Technical drawings and their interpretation

ME 297-1
Fall 2011
Eradat
SJSU
Based on notes on Jim Burge and other online resources

Technical drawings

- Models
  - Used for design and analysis
  - SolidWorks, I-DEAS, Pro-E, …
- Component Drawings
  - Used to specify fabrication or procurement of parts
  - AutoCad, Pro-E, Solid Works …
- Assembly Drawings
  - Used to specify assembly of parts
  - AutoCad, Pro-E, SolidWorks …

- **This lecture covers component drawings and tolerances**

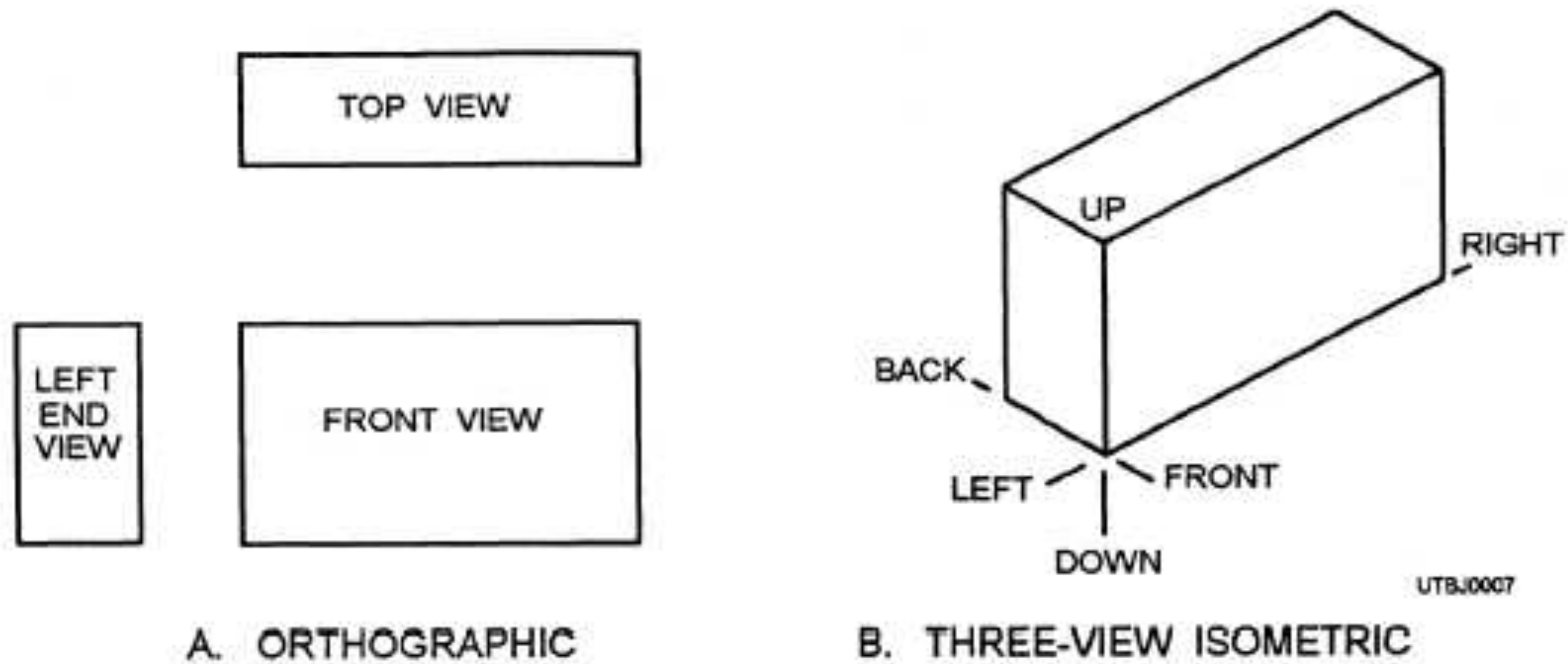
- References

- Earle, J. H., *Engineering Design Graphics* (Addison-Wesley, 1983)
- ASME Y14.5M Dimensioning and tolerancing

Technical drawings

- Orthographic projection
- Isometric layout
- Dimensioning
- Tolerancing

Orthographic vs. isometric drawings



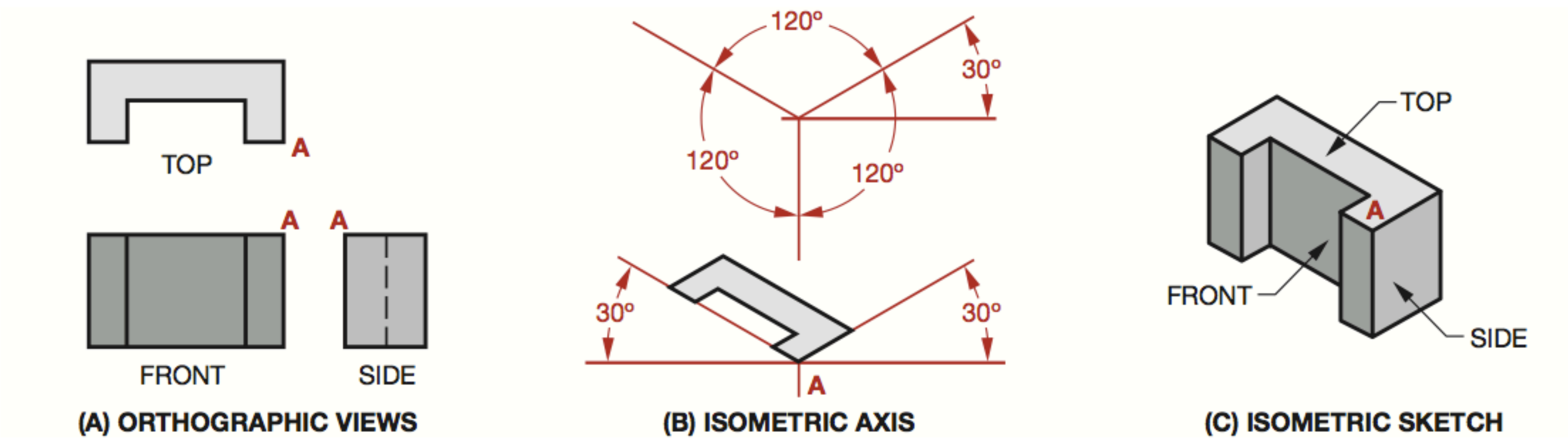
http://webtools.delmarlearning.com/sample_chapters/1418055735_ch07.pdf

Orthographic projections

- A method of projection in which an object is depicted (or a surface is mapped) using parallel lines to project its shape onto a plane.

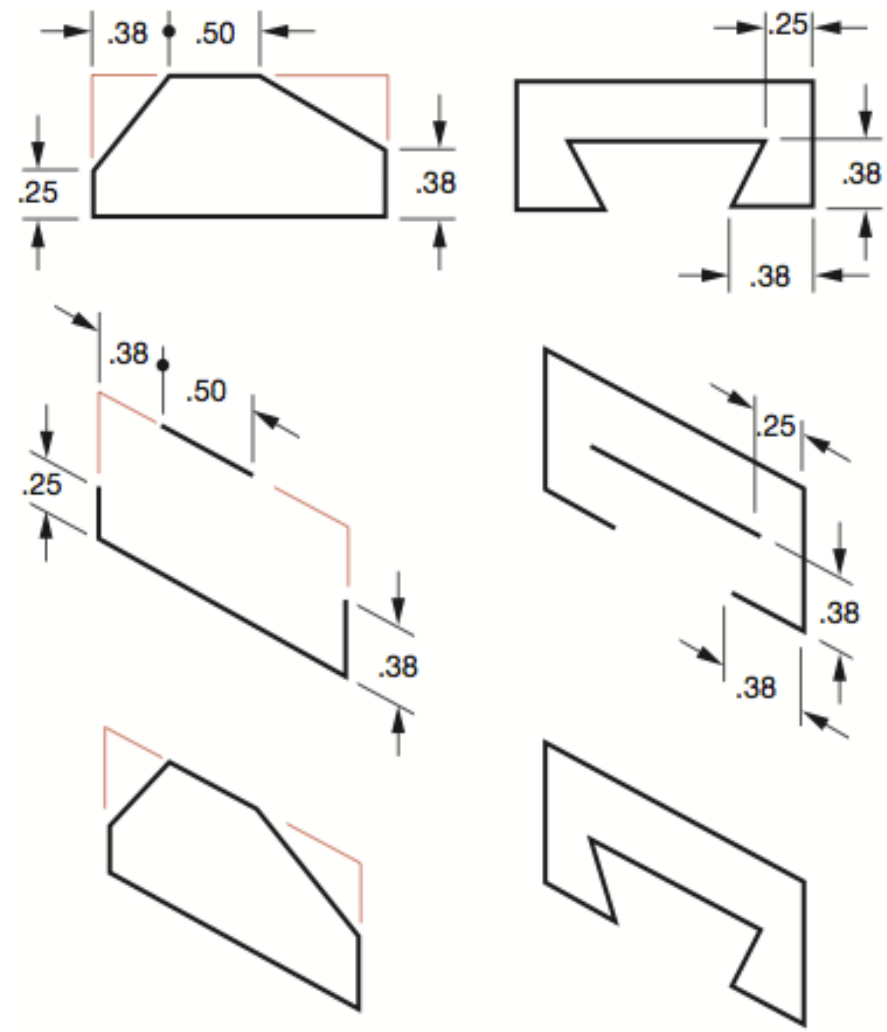
Isometric drawings

- All isometric sketches start by constructing the isometric axes, which includes a vertical line for height and isometric lines to the left and right, at angle angle of 30° from the horizon, for width and depth.
- The three faces seen in the isometric view are the same faces that would be seen in the normal orthographic views: top, front, and side



Inclined surfaces in isometric drawings

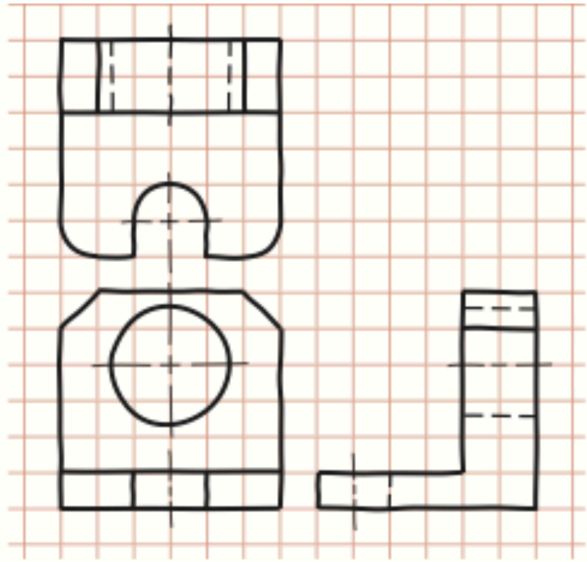
- Many objects have inclined surfaces that are represented by sloping lines in orthographic views.
- In isometric drawings, sloping surfaces appear as non-isometric lines.
- To create them, their endpoints, which are found on the ends of isometric lines, are joined with a straight line.



EXAMPLE A

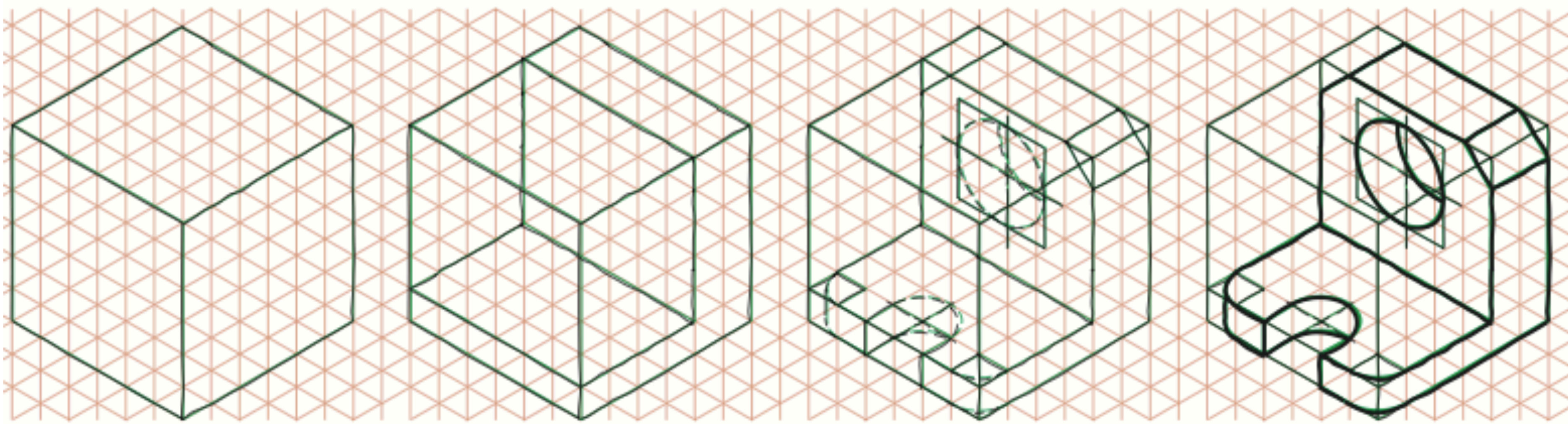
EXAMPLE B

Basic steps for isometric drawing



Isometric grid

(A) THE PART



STEP 1
BUILD THE FRAME

STEP 2
BLOCK IN THE DETAILS

STEP 3
ADD THE DETAILS

STEP 4
DARKEN THE LINES

Cylinders in isometric drawings

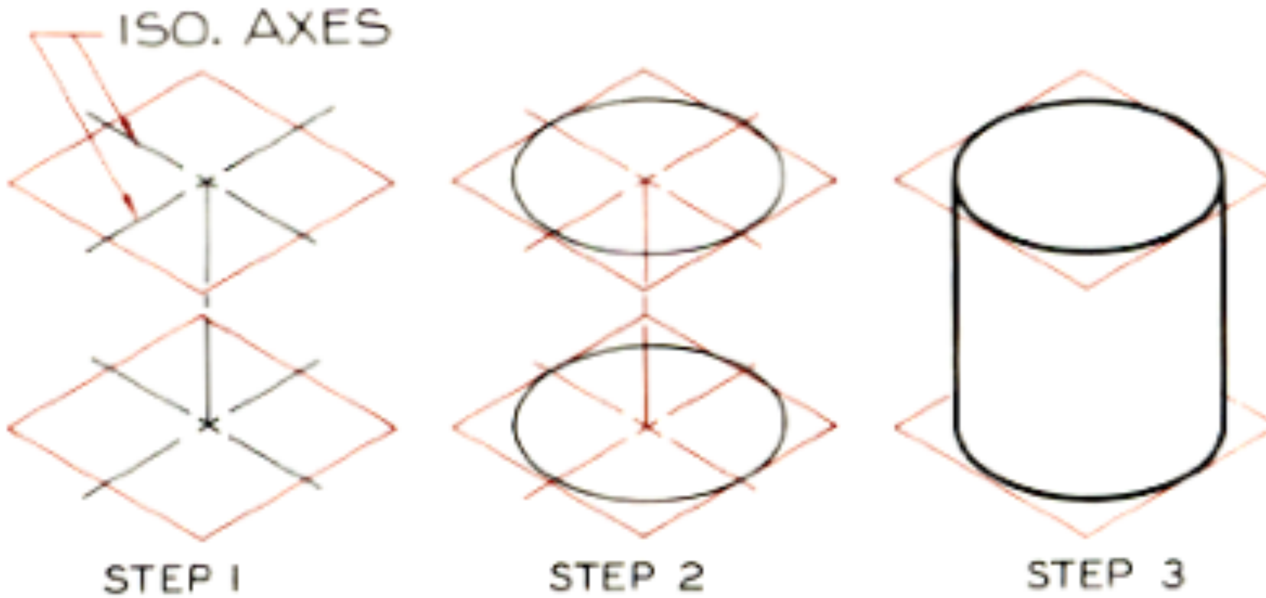


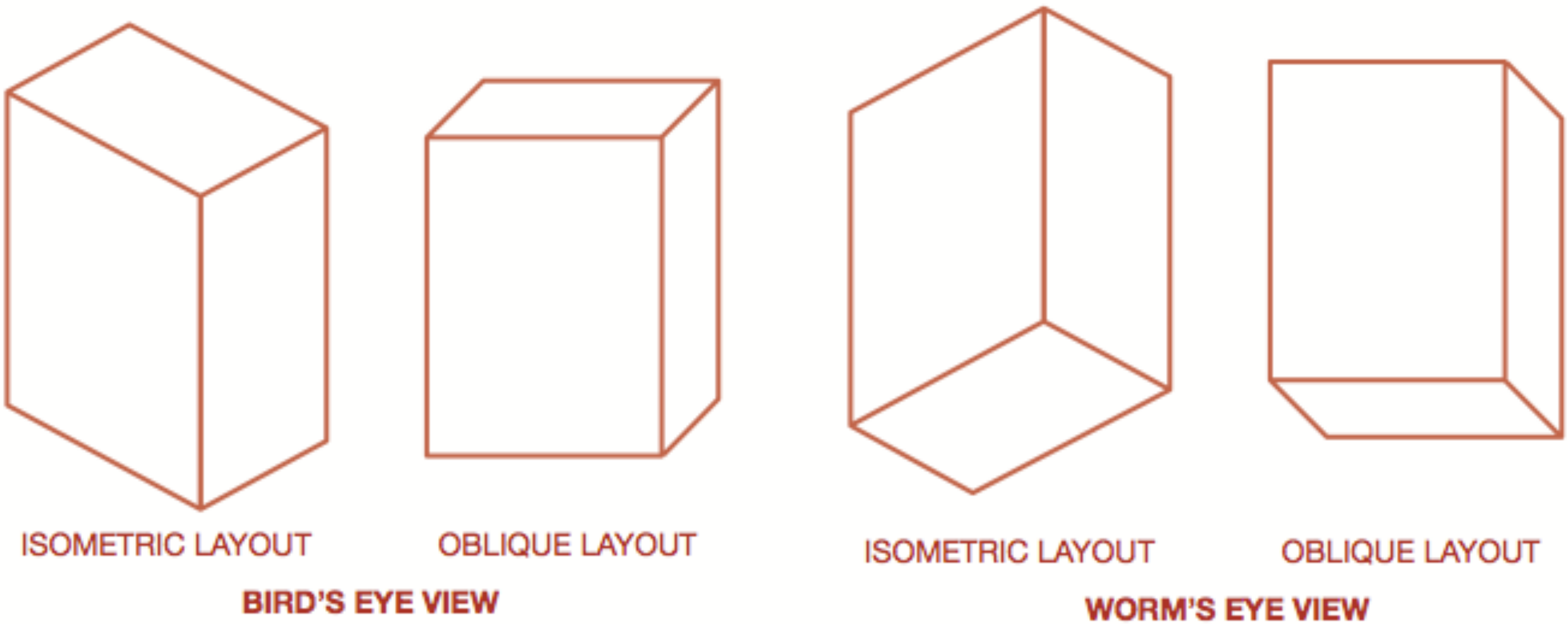
Fig. 25.40 Cylinder: four-center method

Step 1 A rhombus is drawn in isometric at each end of the cylinder's axis.

Step 2 A four-center ellipse is drawn within each rhombus.

Step 3 Lines are drawn tangent to each rhombus to complete the isometric drawing.

Bird's eye vie vs. worm's eye view



Creating an orthographic drawing from

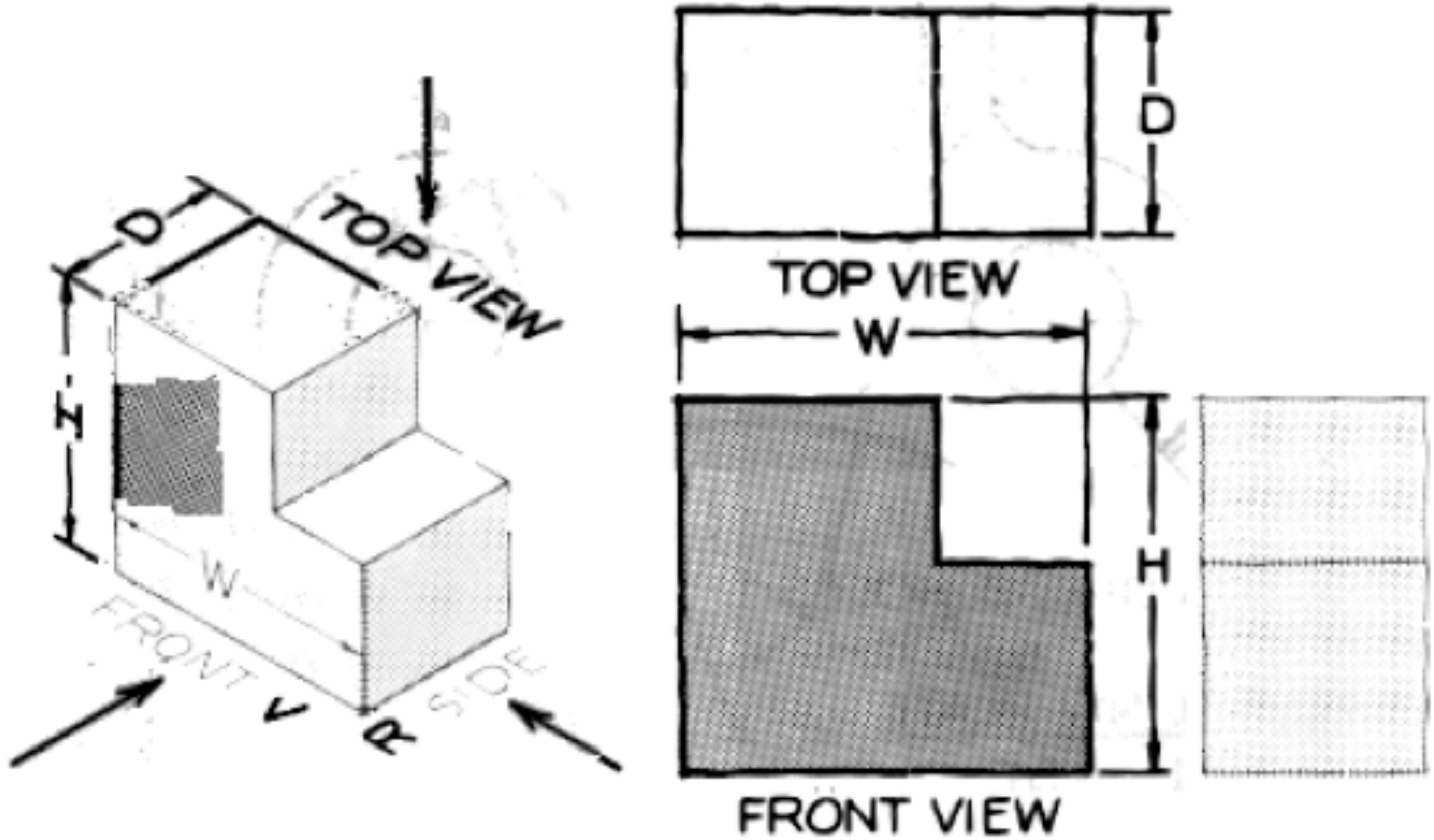
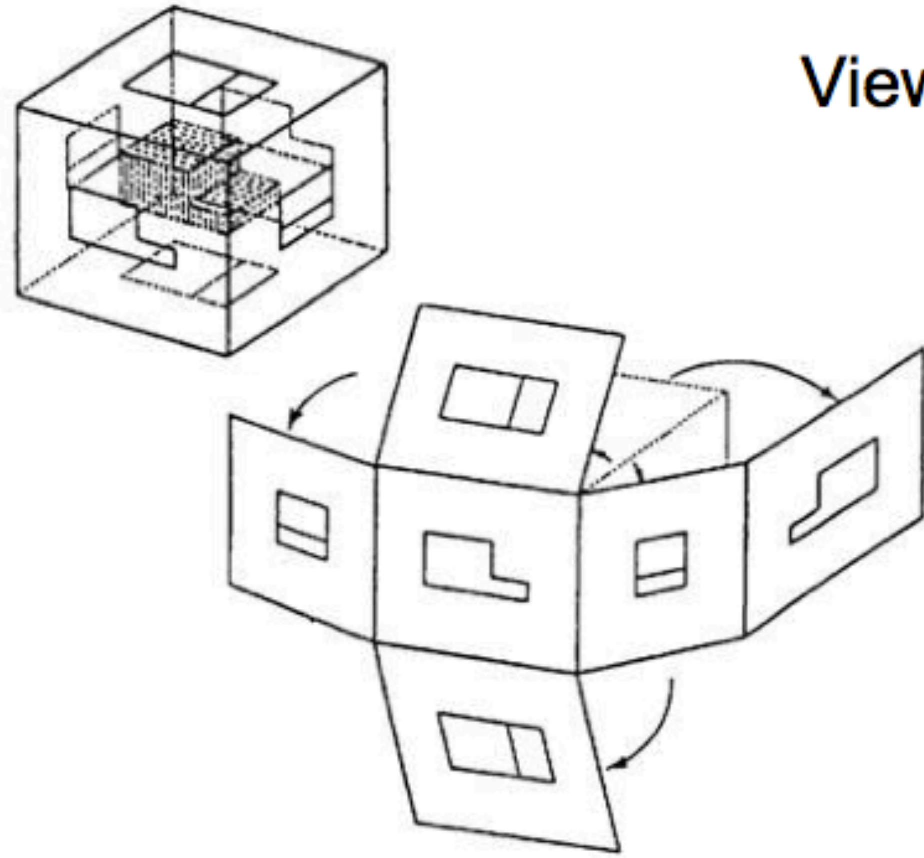
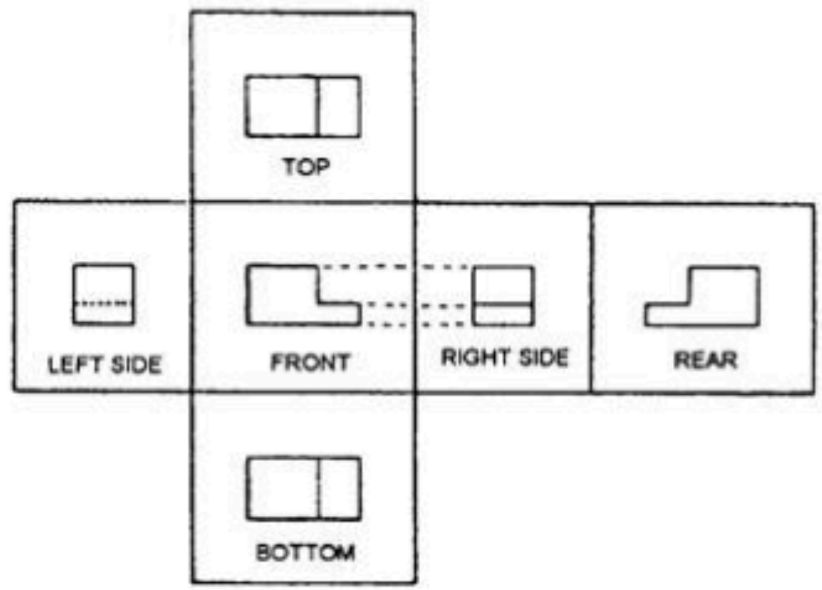


Fig. 13.1 Three views of an object can be found by looking at the object in this manner. The three views· the top, front, and right side—describe the object.

Views and projections

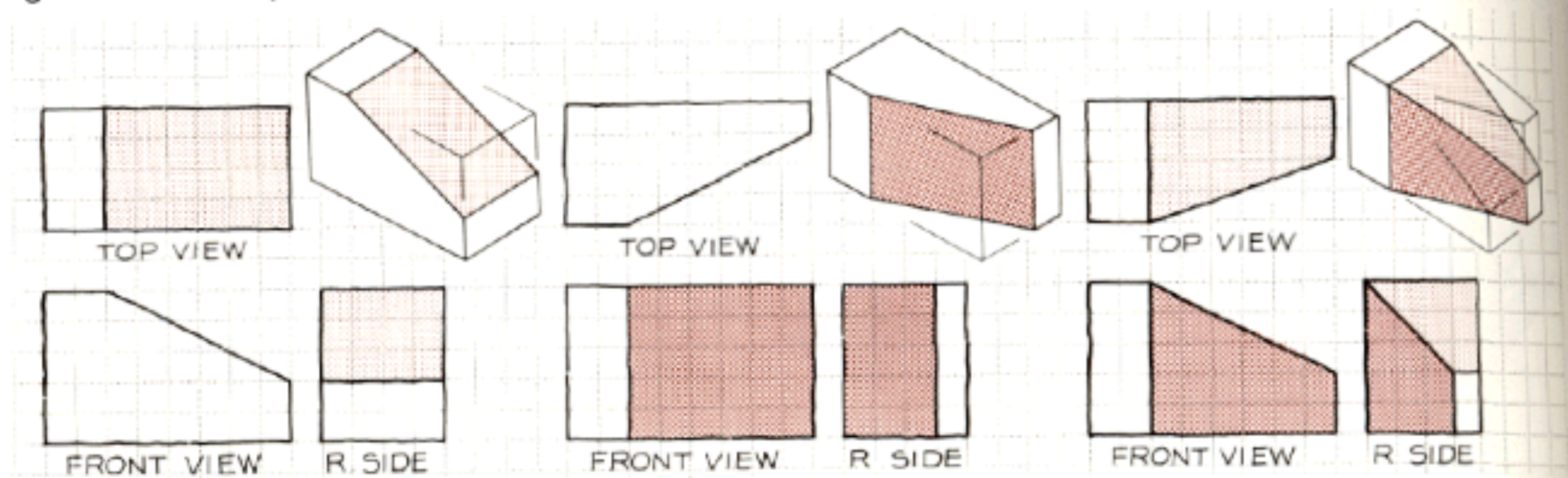




J. H. Burge
University of Arizona

Inclined planes

Fig. 13.10 Views of planes

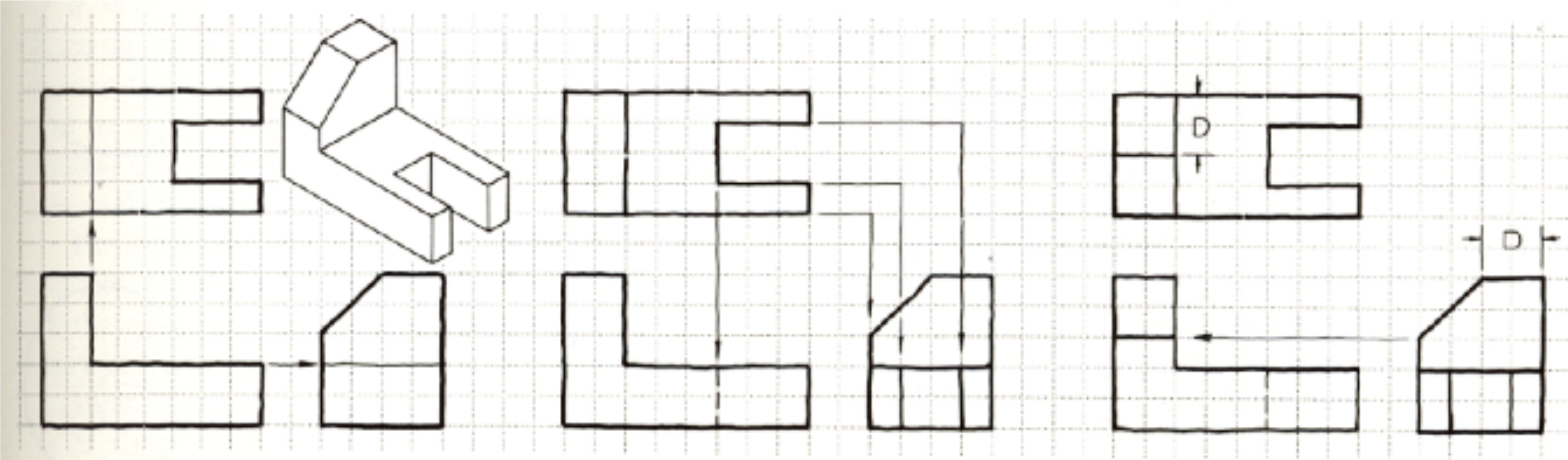


A. The plane appears as an edge in the front view and it is foreshortened in the top and side views.

B. The plane is an edge in the top view and it is foreshortened in the front and side views.

C. These two planes appear foreshortened in the right-side view. Each appears as an edge in either the top or front views.

Missing lines



Step 1 Lines may be missing in all views in this type of problem. The first missing line is found by projecting the edges of the planes from the front to the top and side views.

Step 2 The notch in the top view is projected to the front and side views. The line in the front view is a hidden line.

Step 3 The line formed by the beveled surface is found in the front view by projecting from the side view.

Holes and cylinders

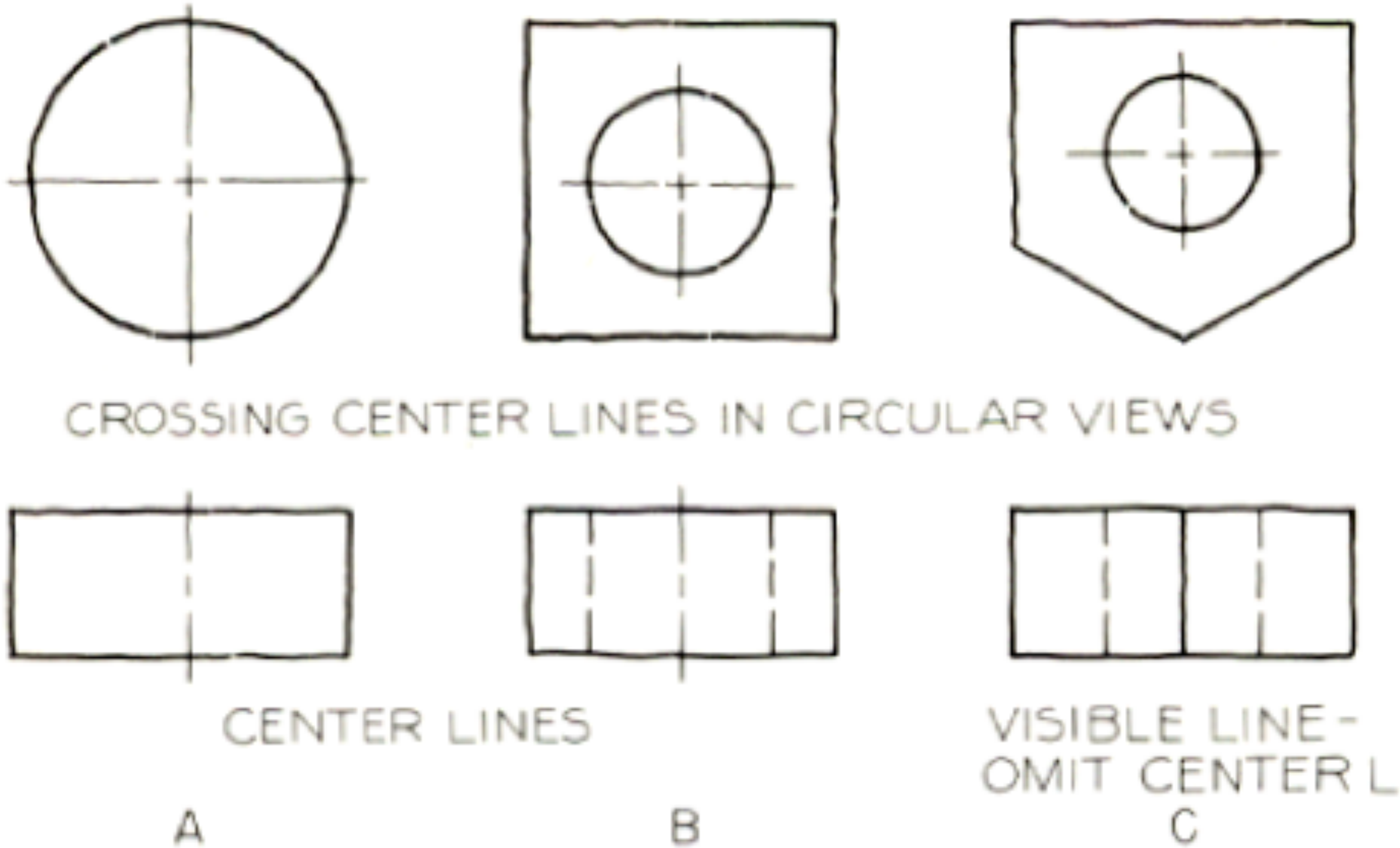


Fig. 13.14 Center lines are used to indicate the centers of circles and the axes of cylinders. These are drawn as very thin lines. When they coincide with visible or hidden lines, center lines are omitted.

Concentric cylinders

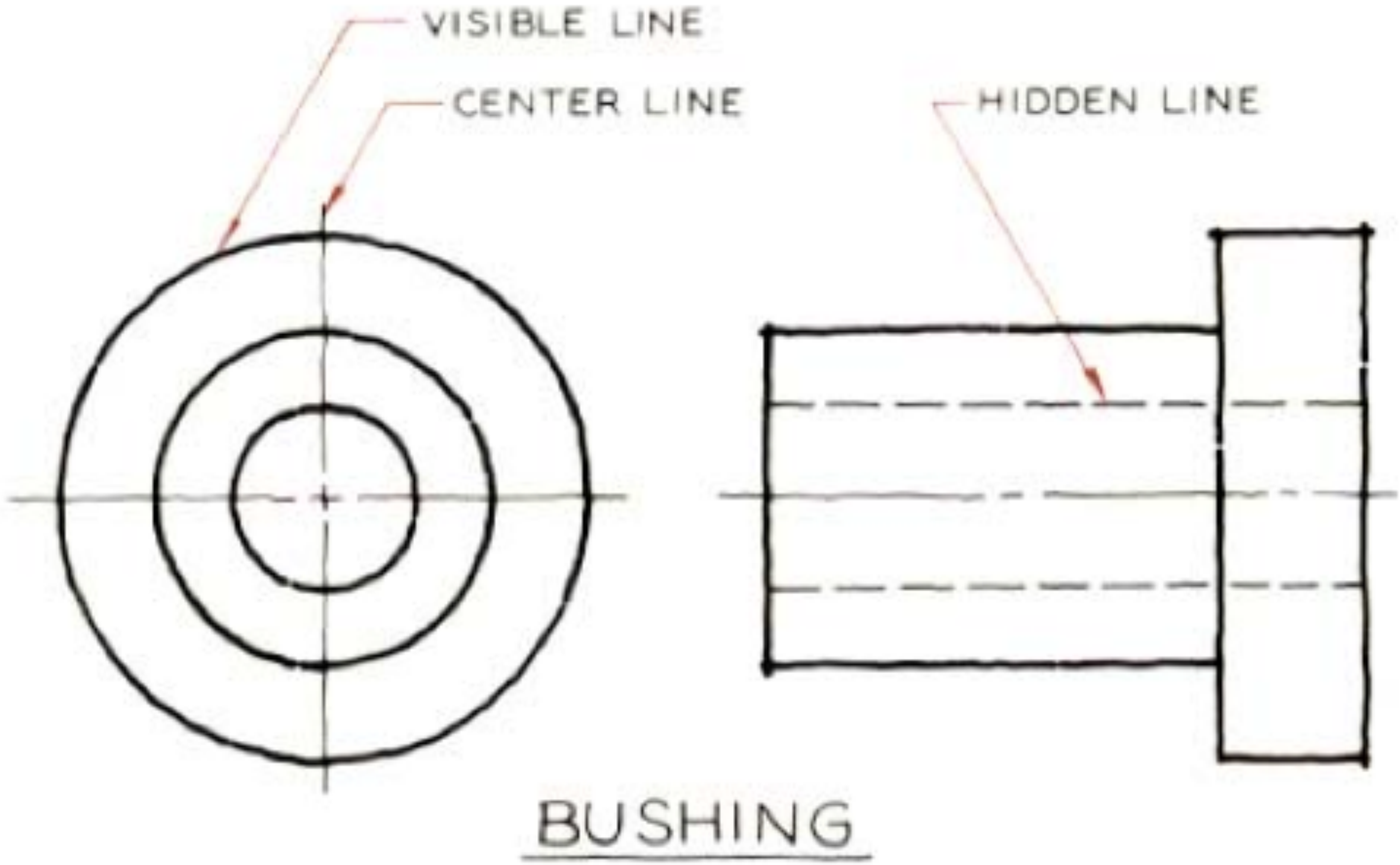


Fig. 13.16 Here you can see the application of center lines of concentric cylinders, and the relative weight of hidden, visible, and center lines.

Auxiliary views

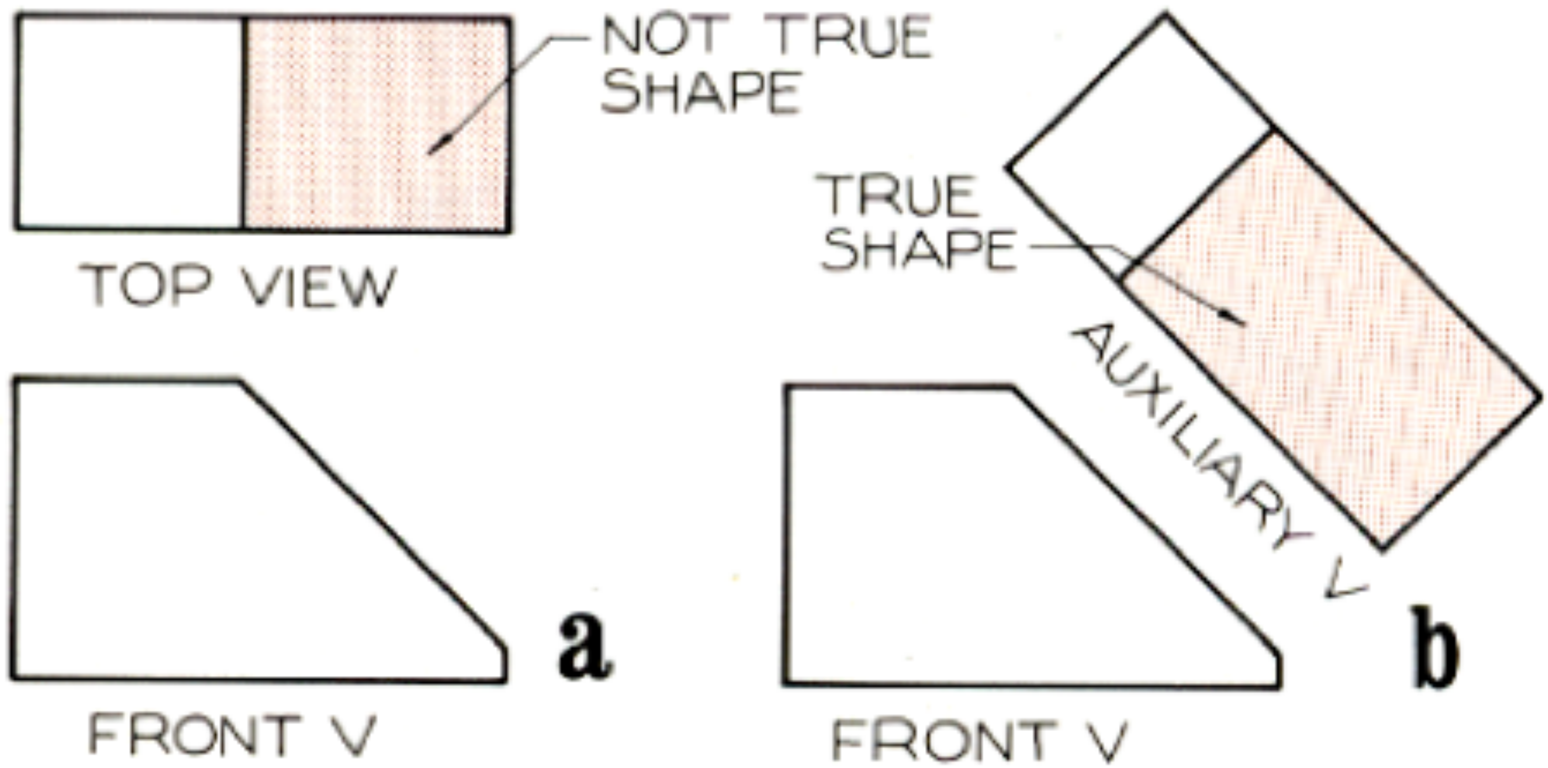


Fig. 15.1 When a surface appears as an inclined edge in a principal view, it can be found true size by an auxiliary view. The top view at a is foreshortened, but this plane is true size in an auxiliary view at b.

Auxiliary views

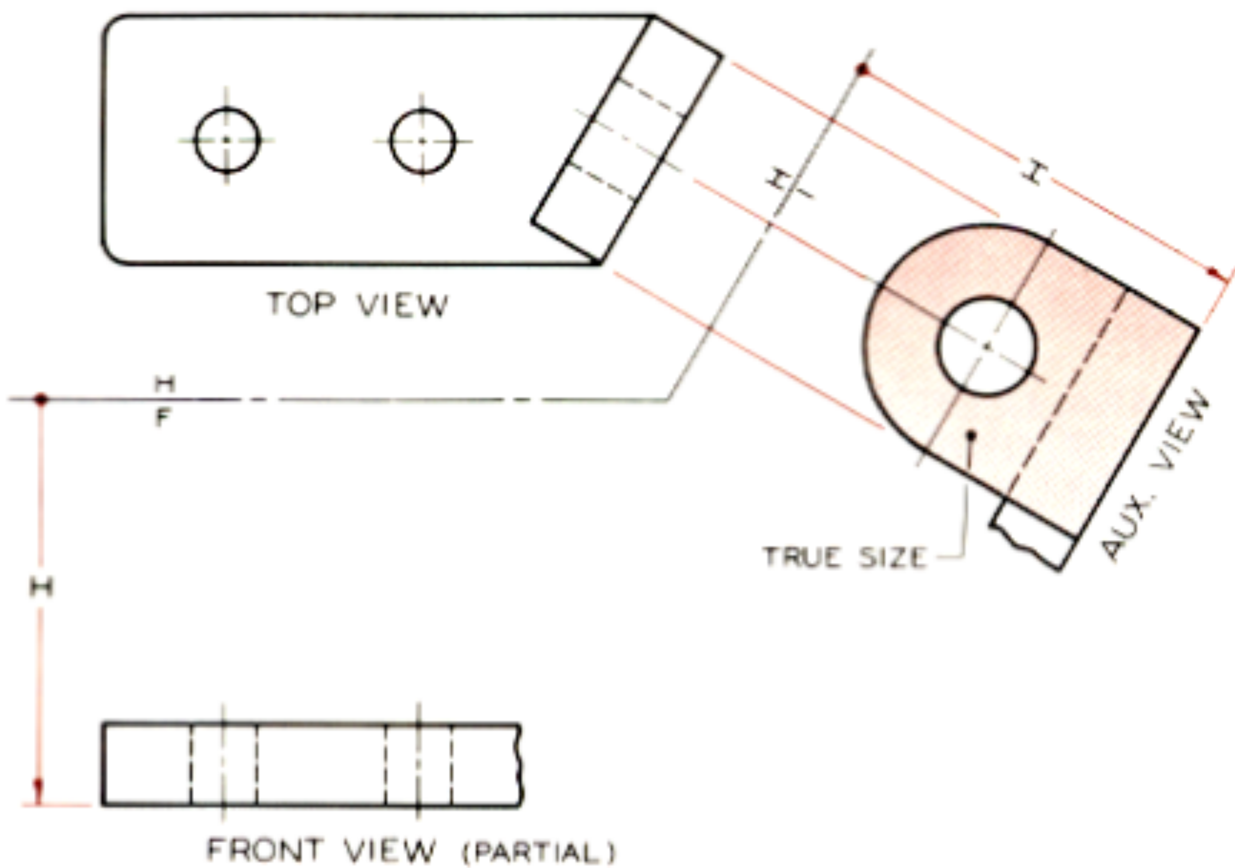


Fig. 15.6 When the object is drawn on a sheet of paper, it would be laid out in this manner. The front view is drawn as a partial view since the omitted part is shown true size in the auxiliary view.

Full section view vs. standard orthographic view

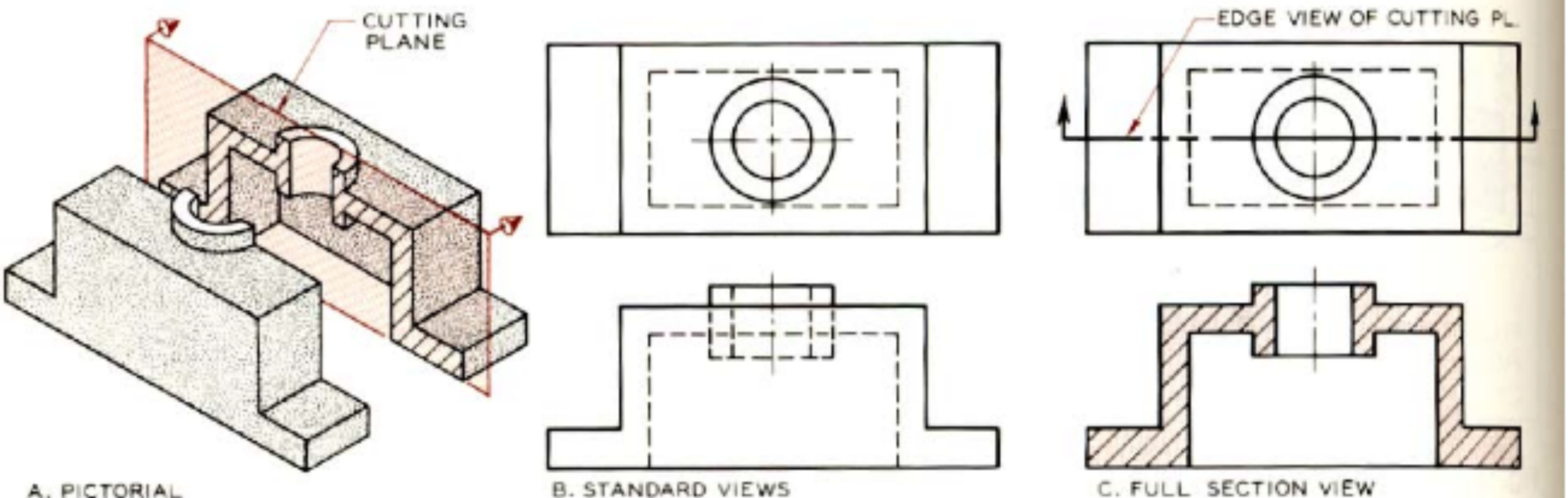


Fig. 16.1 A comparison of a regular orthographic view with a full-section view of the same object to show the internal features as well as the external features.

Half section view

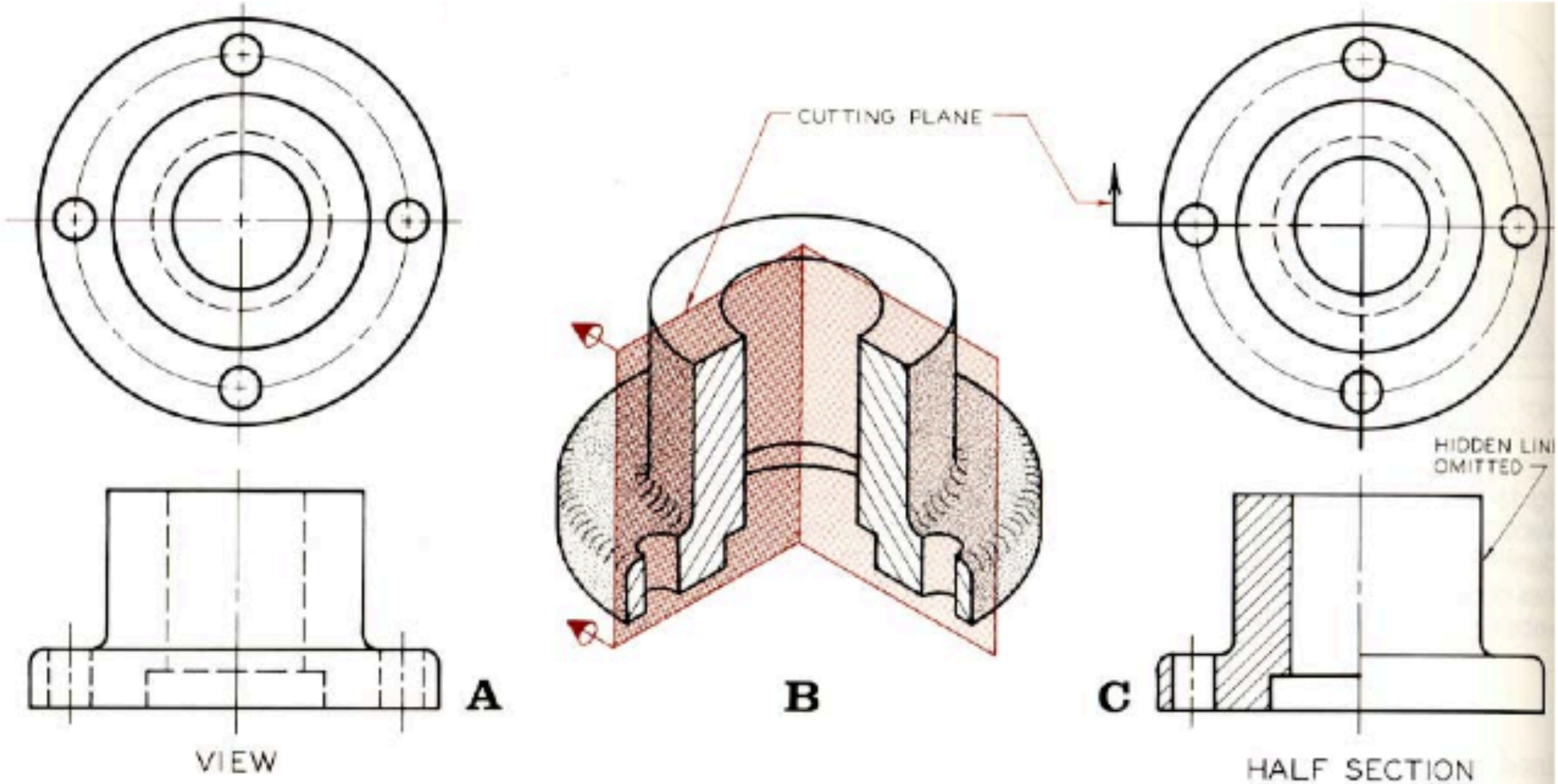
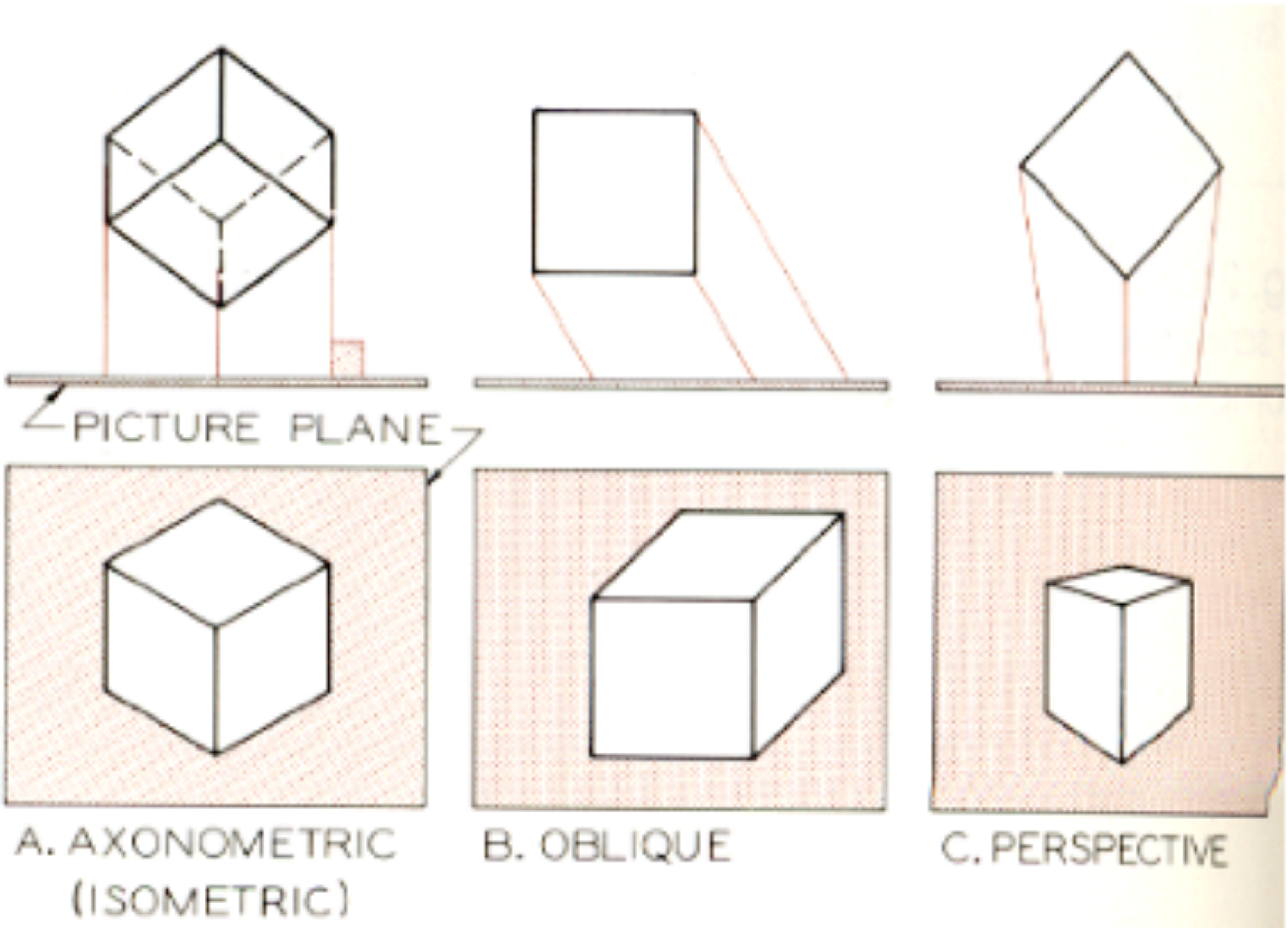


Fig. 16.19 The cutting plane of a half-section passes halfway through the object, which results in a sectional view that shows half of the outside and half of the inside of the object. Hidden lines are omitted unless they are necessary to clarify the view.

Types of projection systems for 3D pictorial drawings

- Axonometric pictorials formed by parallel projectors that are perpendicular to the picture plane
- Obliques are formed by parallel projectors that are oblique to the picture plane
- Perspectives are formed by converging projectors that make varying angles with the picture plane

Pictorial 3D drawings



Geometric dimensioning and tolerancing (GD&T)

- is a system for defining and communicating engineering tolerances.
- It uses a symbolic language on engineering drawings and computer-generated three-dimensional solid models for explicitly describing nominal geometry and its allowable variation.
- It tells the manufacturing staff and machines what degree of accuracy and precision is needed on each facet of the part ?

Linear dimensions

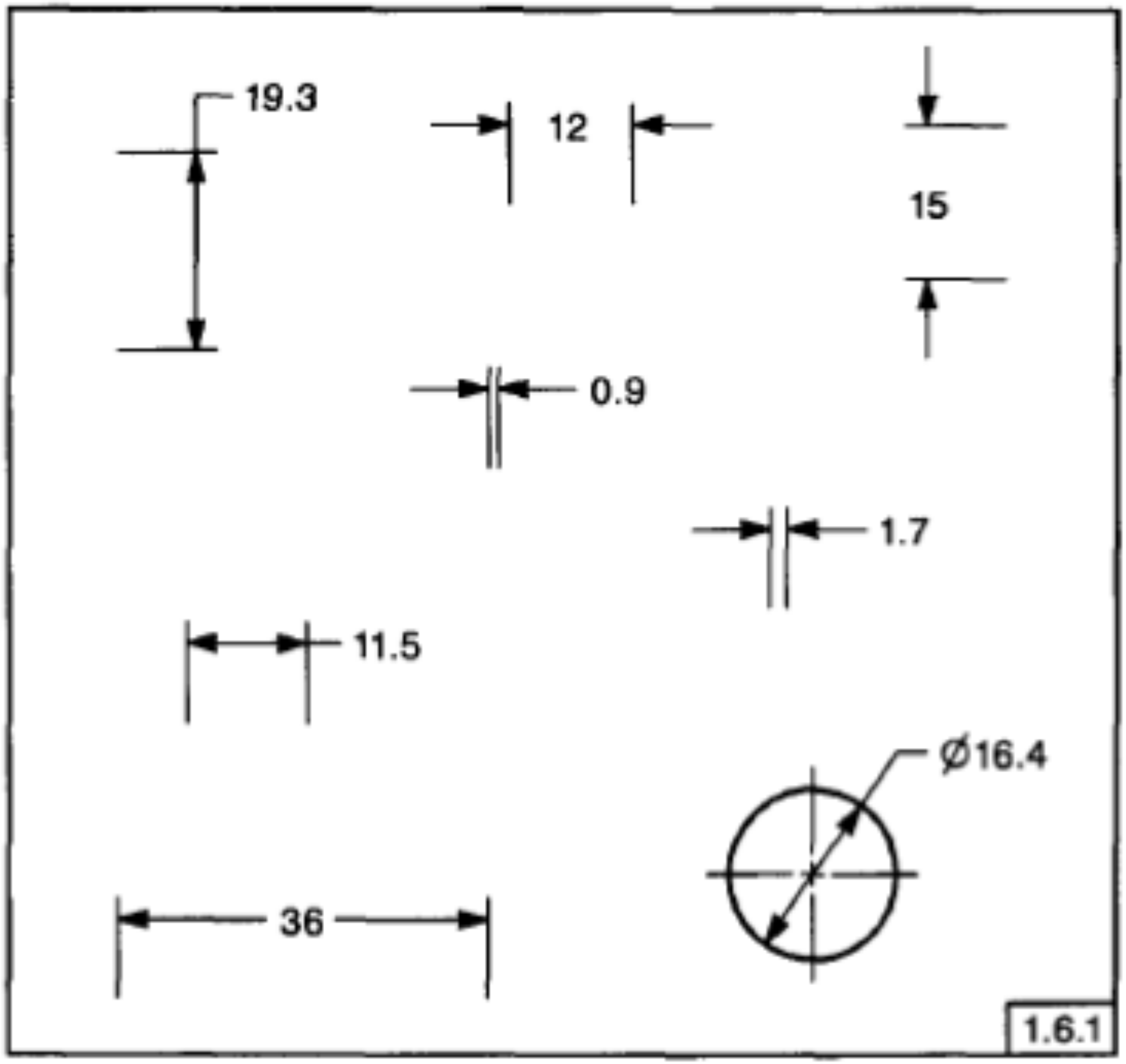


FIG. 1-2 MILLIMETER DIMENSIONS

Angular dimensions

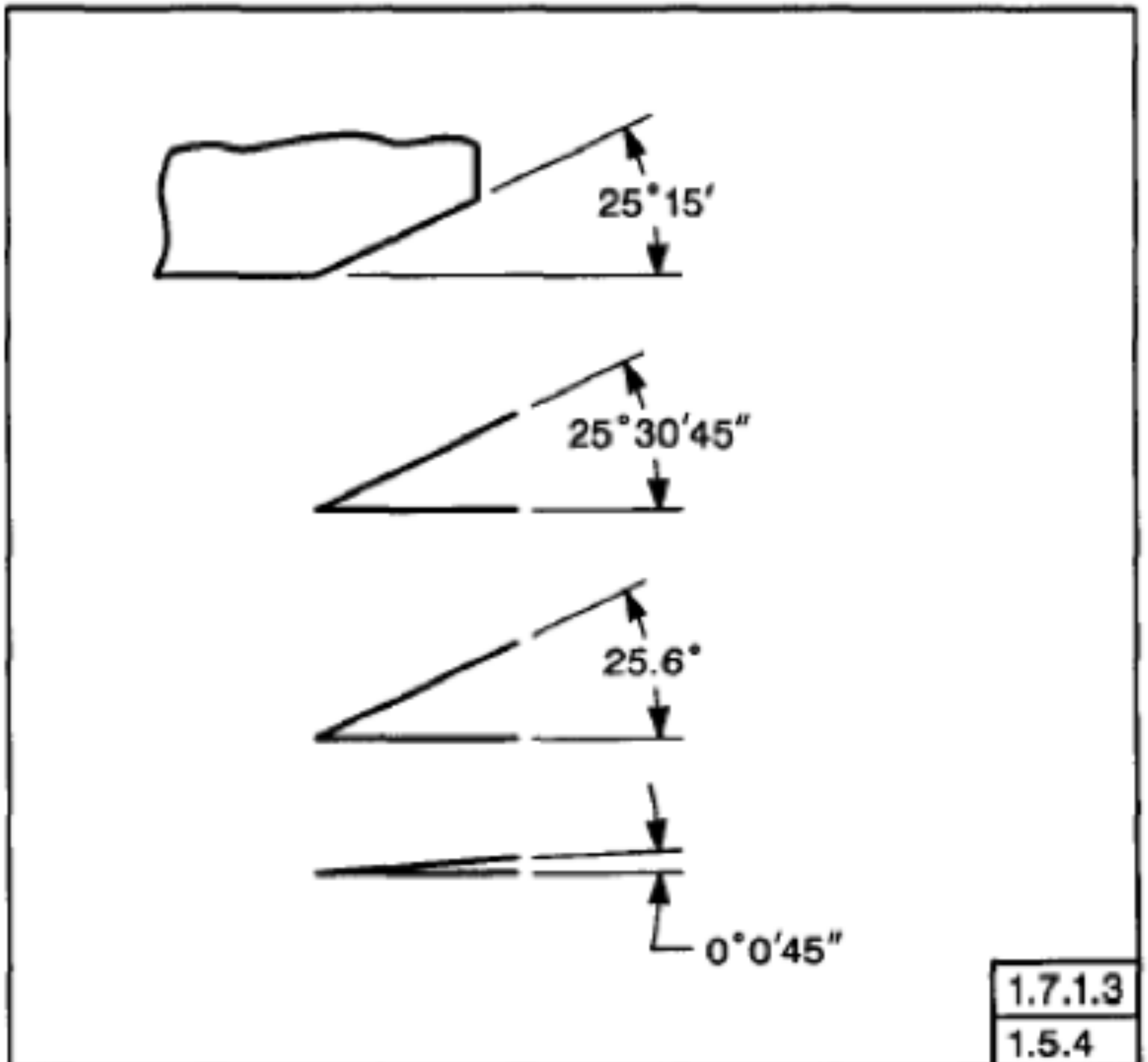


FIG. 1-1 ANGULAR UNITS

Grouping of dimensions

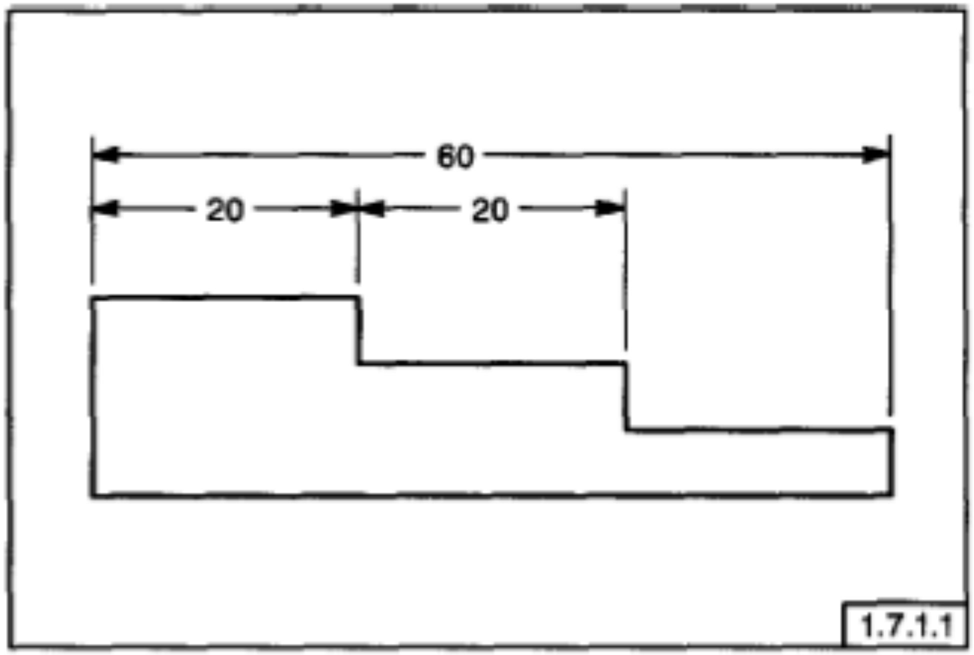


FIG. 1-5 GROUPING OF DIMENSIONS

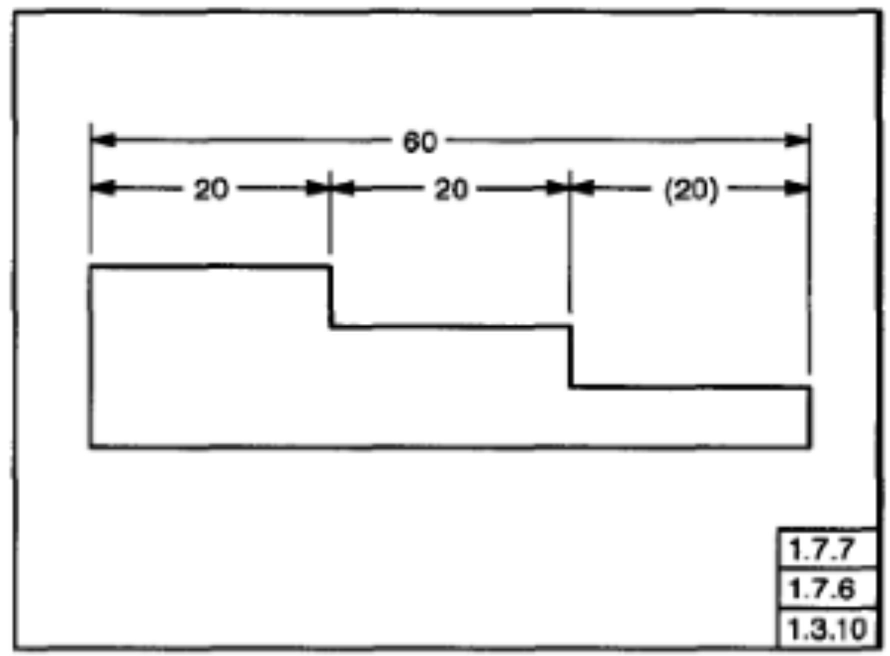


FIG. 1-17 INTERMEDIATE REFERENCE DIMENSION

Application and spacing of dimensions

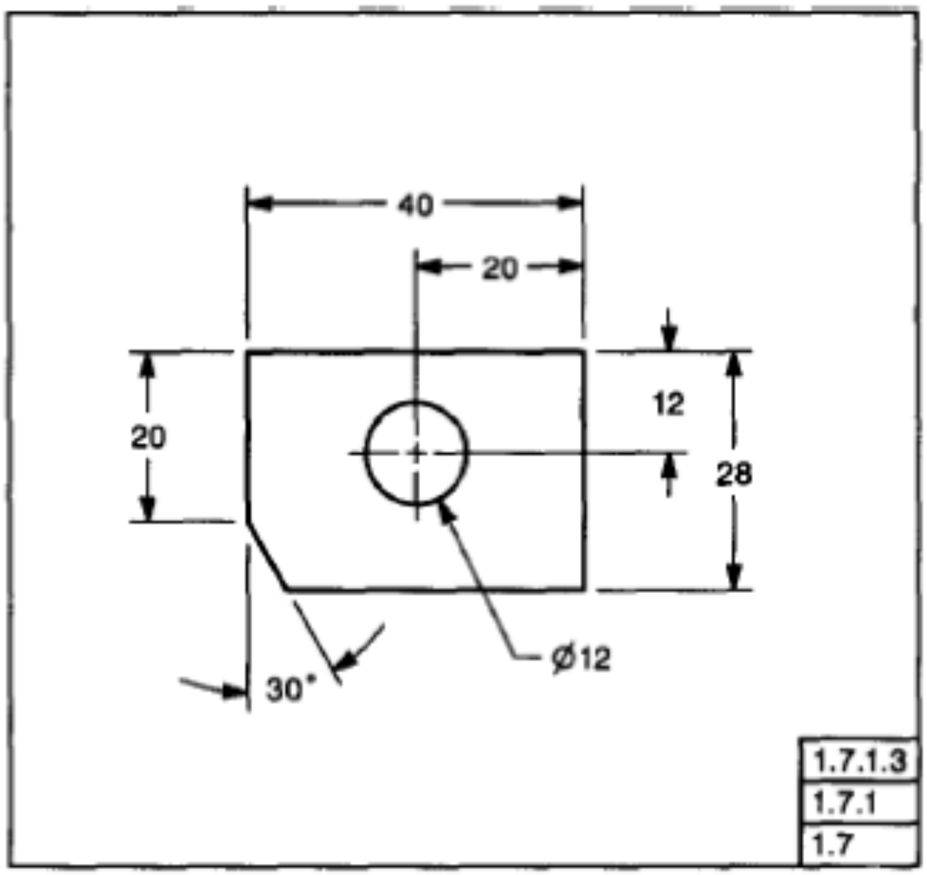


FIG. 1-4 APPLICATION OF DIMENSIONS

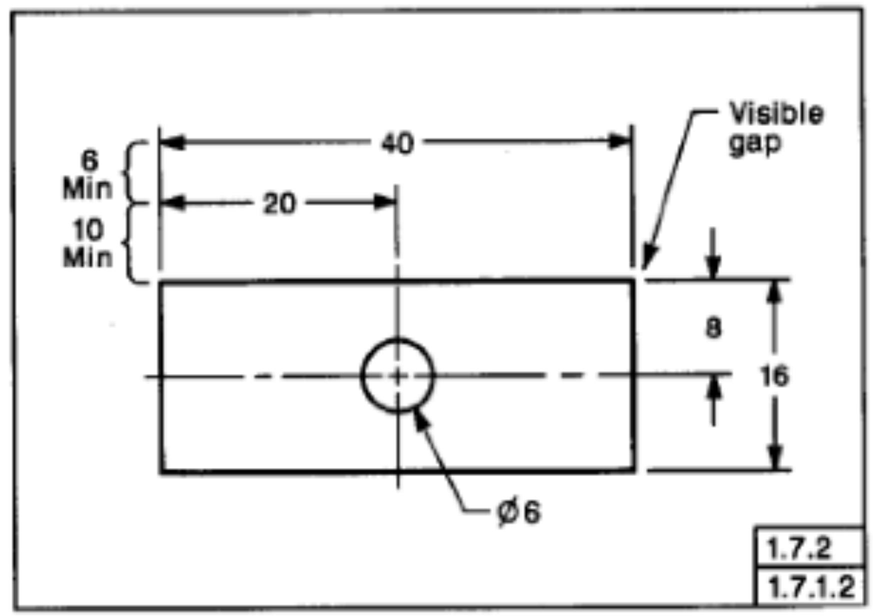


FIG. 1-6 SPACING OF DIMENSION LINES

Staggered dimensions

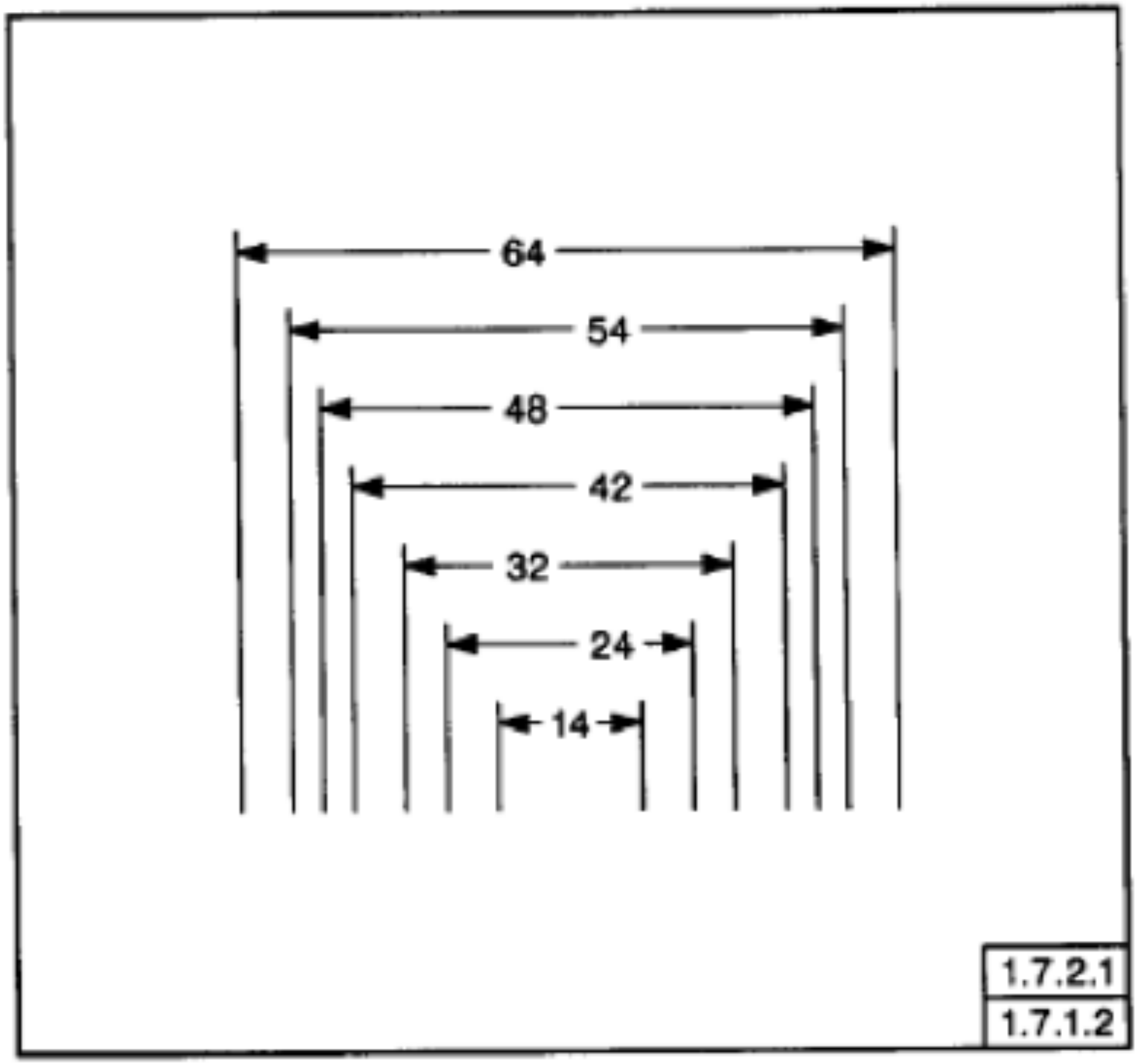


FIG. 1-7 STAGGERED DIMENSIONS

Leaders

- Leaders used to indicate where dimensions or notes are intended to apply.
- Leaders should be thin full lines, terminating in arrowheads or dots.
- Arrowheads always should terminate on a line
- Dots should be within the outline of the object.
- The use of long leaders should be avoided.

Leaders and minimizing them

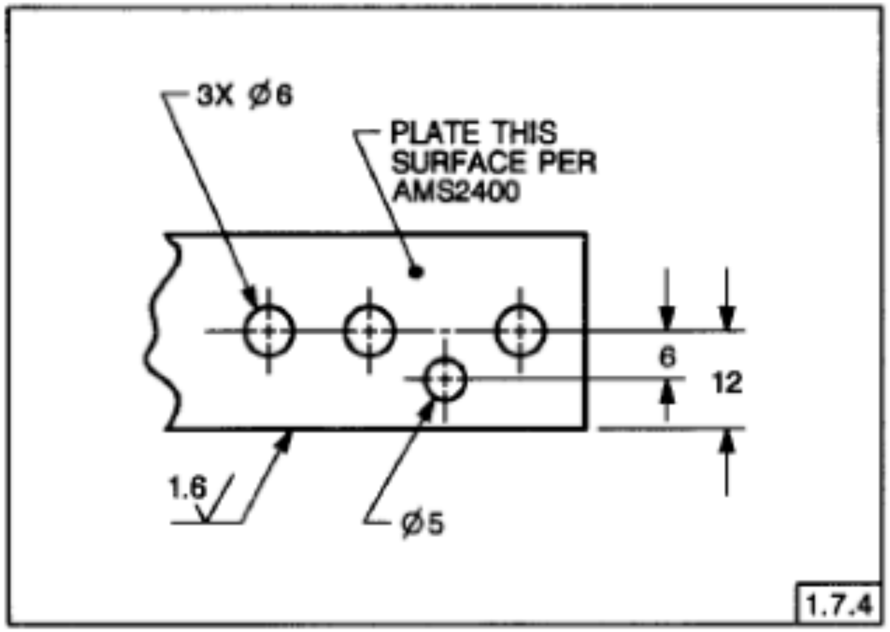


FIG. 1-12 LEADERS

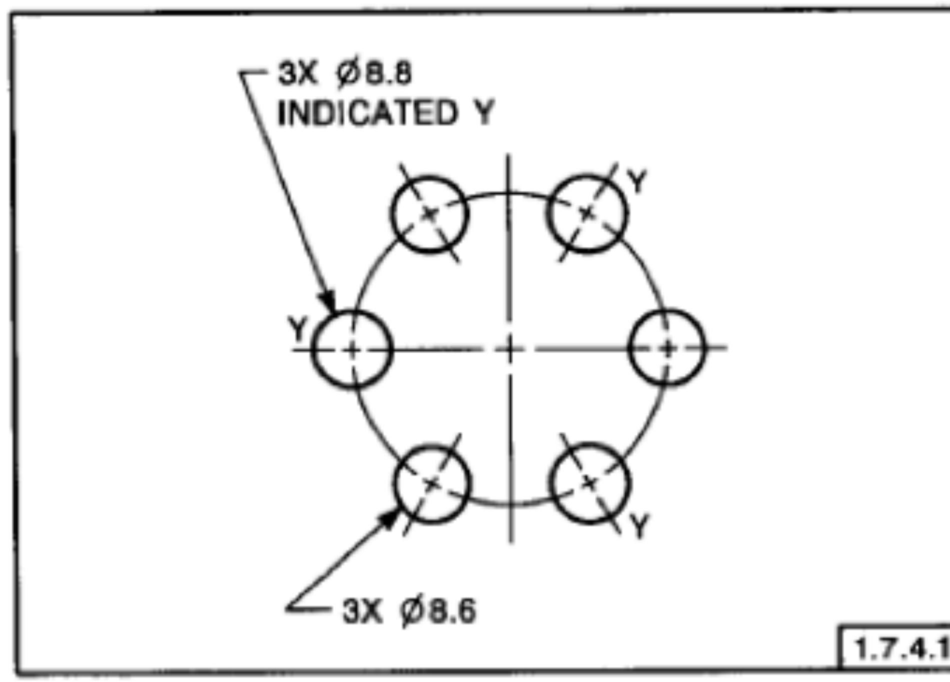


FIG. 1-14 MINIMIZING LEADERS

Diameters, radii

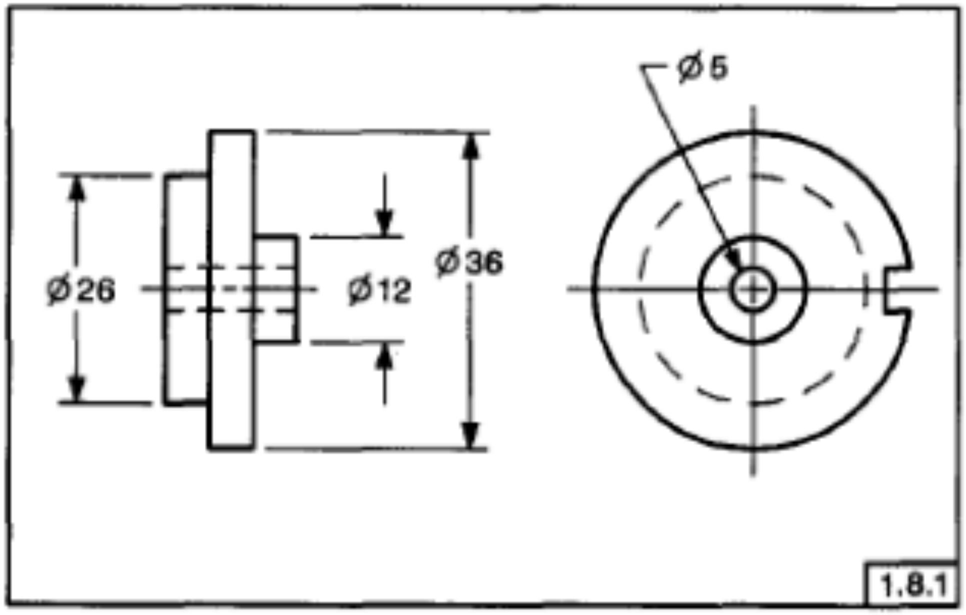


FIG. 1-19 DIAMETERS

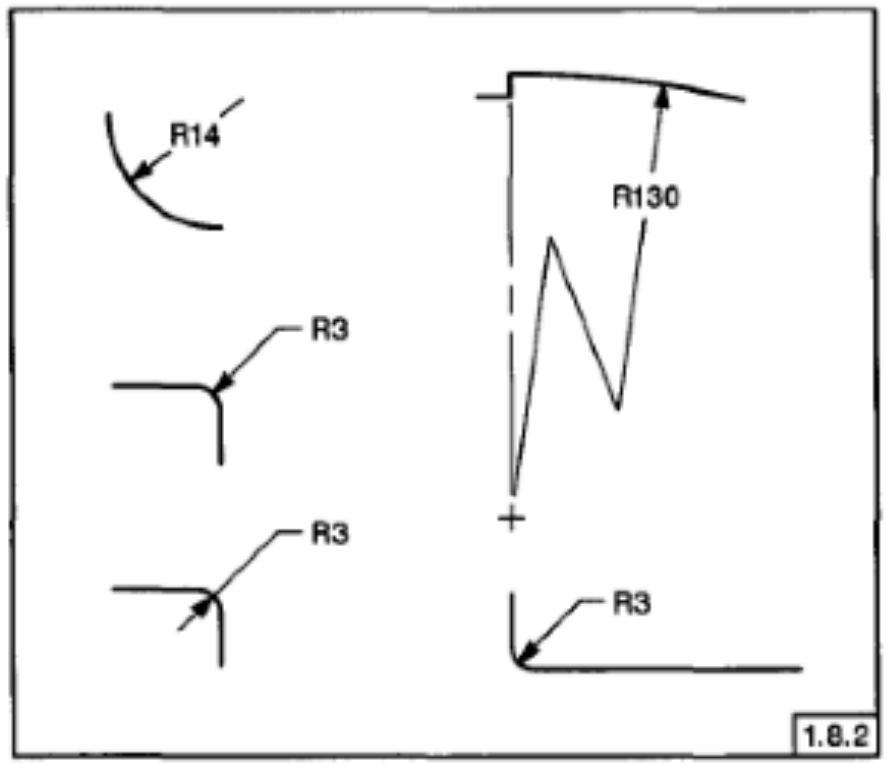
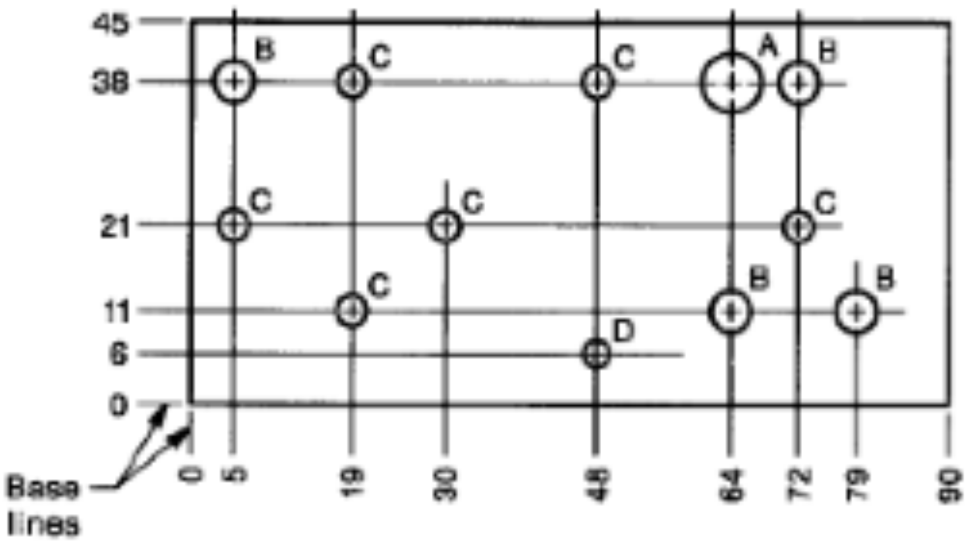


FIG. 1-20 RADII

Tabular dimensions

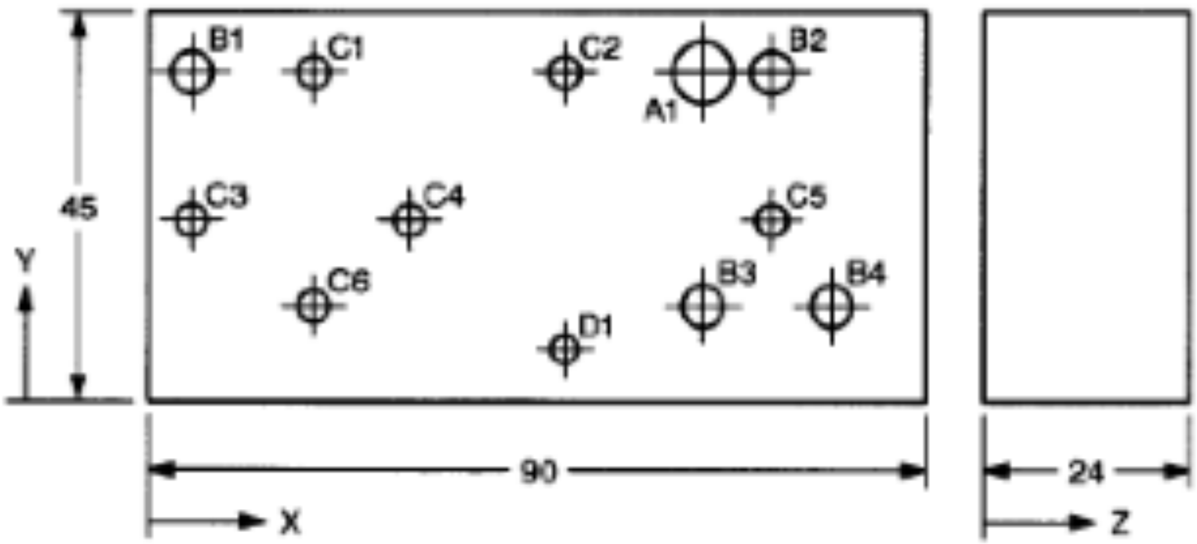


SIZE SYMBOL	A	B	C	D
HOLE Ø	7	4.8	3.6	3.1

1.9.2
1.9
1.7.5.3

Tabular dimensions

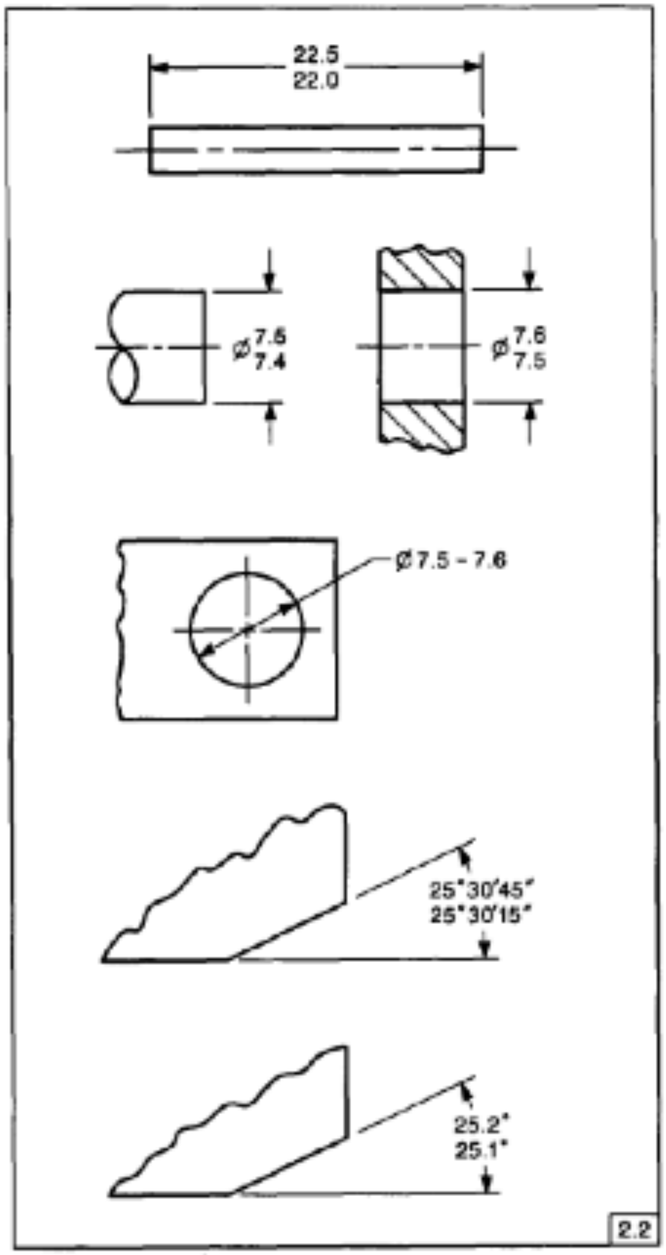
HOLE	DESCRIPTION	QTY
A	Ø7	1
B	Ø4.8	4
C	Ø3.6	6
D	Ø3.1	1



HOLE	FROM	X	Y	Z
A1	X,Y	64	38	18
B1	X,Y	5	38	THRU
B2	X,Y	72	38	THRU
B3	X,Y	64	11	THRU
B4	X,Y	79	11	THRU
C1	X,Y	19	38	THRU
C2	X,Y	48	38	THRU
C3	X,Y	5	21	THRU
C4	X,Y	30	21	THRU
C5	X,Y	72	21	THRU
C6	X,Y	19	11	THRU
D1	X,Y	48	6	12

1.9.3
1.9.2
1.9

Tolerances



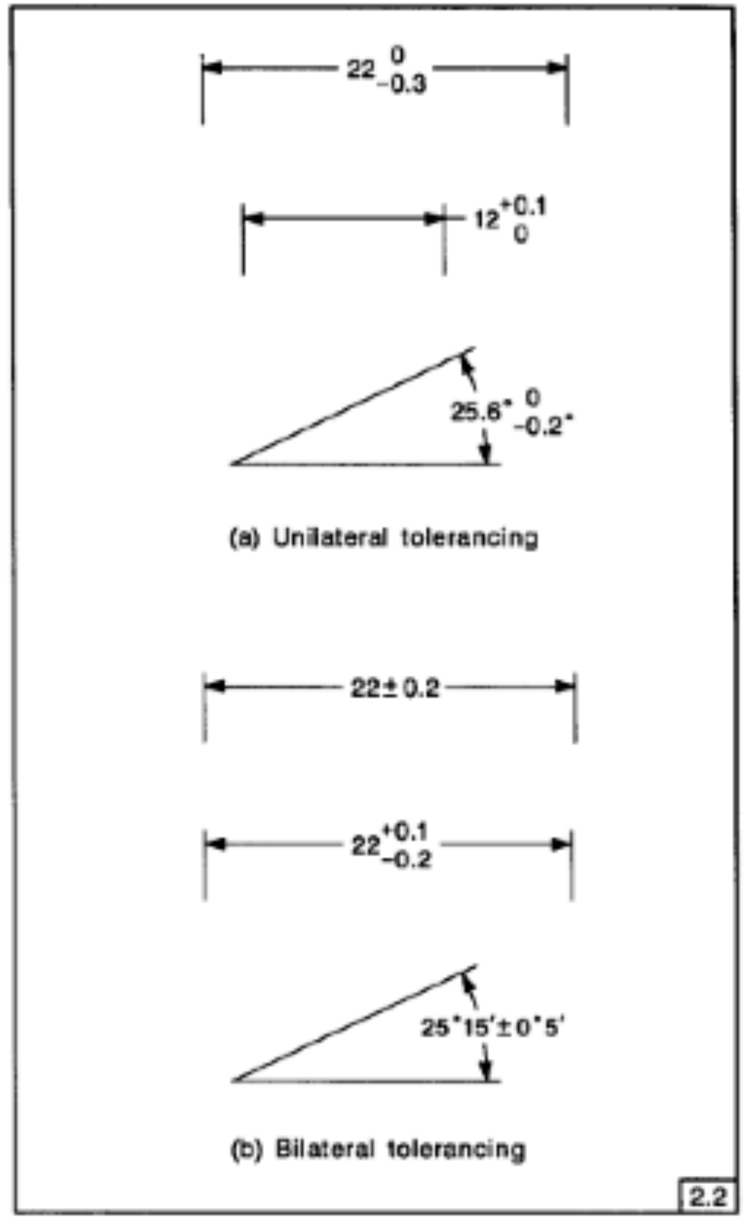
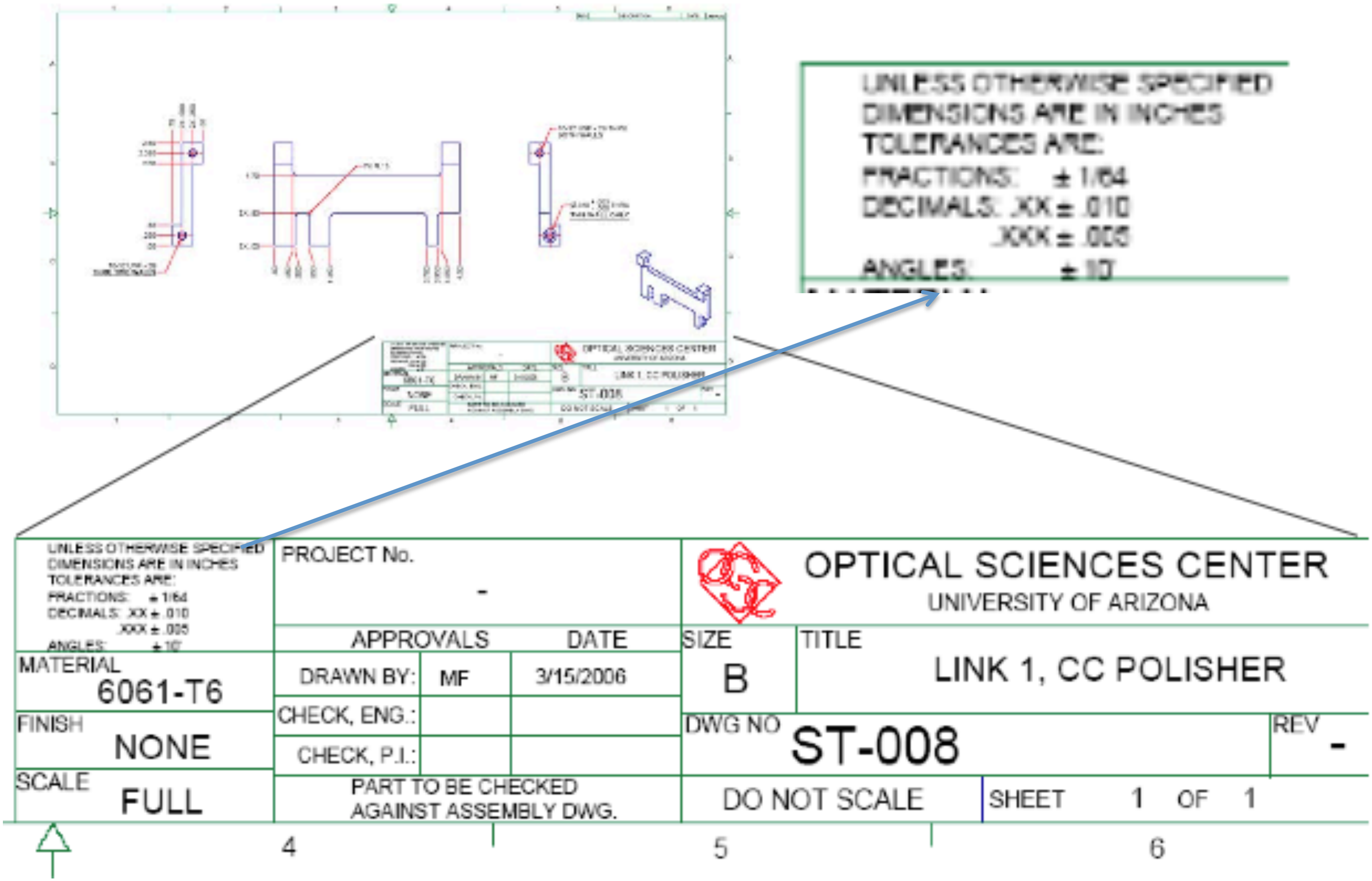


FIG. 2-2 PLUS AND MINUS TOLERANCING

Default tolerance (mentioned on title box)



Datum

- In engineering and drafting, a **datum is a reference point, surface, or axis on an object against which measurements are made.**

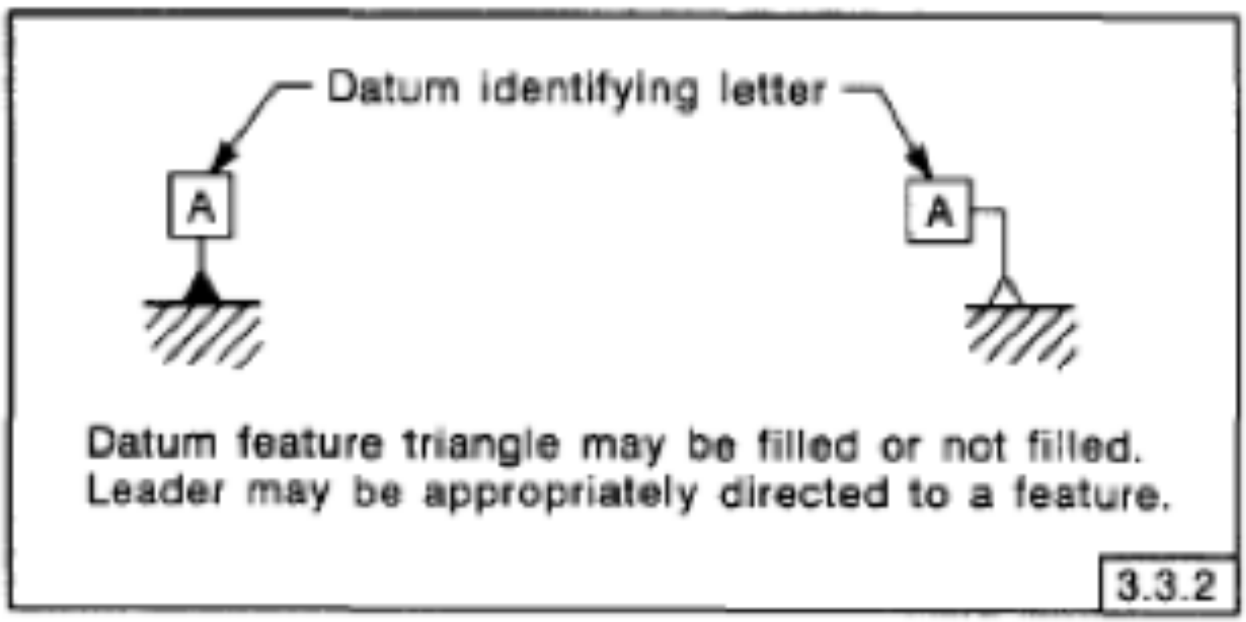


FIG. 3-2 DATUM FEATURE SYMBOL

Datum symbols

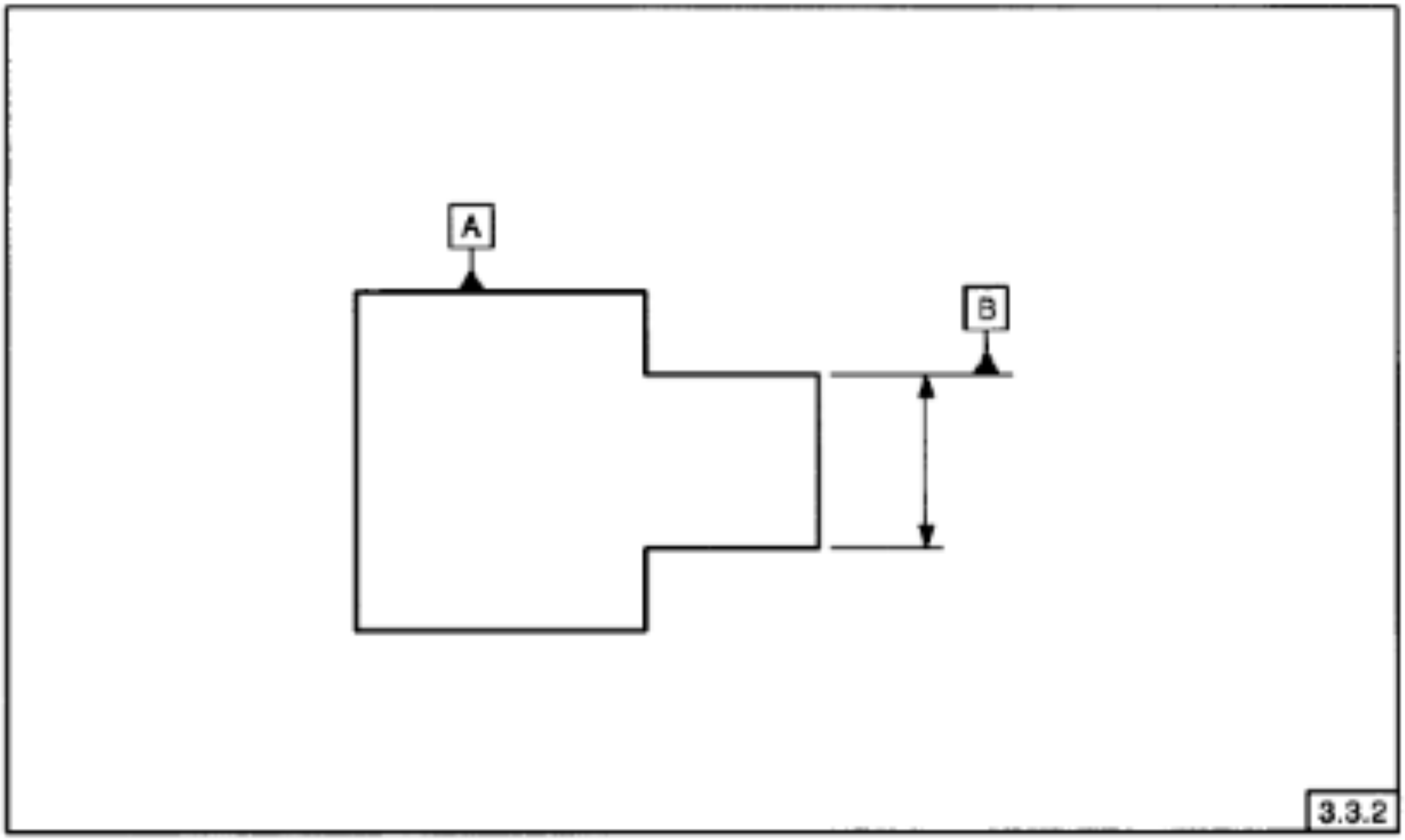


FIG. 3-3 DATUM FEATURE SYMBOLS ON A FEATURE SURFACE AND AN EXTENSION LINE

Datum reference frame

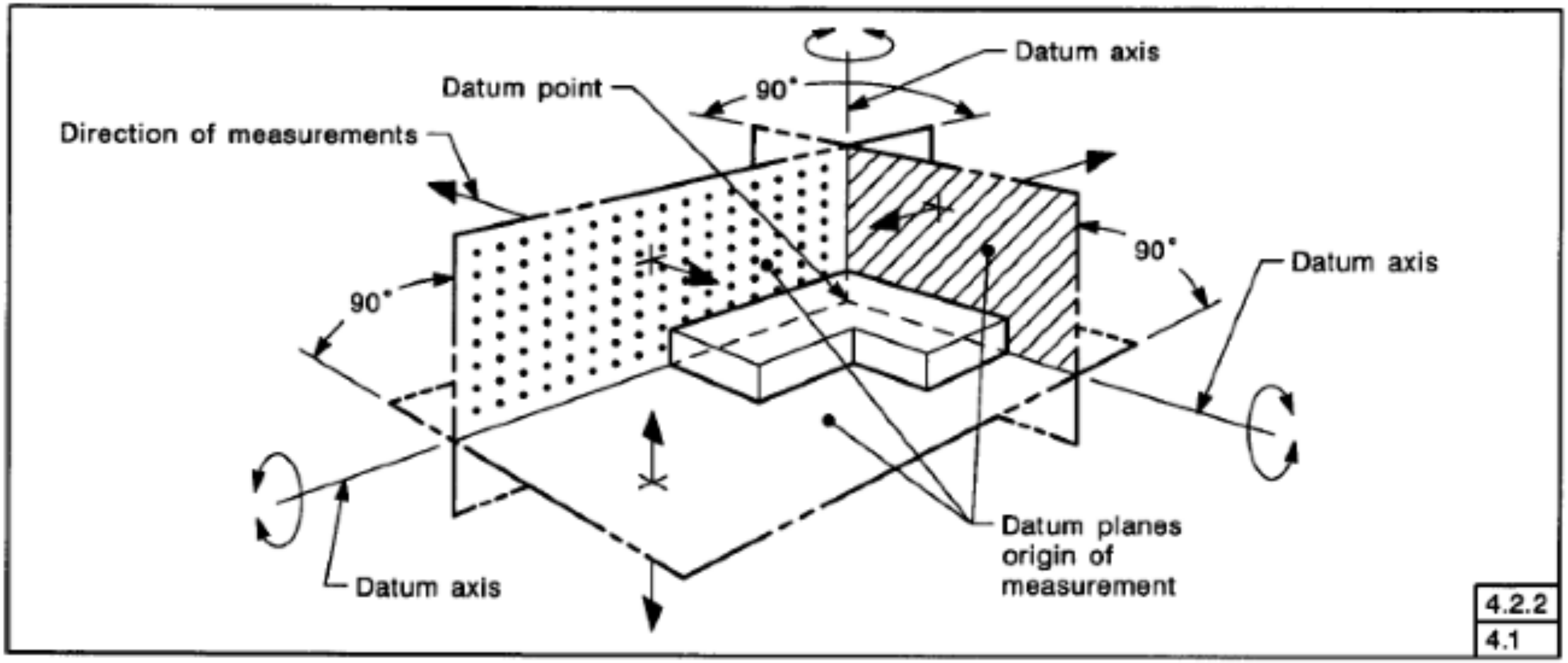


FIG. 4-1 DATUM REFERENCE FRAME

Reference to Datum

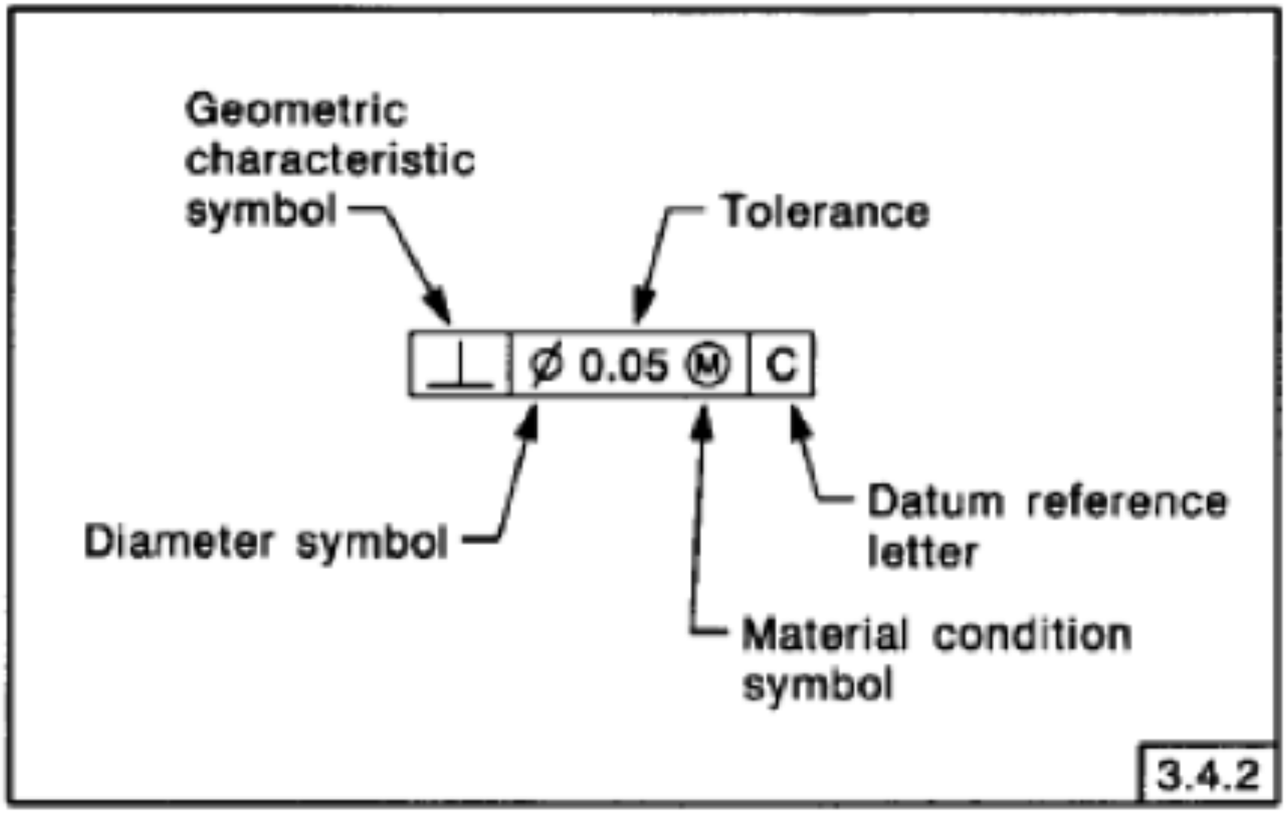


FIG. 3-20 FEATURE CONTROL FRAME INCORPORATING A DATUM REFERENCE

MMC = Maximum Material Condition

LMC = Least Material Condition

Modifying symbol

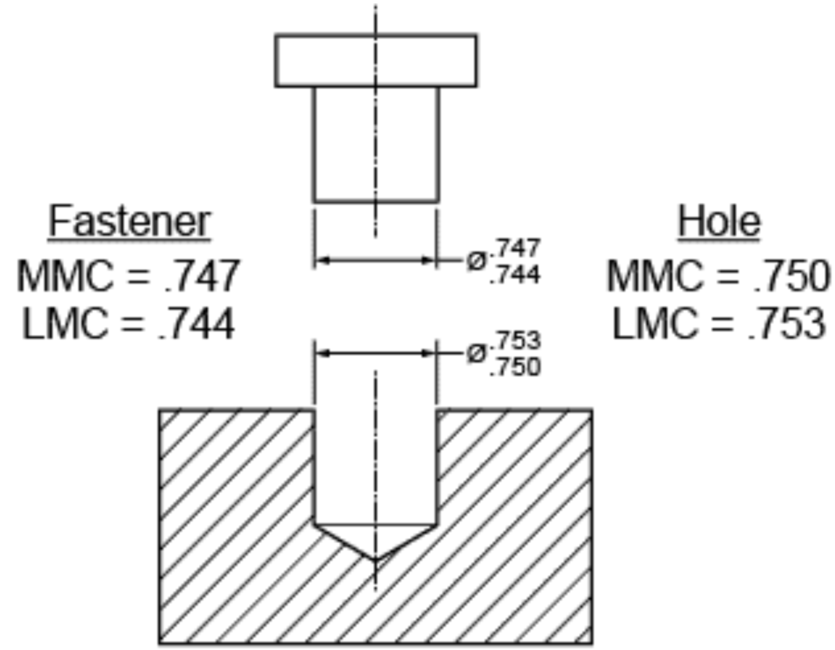
TERM	SYMBOL	SEE:
AT MAXIMUM MATERIAL CONDITION	Ⓜ	3.3.5
AT LEAST MATERIAL CONDITION	Ⓛ	3.3.5
PROJECTED TOLERANCE ZONE	Ⓟ	3.3.6
FREE STATE	Ⓕ	3.3.19
TANGENT PLANE	Ⓣ	3.3.20
DIAMETER	Ø	3.3.7
SPHERICAL DIAMETER	SØ	3.3.7
RADIUS	R	3.3.7
SPHERICAL RADIUS	SR	3.3.7
CONTROLLED RADIUS	CR	3.3.7
REFERENCE	( )	3.3.8
ARC LENGTH	⌒	3.3.9
STATISTICAL TOLERANCE	⟨ST⟩	3.3.10
BETWEEN	◄─►	3.3.11

Maximum Material Condition (MMC) & Least Material Condition (LMC)

- When a part feature contains the **maximum amount of material allowed within the specified size limits**, it's said to be in its maximum material condition.
- *When a part feature contains the **least amount of material allowed within the specified size limits**, it's said to be in its least material condition.*
- *The material condition of the part is significant in geometric dimensioning and tolerancing.*

Example for MMC & LMC

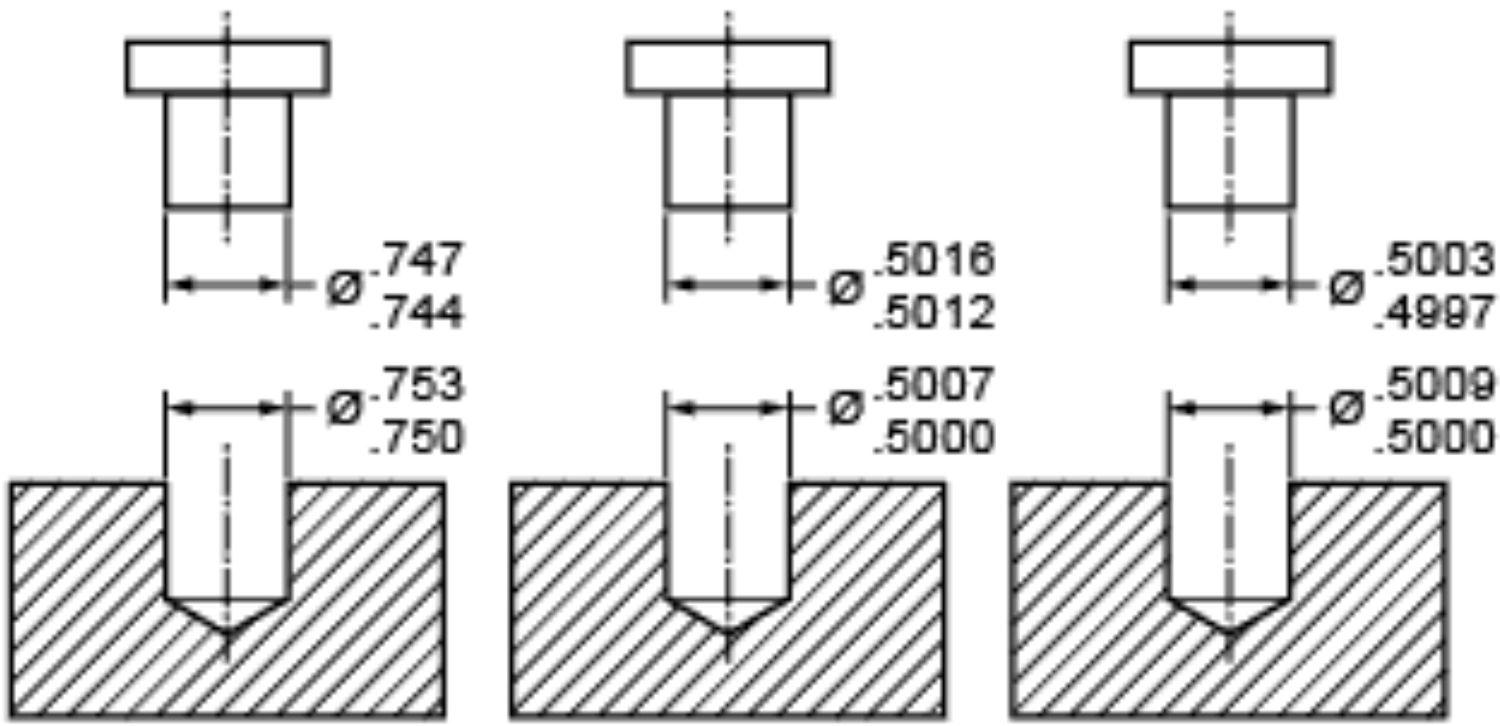
- An external feature, such as a fastener, is in its maximum material condition when it's at its upper size limit.
- *EXAMPLE: The MMC of this fastener is.747.*
- *An internal feature, such as a hole, is in its maximum material condition when it's at its lower size limit.*
- *EXAMPLE: The MMC of this hole is .750.*
- *EXAMPLE: The fastener is in its least material condition when it's at its lower size limit of .744.*
- *The hole is in its least material condition when it's at its upper size limit of .753.*



Types of fit: Clearance fit

- When the specified size limits of mating part features **always result in clearance at assembly,** the parts are said to have a clearance fit.

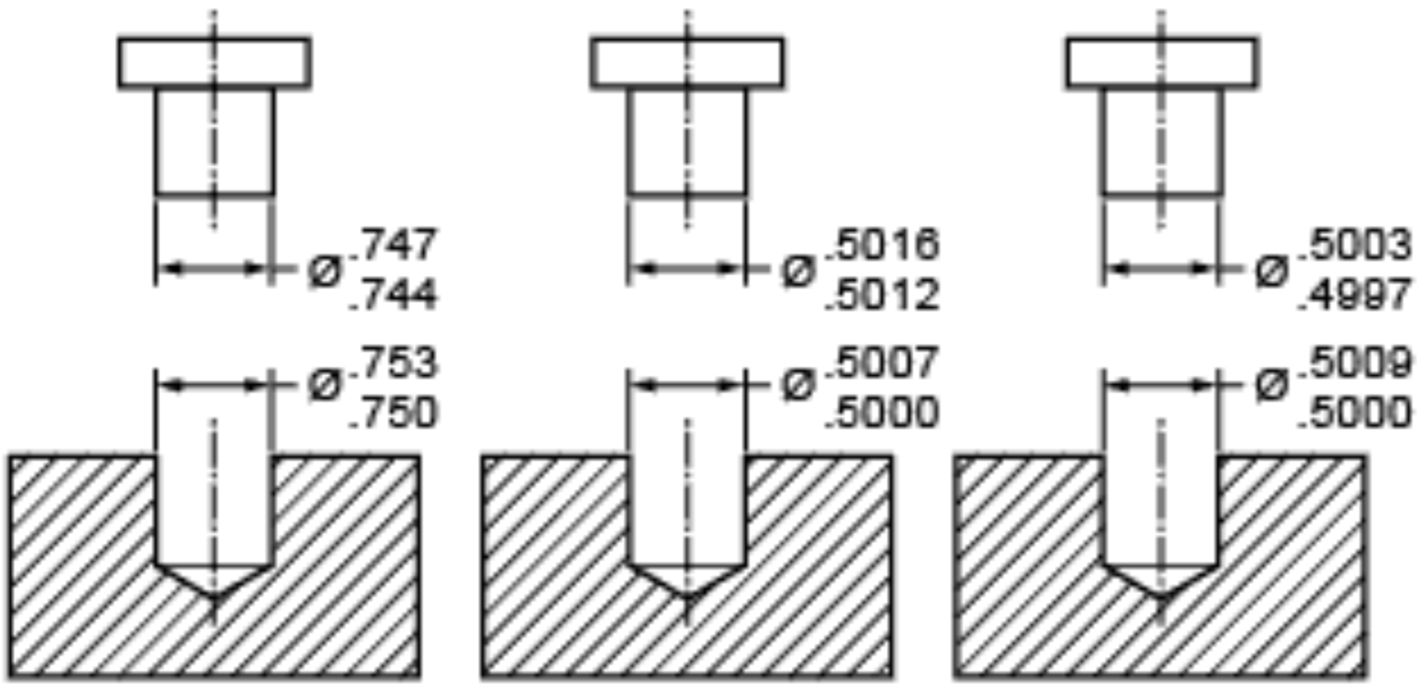
- *EXAMPLE: In this drawing, even when the fastener is at its MMC size of .747 and the hole is at its MMC size of .750, there is clearance*



Types of fit: Interference fit

- *When the specified size* ***limits always produce interference at assembly****, mating part features are said to have an* ***interference fit.***

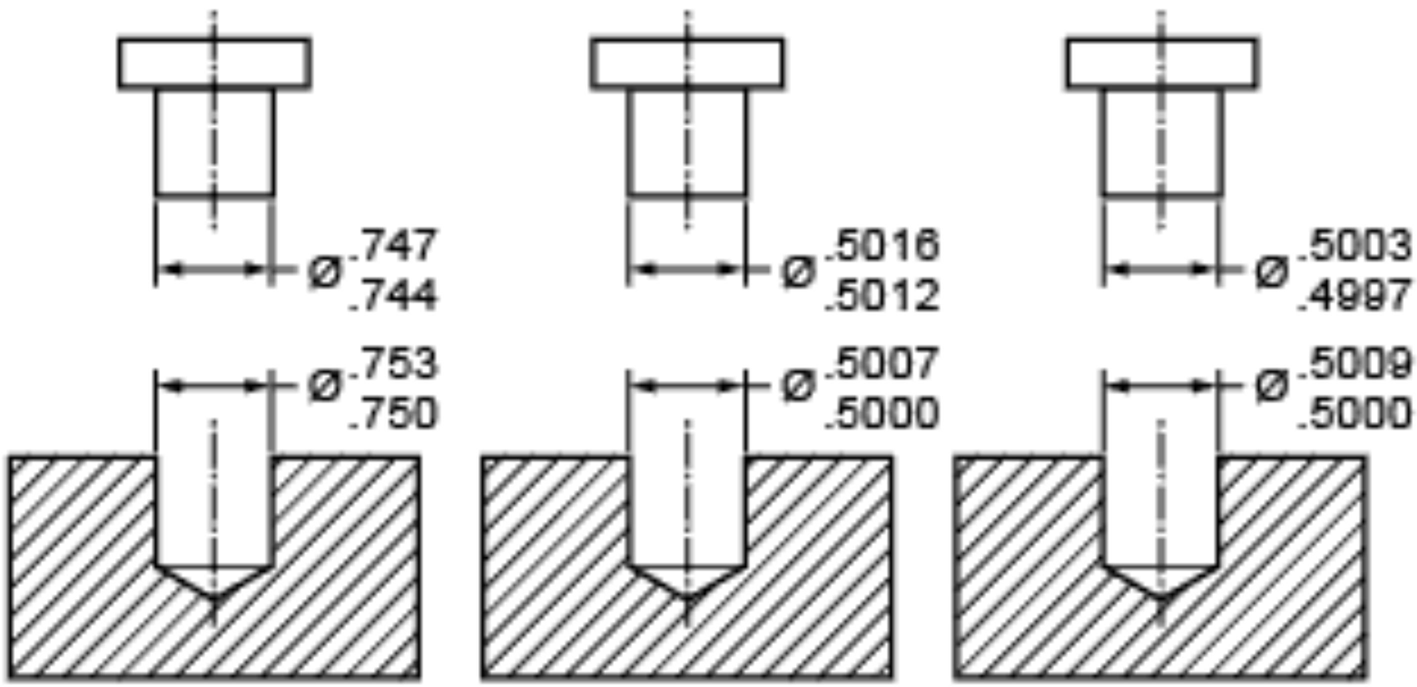
- *EXAMPLE: In the center drawing, even when the fastener is at its LMC size of .5012 and the hole is at its LMC size of .5007, there is interference.*



Types of fit: Transition fit

- *When mating part features do **not fit together in their maximum material condition, but do fit at some point as they approach their least material condition**, they are said to have a transition fit.*

- *EXAMPLE: In the drawing on the right, when the fastener is at its maximum material condition size of .5003, it will not fit the hole at its MMC size of .5000. However, when both features are manufactured at their least material condition size, they will fit together.*



Geometric characteristic symbol

	TYPE OF TOLERANCE	CHARACTERISTIC	SYMBOL	SEE:
FOR INDIVIDUAL FEATURES	FORM	STRAIGHTNESS	⏤	6.4.1
		FLATNESS	⏥	6.4.2
		CIRCULARITY (ROUNDNESS)	○	6.4.3
		CYLINDRICITY	⌭	6.4.4
FOR INDIVIDUAL OR RELATED FEATURES	PROFILE	PROFILE OF A LINE	⌒	6.5.2 (b)
		PROFILE OF A SURFACE	⌓	6.5.2 (a)
FOR RELATED FEATURES	ORIENTATION	ANGULARITY	∠	6.6.2
		PERPENDICULARITY	⊥	6.6.4
		PARALLELISM	//	6.6.3
	LOCATION	POSITION	⌖	5.2
		CONCENTRICITY	◎	5.11.3
		SYMMETRY	⌯	5.13
	RUNOUT	CIRCULAR RUNOUT	↗ •	6.7.1.2.1
		TOTAL RUNOUT	⌰ •	6.7.1.2.2

• ARROWHEADS MAY BE FILLED OR NOT FILLED

3.3.1

Definition of cylindrical OD (outside diameter) Datum

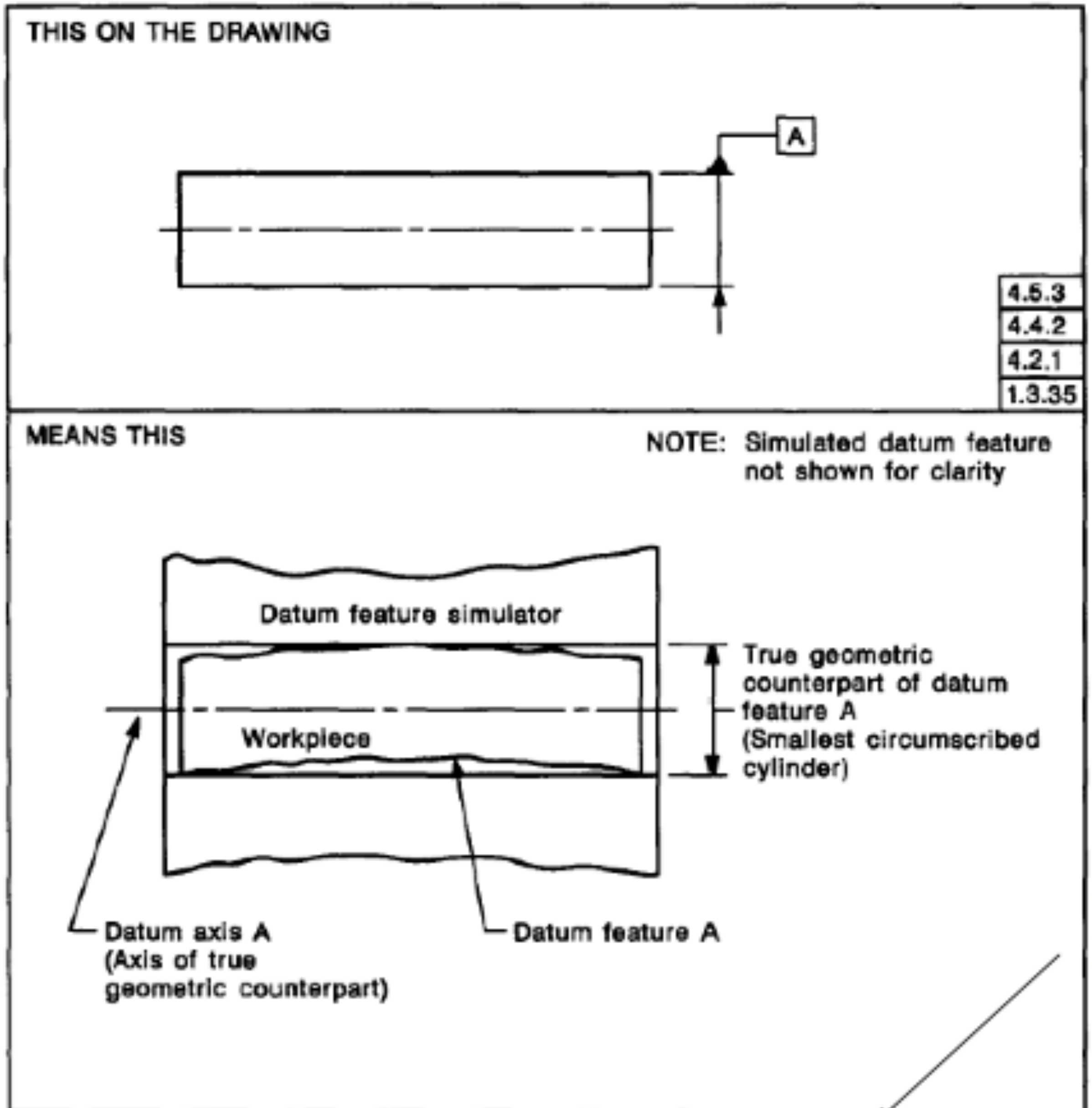


FIG. 4-11 PRIMARY EXTERNAL DATUM DIAMETER — RFS

RFS: regardless of Feature Size

Definition of cylindrical ID (inside diameter) datum

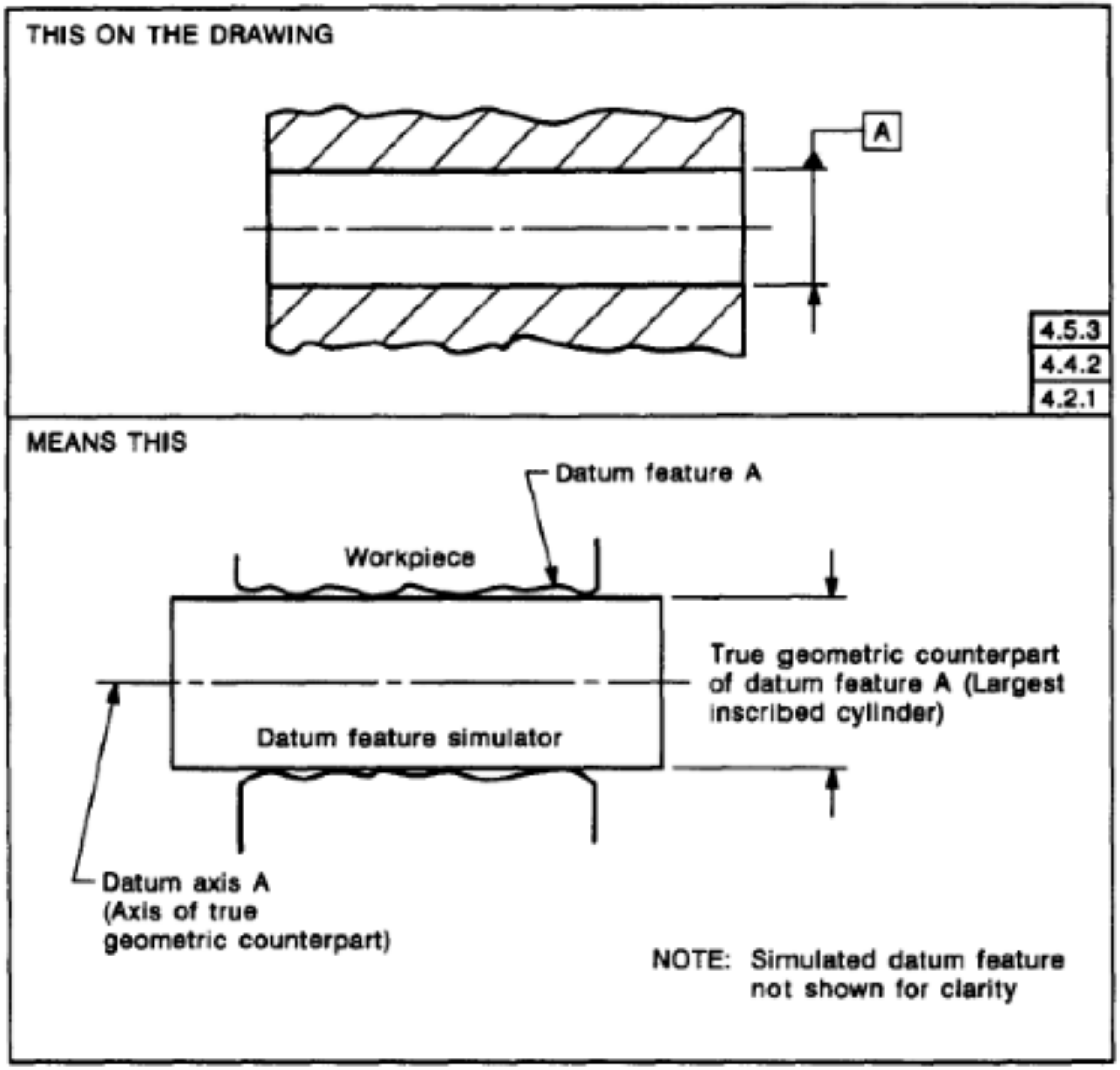


FIG. 4-12 PRIMARY INTERNAL DATUM DIAMETER — RFS

Concentricity

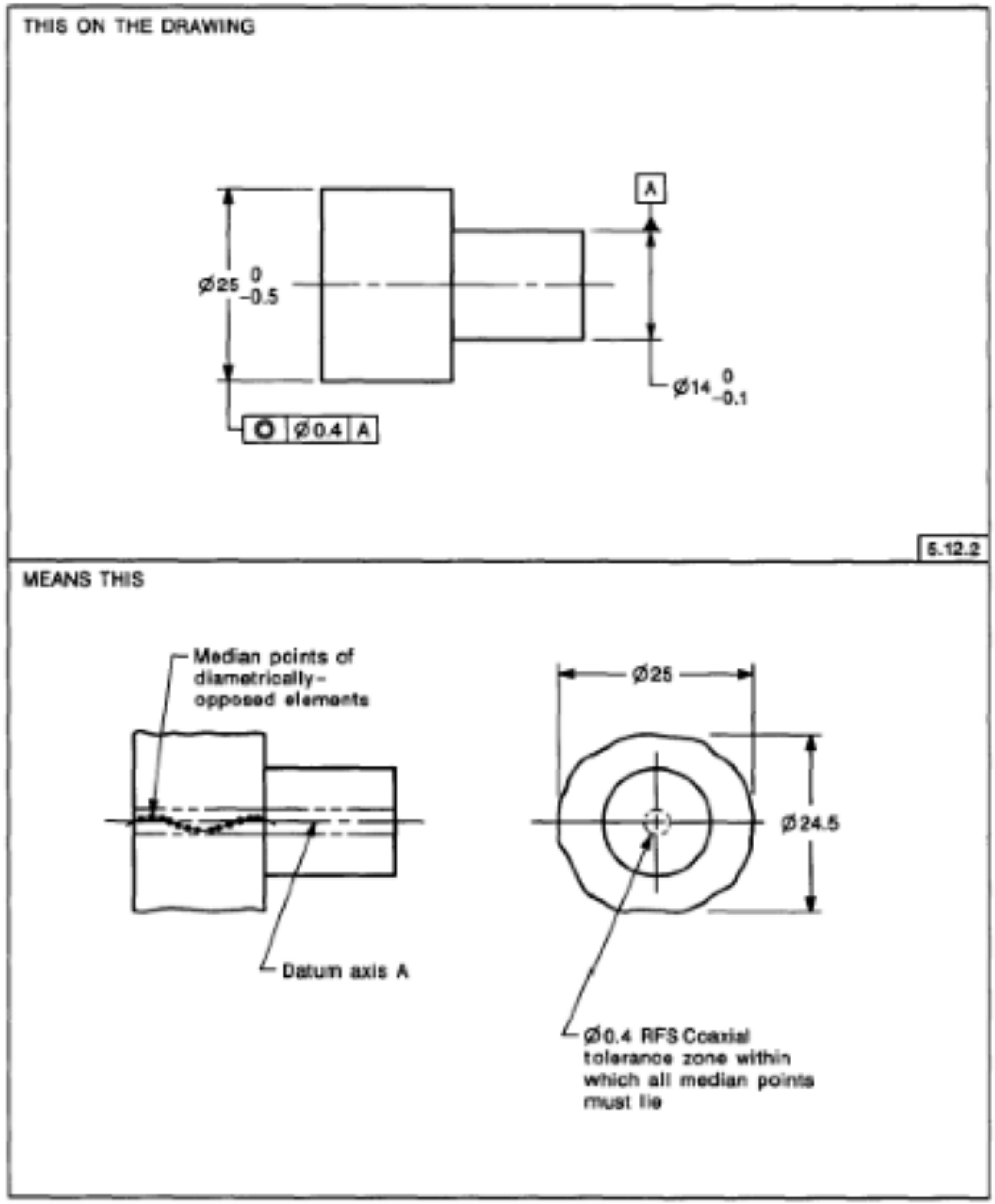


FIG. 5-58 ITEM DEPICTED IN FIG. 5-55 CONTROLLED FOR CONCENTRICITY

Circularity

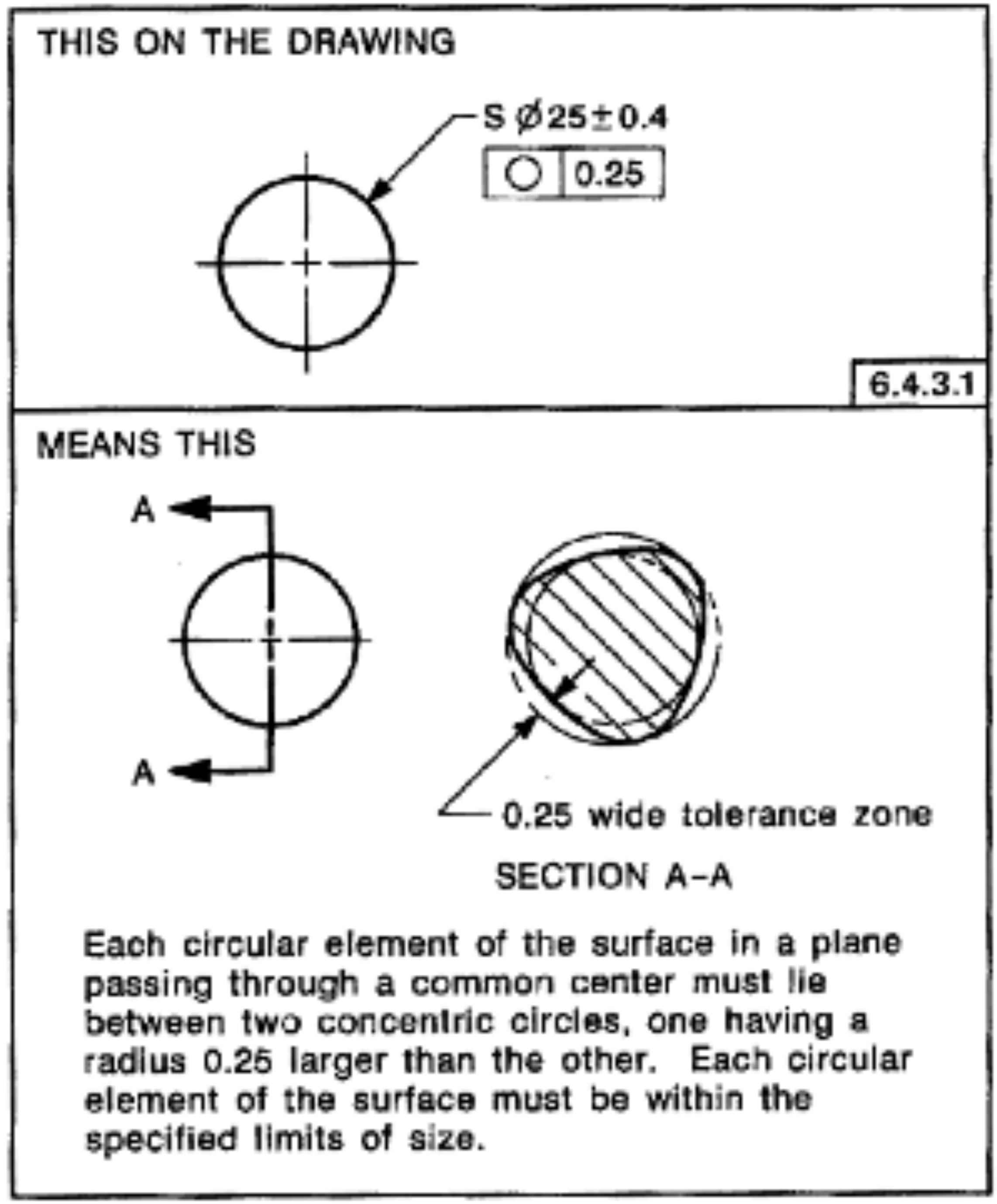


FIG. 6-9 SPECIFYING CIRCULARITY FOR A SPHERE

Cylindricity

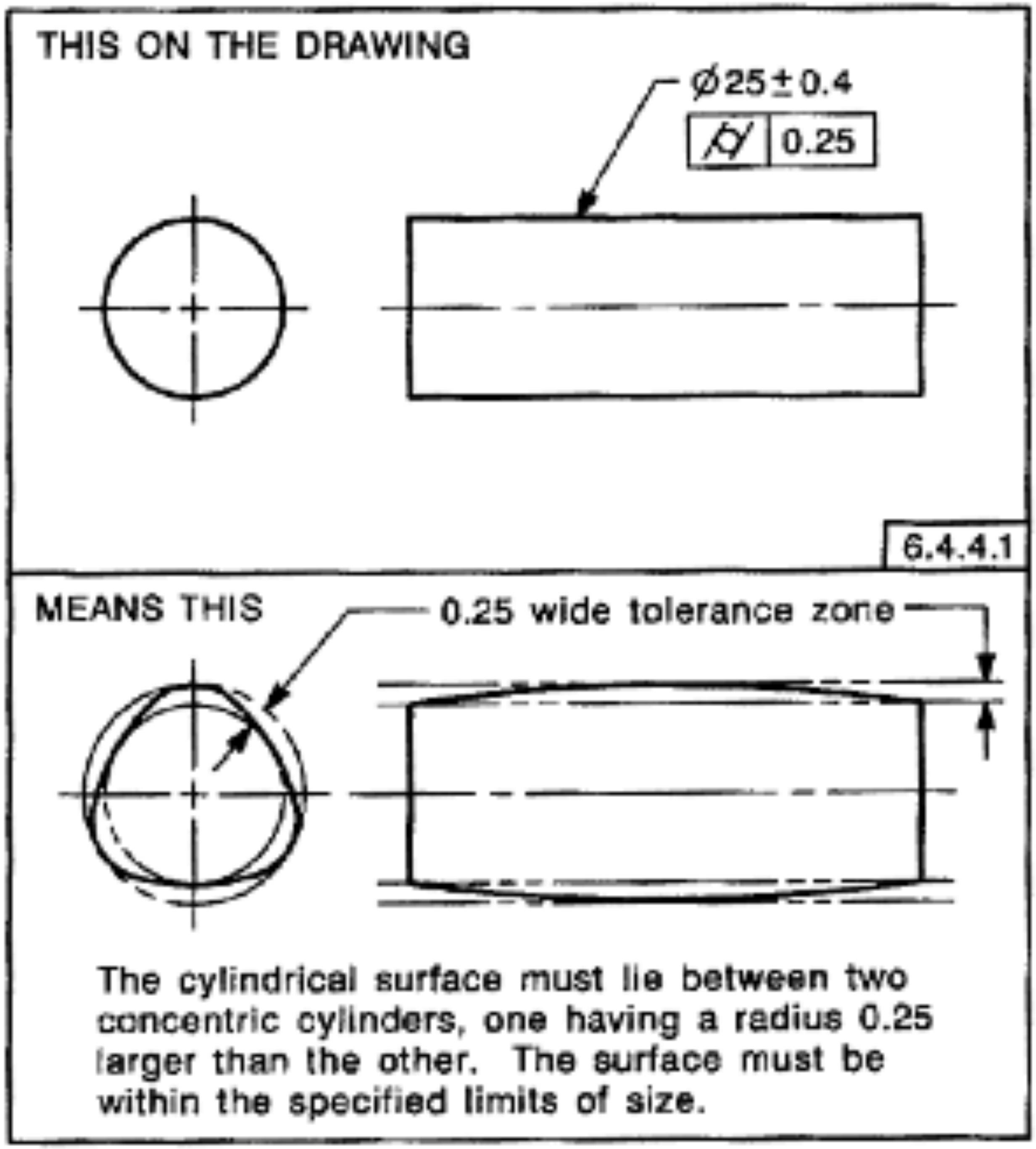


FIG. 6-10 SPECIFYING CYLINDRICITY

Surface flatness

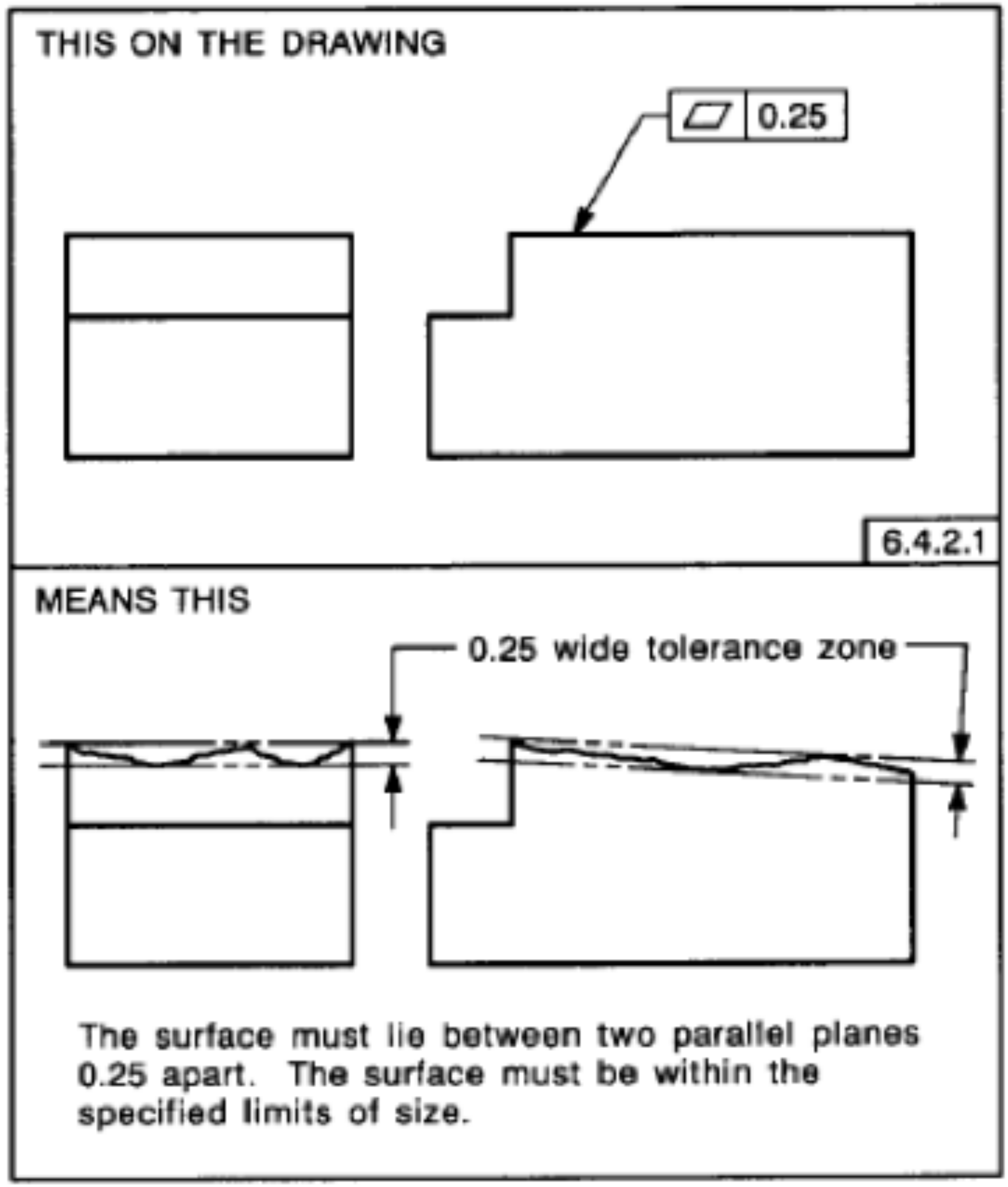
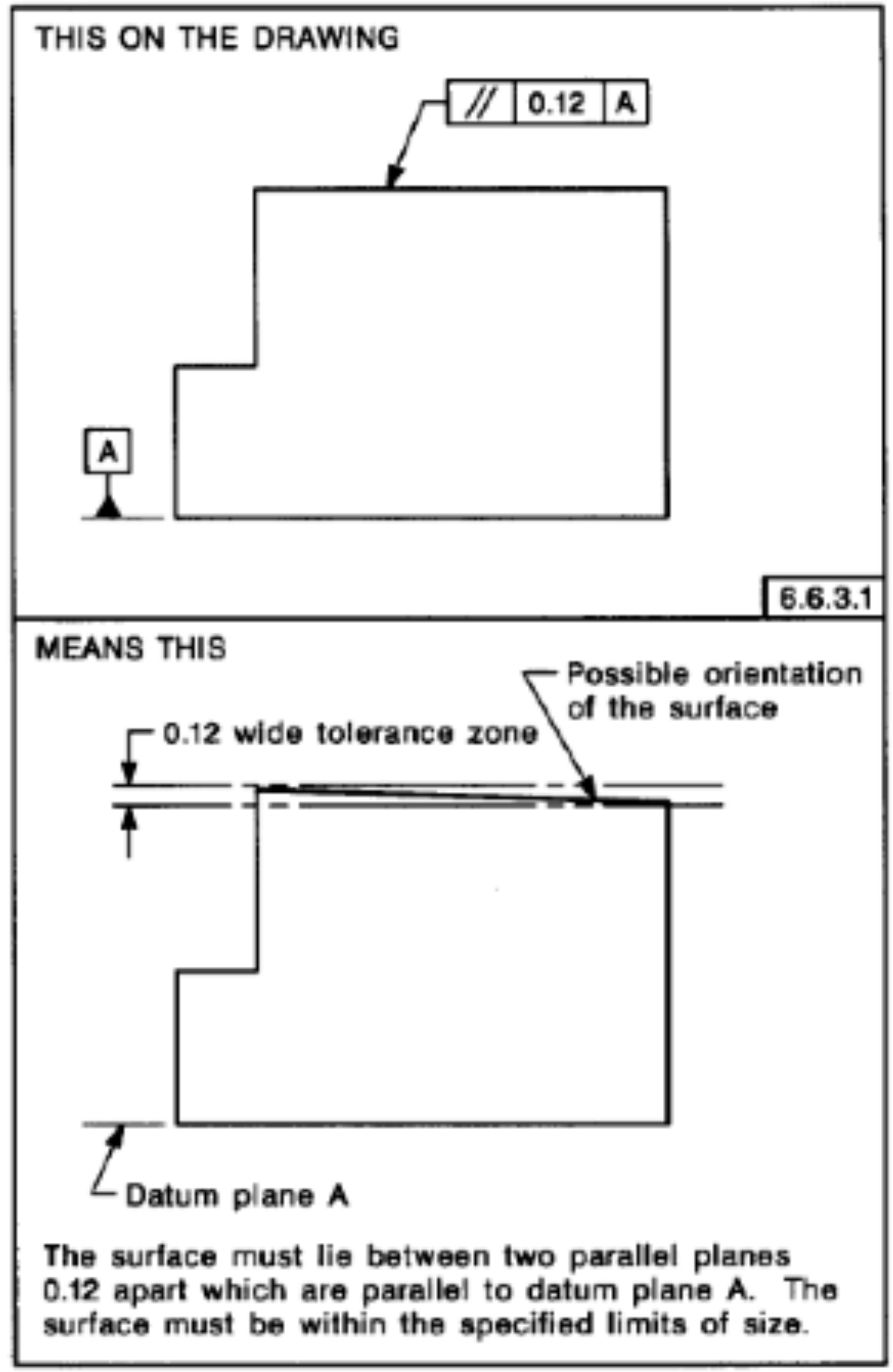


FIG. 6-7 SPECIFYING FLATNESS

Surface parallelism

General



Surface parallelism

Using tangent plane

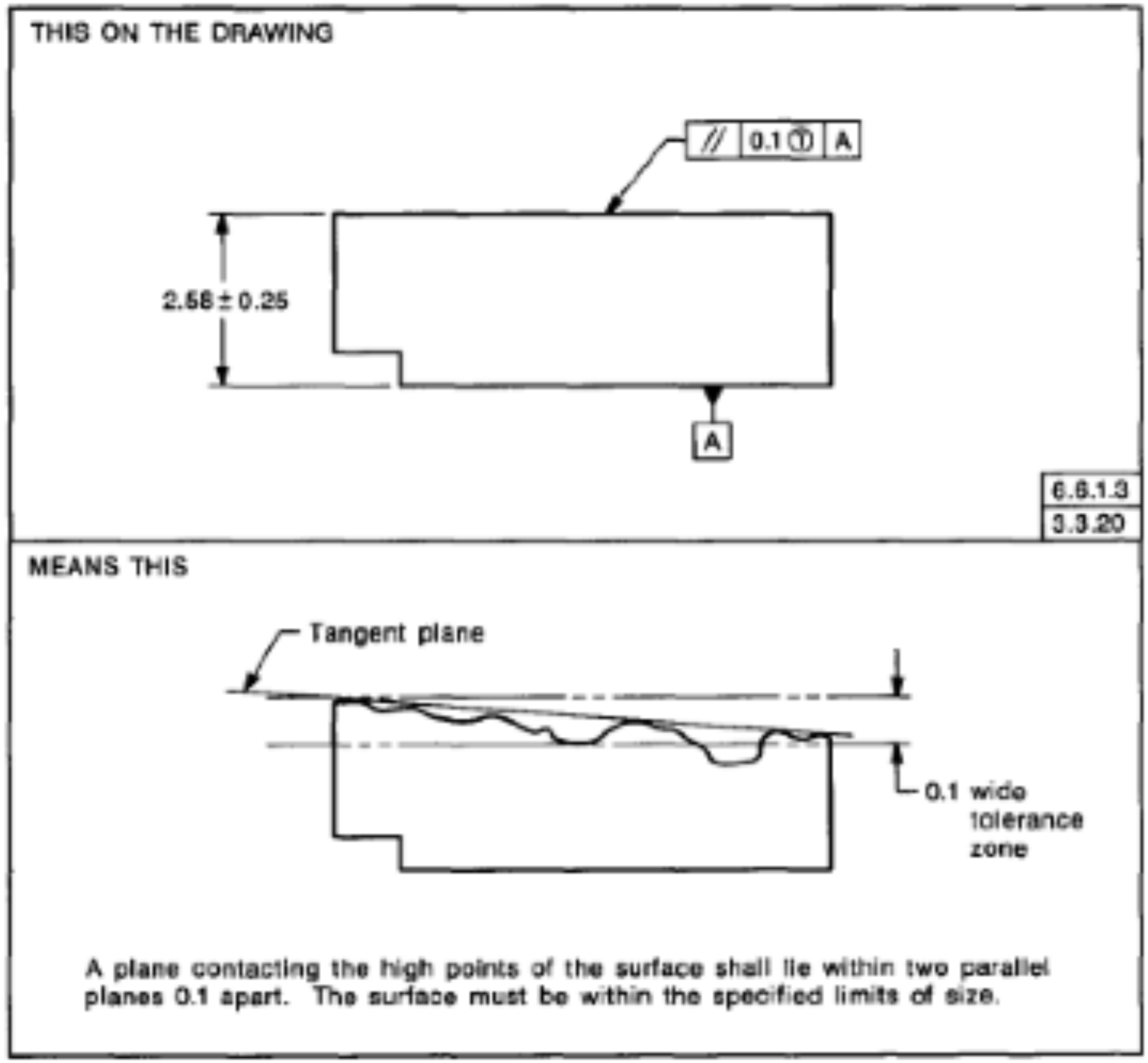


FIG. 6-43 SPECIFYING A TANGENT PLANE

Perpendicularity

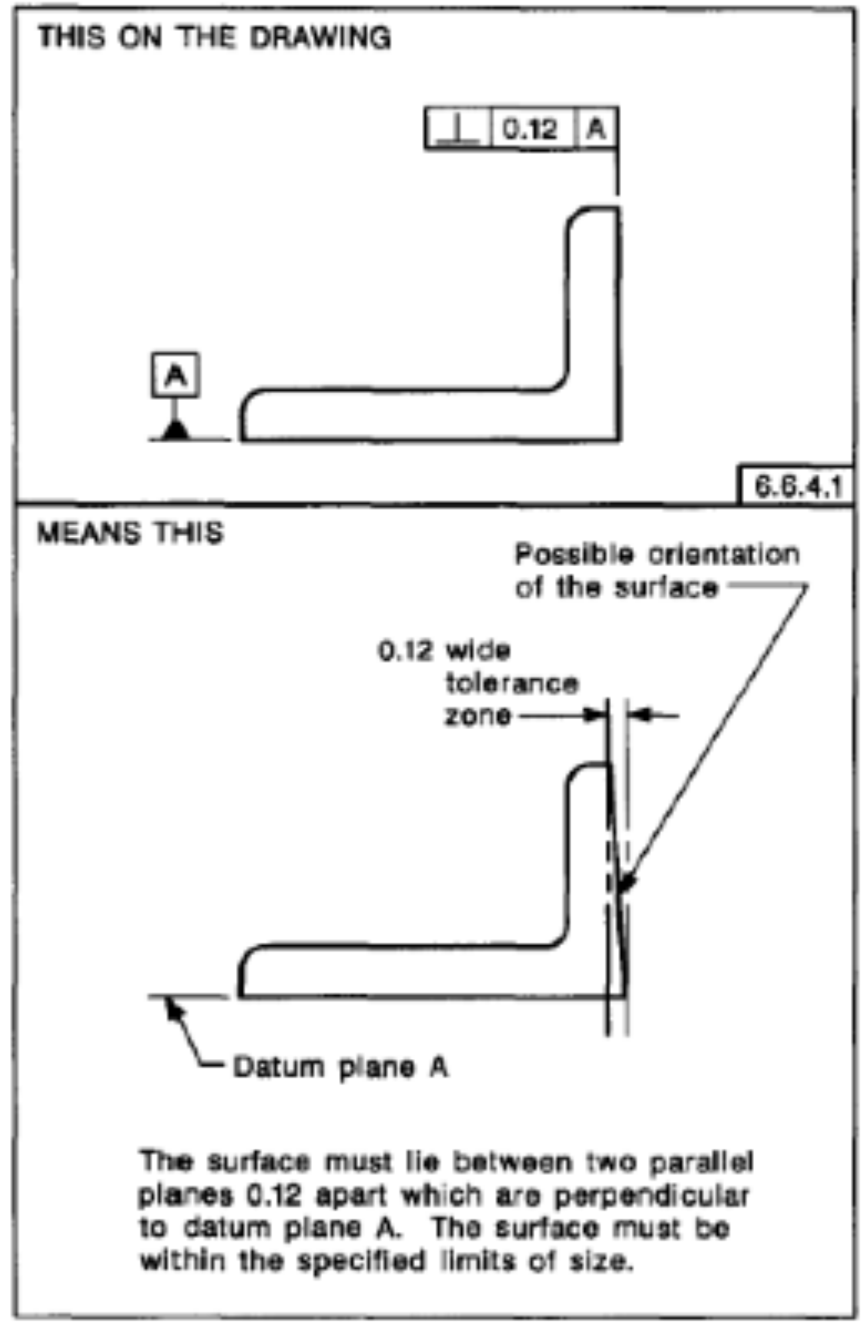


FIG. 6-34 SPECIFYING PERPENDICULARITY FOR A PLANE SURFACE

Perpendicularity

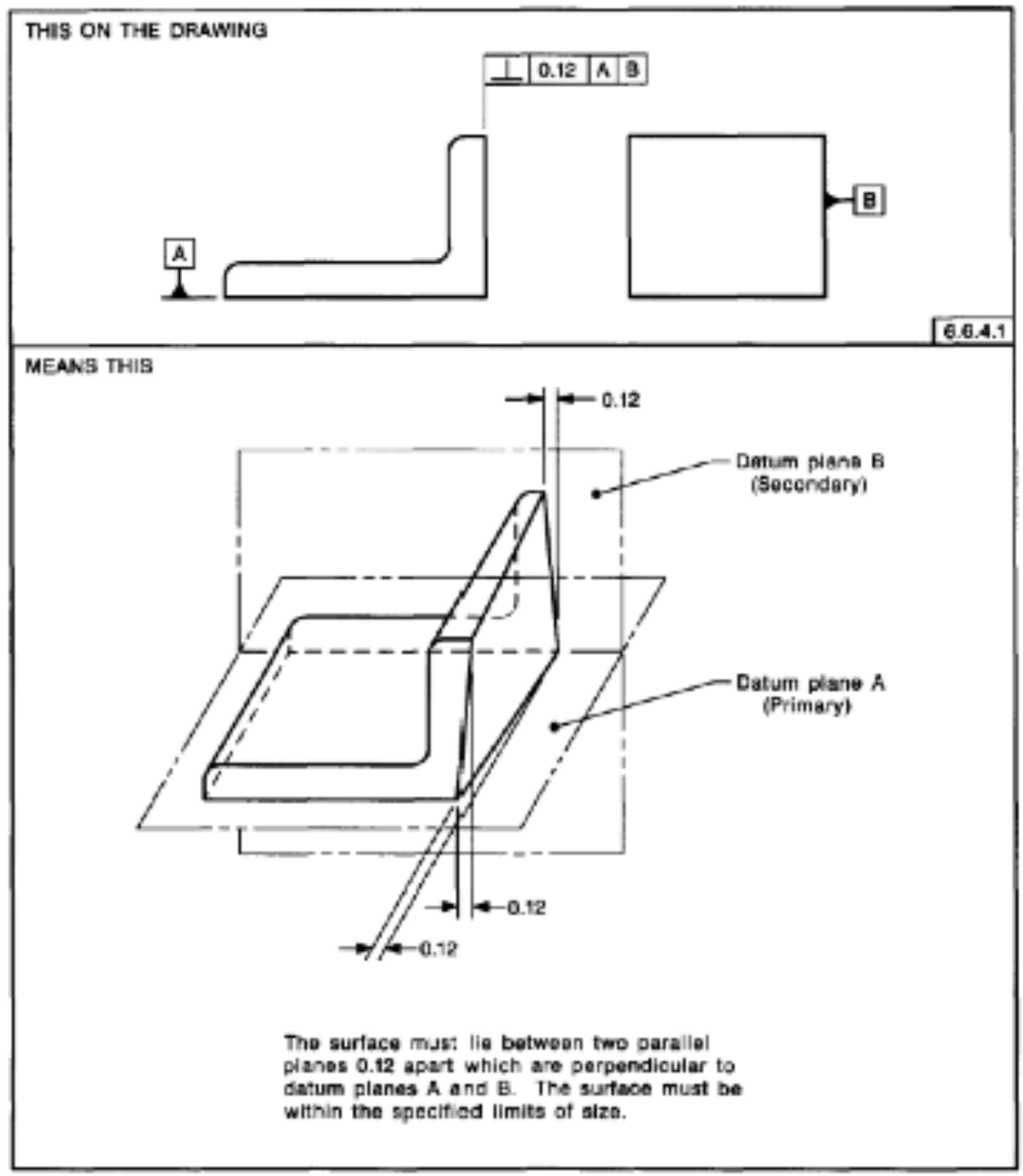
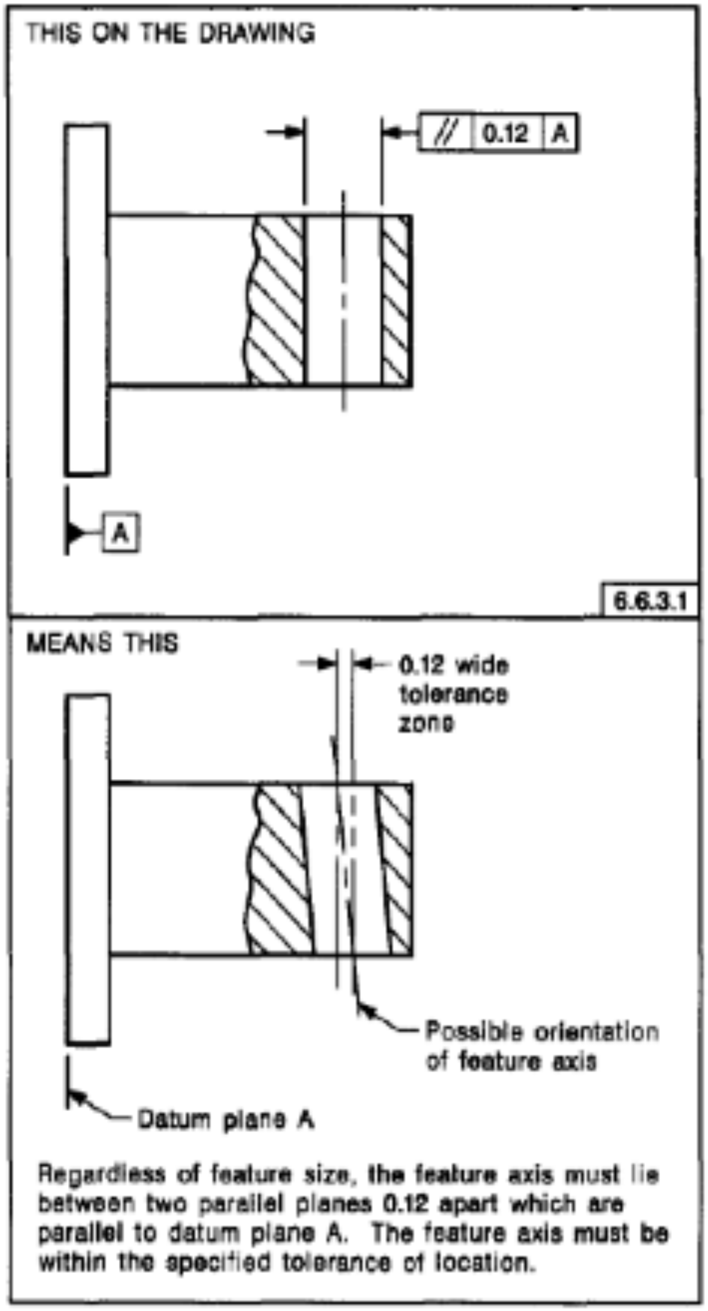
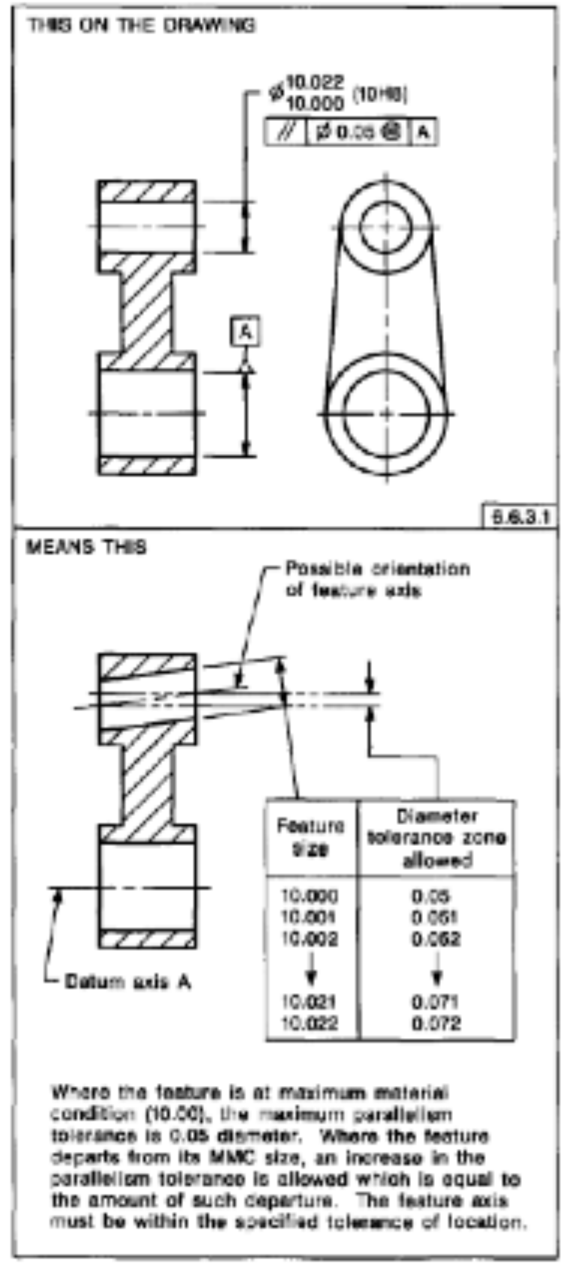


FIG. 6-35 SPECIFYING PERPENDICULARITY FOR A PLANE SURFACE RELATIVE TO TWO DATUMS

Parallelism for axis





Runout

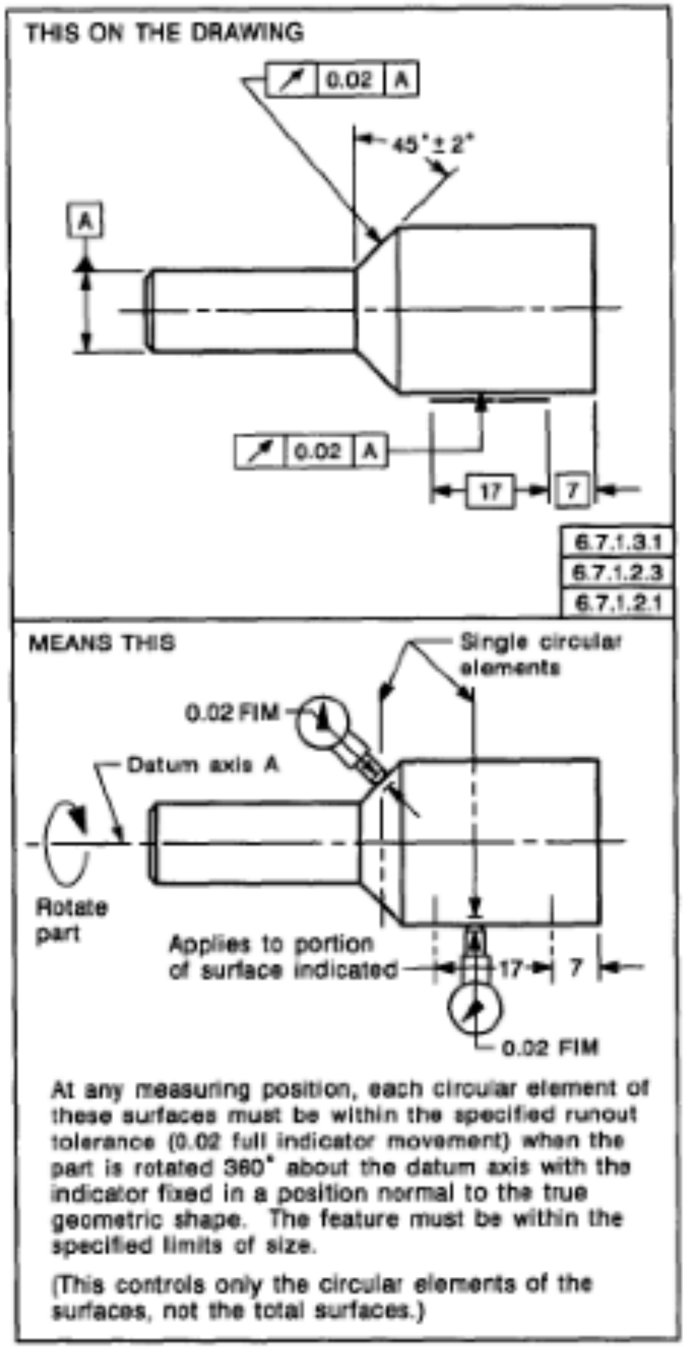


FIG. 6-47 SPECIFYING CIRCULAR RUNOUT RELATIVE TO A DATUM DIAMETER

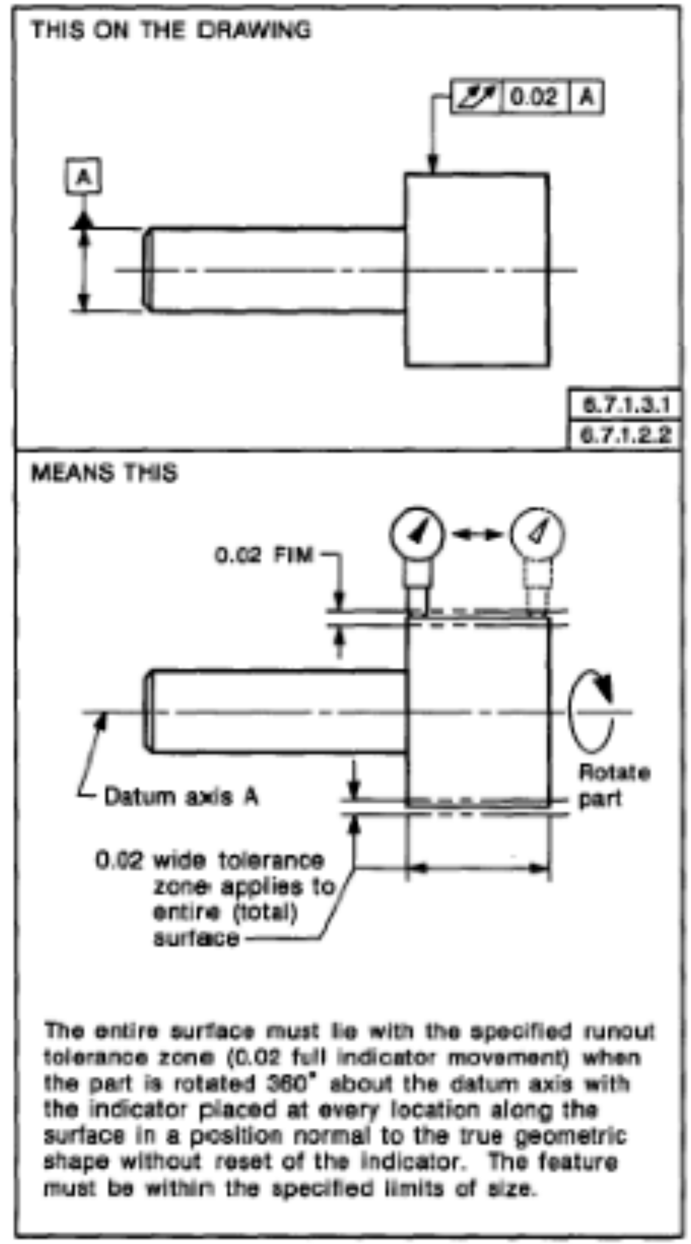
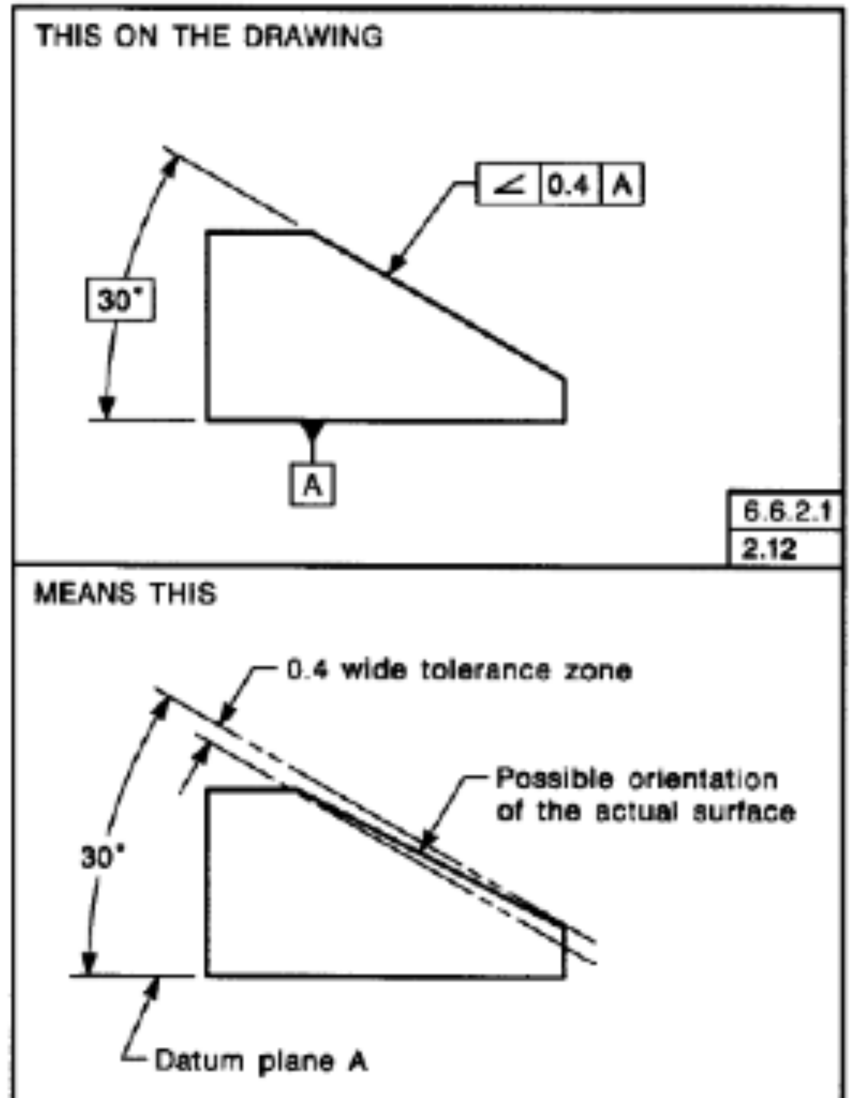


FIG. 6-48 SPECIFYING TOTAL RUNOUT RELATIVE TO A DATUM DIAMETER

Surface orientation

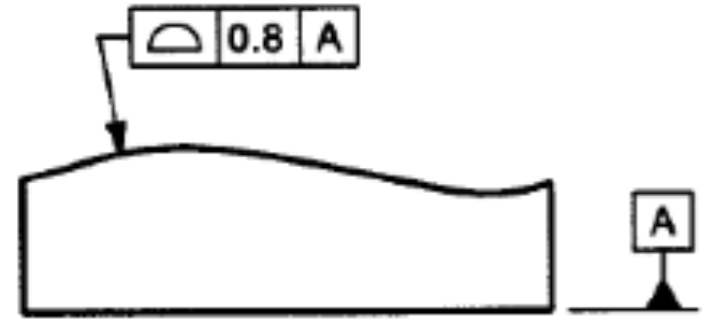


The surface must lie between two parallel planes 0.4 apart which are inclined at 30° to datum plane A. The surface must be within the specified limits of size. See figure 2-14.

FIG. 6-27 SPECIFYING ANGULARITY FOR A PLANE SURFACE

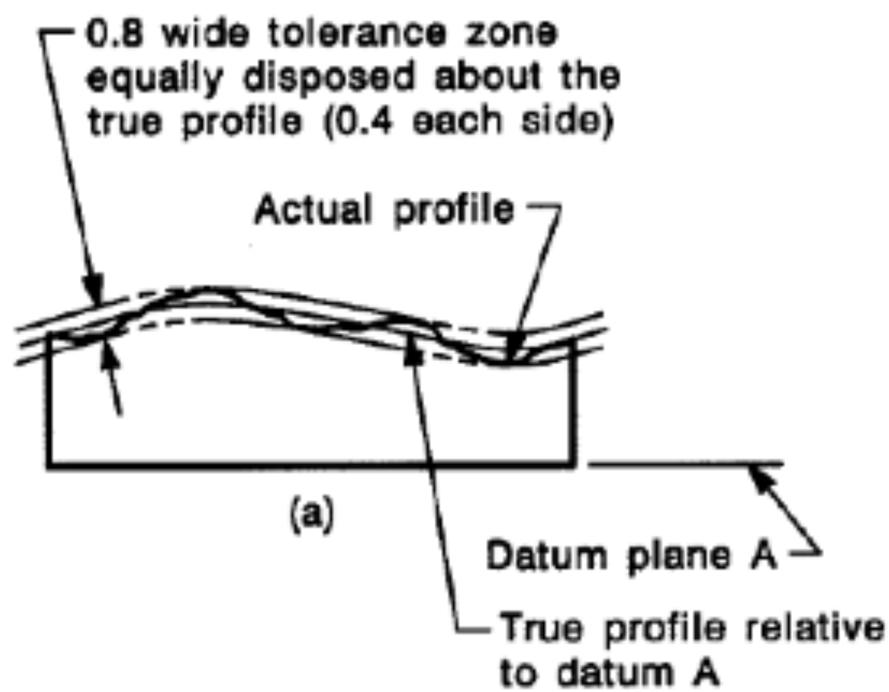
Profile tolerance

THIS ON THE DRAWING



(a) Bilateral tolerance

MEANS THIS



Use of feature control frames

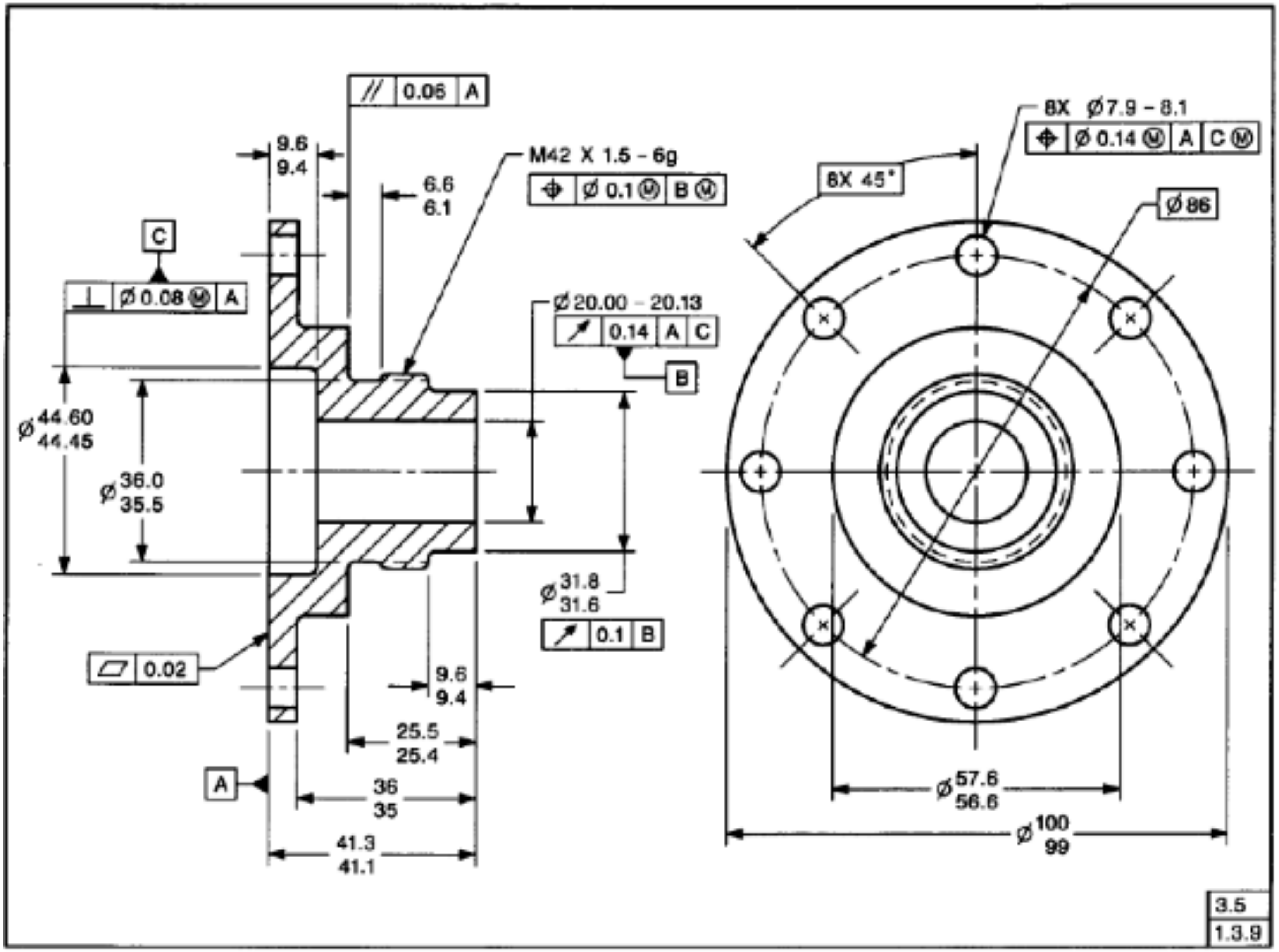
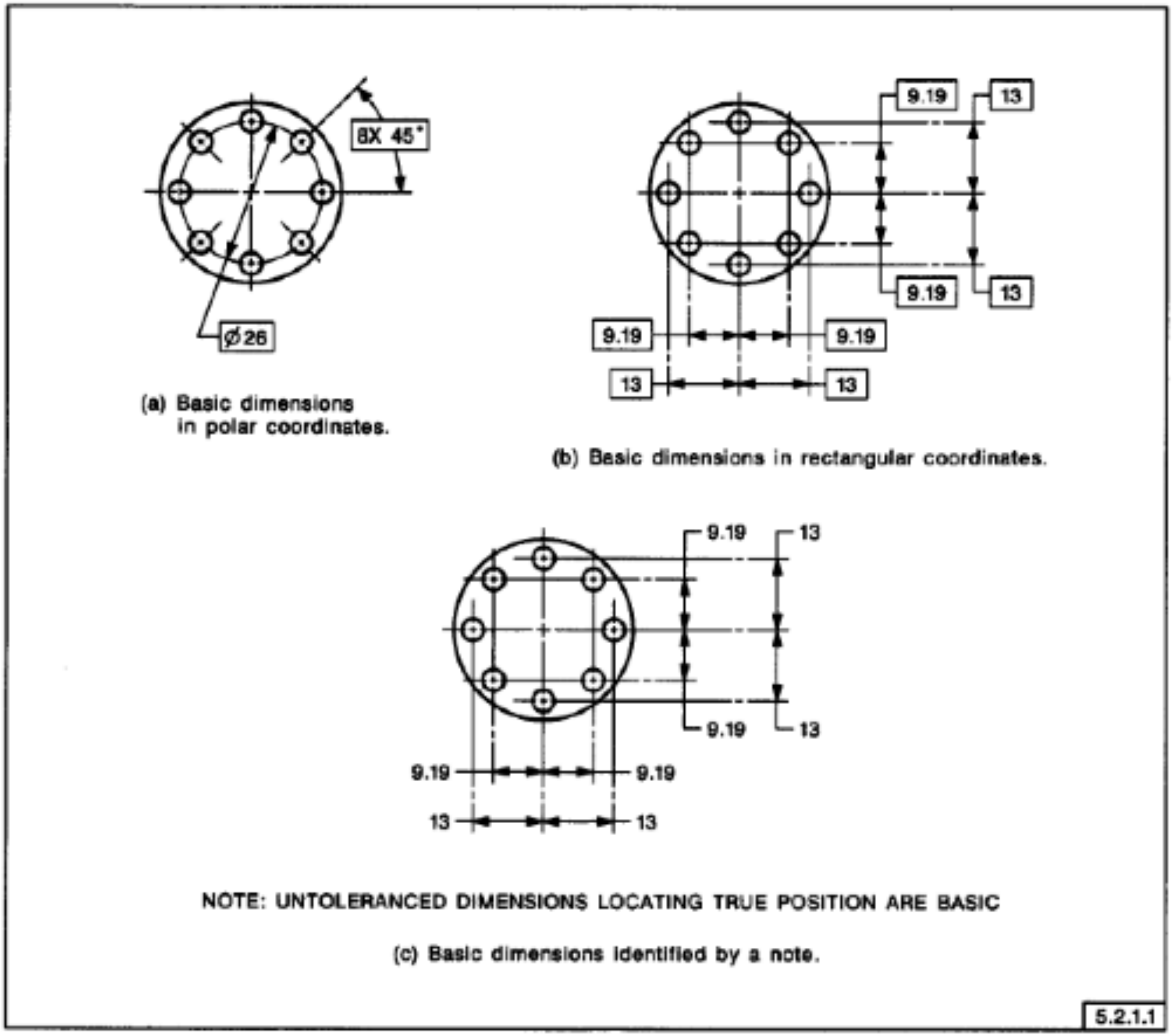


FIG. 3-25 FEATURE CONTROL FRAME PLACEMENT

Basic dimensions



(c) Basic dimensions identified by a note.

Geometric dimensioning & tolerancing
Meaning of basic tolerances

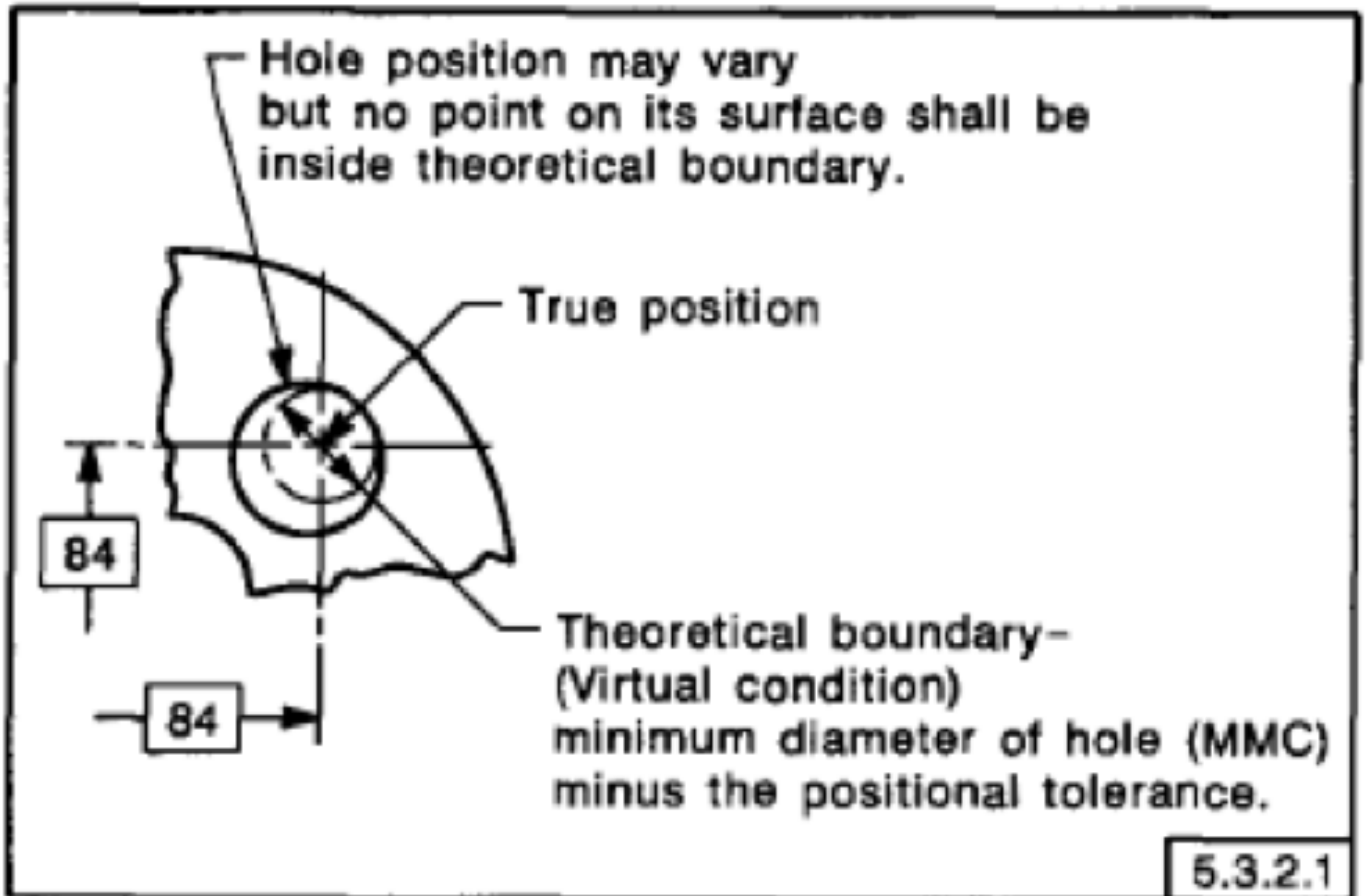


FIG. 5-5 BOUNDARY FOR SURFACE OF HOLE AT MMC

Tolerancing using basic dimensions

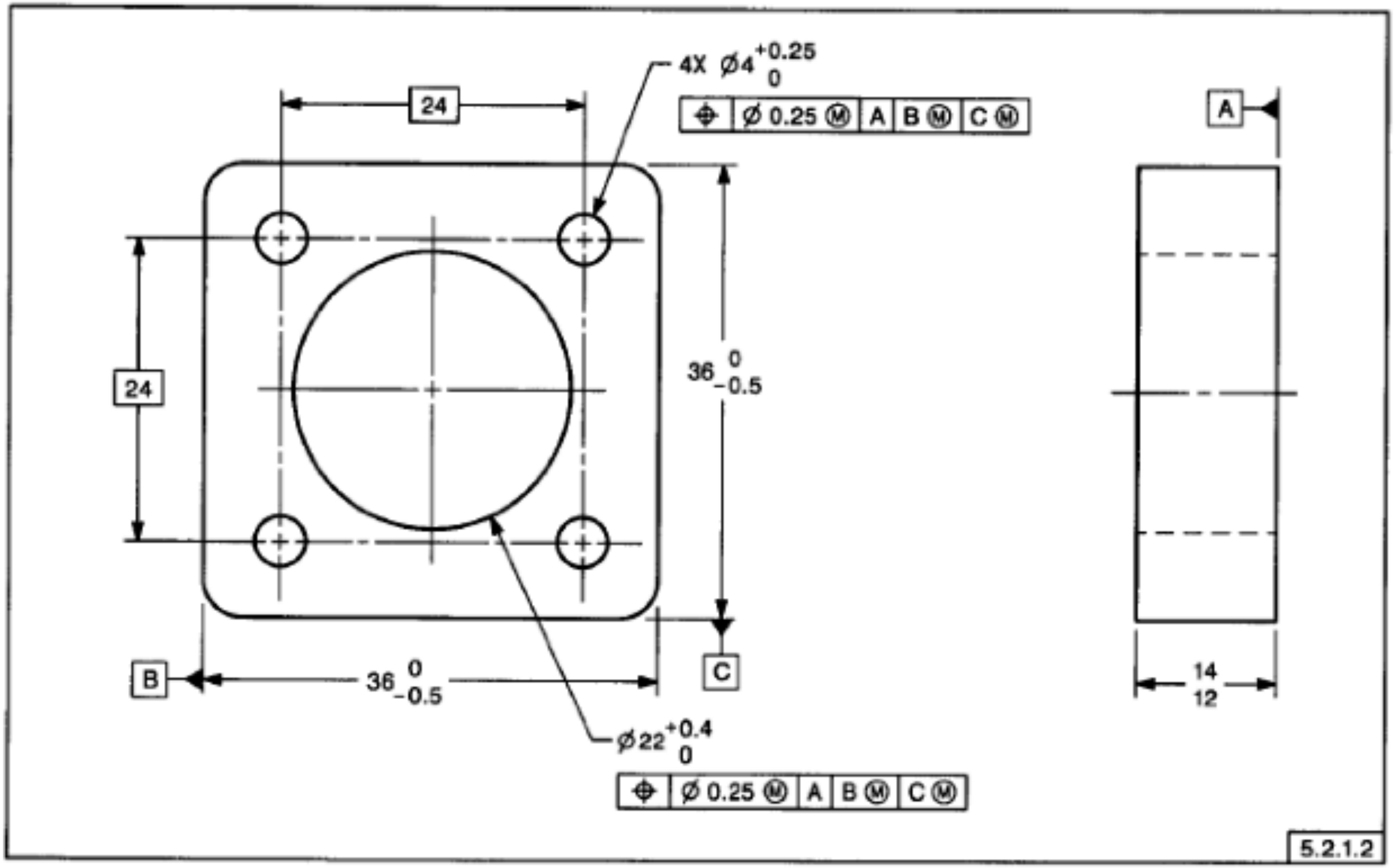


FIG. 5-4 POSITIONAL TOLERANCING AT MMC RELATIVE TO DATUM FEATURE CENTER PLANES

Example of multiple features

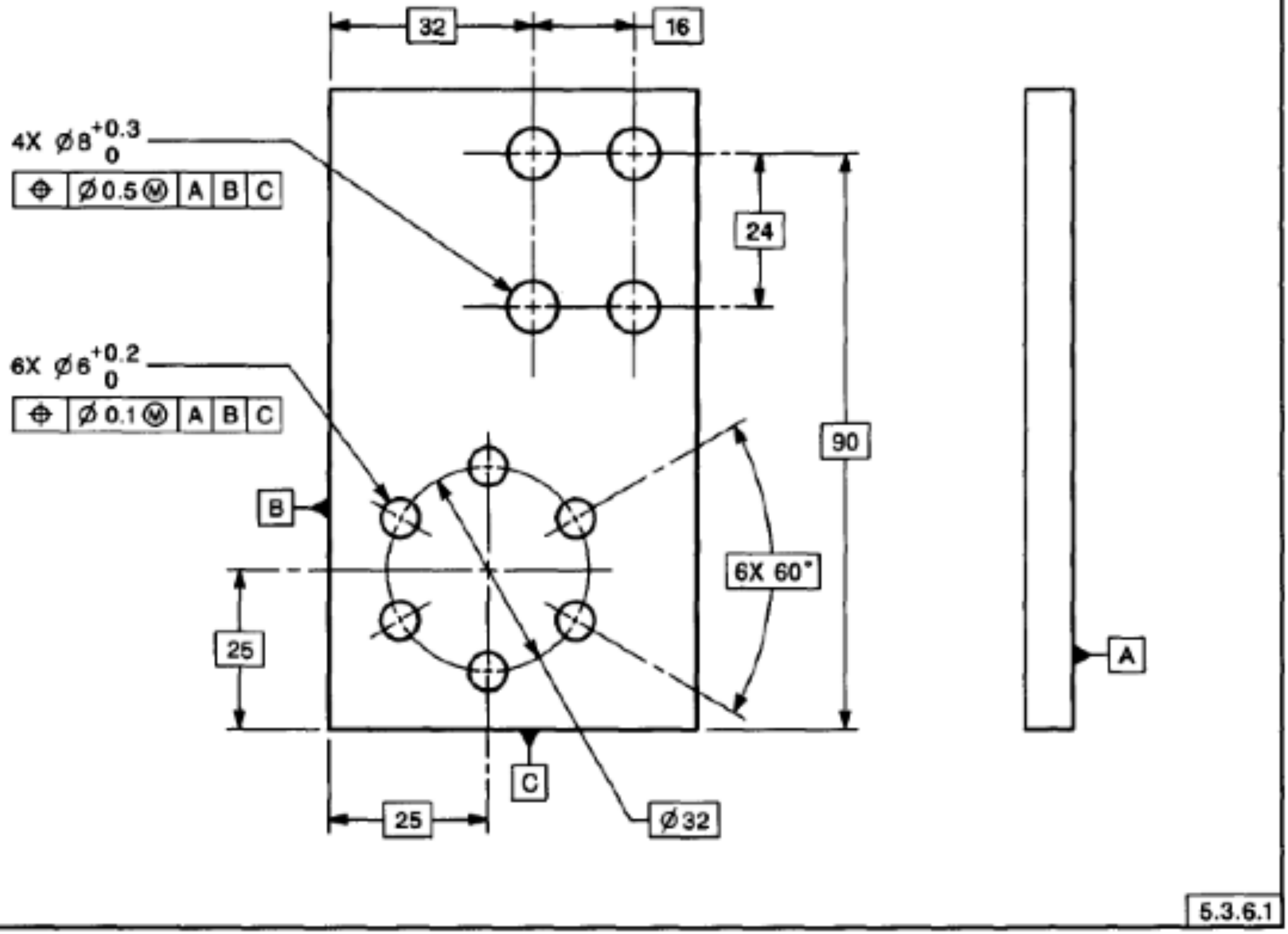


FIG. 5-16 MULTIPLE PATTERNS OF FEATURES

Tolerance Zones