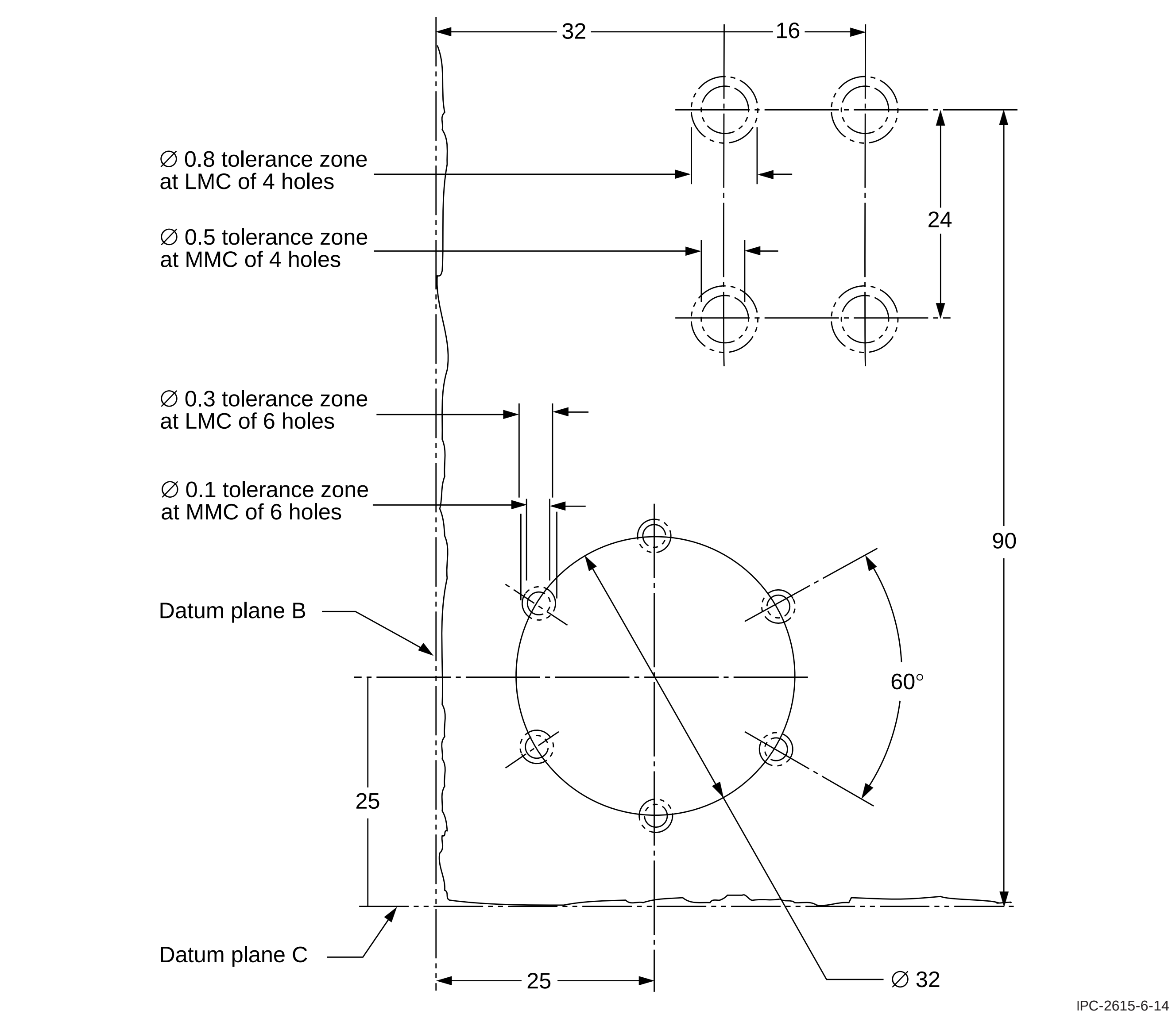


Figure 6-14 Tolerance Zones for Patterns Shown in Figure 6-13

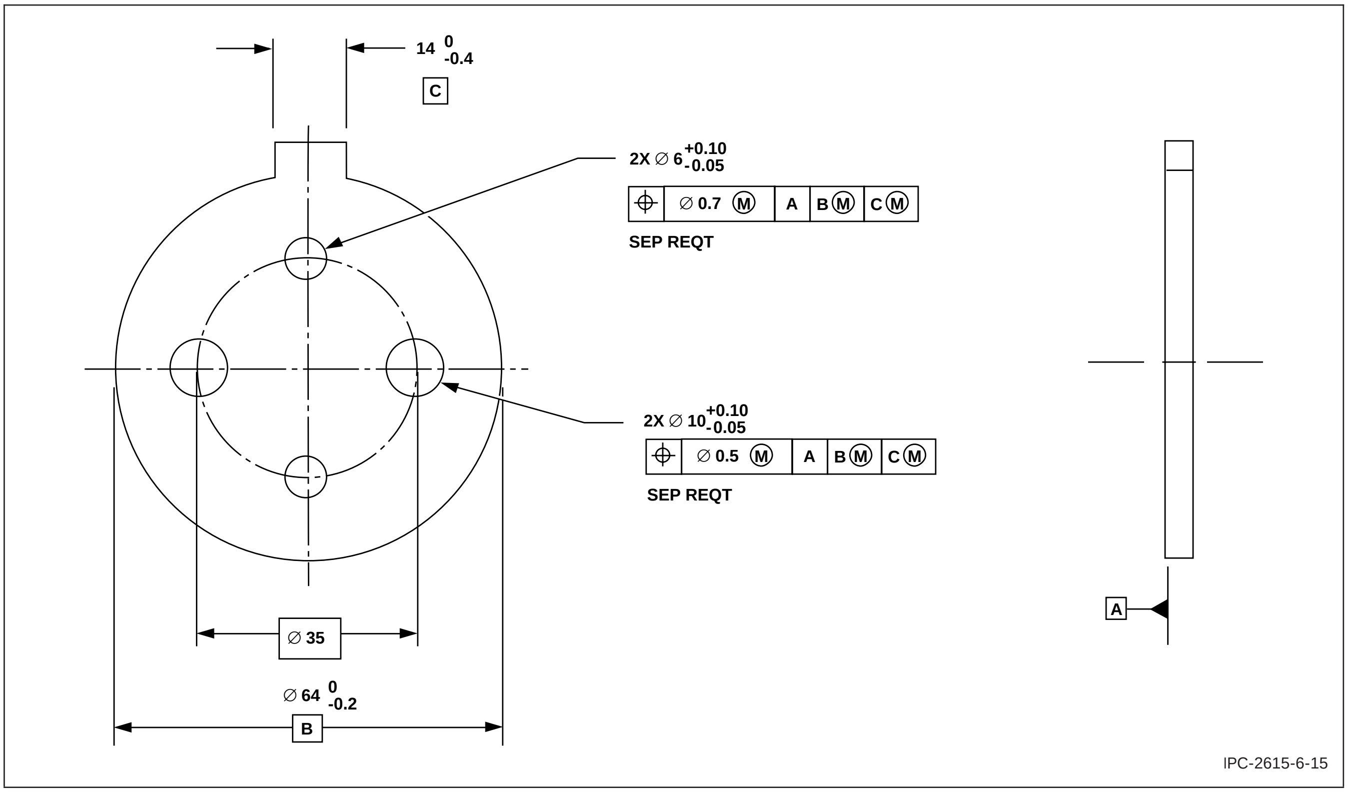


Figure 6-15 Multiple Patterns of Features, Separate Requirement

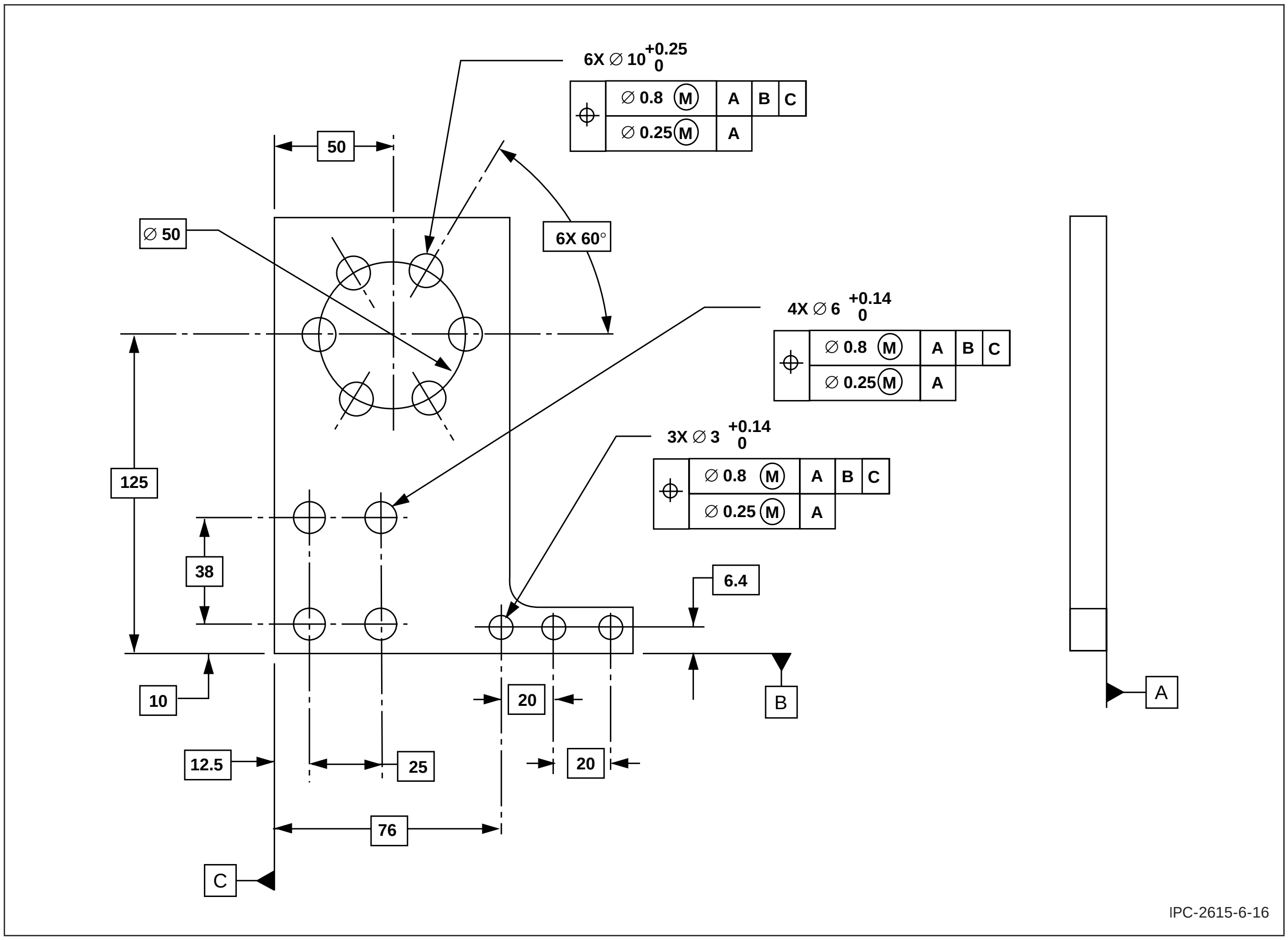


Figure 6-16 Hole Patterns Located By Composite Positional Tolerancing

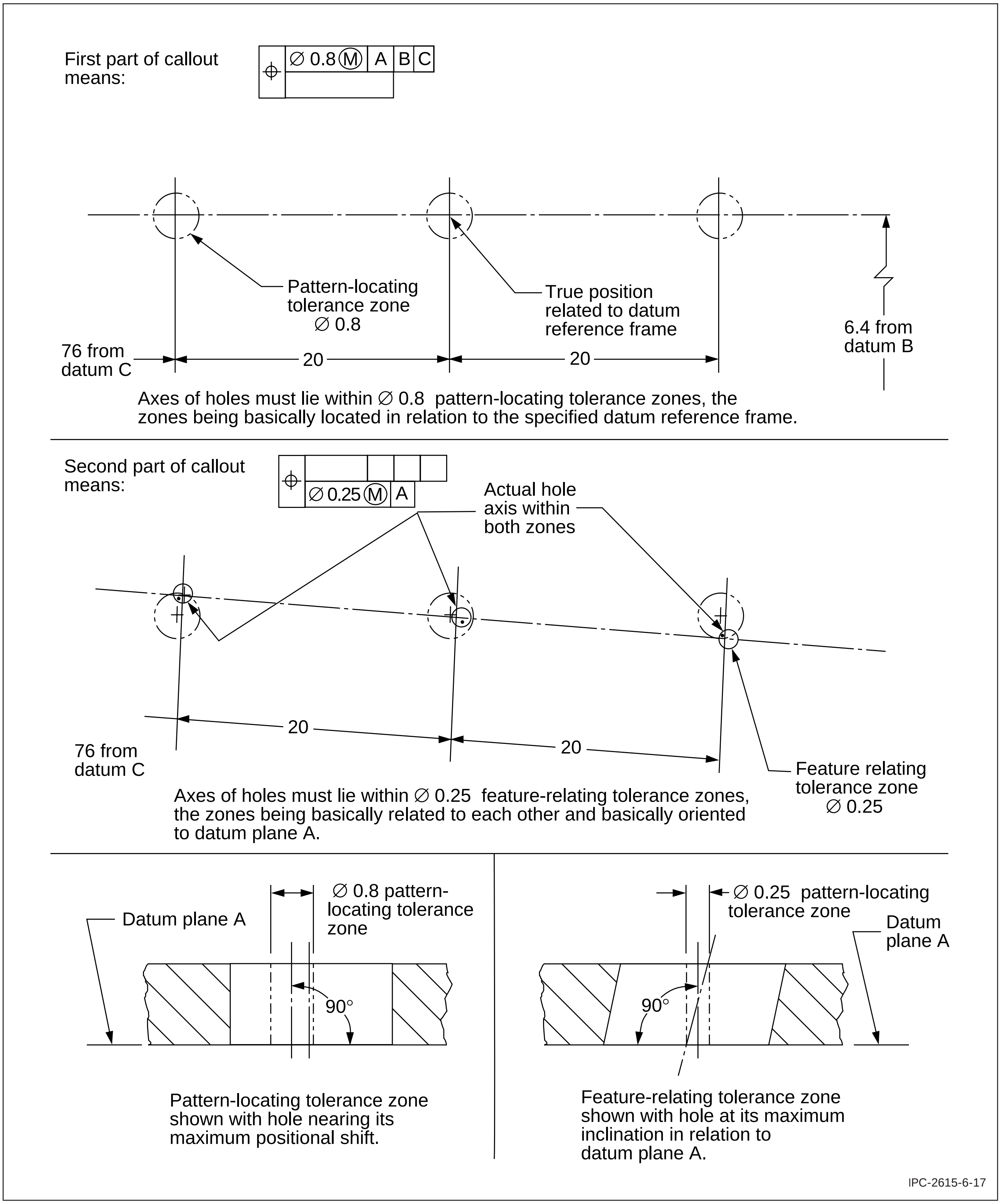


Figure 6-17 Tolerance Zone for Three-Hole Hole Patterns Shown in Figure 6-16.

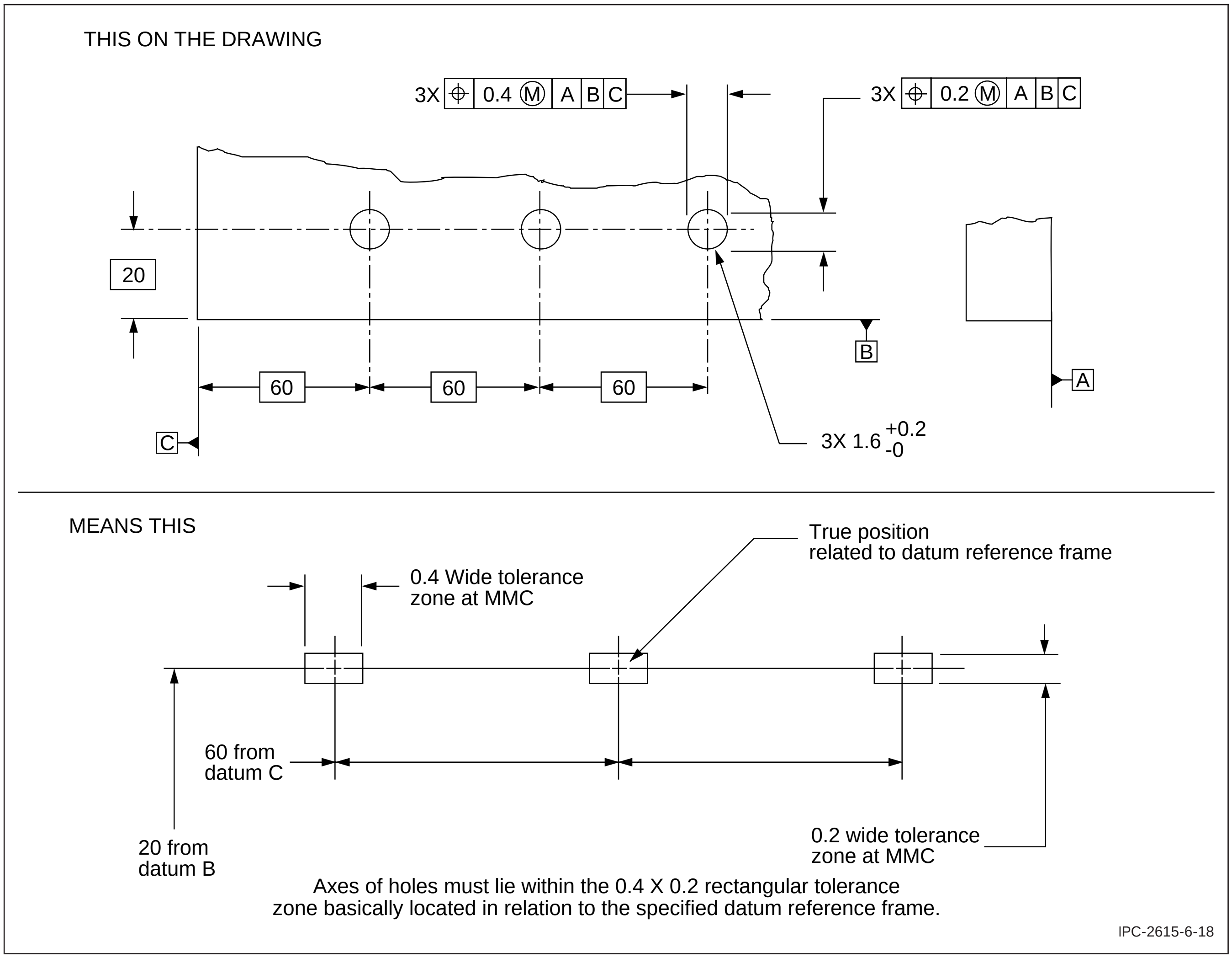


Figure 6-18 Bi-Directional Positional Tolerancing, Rectangular Coordinate Method

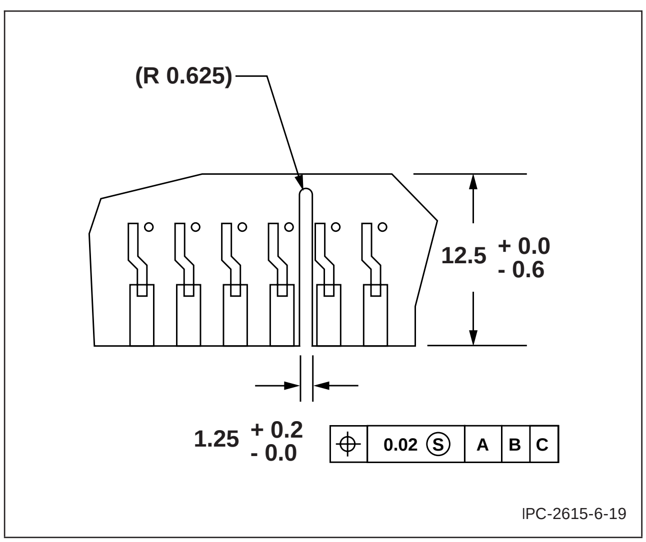


Figure 6-19 Keying Slot Detail

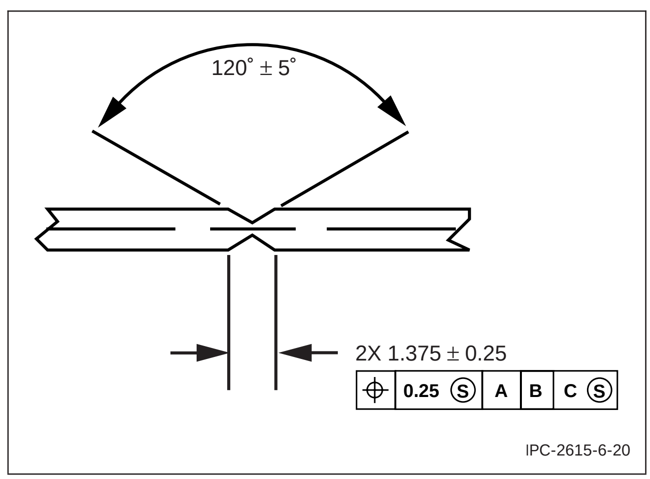


Figure 6-20 “V” Groove

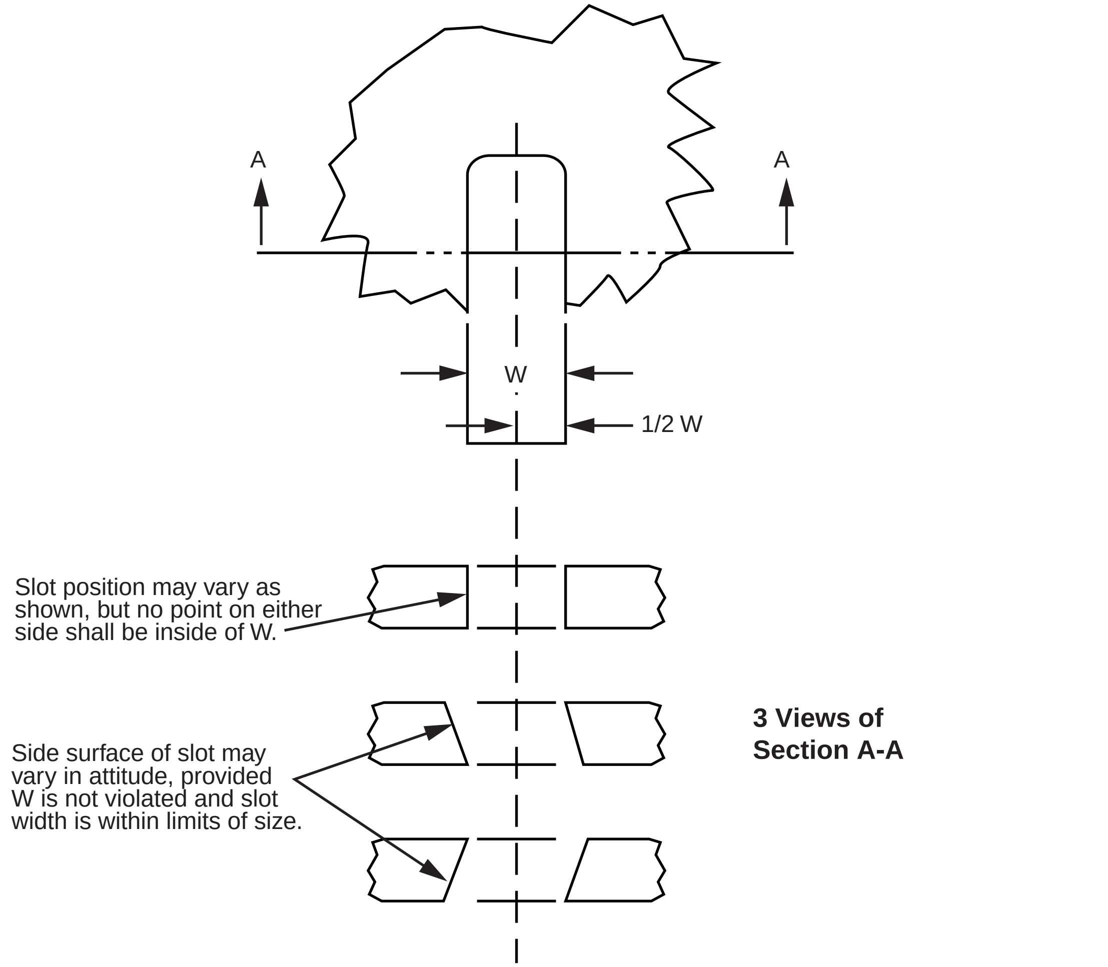


Figure 6-21 Keying Slot Detail

7 TOLERANCES OF FORM, ORIENTATION, PROFILE

7.1 General This Section establishes the principles and methods of dimensioning and tolerancing to control form, profile, and orientation of various geometrical shapes and free state variations.

7.2 Form and Orientation Control Form tolerances control straightness, flatness and circularity. Orientation tolerances control angularity, parallelism and perpendicularity. A profile tolerance may control form, orientation, and size, depending on how it is applied. Since, to a certain degree, the limits of size control form and parallelism, and tolerances of location control orientation, the extent of this control should be considered before specifying form and orientation tolerances.

7.3 Specifying Form and Orientation Tolerances Form and orientation tolerances critical to function and interchangeability are specified where the tolerances of size and location do not provide sufficient control. A tolerance of form or orientation may be specified where no tolerance of size is given; for example, the control of flatness after assembly of the printed board.

7.3.1 Form and Orientation Tolerance Zones A form or orientation tolerance specifies a zone within which the considered feature, its axis, or its center plane must be contained. The tolerance value represents a total linear distance between two geometric boundaries.

7.3.1.1 Tolerance Control Over a Limited Area Certain designs require control over a limited area or length of the surface, rather than control of the total surface. In these instances, the area, or length, and its location are indicated by a heavy chain line drawn adjacent to the surface with appropriate dimensioning. Where so indicated, the specified tolerance applies within these limits instead of to the total surface.

7.4 Profile Control A profile can be the outline of a printed board or the outline of a land or land pattern. The elements of a profile are straight lines, arcs, and other curved lines. If the drawing specifies individual tolerances for segments of a profile, these segments must be individually verified. With profile tolerancing, the true profile may be defined by basic radii, basic angular dimensions, basic coordinate dimensions, and formulas or undimensioned drawings (artwork).

7.4.1 Profile Tolerancing The profile tolerance specifies a uniform boundary along the true profile within which the elements of the surface must lie. It is used to control form, or combinations of size, form, and orientation. Profile tolerances are specified as a tolerance divided bilaterally on both sides of the true profile or applied unilaterally to either side of the true profile. Where an equally disposed bilateral tolerance is intended, it is necessary to show only the feature control frame with a leader directed to the surface.

For an unequally disposed or a unilateral tolerance, phantom lines are drawn parallel to the true profile to indicate the tolerance zone boundary. One end of a dimension line is extended to the feature control frame. The phantom line need extend only a sufficient distance to make its application clear (see Figure 7-1).

Where a profile tolerance applies all around the profile of a printed board, the symbol used to designate ''all around'' is placed on the leader from the feature control frame (see Figure 7-2). Where segments of a profile have different tolerances, the extent of each profile tolerance is indicated by the use of reference letters to identify the extremities or limits of each requirement (see Figure 7-3).

7.4.1.1 Profile Tolerance Implementation The tolerance value represents the distance between two boundaries disposed about the true profile or entirely disposed on one side of the true profile. Profile tolerances apply normal (perpendicular) to the true profile at all points along the profile. The boundaries of the tolerance zone follow the geometric shape of the true profile. The actual surface must lie within the specified tolerance zone and all variations from the true profile must blend. Where a profile tolerance encompasses a sharp corner, the tolerance zone extends to the intersection of the boundary lines (see Figure 7-4). Since the intersecting surfaces may lie anywhere within this converging zone, the actual printed board contour could conceivably be rounded. If this is undesirable, the drawing must indicate the design requirements, such as by specifying the maximum radius (see Figure 7-5).

7.4.1.2 Application of Datums In most cases, profile of a surface tolerance requires reference to datums in order to provide proper orientation of the profile.

7.4.2 Controlled Radius Tolerance A controlled toleranced radius symbol CR creates a tolerance zone defined by two arcs (the minimum and maximum radii) that are tangent to the adjacent surfaces (see Figure 7-5). The printed board contour within the crescent-shaped tolerance zone must be a faired curve without reversals. Additionally, radii taken at all points on the printed board contour shall neither be smaller than the specified minimum limit nor larger than the maximum limit.

7.4.3 Angular Surfaces Where an angular surface, such as the chamfer on an edge connector is defined by a combination of a linear dimension and an angle, the surface must lie within a tolerance zone represented by two nonparallel planes (see Figure 7-6). The tolerance zone will be wider as the distance from the apex of the angle increases. Where a tolerance zone with parallel boundaries is desired, a basic angle may be specified as in Figure 7-7.

7.4.3.1 Angularity Tolerance Angularity is an orientation tolerance applicable to related features. This tolerance controls the orientation of features to one another. Angularity is the condition of a surface or axis at a specified angle (other than 90°) from a datum plane or axis. An angularity tolerance specifies a tolerance zone defined by two parallel planes at the specified basic angle from a datum plane, or axis, within which the surface of the considered feature must lie.

7.4.3.2 Implied 90° Angle By convention, where center lines and surfaces of features of a part are depicted on engineering drawings intersection at right angles, a 90° angle is not specified. Implied 90° angles are understood to apply.

7.4.3.3 Chamfers Specified By Note A note may be used to specify 45° chamfers as in Figure 7-8. This method is used only with 45° chamfers as the linear value applies in either direction.

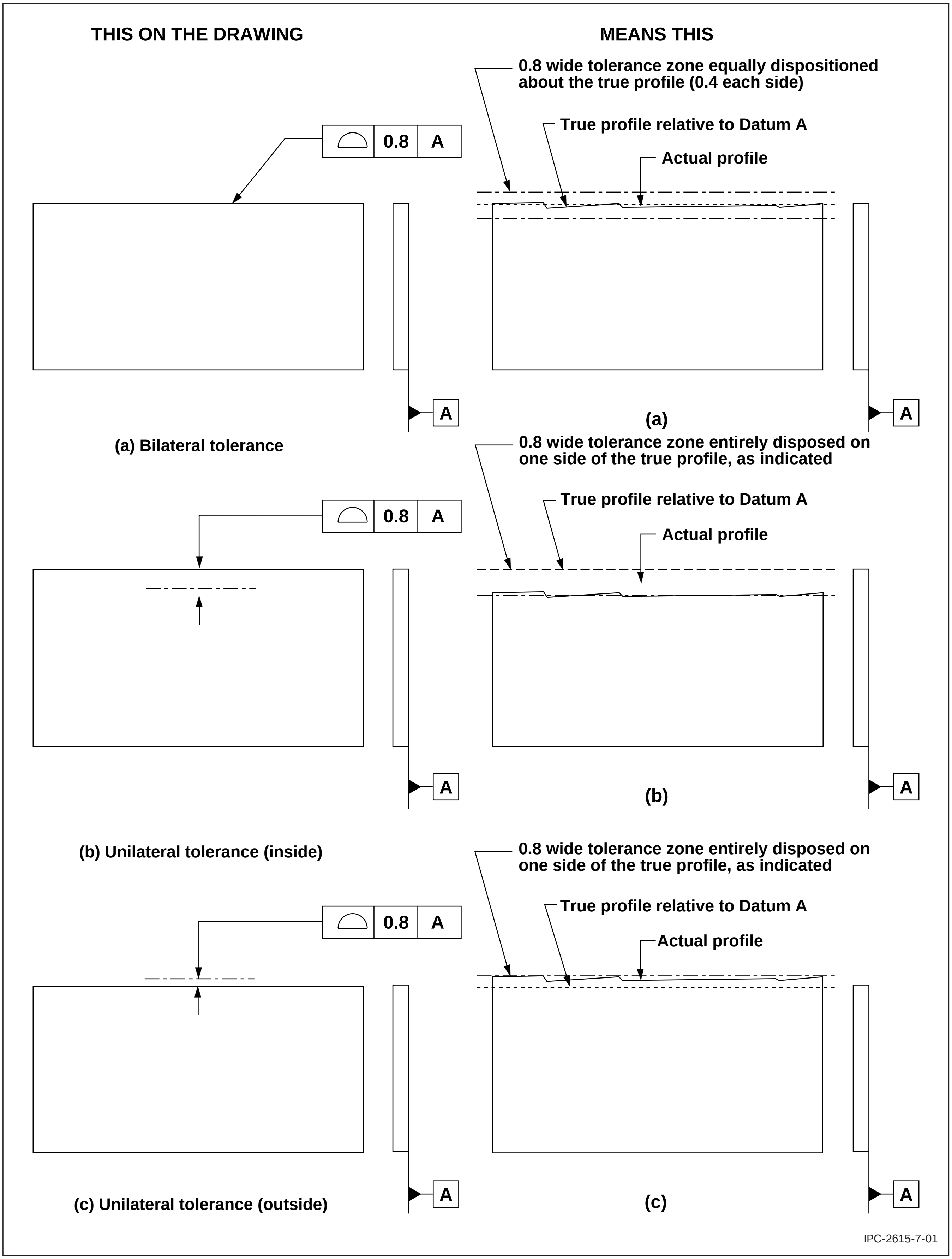


Figure 7-1 Application of A Profile of A Surface to A Contour

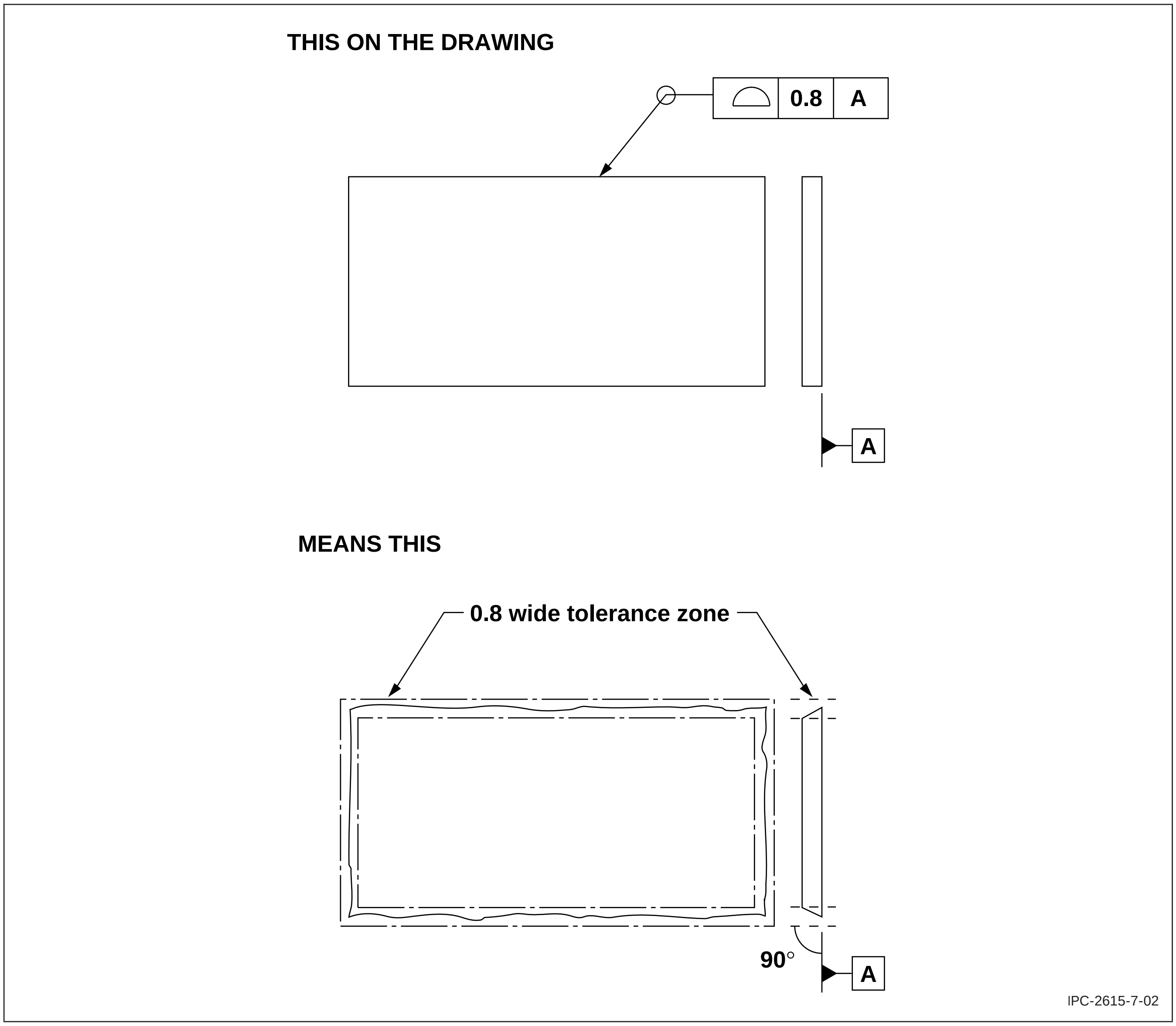


Figure 7-2 Specifying Profile of A Surface All Around

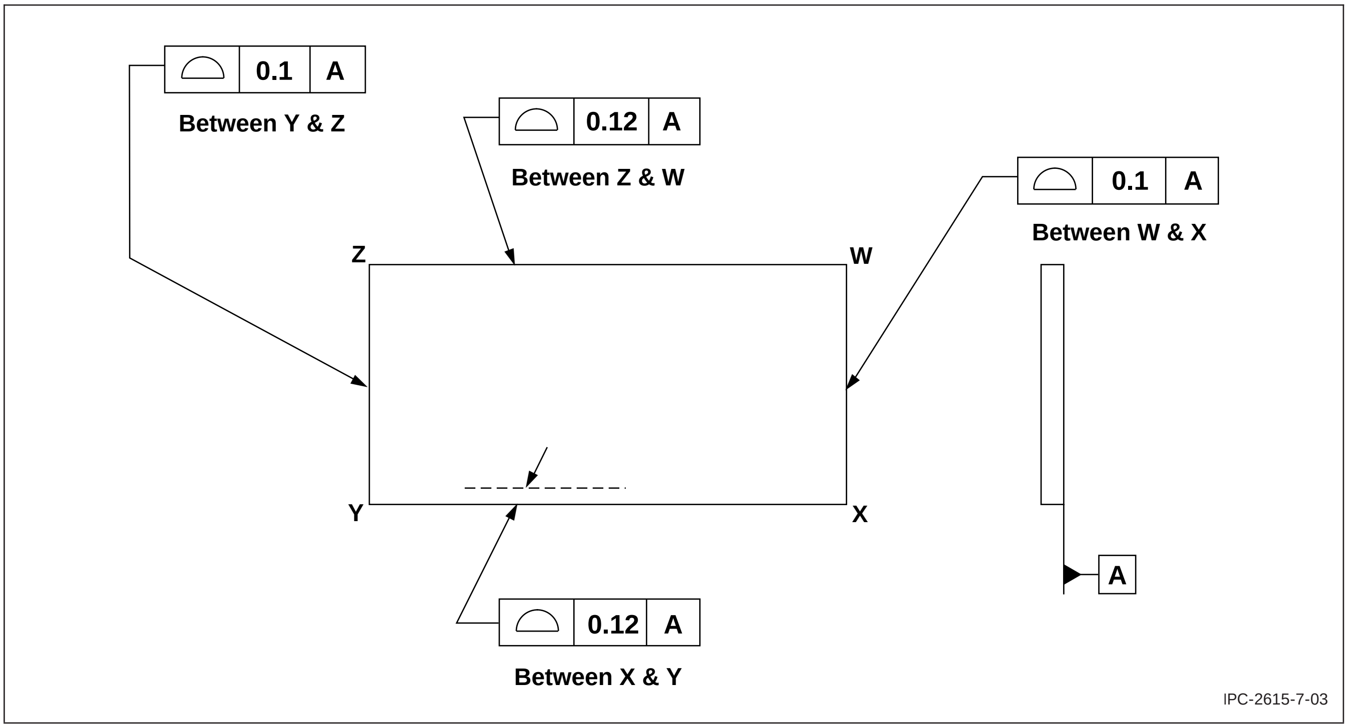


Figure 7-3 Specifying Different Profile Tolerance

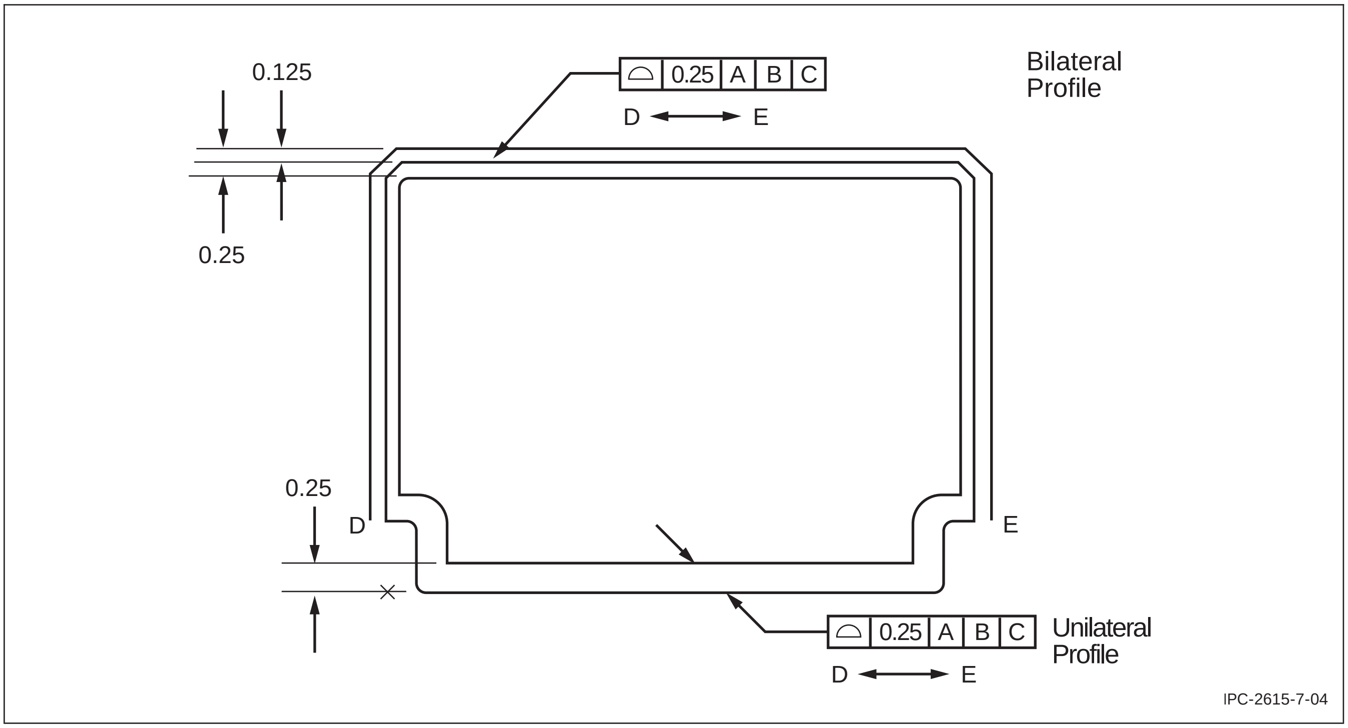


Figure 7-4 Profile Implementation

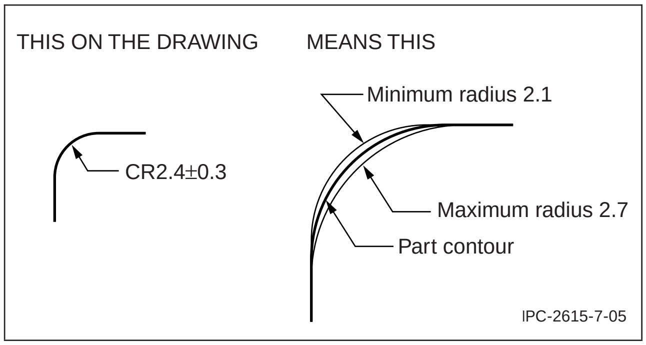


Figure 7-5 Specifying A Controlled Radius

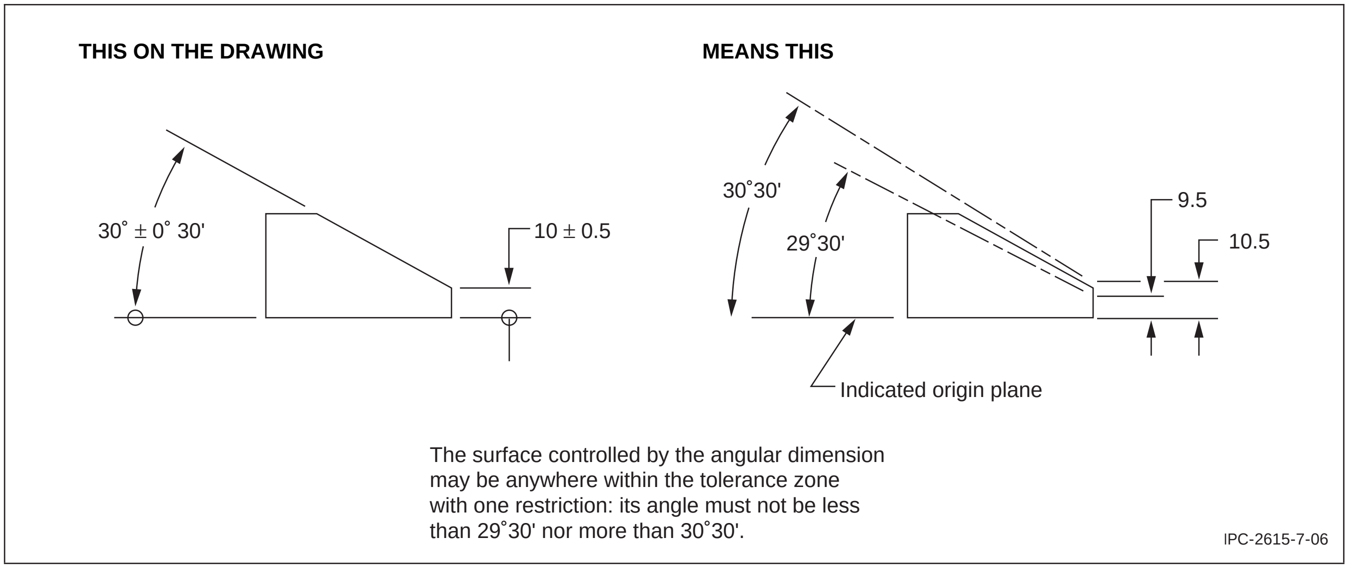


Figure 7-6 Tolerancing An Angular Surface Using A Combination of Linear and Angular Dimensions

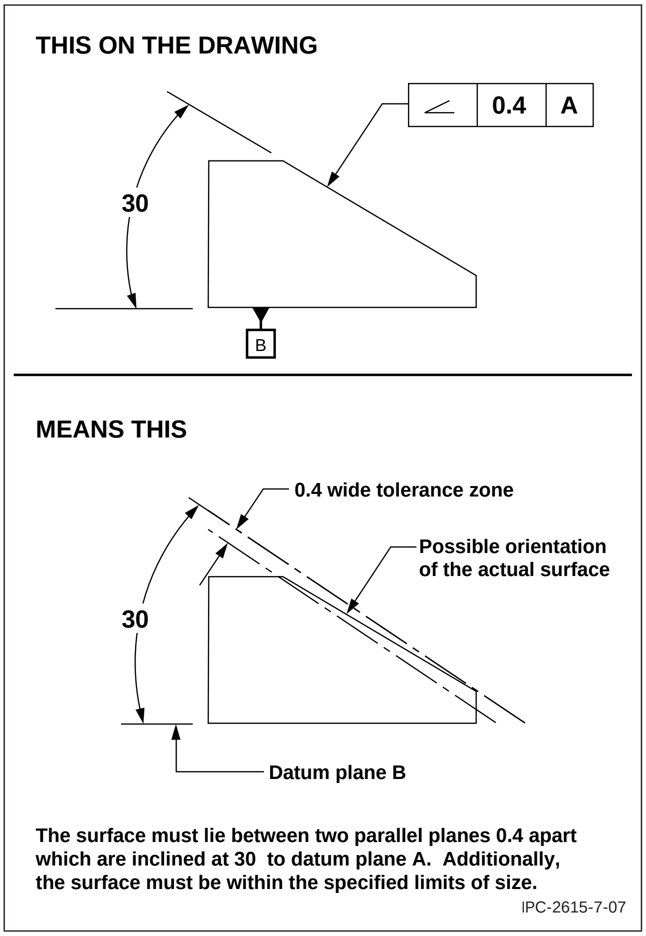


Figure 7-7 Interpreting Angularity Tolerances

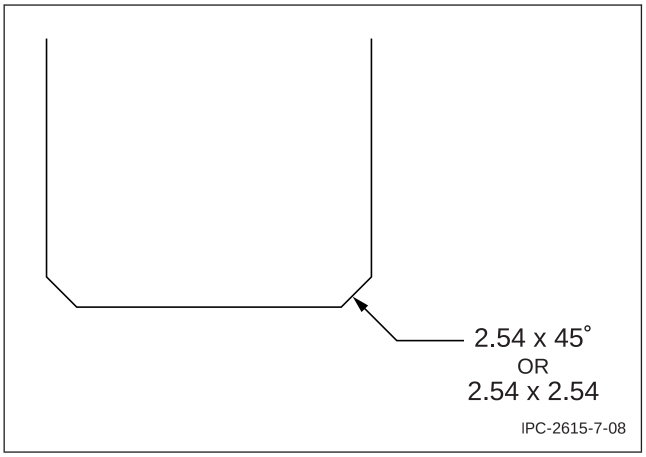


Figure 7-8 45 Degree Chamfer

Appendix A: FUNDAMENTAL DIMENSIONING AND TOLERANCING RULES

A1.0 Dimensioning and Tolerancing Fundamentals

Dimensioning and tolerancing shall clearly define engineering intent and shall conform to the following.

a) Each dimension shall have a tolerance, except for those dimensions specifically identified as reference, maximum, minimum, or stock (commercial stock size). The tolerance may be applied directly to the dimension (or indirectly in the case of basic dimensions), indicated by a general note, or located in a supplementary block of the drawing format (see ANSI Y14.1).

b) Dimensions for size, form, and location of features shall be complete to the extent that there is full understanding of the characteristics of each feature. Neither scaling (measuring the size of a feature directly from an engineering drawing) nor assumption of a distance or size is permitted. **Note:** Undimensioned drawings, such as printed board artwork prepared on stable material, are excluded, provided the necessary control dimensions are specified.

c) Each necessary dimension of an end product shall be shown. No more dimensions than those necessary for complete definition shall be given. The use of reference dimensions on a drawing should be minimized.

d) Dimensions shall be selected and arranged to suit the function and mating relationship of a printed board and shall not be subject to more than one interpretation.

e) The drawing should define a printed board without specifying manufacturing methods. Thus, only the finished diameter of a hole is given without indicating whether it is to be drilled, punched, or made by any other operation. However, in those instances where manufacturing, processing, quality assurance, or environmental information is essential to the definition of engineering requirements, it shall be specified on the drawing or in a document referenced on the drawing i.e., drilled hole considerations used to determine land size.

f) It is permissible to identify as nonmandatory certain processing dimensions that provide for finish allowance, shrink allowance, and other requirements, provided the final dimensions are given on the drawing. Nonmandatory processing dimensions shall be identified by an appropriate note, such as NONMANDATORY (MFG DATA).

g) Dimensions should be arranged to provide required information for optimum readability. Dimensions should be shown in true profile views and refer to visible outlines.

h) Wires, cables, laminate sheets, rods, and other materials manufactured to gauge or code numbers shall be specified by linear dimensions indicating the diameter or thickness. Gauge or code numbers may be shown in parentheses following the dimension.

i) A 90° angle is implied where center lines and lines depicting features are shown on a drawing at right angles and no angle is specified.

j) A 90° BASIC angle applies where center lines of features in a pattern or surfaces shown at right angles on the drawing are located or defined by basic dimensions and no angle is specified.

k) Unless otherwise specified, all dimensions are applicable at 25°C (68°F). Compensation may be made for measurements made at other temperatures.

A1.2 Units of Measurement For uniformity, all dimensions in this Standard are given in SI units. However, the unit of measurement selected should be in accordance with the policy of the user.

A1.2.1 SI Metric Linear Units The commonly used SI linear unit used on engineering drawings is the millimeter.

A1.2.2 U.S. Customary Linear Units Historically, the common U.S. customary linear unit used on engineering drawings was the decimal inch. However, for electronics the dominate and preferred system is now the metric system.

A1.2.3 Identification of Linear Units On drawings where all dimensions are either in millimeters or inches, individual identification of linear units is not required. However, the drawing shall contain a note stating UNLESS OTHERWISE SPECIFIED, ALL DIMENSIONS ARE IN MILLIMETERS (or IN INCHES, as applicable).

A1.2.3.1 Unit Abbreviations Where some inch dimensions are shown on a millimeter-dimensioned drawing, the abbreviation IN. shall follow the inch values. Where some millimeter dimensions are shown on an inch-dimensioned drawing, the symbol mm shall follow the millimeter values.

A1.2.3.2 Angular Units Angular dimensions are expressed in either degrees and decimal parts of a degree or in degrees, minutes, and seconds. These latter dimensions are expressed by symbols: for degrees °, for minutes ', and for seconds.'' Where degrees are indicated alone, the numerical value shall be followed by the symbol °. Where only minutes or seconds are specified, the number of minutes or seconds shall be preceded by ''0' or ''00,'' as applicable (see Figure A-1).

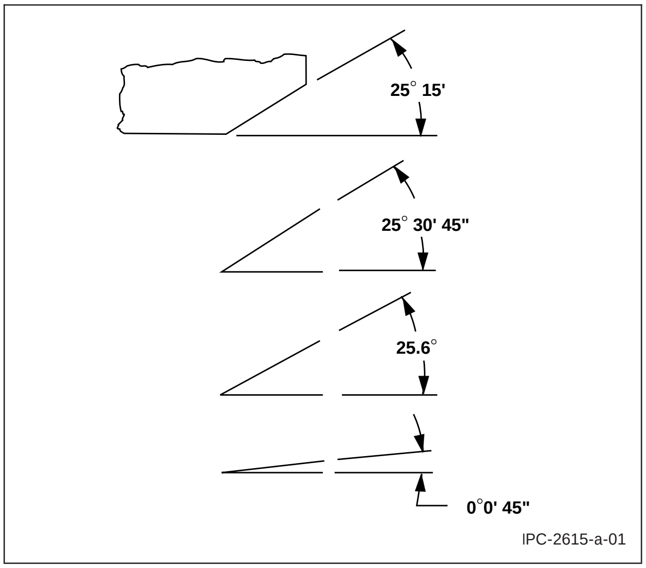


Figure A-1 Angular Units

A1.3 Types of Dimensioning Decimal dimensioning shall be used on drawings.

A1.3.1 Millimeter Dimensioning The following shall be observed when specifying millimeter dimensions on drawings.

a) Where the dimension is less than one millimeter, a zero precedes the decimal point (see Figure A-2).

b) Where the dimension is a whole number, neither the decimal point nor a zero is shown (see Figure A-2).

c) Where the dimension exceeds a whole number by a decimal fraction of one millimeter, the last digit to the right of the decimal point is not followed by a zero (see Figure A-2). **Note:** This practice differs for tolerances expressed bilaterally or as limits (see B1.3).

d) Neither commas nor spaces shall be used to separate digits into groups in specifying millimeter dimensions on drawings.

A1.3.2 Decimal Inch Dimensioning The decimal inch system is explained in ANSI B87.1. The following shall be observed when specifying decimal inch dimensions on drawings.

a) A zero is not used before the decimal point for values less than one inch.

b) A dimension is expressed to the same number of decimal places as its tolerance. Zeros are added to the right of the decimal point where necessary (see Figure A-3 and B1.3).

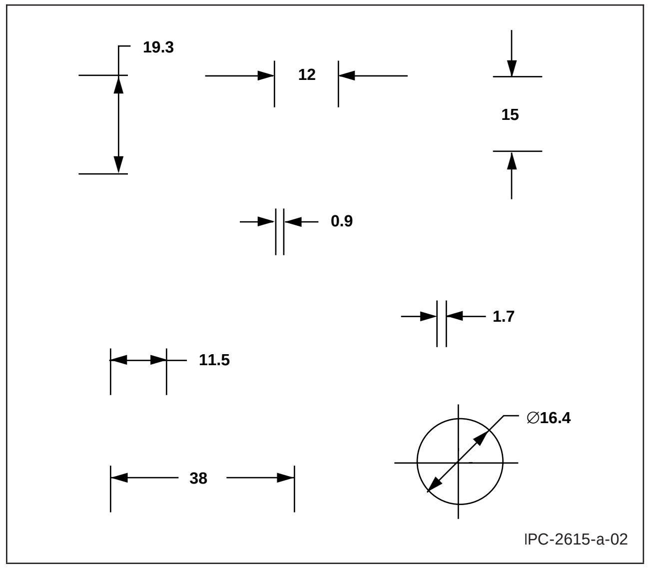


Figure A-2 Millimeter Dimensioning

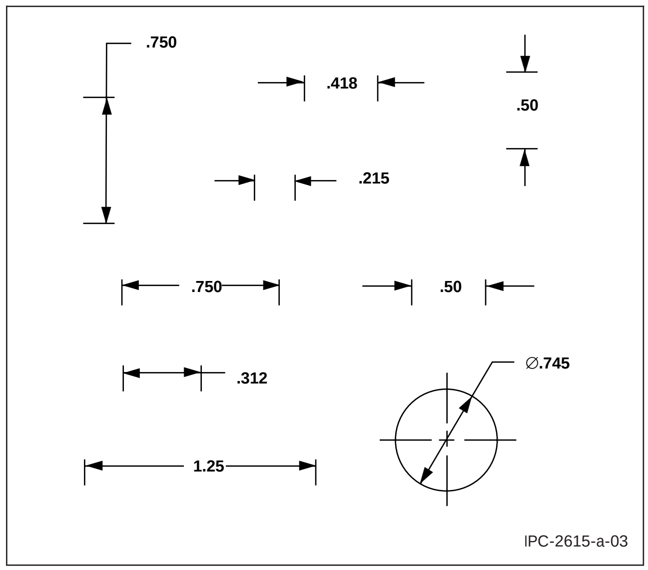


Figure A-3 Decimal Inch Dimensioning

A1.3.3 Decimal Points Decimal points must be uniform, dense, and large enough to be clearly visible and meet the reproduction requirements of ANSI Y14.2M. Decimal points are placed in line with the bottom of the associated digits.

A1.3.4 Conversion and Rounding of Linear Units For conversion and rounding of linear units, see appendix B B1.5.2.

A1.4 Application of Dimensions Dimensions are applied by means of dimension lines, extension lines, chain lines, or a leader from a dimension, note, or specification directed to the appropriate feature (see Figure A-4). General notes are used to convey additional information. For further information on dimension lines, extension lines, chain lines, and leaders. see ANSI Y14.2M.

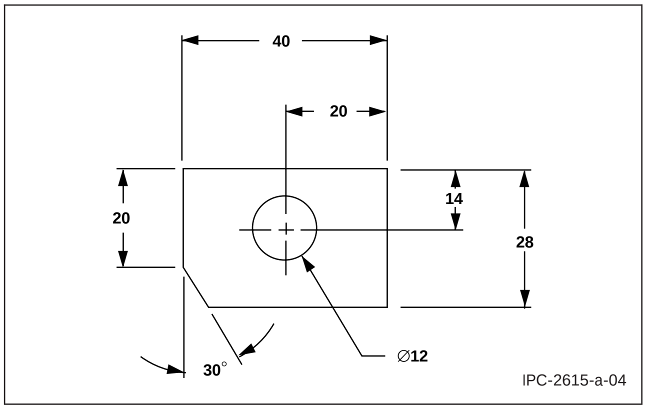


Figure A-4 Application of Dimensions

A1.4.1 Dimension Lines A dimension line, with its arrowheads, shows the direction and extent of a dimension. Numerals indicate the number of units of a measurement. Preferably, dimension lines should be broken for insertion of numerals as shown in Figure A-4. Where horizontal dimension lines are not broken, numerals are placed above and parallel to the dimension lines.

A1.4.1.1 Dimension Lines Alignment Dimension lines shall be aligned if practical and grouped for uniform appearance (see Figure A-5).

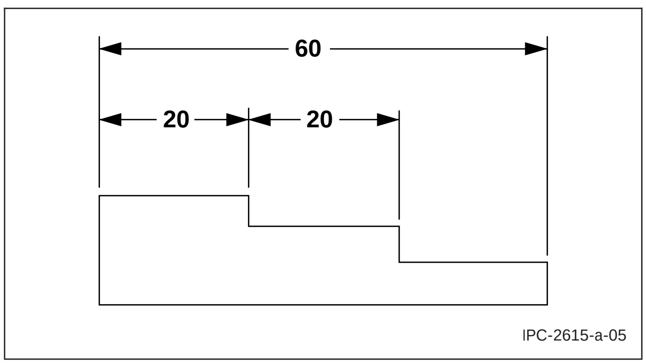


Figure A-5 Grouping of Dimensions

A1.4.1.2 Orientation of Dimension Lines to Measured Feature Dimension lines are drawn parallel to the direction of measurement. The space between the first dimension line and the printed board outline should be not less than 10 mm; the space between succeeding parallel dimension lines should be not less than 6 mm (see Figure A-6).

Note: These spacings are intended as guides only. If the drawing meets the reproduction requirements of the accepted industry or military reproduction specification, non conformance to these spacing requirements is not a basis for rejection of the drawing.

Where there are several parallel dimension lines, the numerals should be staggered for easier reading (see Figure A-7).

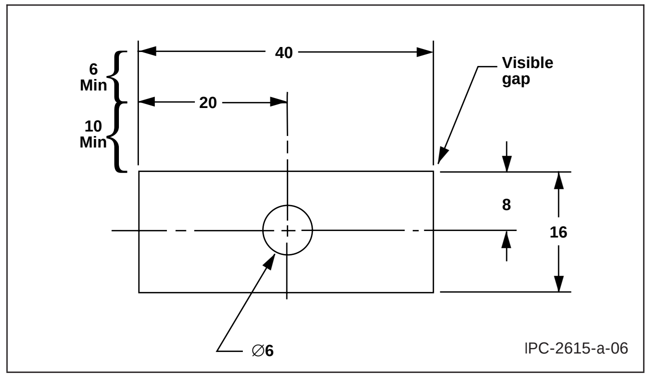


Figure A-6 Spacing of Dimensions

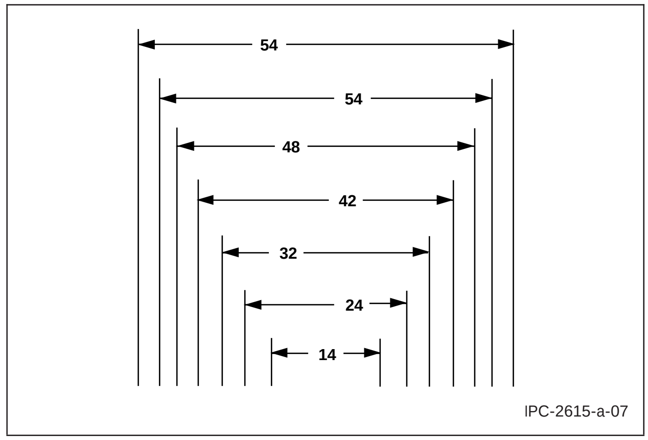


Figure A-7 Staggered Dimensions

A1.4.1.3 The following shall not be used as a dimension line:

- a center line,
- an extension line,
- a phantom line,
- a line that is part of the outline of the object, or
- a continuation of any of these lines.

A dimension line is not used as an extension line, except where a simplified method of coordinate dimensioning is used to define curved outlines (see Figure A-8).

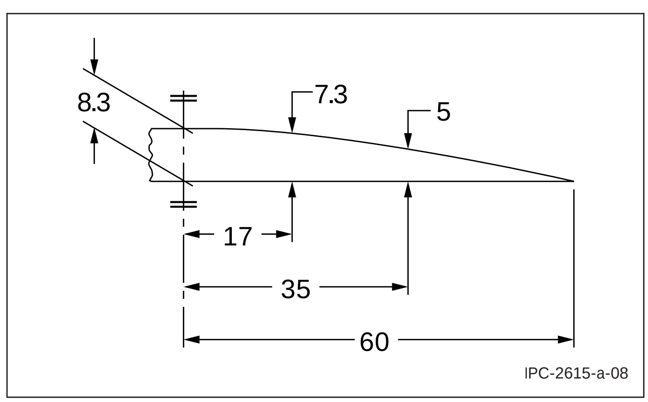


Figure A-8 Dimension Line/Extension Line

A1.4.1.4 Dimension Line of an Angle The dimension line of an angle is an arc drawn with its center at the apex of the angle. The arrowheads terminate at the extensions of the two sides (see Figures A-1 and A-4).

A1.4.1.5 Crossing Dimension Lines Crossing dimension lines should be avoided. Where unavoidable, the dimension lines are unbroken.

A1.4.2 Extension (Projection) Lines Extension lines are used to indicate the extension of a surface or point to a location outside the printed board outline. Normally, extension lines start with a short visible gap from the outline of the printed board and extend beyond the outermost related dimension line (see Figure A-6). Extension lines are usually drawn perpendicular to dimension lines. Where space is limited, extension lines may be drawn at an oblique angle to clearly illustrate where they apply. Where oblique lines are used, the dimension lines are shown in the direction in which they apply (see Figure A-9).

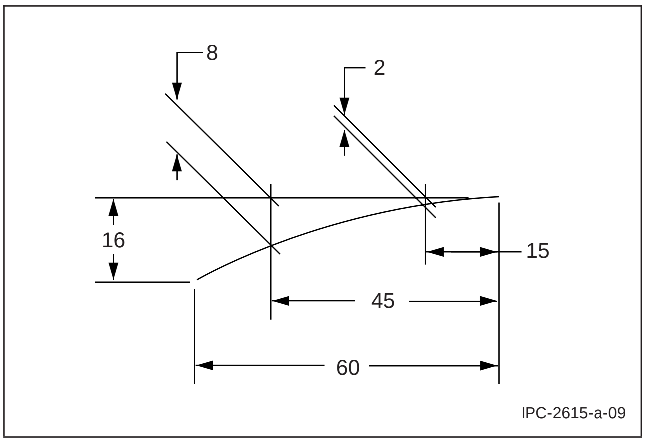


Figure A-9 Oblique Extension Lines

A1.4.2.1 Crossing Extension Lines Wherever practical, extension lines should neither cross one another nor cross dimension lines. To minimize such crossings, the shortest dimension line is shown nearest the outline of the object (see Figure A-7). Where extension lines must cross other extension lines, dimension lines, or lines depicting features, they are not broken. Where extension lines cross arrowheads or dimension lines close to arrowheads, a break in the extension line is advisable (see Figure A-10).

A1.4.2.2 Point Location by Extension Lines Where a point is located by extension lines only, the extension lines from surfaces should pass through the point (see Figure A-11).

A1.4.3 Limited Length or Area Indication Where it is desired to indicate that a limited length or area of a surface is to receive additional treatment or consideration within limits specified on the drawing, the extent of these limits

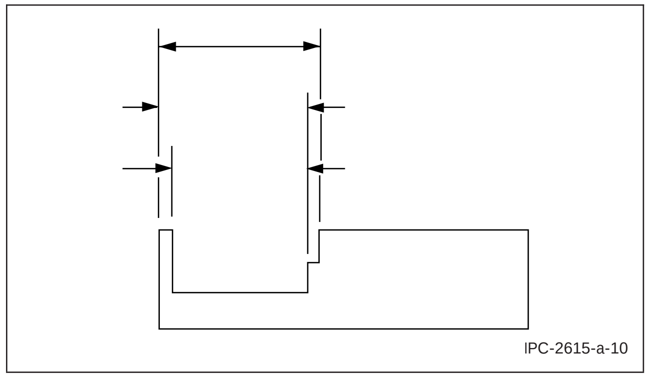


Figure A-10 Breaks In Extension Lines

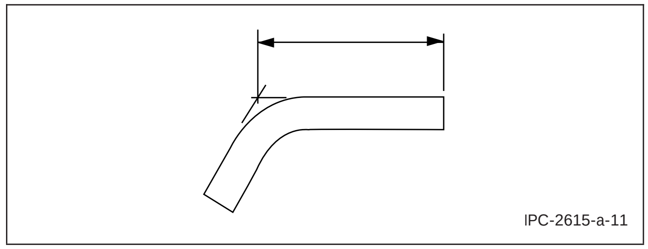


Figure A-11 Point Location

may be indicated by use of a chain line. Where the desired area is shown on a direct view of the surface, the area is section lined within the chain line boundary and appropriately dimensioned (see Figure A-12).

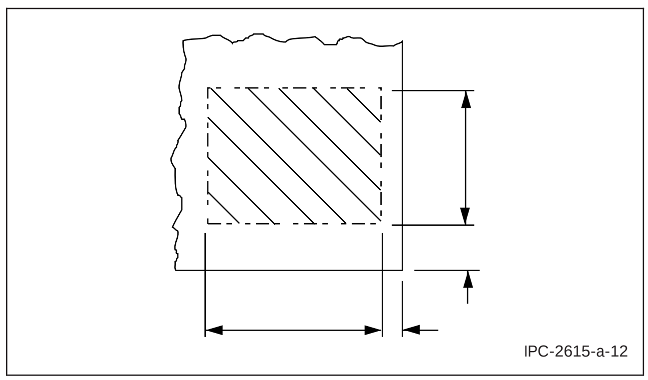


Figure A-12 Limited Length or Area Indication

A1.4.4 Leaders (Leader Lines) A leader is used to direct a dimension, note, or symbol to the intended place on the drawing (see Figure A-13).

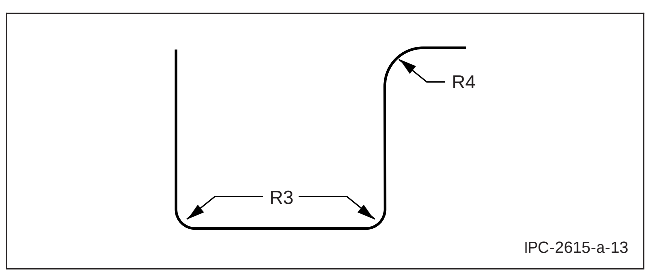


Figure A-13 Leader-Directed Dimension

A1.4.4.1 Leader-Directed Dimensions Leader-directed dimensions are specified individually to avoid complicated leaders. If too many leaders would impair the legibility of the drawing, letters or symbols should be used to identify features (see Figure A-14).

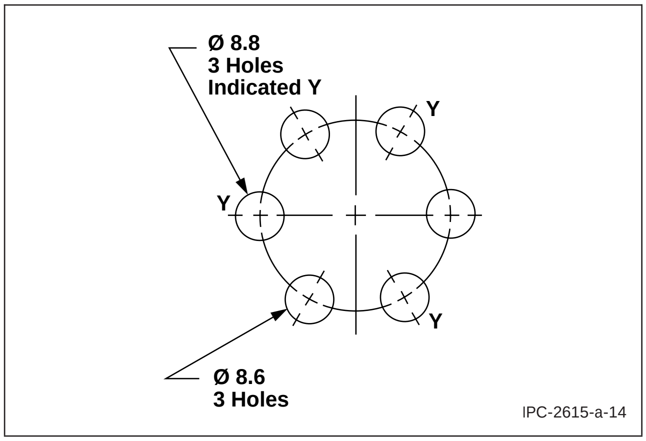


Figure A-14 Minimizing Leaders

A1.4.4.2 Leaders Directed to Circles or Arcs Where the leader is directed to a circle or an arc, its direction should be radial (see Figure A-15).

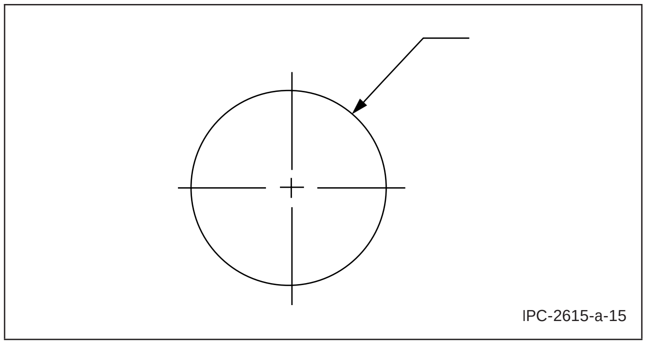


Figure A-15 Leader Directed to Circle

A1.4.5 Reading Direction Dimensions and notes should be placed to be read from the bottom of the drawing with regard to orientation of the drawing format (see Figure A-16).

A1.4.6 Reference Dimensions The method for identifying a reference dimension (or reference data) on drawings is to enclose the dimension (or data) within parentheses.

A1.4.7 Overall Dimensions Where an overall dimension is specified, one intermediate dimension is omitted or identified as a reference dimension. Where the intermediate dimensions are more important than the overall dimension, the overall dimension, if used, is identified as a reference dimension (see Figure A-17).

A1.4.8 Dimensioning Within the Outline of a View Dimensions are usually placed outside the outline of a view. Where directness of application makes it desirable, or

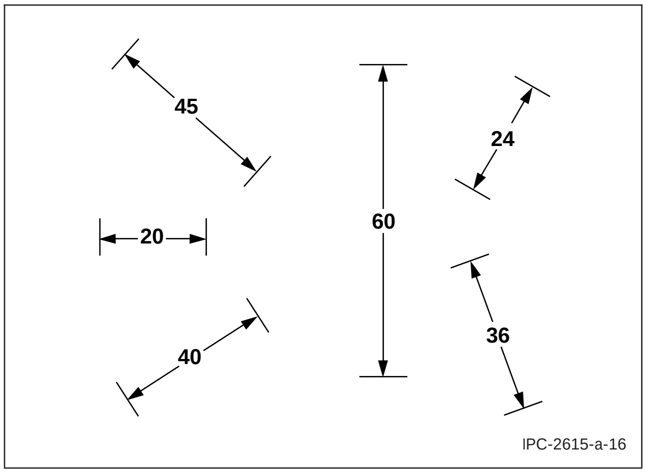


Figure A-16 Reading Direction

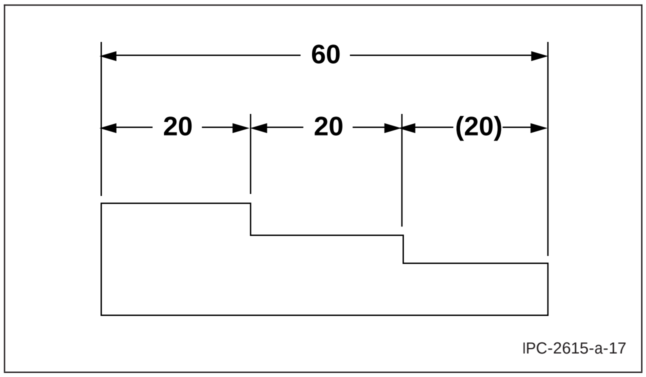


Figure A-17 Intermediate Reference Dimension

where extension lines or leader lines would be excessively long, dimensions may be placed within the outline of a view

A1.4.9 Dimensions Not to Scale Where it is necessary or desirable to indicate that a particular feature is not to scale, the dimension should be underlined with a straight thick line.

A1.5 Dimensioning Features Various characteristics and features of printed board require unique methods of dimensioning.

A1.5.1 Diameters The diameter symbol precedes all diametrical values. Where the diameters of a number of concentric cylindrical features are specified, such diameters should be dimensioned in a longitudinal view if practical.

A1.5.2 Radii Each radius value is preceded by the appropriate radius symbol. A radius dimension line uses one arrowhead, at the arc end. An arrowhead is never used at the radius center. Where location of the center is important and space permits, a dimension line is drawn from the radius center with the arrowhead touching the arc, and the dimension is placed between the arrowhead and the center. Where space is limited, the dimension line is extended through the radius center. Where it is inconvenient to place the arrowhead between the radius center and the arc, it may be placed outside the arc with a leader. Where the center of a radius is not dimensionally located, the center shall not be indicated (see Figure A-18).

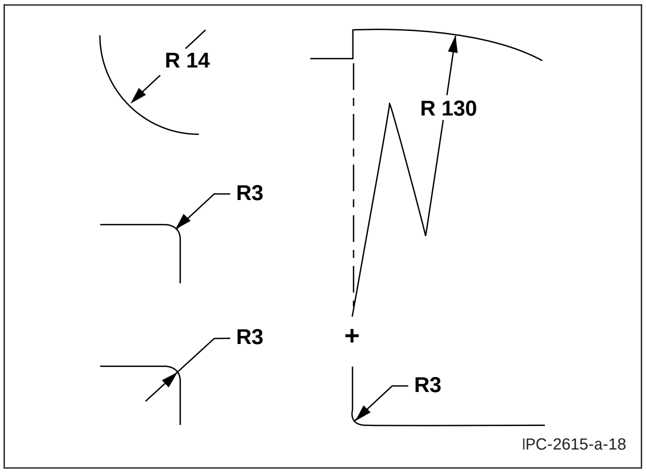


Figure A-18 Radii

A1.5.2.1 Dimensions Given to the Center of a Radius Where a dimension is given to the center of a radius, a small cross is drawn at the center. Extension lines and dimension lines are used to locate the center (see Figure A-19). Where location of the center is unimportant, the drawing must clearly show that the arc location is controlled by other dimensioned features such as tangent surfaces (see Figure A-20).

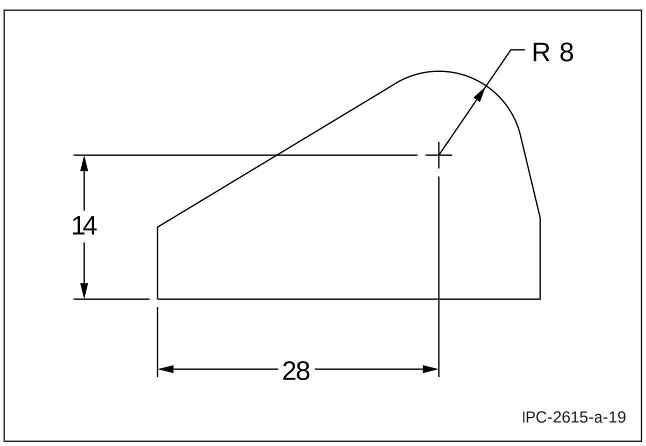


Figure A-19 Radius With Locating Center

A1.5.2.2 Center of a Radius Outside or Interfering with the Drawing Where the center of a radius is outside the drawing or interferes with another view, the radius dimension line may be foreshortened (see Figure A-20). That portion of the dimension line extending from the arrowhead is radial relative to the arc. Where the radius dimension line is foreshortened and the center located by coordinate dimensions, the dimension line locating the center is also foreshortened.

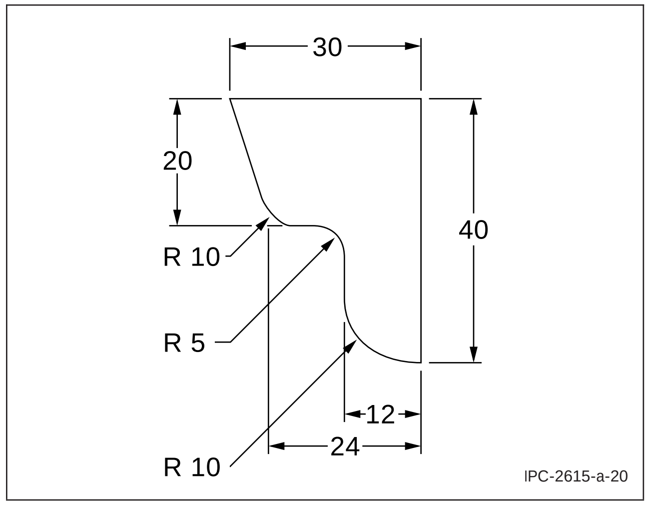


Figure A-20 Radii With Unlocated Center

A1.5.2.3 Multiple Radii of the Same Dimension Where a printed board has a number of radii of the same dimension, a note may be used instead of dimensioning each radius separately.

A1.5.3 Chords, Arcs, and Angles The dimensioning of chords, arcs, and angles shall be as shown in Figure A-21.

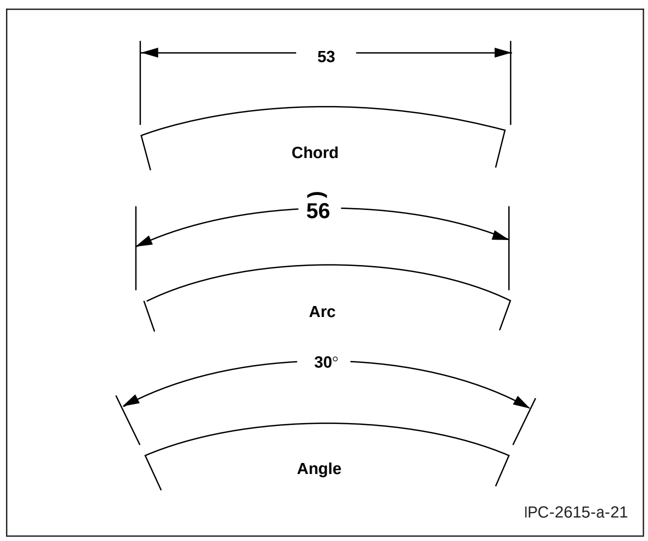


Figure A-21 Dimensioning Chords, Arcs, and Angles

A1.5.4 Rounded Ends Overall dimensions are used for printed boards having rounded ends. For fully rounded ends, the radii are indicated but not dimensioned (see Figure A-22). For printed board with partially rounded ends, the radii are dimensioned (see Figure A-23).

A1.5.5 Rounded Corner Where corners are rounded, dimensions define the edges and the arcs are tangent (see Figure A-24).

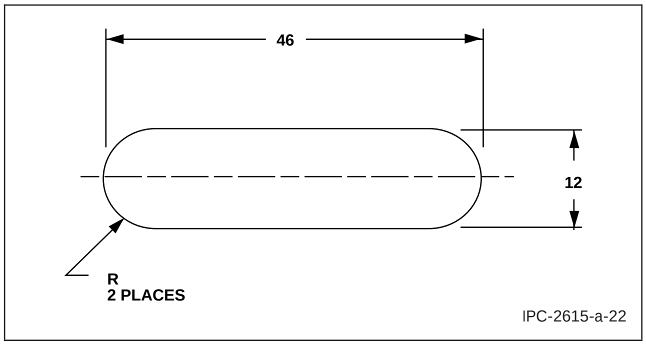


Figure A-22 Fully Rounded Ends

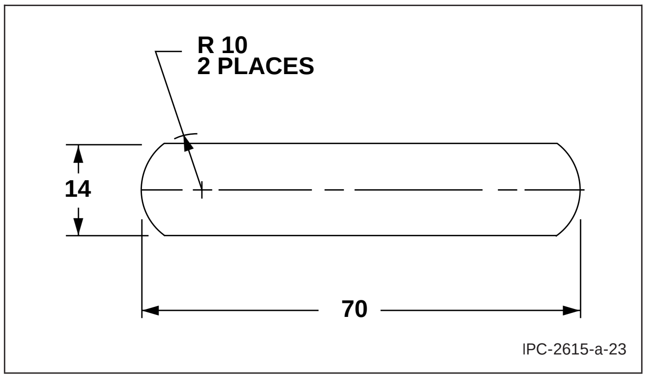


Figure A-23 Partially Rounded Ends

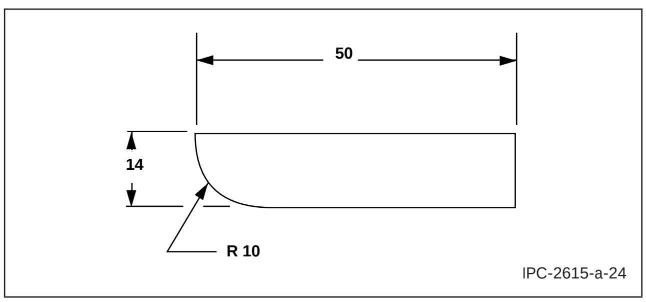


Figure A-24 Rounded Corners

A1.5.6 Outlines Consisting of Arcs A curved outline composed of two or more arcs is dimensioned by giving the radii of all arcs and locating the necessary centers with coordinate dimensions. Other radii are located on the basis of their points of tangency (see Figure A-25).

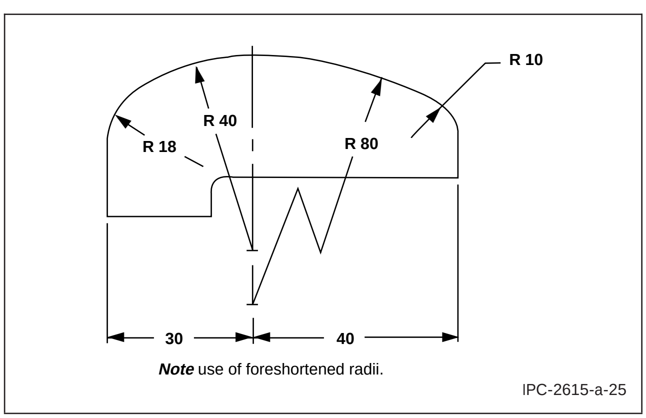


Figure A-25 Circular Arc Outline

A1.5.7 Irregular Outlines Irregular outlines may be dimensioned as shown in Figures A-26 and A-27.

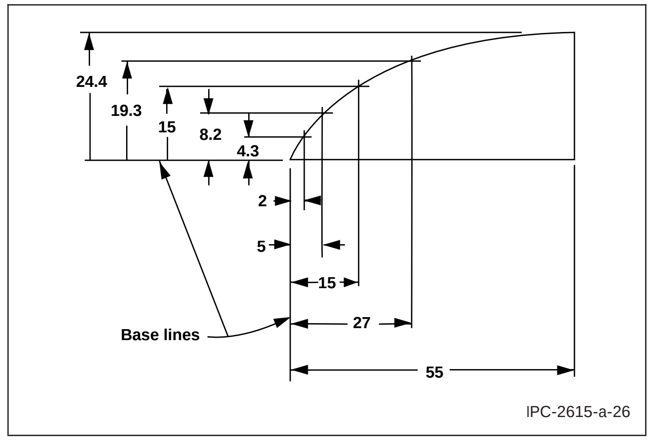
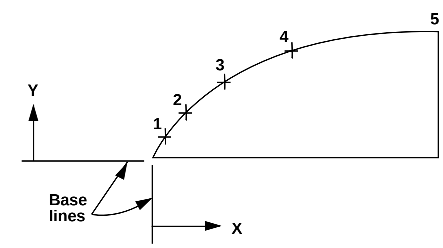


Figure A-26 Coordinate or Offset Outline



Station	1	2	3	4	5
X	2	5	15	27	55
Y	4.3	8.2	15	19.3	24.4

IPC-2615-a-27

Figure A-27 Tabulated Outline

A1.5.7.1 Circular or Non-Circular Outlines Circular or noncircular outlines may be dimensioned by the rectangular coordinate or offset method (see Figure A-26). Coordinates are dimensioned from base lines. Where many coordinates are required to define an outline, the vertical and horizontal coordinate dimensions may be tabulated, as in Figure A-27.

A1.5.7.2 Curved Patterns Curved patterns may be defined by a grid system with numbered grid lines.

A1.5.8 Round Holes Round holes are dimensioned as shown in Figure A-28. Where it is not clear that a hole goes through, the abbreviation THRU follows a dimension. The depth dimension of a blind hole is the depth of the full diameter from the surface of the printed board. Where a blind hole is also counterbored or counterdrilled, the depth dimension applies from the outer surface. For clarity, it is recommended that a fully dimensioned, side view detail be provided.

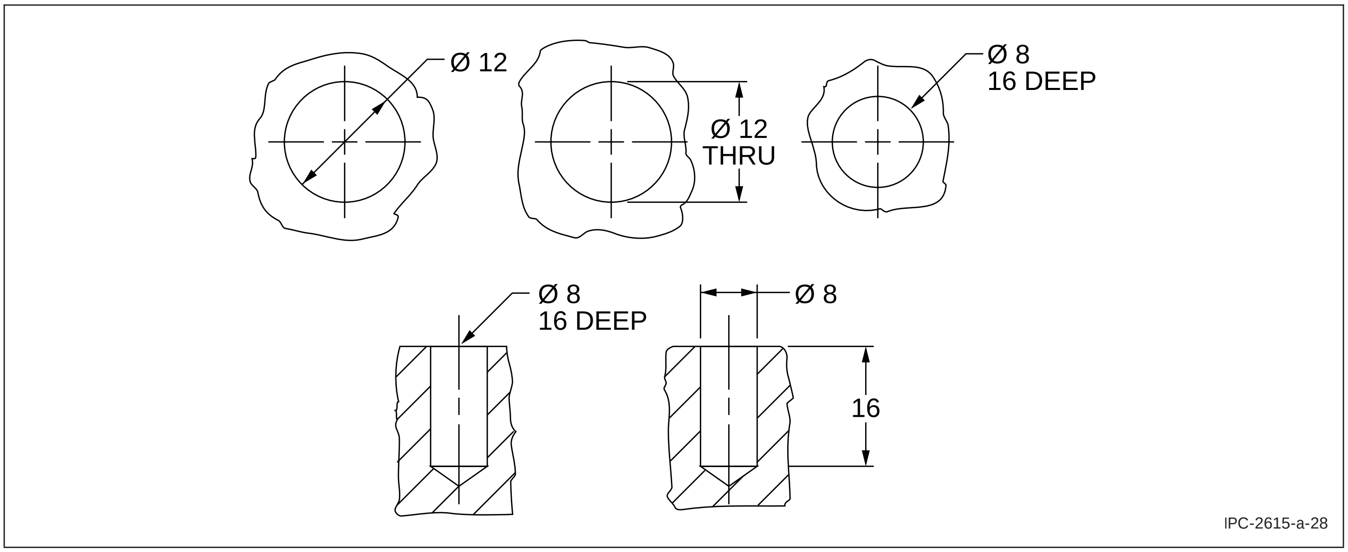


Figure A-28 Round Holes

A1.5.9 Slotted Holes Slotted holes are dimensioned as shown in Figure A-29. The end radii are indicated but not dimensioned.

A1.5.12 Chamfers Chamfers are dimensioned by an angle and a linear dimension, or by two linear dimensions (see Figure A-30). Where an angle and a linear dimension are specified, the linear dimension is the distance from the indicated surface of the printed board to the start of the chamfer. A note may be used to specify 45° chamfers. This method is used only with 45° chamfers, as the linear value applies in either the longitudinal or radial direction. Where chamfers are required for surfaces intersecting at other than right angles, the methods shown in Figure A-31 are used.

A1.5.13 Edge Card Connector Dimensions Edge card connectors are dimensioned by length, width, and location to the critical mating feature (see Figure A-32). Tolerances for card edges and keying slots shall be such that keying slots do not cut into or damage contact finger.

A1.5.14 Surface Texture Methods of specifying surface texture requirements are covered in ANSI Y14.36. For additional information, see ANSI B46.1.

A1.2 Location of Features Rectangular coordinate or polar coordinate dimensions locate features with respect to one another and, as a group or individually, from a datum or an origin. The features that establish this datum or origin must be identified. See 6.2.3. Round holes or other features of symmetrical contour are located by giving distances or distances and directions to the features' centers (see Figures A-33 through A-35).

A1.2.1 Rectangular Coordinate Dimensioning Where rectangular coordinate dimensioning is used to locate features, linear dimensions specify distances in coordinate

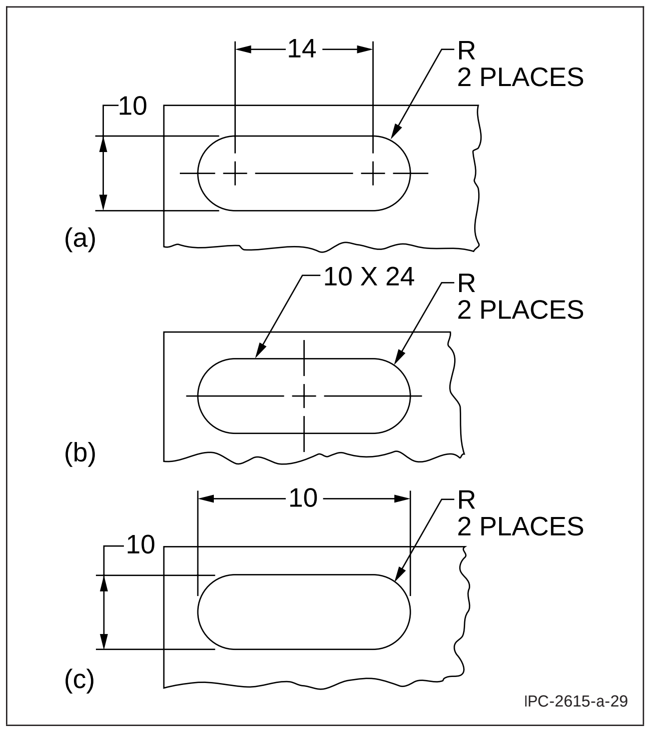


Figure A-29 Slotted Holes

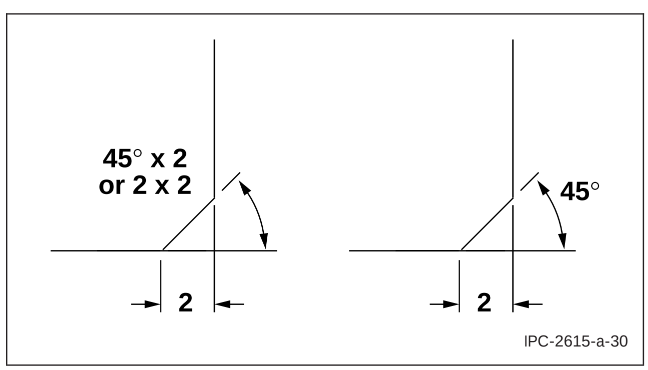


Figure A-30 Equalized Chamfers

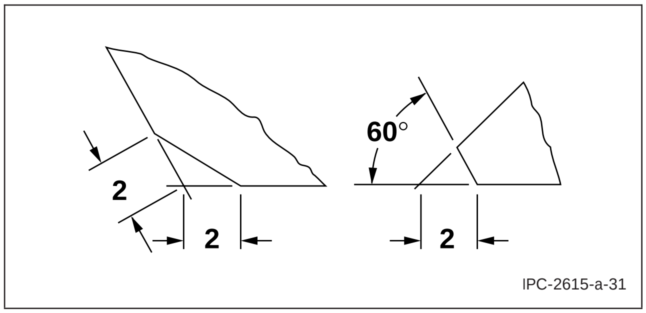


Figure A-31 Chamfers at Other Than 90°

directions from two or three mutually perpendicular planes (see Figure A-33). Coordinate dimensioning must clearly indicate which features of the printed board establish these planes.

A1.2.2 Rectangular Coordinate Dimensioning Without Dimension Lines Dimensions may be shown on extension lines without the use of dimension lines or arrowheads. The base lines are indicated as zero coordinates, or they may be labelled as X, Y, and occasionally Z (see Figure A-34).

A1.2.3 Tabular Dimensioning Tabular dimensioning is a type of rectangular coordinate dimensioning in which dimensions from mutually perpendicular planes are listed in a table on the drawing rather than on the pictorial delineation. This method is used on drawings that require the location of a large number of similarly shaped features. Tables are prepared in any suitable manner that adequately locates the features.

A1.2.4 Polar Coordinate Dimensioning Where polar coordinate dimensioning is used to locate features, a linear and an angular dimension specify a distance from a fixed point at an angular direction from two or three mutually perpendicular planes. The fixed point is the intersection of these planes. Components shall not be located on polar grids because of manufacturing equipment incompatibility (see Figure A-35).

A1.2.5 Repetitive Features or Dimensions Repetitive features or dimensions may be specified by the use of an ''X'' in conjunction with a numeral to indicate the ''number of times'' or ''places'' required.

A1.2.5.1 Repeated Features Features, such as holes and slots which are repeated in a series or pattern, may be specified by giving the required number of features and an ''X,'' followed by the size dimension of the feature. A space is used between the ''X'' and the dimension (see Figure A-36).

A1.2.5.2 Equal Spacing of Features Equal spacing of features in a series or pattern may be specified by giving the required number of spaces and an ''X,'' followed by the

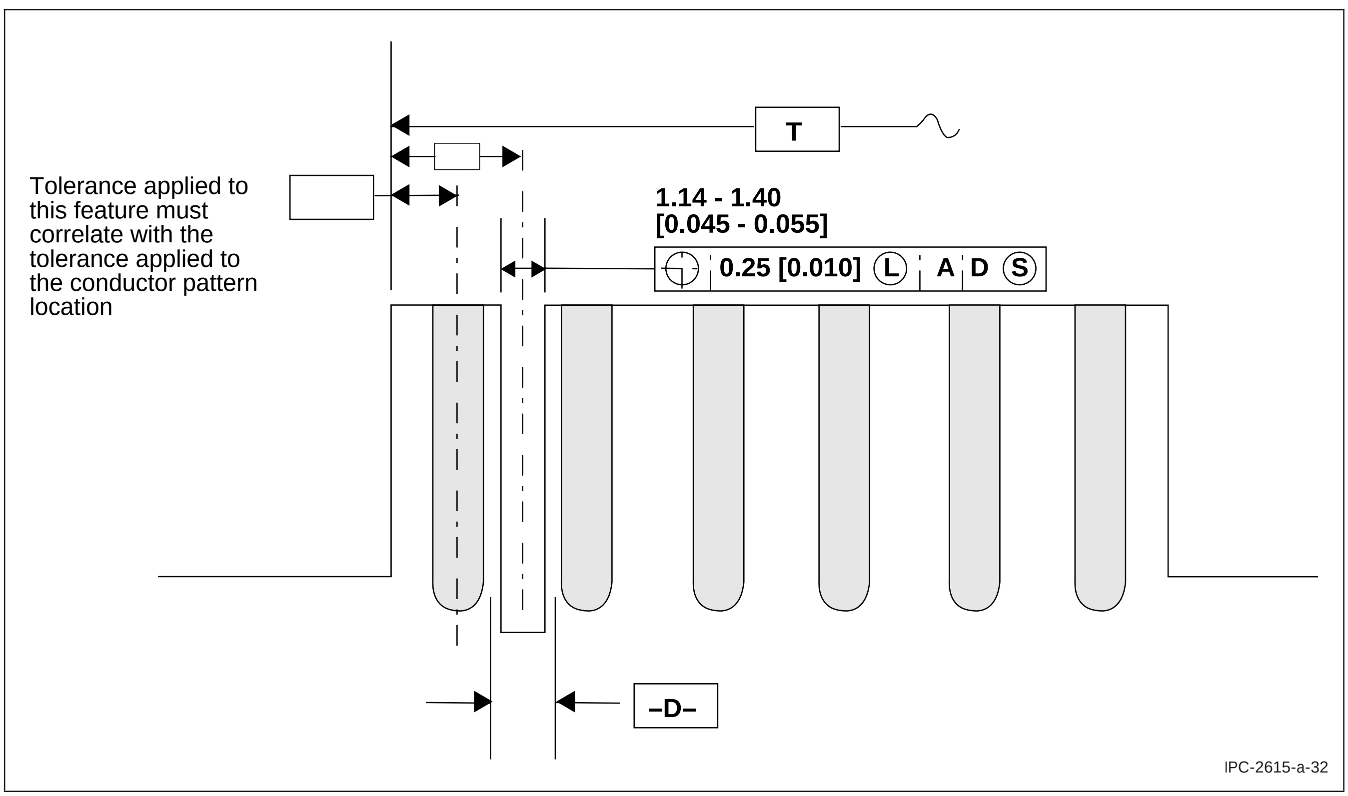


Figure A-32 Edge Card Connector

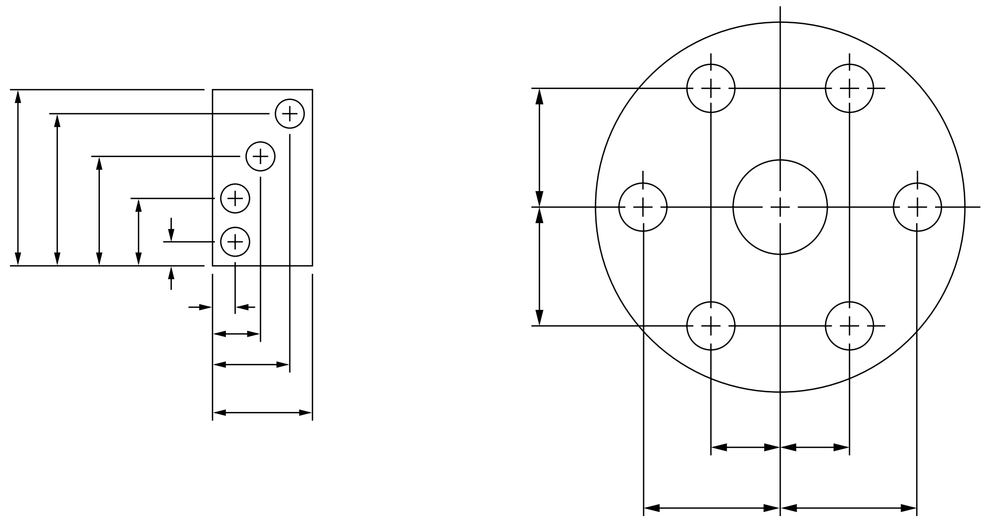
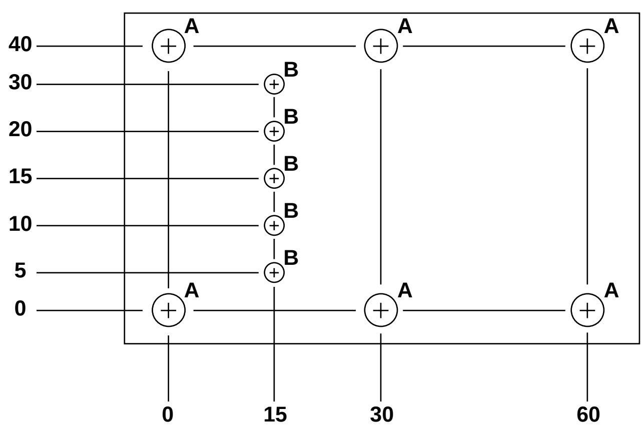


Figure A-33 Rectangular Coordinate Dimension



Hole List		
Symbol	**Qty.**	**Dia.**
A	**6**	**.20 – .25**
B	**5**	**.10 – .15**

IPC-2615-a-34

Figure A-34 Rectangular Coordinate Dimensions Without Dimension Lines

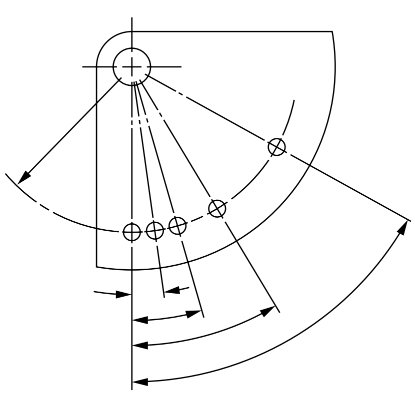


Figure A-35 Polar Coordinate Dimensions

applicable dimension. A space is used between the ''X'' and the dimension (see Figure A-37). Where it is difficult to distinguish between the dimension and the number of spaces, one space is dimensioned and identified as reference.

A1.2.6 Use of ''X'' to Indicate ''BY'' ''X'' may be used to indicate ''BY'' between coordinate dimensions. In such cases, the ''X'' shall be preceded and followed by one character space.

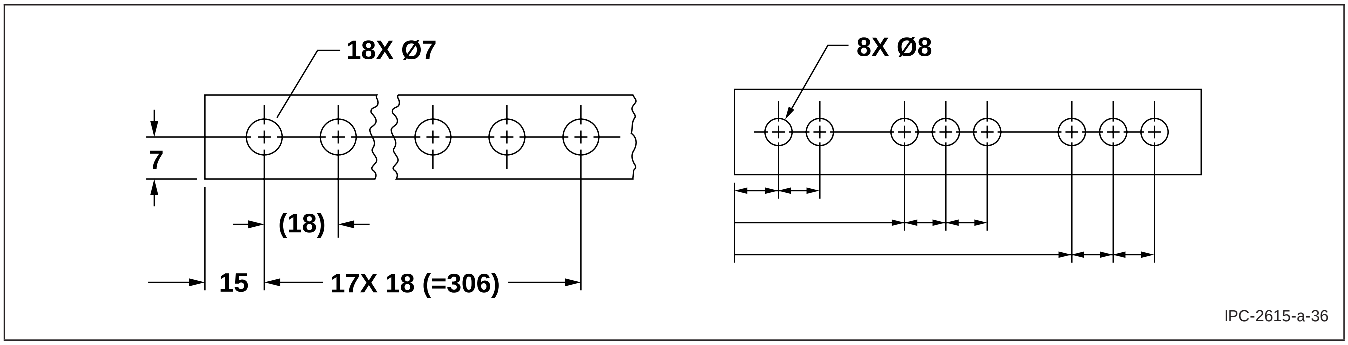


Figure A-36 Repetitive Features and Dimensions

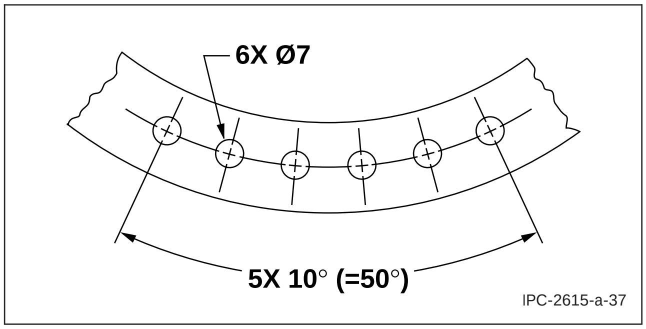


Figure A-37 Equal Spacing of Feature

Appendix B: GENERAL TOLERANCING AND RELATED PRINCIPLES

B1 General This Section establishes practices for expressing tolerances on linear and angular dimensions, applicability for material condition modifiers, and interpretations governing limits and tolerances.

B1.2 Application of Tolerances Tolerances may be expressed as follows:

a) as direct limits or as tolerance values applied directly to a dimension;

b) as a geometric tolerance,

c) in a note referring to specific dimensions;

d) as specified in other documents referenced on the drawing for specific features or processes;

e) in a general tolerance block referring to all dimensions on a drawing for which tolerances are not otherwise specified; see ANSI Y14.1.

B1.2.1 Tolerances on Feature Location Tolerances on dimensions that locate features of size may be applied directly to the locating dimensions or specified by the positional tolerancing method described in (Section 5).

B1.2.2 Angular Tolerances Within General Tolerance Notes Unless otherwise specified, where a general tolerance note on the drawing includes angular tolerances, it applies to features shown at specified angles and at implied 90°angles.

B1.3 Direct Tolerancing Methods Limits and directly applied tolerance values are specified as follows.

a) Limit Dimensioning. The high limit (maximum value) is placed above the low limit (minimum value). When expressed in a single line, the low limit precedes the high limit and a dash separates the two values (see Figure B-1).

b) Plus and Minus Tolerancing. The dimension is given first and is followed by a plus and minus expression of tolerance (see Figure B-2).

B1.4 Tolerance Expression The conventions pertaining to the number of decimal places carried in the tolerance shown in the following paragraphs shall be observed.

B1.4.1 Millimeter Tolerances Where millimeter dimensions are used on the drawings, the following applies.

a) Where unilateral tolerancing is used and either the plus or minus value is nil, a single zero is shown without a plus or minus sign.

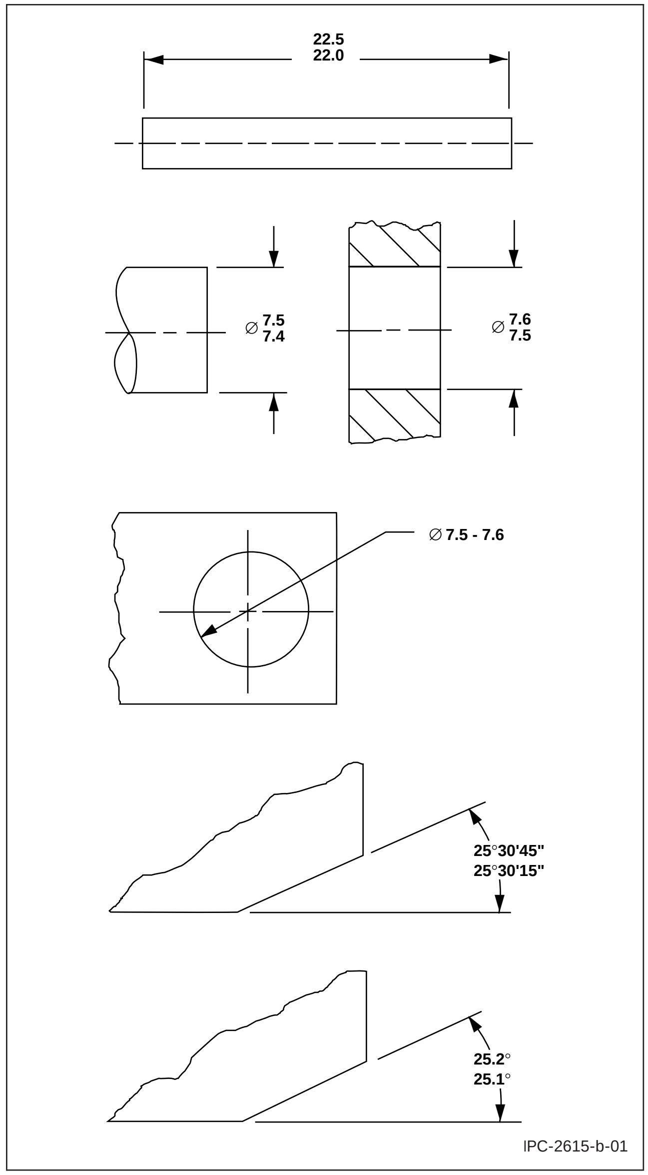


Figure B-1 Limit Dimensions

EXAMPLE:

$32^{\;0}_{-0.02}$ *or* $32^{+0.02}_{\;0}$

b) Where bilateral tolerancing is used, both the plus and minus values have the same number of decimal places, using zeros where necessary.

EXAMPLE:

$32^{+0.25}_{-0.10}$ *not* $32^{+0.25}_{-0.1}$

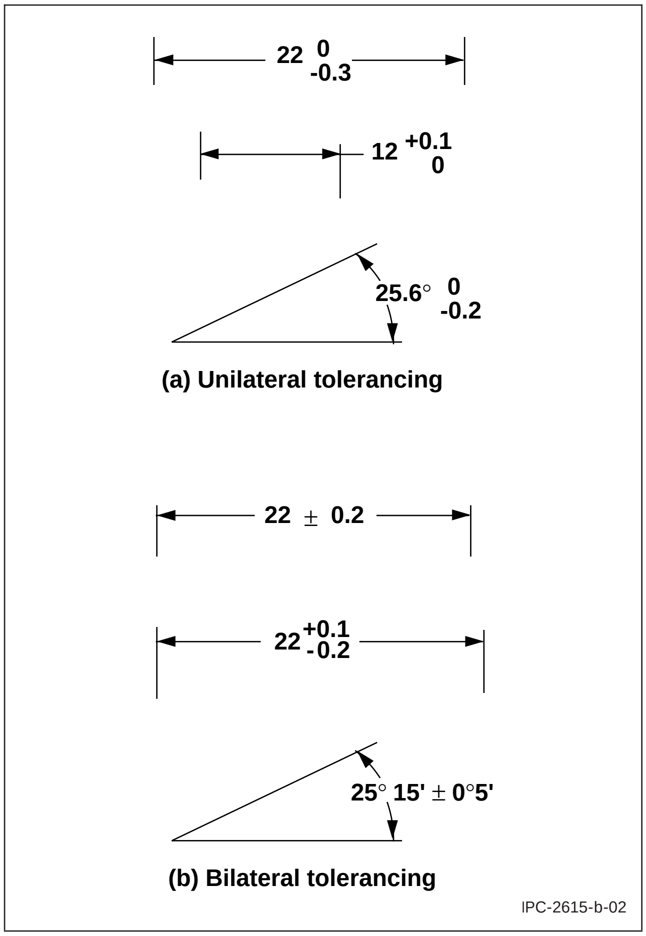


Figure B-2 Plus or Minus Tolerances

c) Where limit dimensioning is used and either the maximum or minimum value has digits following a decimal point, the other value has zeros added for uniformity.

EXAMPLE:

25.45		25.45
	not	
25.00		25

B1.4.2 Inch Tolerances Where inch dimensions are used on the drawing, both limit dimensions or the plus and minus tolerance and its dimension shall be expressed with the same number of decimal places.

EXAMPLES:

.750		.75
	not	
.748		.748

B1.5 Interpretation of Limits

B1.5.1 Acceptability When Limiting Values Are Specified Specified limiting values of 63.5 mm [2.5 in.] maximum, 63.50 mm [2.50 in.] maximum, 63.500 mm [2.500 in.] maximum are taken to mean that, for the purposes of determining conformance to this specification, an observed value shall be rounded off to the nearest .1 mm [0.1 in.], 0.01 mm [0.01 in] and .001 mm [0.001 in], respectively, and then compared to the specified limiting value. (See ASTM E29-67).

B1.5.2 Rounding Convention When measurements are made to greater precision than is required by this specification, it becomes necessary to round results in order to determine conformance. The following rounding convention shall be used (See ANSI Z25.1 - 1940):

B1.5.2.1 Rules for Not Changing Last Place Figure The figure in the last place to be retained shall be kept unchanged when the figure in the next place

- is less than 5; or
- is 5 followed by no other figures or only by zeroes and the next figure in the next place is even.

B1.5.2.2 Rules for Increasing by One the Last Place Figure The figure in the last place to be retained shall be increased by 1 when the figure in the next place

- is less than 5; or
- is followed by no other figure or only by zeroes, and the figure in the last place to be retained is odd; or
- is 5 followed by any figure or figures other than zero.

B1.5.2.3 Rounding the Last Figure The final rounded figure shall be obtained from the most precise value available and not from a series of successive roundings.

B1.6 Dimensions for Plated or Coated Printed Boards When the printed board is to be plated or coated, the drawing or referenced document shall specify whether the dimensions are before or after plating. Typical examples of notes are the following:

a) DIMENSIONAL LIMITS APPLY AFTER PLATING.

b) DIMENSIONAL LIMITS APPLY BEFORE PLATING.

c) (For coatings other than plating, substitute the appropriate term.)

B1.7 Single Limits MIN or MAX is placed after a dimension where other elements of the design definitely determine the other unspecified limit. Features such as depth of holes, corner radii, chamfers, etc., may be limited in this way. Single limits are used where the intent will be clear, and the unspecified limit can be zero or approach infinity and will not result in a condition detrimental to the design.

Appendix C: DIMENSIONING FOR COMPUTER-AIDED DESIGN AND MANUFACTURING

C1 General. Industry Acceptance of CAD and CAM General. Industry acceptance of Computer-Aided Design (CAD) and Computer-Aided Manufacturing (CAM) systems for use in component design and fabrication is rapidly accelerating. Collectively, these highly sophisticated systems can be used to describe the desired part as a geometric model, interactively interject manufacturing data, and deliver this information to a designated machine tool for execution of the finished part. Although computer-aided systems continue to require dimensions and tolerances for printed board definition, in many cases the dimensioning is accomplished by means of algorithms which emulate manual dimensioning practices. In view of the changing state-of-the-art, it is important that the designer understand where certain practices can be employed for expressing dimensional requirements most effectively. The purpose of this Appendix is to contribute to that understanding by first iterating the standard coordinate system and then providing guidelines applicable to the CAD/CAM (data base) mode as well as the manual (conventional drawing) mode. This information will assist the designer in developing dimensioning and tolerancing practices common to these modes.

C2 Coordinate System The coordinate system is the same for both the geometric model created by CAD and the graphic definition found on conventional drawings. It is the standard system of rectangular or Cartesian coordinates wherein a point is located by its distance from each of two or three mutually perpendicular intersecting planes. Two-dimensional coordinates (in X and Y directions) locate points on a plane, while three-dimensional coordinates (in X, Y, and Z directions) locate points in space. Once a geometric model is defined, it is the basis for interactive programming of commands for the machine tool to execute the required relative motion between cutting tool and workpiece. For CAM usage, dimensional coordinates translate into point locations relative to coordinate axes since linear and rotary motion is involved.

C3 Reference Planes For CAD, three mutually perpendicular planes are established from which a geometric model of the desired printed board can be constructed. These planes normally coincide with the exterior outline of printed boards having surfaces at right angles. Where the printed board is circular, two of these planes intersect along the axis of the cylinder and the third is perpendicular to it. When viewed from above, as in the top view of Figure C-l, these planes are oriented in accordance with the following.

a) The first plane lies in the plane of projection. It is the plane from which coordinate distances are specified-in the Z direction.

b) The second plane is horizontal and perpendicular to the first. It is the plane from which coordinate distances are specified in the Y direction.

c) The third plane is vertical and perpendicular to the other two. It is the plane from which coordinate distances are specified in the X direction.

C4 Reference Axes For CAM, three mutually perpendicular axes are established along which linear and rotary motions occur in the machine tool used for producing the desired printed board. Generally, these axes are designated as the basic coordinate axes of the equipment. Additional (secondary) axes may also be designated, depending on machine capability and printed board configuration. The basic axes, when viewed from above, or the top/primary side. are oriented in accordance with the following.

a) The first axis is horizontal in the plane of projection. It is the X axis of motion.

b) The second axis is vertical in the plane of projection and perpendicular to the X axis. It is the Y axis of motion.

c) The third axis is perpendicular to the plane of projection and perpendicular to the X and Y axes. It is the Z axis of motion.

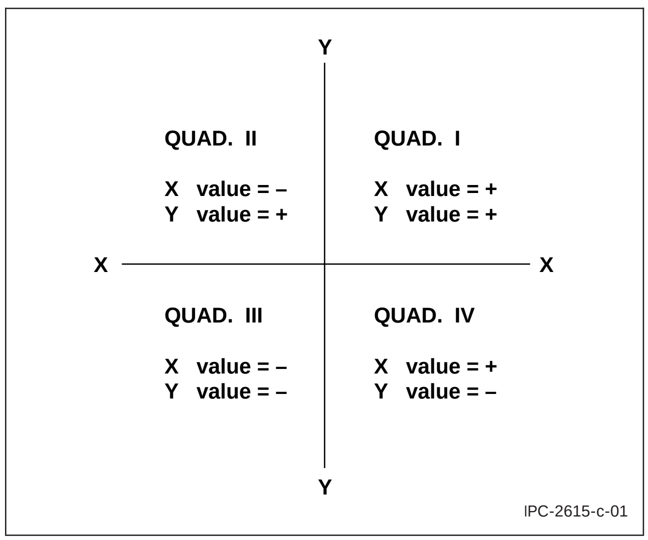


Figure C-1 Mathematical Quadrants

C5 Mathematical Quadrants The intersection of the X axis and Y axis forms quadrants described in Figure C-1. These axes are normally aligned or coincident with appropriate surfaces or features on the desired printed board. When programming commands for the machine tool, the workpiece should be positioned in a quadrant in such a way that a maximum of positive values will result. For example, if the workpiece is positioned in Quadrant 1, positive values will result. If the workpiece is positioned in two or more quadrants, positive and negative values will result, and the potential for error is greater. This precaution is generally not necessary when programming on the computer, but helpful when programming without computer assistance. The considerations described above also apply to quadrants formed by intersections of the X-Z and Y-Z axes.

C6 Dimensioning and Tolerancing

C6.1 Locating and Tolerancing of Patterns Produced in A Common Fabrication Operation To maximize producibility, a good practice is to separately locate and tolerance as patterns those features that are produced in the same fabrication operations. Applicable patterns are as follows:

a) *Unplated-Through Hole Patterns* Two or more of these holes are typically used as datum features.

b) *Plated-Through Hole Patterns* The plated-through hole pattern is generally the first drilling operation and as such is the first operation to define the printed board. The holes are typically located on a basic grid. The hole location tolerance is specified either on the hole list, by drawing note, or in a separate specification. The basic grid must be located in relation to the datum reference frame.

c) *Conductor Pattern* The conductor pattern does not need a separate datum reference frame provided a minimum annular ring is specified. The fabrication allowance in essence becomes the location tolerance for the conductor pattern. Fiducials may also be used to locate the conductor pattern. The centerlines of two orthogonal conductors may also be used to locate the conductor pattern (see Figure C-2).

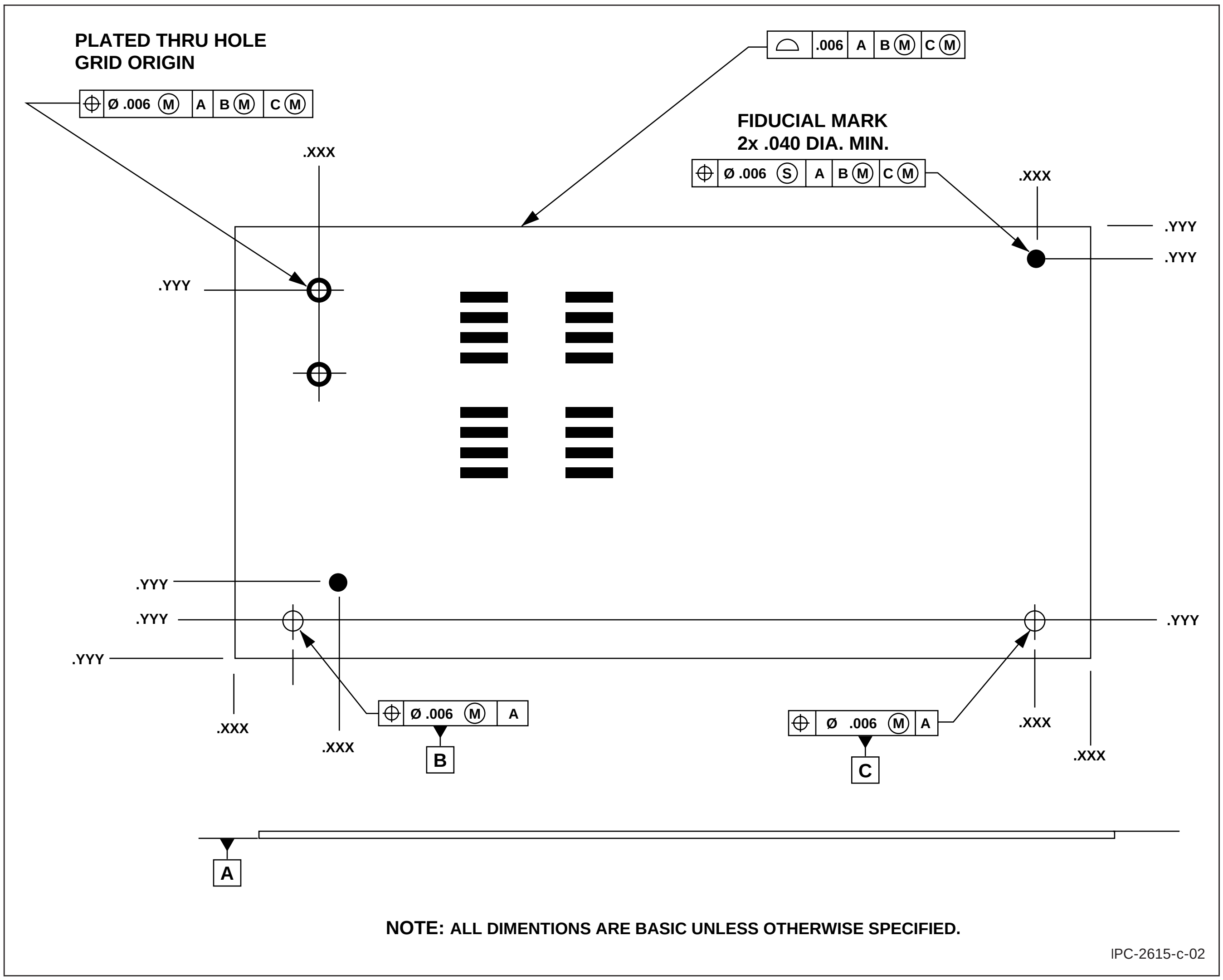


Figure C-2 Locating A Circuit Pattern Using Fiducials Relative to Plated-Through Holes

d) *Printed Board Profile* The printed board profile, including cutouts and slots, require a minimum of one datum reference to the primary datum. The use of three datum references maximizes producibility and allows use of hard tool gauging.

e) *Solder Resist Coatings* The solder resist coating may be located by specifying a minimum land clearance or targets may be provided which serve the same function as fiducials do for the conductor pattern. A minimum land clearance serves the same function as a minimum annular ring requirement in that it tolerances the solder-mask pattern location with respect to the conductor pattern.

C6.2 Recommended Guidelines for Dimensioning and Tolerancing PCBs Recommended guidelines for dimensioning and tolerancing practices for use in defining printed board for the CAD/CAM mode are as follows.

a) Major features of a printed board are used to establish the basic coordinate system for initial printed board definition. These features may or may not be subsequently identified as datum features.

b) For final printed board definition, any number of sub-coordinate systems may be used to locate and orient features of a printed board. These systems, however, must be geometrically related to the basic coordinate system of the given printed board.

c) Define printed board features in relation to three mutually perpendicular reference planes. Establish these planes along features that parallel the axes and motions of CAM equipment, wherever possible.

d) The assignment of datum features is based primarily on the functional requirements of the printed board.

e) Dimension the printed board so that its geometric shape is completely defined and mathematically precise.

f) Regular geometric profiles such as ellipses, parabolas, hyperbolas, etc., may be defined on the drawing by mathematical formulas. CAM equipment can be programmed to generate these profiles by linear interpolation, that is, a series of short straight lines whose end points are spaced close enough to approximate the desired profile within the specified profile tolerance.

g) A printed board surface whose profile is defined on the drawing by a mathematical formula should not be coordinately dimensioned unless specific dimensions are required for inspection or identified as reference information.

h) For arbitrary profiles, the drawing should specify appropriate points on the profile by coordinate dimensions. or provide a table of coordinates. When determining the number of points needed to define the profile. keep in mind that the tighter the tolerance or the smaller the radius of curvature, the closer together the points should be. Terms such as ‘‘blend smoothly’’ and ‘‘faired curve’’ are not specified.

i) Profiles may also be defined by other coordinate systems, such as polar or cylindrical, as applicable. However, it is desirable to use the same coordinate system on a given drawing.

j) Any change in profile (points of inflection or tangency) should be clearly defined, with prime consideration given to design intent. Precise continuity of the profile is necessary for CAD.

k) A circular pattern of holes may be defined by polar coordinate dimensions. Location and orientation of the pattern must be clearly shown.

l) Where possible, express angular dimensions in degrees and decimal parts of a degree.

m) Where plus and minus tolerancing is used, the tolerance should be bilateral and not unilateral. Equal plus and minus values are preferred. Positional tolerancing is recommended for locating features of size.

n) Geometric tolerances are specified in all cases where the control of specific geometric characteristics of printed board features is required. Where applicable, identifying datum features on the drawing and referencing them in an order of precedence will clearly indicate their usage for CAM.

o) Tolerances should be specified on the basis of actual design requirements. The accuracy capability of CAM equipment is not a basis for specifying more restrictive tolerances than are functionally required.

ASSOCIATION CONNECTING ELECTRONICS INDUSTRIES

ANSI/IPC-T-50 Terms and Definitions for Interconnecting and Packaging Electronic Circuits Definition Submission/Approval Sheet

The purpose of this form is to keep current with terms routinely used in the industry and their definitions. Individuals or companies are invited to comment. Please complete this form and return to:

IPC
2215 Sanders Road
Northbrook, IL 60062-6135
Fax: 847 509.9798

SUBMITTOR INFORMATION:

Name: ______________________________

Company: ______________________________

City: ______________________________

State/Zip: ______________________________

Telephone: ______________________________

Date: ______________________________

❑ This is a **NEW** term and definition being submitted.
❑ This is an **ADDITION** to an existing term and definition(s).
❑ This is a **CHANGE** to an existing definition.

Term	**Definition**

If space not adequate, use reverse side or attach additional sheet(s).

Artwork: ❑ Not Applicable ❑ Required ❑ To be supplied
❑ Included: Electronic File Name: ______________________________

Document(s) to which this term applies: ______________________________

Committees affected by this term: ______________________________

Office Use

IPC Office	**Committee 2-30**
Date Received: ____________	Date of Initial Review: ____________
Comments Collated: ____________	Comment Resolution: ____________
Returned for Action: ____________	Committee Action: ❑ Accepted ❑ Rejected
Revision Inclusion: ____________	❑ Accept Modify

IEC Classification

Classification Code • Serial Number

Terms and Definition Committee Final Approval Authorization:

Committee 2-30 has approved the above term for release in the next revision.

Name: ____________ Committee: IPC 2-30 Date: ____________

This Page Intentionally Left Blank

Technical Questions

The IPC staff will research your technical question and attempt to find an appropriate specification interpretation or technical response. Please send your technical query to the technical department via:

tel 847/509-9700 fax 847/509-9798
www.ipc.org e-mail: answers@ipc.org

IPC World Wide Web Page www.ipc.org

Our home page provides access to information about upcoming events, publications and videos, membership, and industry activities and services. Visit soon and often.

IPC Technical Forums

IPC technical forums are opportunities to network on the Internet. It’s the best way to get the help you need today! Over 2,500 people are already taking advantage of the excellent peer networking available through e-mail forums provided by IPC. Members use them to get timely, relevant answers to their technical questions.

TechNet@ipc.org

TechNet forum is for discussion of technical help, comments or questions on IPC specifications, or other technical inquiries. IPC also uses TechNet to announce meetings, important technical issues, surveys, etc.

ChipNet@ipc.org

ChipNet forum is for discussion of flip chip and related chip scale semiconductor packaging technologies. It is cosponsored by the National Electronics Manufacturing Initiative (NEMI).

ComplianceNet@ipc.org

ComplianceNet forum covers environmental, safety and related regulations or issues.

DesignerCouncil@ipc.org

Designers Council forum covers information on upcoming IPC Designers Council activities as well as information, comment, and feedback on current design issues, local chapter meetings, new chapters forming, and other design topics.

Roadmap@ipc.org

The IPC Roadmap forum is the communication vehicle used by members of the Technical Working Groups (TWGs) who develop the IPC National Technology Roadmap for Electronic Interconnections.

LeadFree@ipc.org

This forum acts as a peer interaction resource for staying on top of lead elimination activities worldwide and within IPC.

ADMINISTERING YOUR SUBSCRIPTION STATUS:

All commands (such as subscribe and signoff) must be sent to listserv@ipc.org. Please DO NOT send any command to the mail list address, (i.e.<mail list> @ipc.org), as it would be distributed to all the subscribers.

Example for subscribing:
To: LISTSERV@IPC.ORG
Subject:
Message: subscribe TechNet Joseph H. Smith

Example for signing off:
To: LISTSERV@IPC.ORG
Subject:
Message: sign off DesignerCouncil

Please note you must send messages to the mail list address ONLY from the e-mail address to which you want to apply changes. In other words, if you want to sign off the mail list, you must send the signoff command from the address that you want removed from the mail list. Many participants find it helpful to signoff a list when travelling or on vacation and to resubscribe when back in the office.

How to post to a forum:

To send a message to all the people currently subscribed to the list, just send to <mail list>@ipc.org. Please note, use the mail list address that you want to reach in place of the <mail list> string in the above instructions.

Example:
To: TechNet@IPC.ORG
Subject: <your subject>
Message: <your message>

The associated e-mail message text will be distributed to everyone on the list, including the sender. Further information on how to access previous messages sent to the forums will be provided upon subscribing.

For more information, contact Hugo Scaramuzza
tel 847/790-5312 fax 847/509-9798
e-mail: scarhu@ipc.org www.ipc.org/html/forum.htm

Education and Training

IPC conducts local educational workshops and national conferences to help you better understand emerging technologies. National conferences have covered Ball Grid Array and Flip Chip/Chip Scale Packaging. Some workshop topics include:

- Printed Wiring Board Fundamentals
- Troubleshooting the PWB Manufacturing Process
- Choosing the Right Base Material Laminate
- Acceptability of Printed Boards
- Designers Certification Preparation
- New Design Standards
- High Speed Design
- Design for Manufacturability
- Design for Assembly

IPC-A-610 Training and Certification Program

"The Acceptability of Electronic Assemblies" (ANSI/IPC-A-610) is the most widely used specification for the PWB assembly industry. An industry consensus Training and Certification program based on the IPC-A-610 is available to your company.

For more information on programs, contact John Riley
tel 847/790-5308 fax 847/509-9798
e-mail: rilejo@ipc.org www.ipc.org

IPC Video Tapes and CD-ROMs

IPC video tapes and CD-ROMs can increase your industry know-how and on the job effectiveness.

For more information on IPC Video/CD Training, contact Mark Pritchard
tel 505/758-7937 ext. 202 fax 505/758-7938
e-mail: markp@ipcvideo.com www.ipc.org
www.ipc.org

IPC Printed Circuits Expo℠

IPC Printed Circuits Expo is the largest trade exhibition in North America devoted to the PWB industry. Over 90 technical presentations make up this superior technical conference.

April 4-6, 2000 **San Diego, California**	**April 3-5, 2001** **Anaheim, California**	**March 26-28, 2002** **Long Beach, California**

Exhibitor information:
Contact: Jeff Naccarato
tel 630/434-7779

Registration information:
tel 847/790-5361
fax 847/509-9798
e-mail: registration@ipc.org
www.ipcprintedcircuitexpo.org

APEX℠ / IPC SMEMA Council Electronics Assembly Process Exhibition & Conference

APEX is the premier technical conference and exhibition dedicated entirely to the PWB assembly industry.

March 14-16, 2000 **Long Beach, California**	**January 16-18, 2001** **San Diego, California**	**Spring 2002** **TBA**

Exhibitor information:
Contact: Mary MacKinnon
tel 847/790-5386

Registration information:
APEX Hotline: tel 877/472-4724
fax 847/790-5361
e-mail: apex2000@ipc.org
www.apex2000.org

How to Get Involved

The first step is to join IPC. An application for membership can be found in the back of this publication. Once you become a member, the opportunities to enhance your competitiveness are vast. Join a technical committee and learn from our industry's best while you help develop the standards for our industry. Participate in market research programs which forecast the future of our industry. Participate in Capitol Hill Day and lobby your Congressmen and Senators for better industry support. Pick from a wide variety of educational opportunities: workshops, tutorials, and conferences. More up-to-date details on IPC opportunities can be found on our web page: www.ipc.org.

For information on how to get involved, contact:
Jeanette Ferdman, Membership Manager
tel 847/790-5309 fax 847/509-9798
e-mail: JeanetteFerdman@ipc.org www.ipc.org

ASSOCIATION CONNECTING ELECTRONICS INDUSTRIES

Application for Site Membership

PLEASE CHECK APPROPRIATE CATEGORY

Thank you for your decision to join IPC. IPC Membership is **site specific,** which means that IPC member benefits are available to all individuals employed at the site designated on the other side of this application.

To help IPC serve your member site in the most efficient manner possible, please tell us what your facility does by choosing the most appropriate member category.

■ **INDEPENDENT PRINTED BOARD MANUFACTURERS**

Our facility manufactures and sells to other companies, printed wiring boards or other electronic interconnection products on the merchant market.

WHAT PRODUCTS DO YOU MAKE FOR SALE?

- ☐ One-sided and two-sided rigid printed boards
- ☐ Multilayer printed boards
- ☐ Flexible printed boards
- ☐ Flat cable
- ☐ Hybrid circuits
- ☐ Discrete wiring devices
- ☐ Other interconnections

Name of Chief Executive Officer/President____

■ **INDEPENDENT PRINTED BOARD ASSEMBLERS EMSI COMPANIES**

Our facility assembles printed wiring boards on a contract basis and/or offers other electronic interconnection products for sale.

- ☐ Turnkey
- ☐ SMT
- ☐ Chip Scale Technology
- ☐ Through-hole
- ☐ Mixed Technology
- ☐ Consignment
- ☐ BGA

Name of Chief Executive Officer/President____

■ **OEM – MANUFACTURERS OF ANY END PRODUCT USING PCB/PCAS OR CAPTIVE MANUFACTURERS OF PCBS/PCAS**

Our facility purchases, uses and/or manufactures printed wiring boards or other electronic interconnection products for our own use in a final product. Also known as original equipment manufacturers (OEM).

IS YOUR INTEREST IN:

- ☐ purchasing/manufacture of printed circuit boards
- ☐ purchasing/manufacturing printed circuit assemblies

What is your company's main product line?

____

■ **INDUSTRY SUPPLIERS**

Our facility supplies raw materials, machinery, equipment or services used in the manufacture or assembly of electronic interconnection products.

What products do you supply?

____

■ **GOVERNMENT AGENCIES/ ACADEMIC TECHNICAL LIAISONS**

We are representatives of a government agency, university, college, technical institute who are directly concerned with design, research, and utilization of electronic interconnection devices. (Must be a non-profit or not-for-profit organization.)

Please be sure to complete both pages of application.

Application for Site Membership

IPC
ASSOCIATION CONNECTING ELECTRONICS INDUSTRIES

Company Name

Street Address

City State Zip Country

Main Phone No. Fax

Primary Contact Name

Title Mail Stop

Phone Fax e-mail

Senior Management Contact

Title Mail Stop

Phone Fax e-mail

Please check one:

❑ $1,000.00 Annual dues for Primary Site Membership (Twelve months of IPC membership begins from the time the application and payment are received)

❑ $800.00 Annual dues for Additional Facility Membership: Additional membership for a site within an organization where another site is considered to be the primary IPC member.

❑ $600.00** Annual dues for an independent PCB/PWA fabricator or independent EMSI provider with annual sales of less than $1,000,000.00. **Please provide proof of annual sales.

❑ $250.00 Annual dues for Government Agency/University/not-for-profit organization

TMRC Membership ❑ Please send me information on Membership in the Technology Marketing Research Council (TMRC)

AMRC Membership ❑ Please send me information for Membership in the Assembly Marketing Research Council (AMRC)

Payment Information

Enclosed is our check for $___________________

Please bill my credit card: (circle one) MC AMEX VISA DINERS

Card No. ____Exp date ______________

Authorized Signature ____

Mail application with check or money order to:

IPC
Dept. 851-0117W
P.O. Box 94020
Palatine, IL 60094-4020

Fax/Mail application with credit card payment to:

IPC
2215 Sanders Road
Northbrook, IL 60062-6135
Tel: 847 509.9700
Fax: 847 509.9798

PLEASE ATTACH BUSINESS CARD OF OFFICIAL REPRESENTATIVE HERE

ASSOCIATION CONNECTING ELECTRONICS INDUSTRIES

Standard Improvement Form

IPC-2615

The purpose of this form is to provide the Technical Committee of IPC with input from the industry regarding usage of the subject standard.

Individuals or companies are invited to submit comments to IPC. All comments will be collected and dispersed to the appropriate committee(s).

If you can provide input, please complete this form and return to:

IPC
2215 Sanders Road
Northbrook, IL 60062-6135
Fax 847 509.9798
answers@ipc.org

1. I recommend changes to the following:

___ Requirement, paragraph number __________

___ Test Method number __________, paragraph number __________

The referenced paragraph number has proven to be:

___ Unclear ___ Too Rigid ___ In Error

___ Other ____

2. Recommendations for correction:

____

____

____

____

3. Other suggestions for document improvement:

____

____

____

____

Submitted by:

Name ________________________ Telephone ________________________

Company ________________________ E-mail ________________________

Address ____

City/State/Zip ________________________ Date ________________________

ASSOCIATION CONNECTING
ELECTRONICS INDUSTRIES

ISBN #1-580982-49-2

2215 Sanders Road, Northbrook, IL 60062-6135
Tel. 847.509.9700 Fax 847.509.9798
www.ipc.org