Dimensioning and Tolerancing

Dimensioning

- Before an object can be built, complete information about both the size and shape of the object must be available. The exact shape of an object is communicated through orthographic drawings, which are developed following standard drawing practices. The process of adding size information to a drawing is known as dimensioning the drawing.

Dimensioning

- **Geometrics** is the science of specifying and tolerancing the shapes and locations of features on objects. Once the shape of a part is defined with an orthographic drawings, the size information is added also in the form of **dimensions**.
- Dimensioning a drawing also identifies the tolerance (or accuracy) required for each dimension.

Dimensioning

- If a part is dimensioned properly, then the intent of the designer is clear to both the person making the part and the inspector checking the part.
- A fully defined part has three elements: graphics, dimensions, and words (notes).

Size and Location Dimensions

- A well dimensioned part will *communicate* the size and location requirements for each feature. Communications is the fundamental purpose of dimensions.
- Parts are dimensioned based on two criteria:
  - Basic size and locations of the features.
  - Details of a part's construction and for manufacturing.

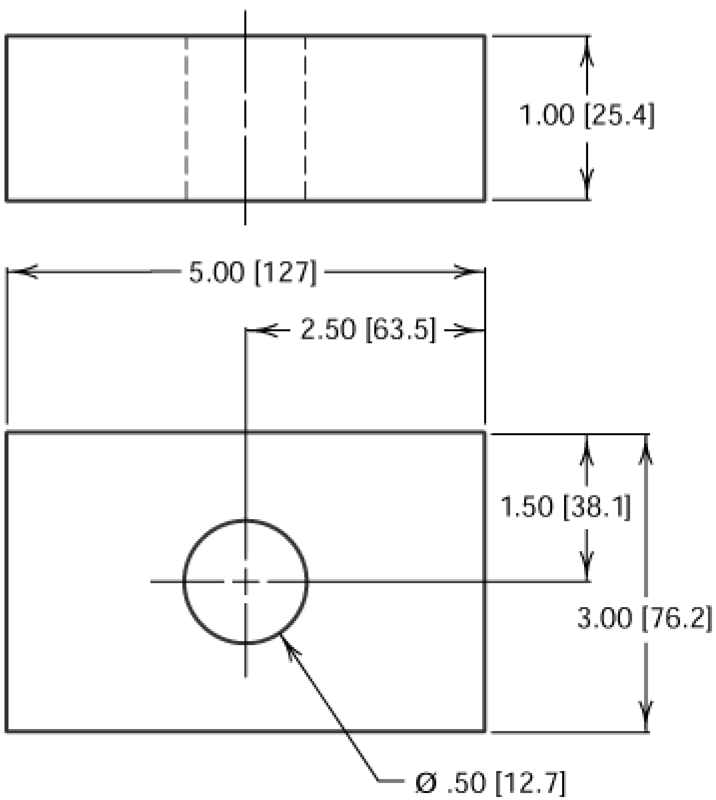
Unit of measure

- On a drawing used in American industry, all dimensions are in inches, unless otherwise stated.
- Most countries outside of the United States use the metric system of measure, or the international system of units (SI), which is based on the meter.
- The SI system is being used more in the United States because of global trade and multinational company affiliations

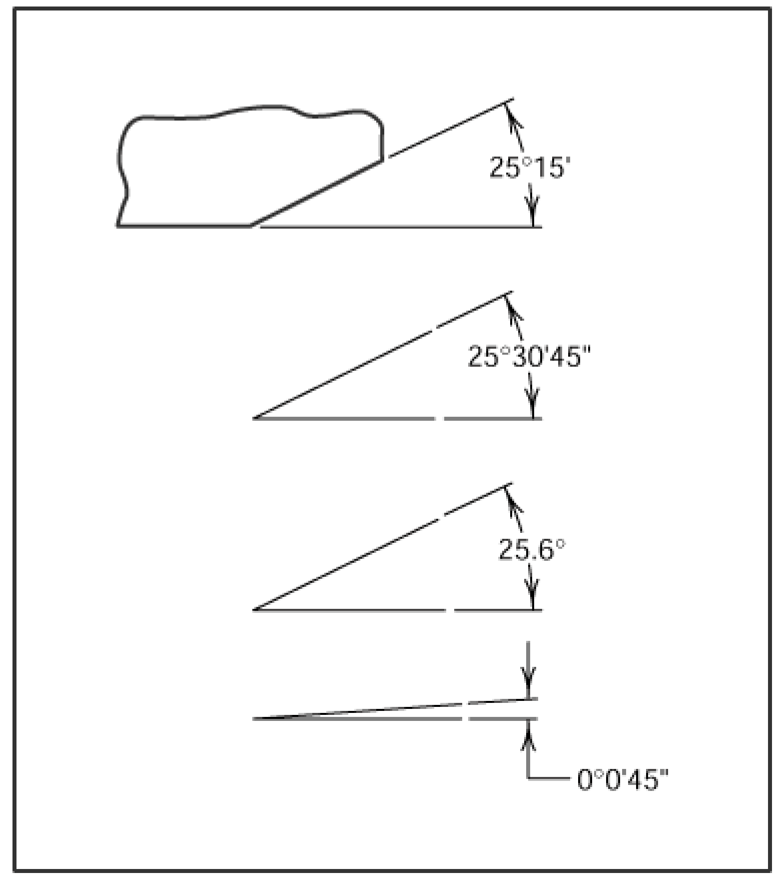
Unit of measure

- Occasionally, a company will used dual dimensioning, that is, both metric and English measurements on a drawing.
- Angular dimensions are shown either in decimal degrees or in degrees, minutes, and seconds.

(A) Position Method

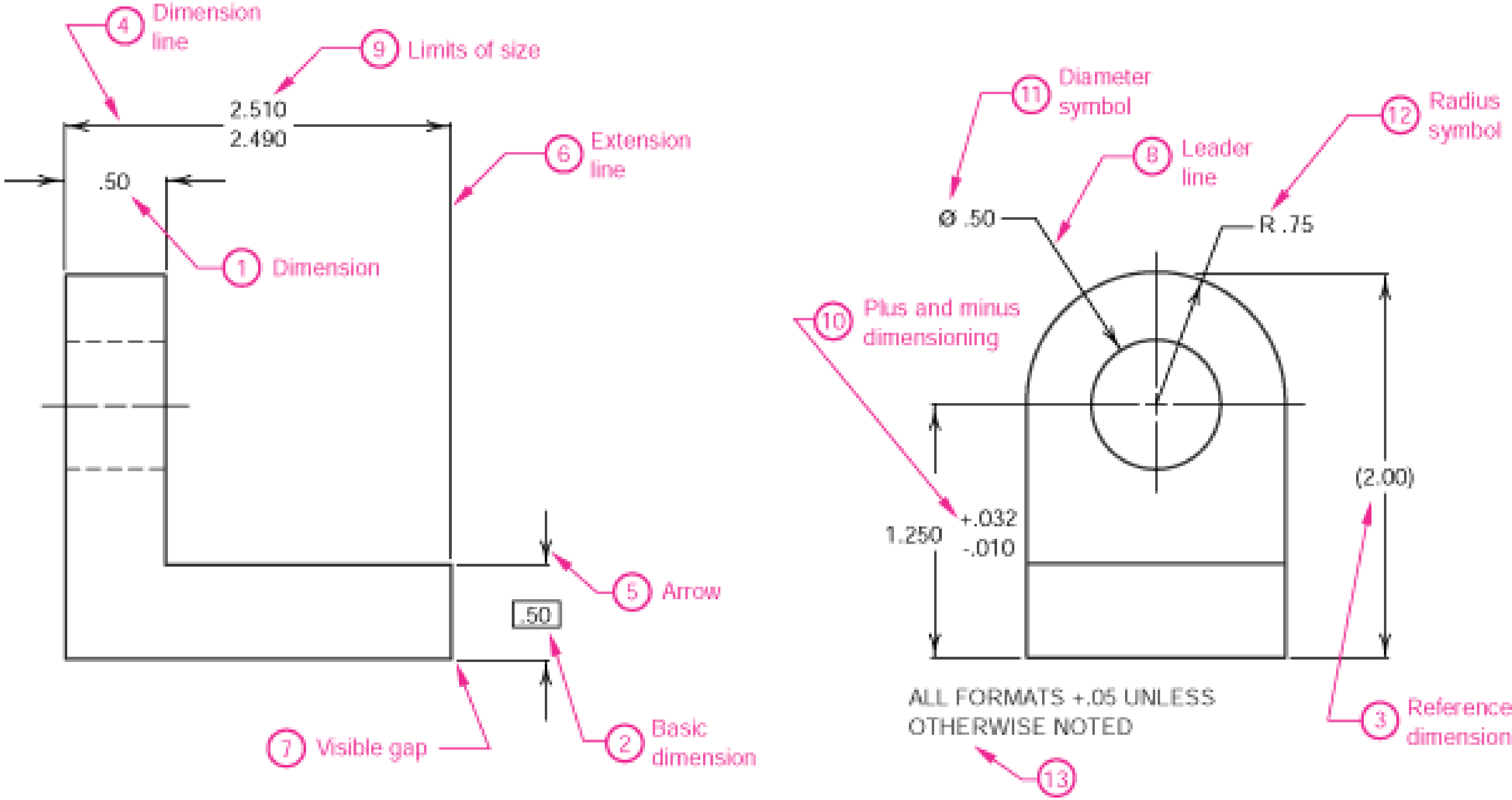


(B) Bracket Method

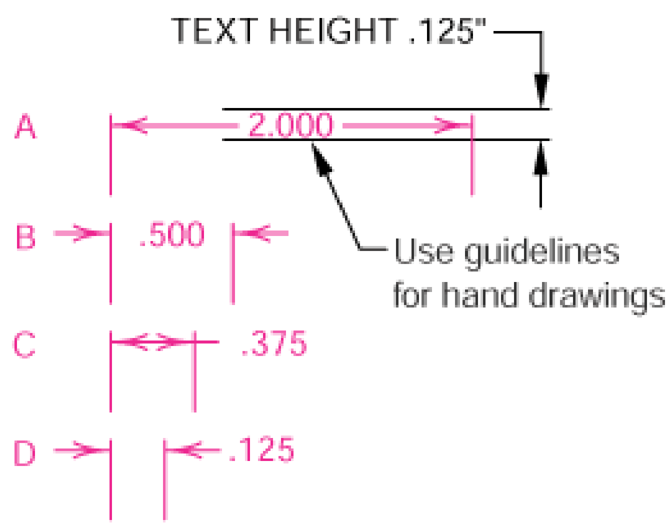


Terminology

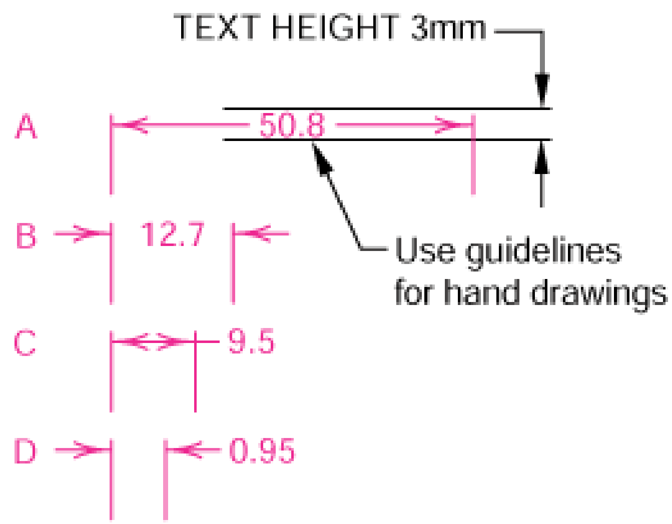
- **Dimension** is the numerical value that defines the size or geometric characteristic of a feature.
- **Basic dimension** is the numerical value defining the theoretically exact size of a feature.
- **Reference dimension** is the numerical value enclosed in parentheses provided for information only and is not used in the fabrication of the part.



4 Dimension line
9 Limits of size
2.510
2.490
6 Extension line
.50
1 Dimension
5 Arrow
.50
7 Visible gap
2 Basic dimension
11 Diameter symbol
12 Radius symbol
8 Leader line
Ø .50
R .75
10 Plus and minus dimensioning
1.250
+.032
-.010
(2.00)
ALL FORMATS +.05 UNLESS OTHERWISE NOTED
13
3 Reference dimension



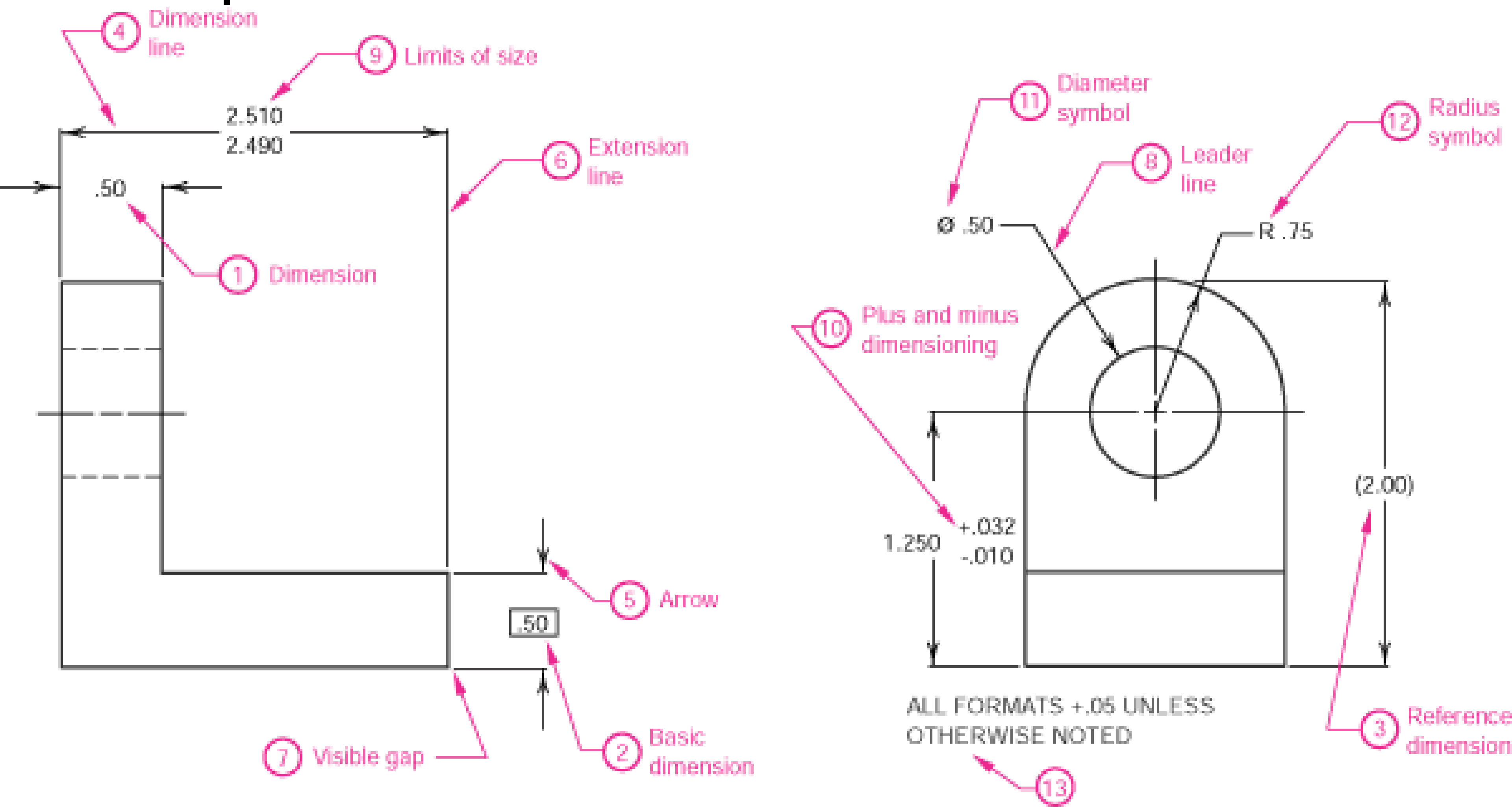
Decimal dimensioning



Millimeter dimensioning

Terminology

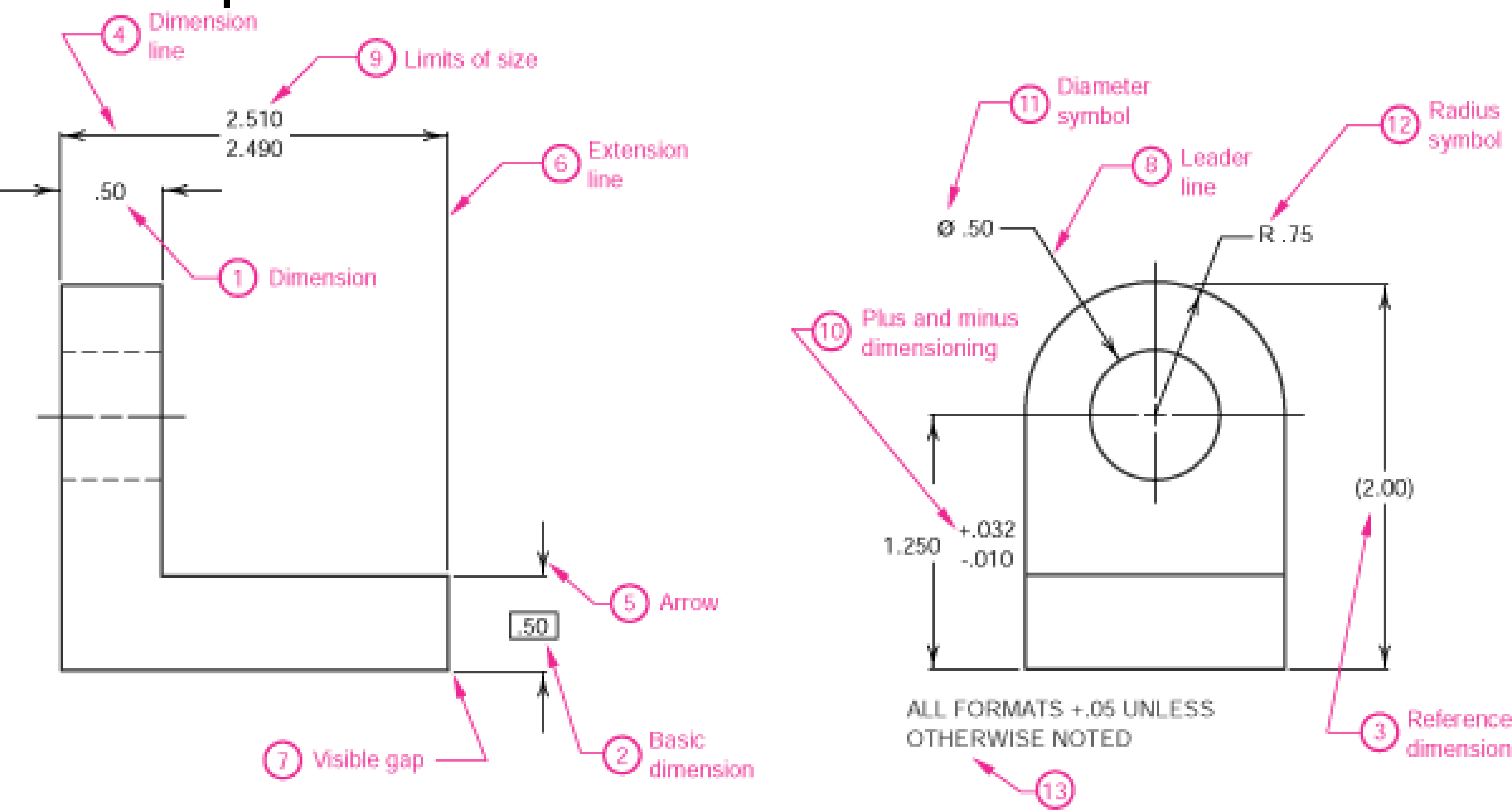
- **Dimension line** is the thin solid line which shows the extent and direction of a dimension.
- **Arrows** are placed at the ends of dimension lines to show the limits of the dimension.
- **Extension line** is the thin solid line perpendicular to a dimension line indicating which feature is associated with the dimension.



4 Dimension line
9 Limits of size
2.510
2.490
6 Extension line
.50
1 Dimension
5 Arrow
.50
7 Visible gap
2 Basic dimension
11 Diameter symbol
8 Leader line
12 Radius symbol
Ø .50
R .75
10 Plus and minus dimensioning
+.032
-.010
1.250
(2.00)
ALL FORMATS ±.05 UNLESS OTHERWISE NOTED
13
3 Reference dimension

Terminology

- **Leader line** is the thin solid line used to indicate the feature with which a dimension, note, or symbol is associated.
- **Tolerance** is the amount a particular dimension is allowed to vary.
- **Plus and minus dimensioning** is the allowable positive and negative variance from the dimension specified.

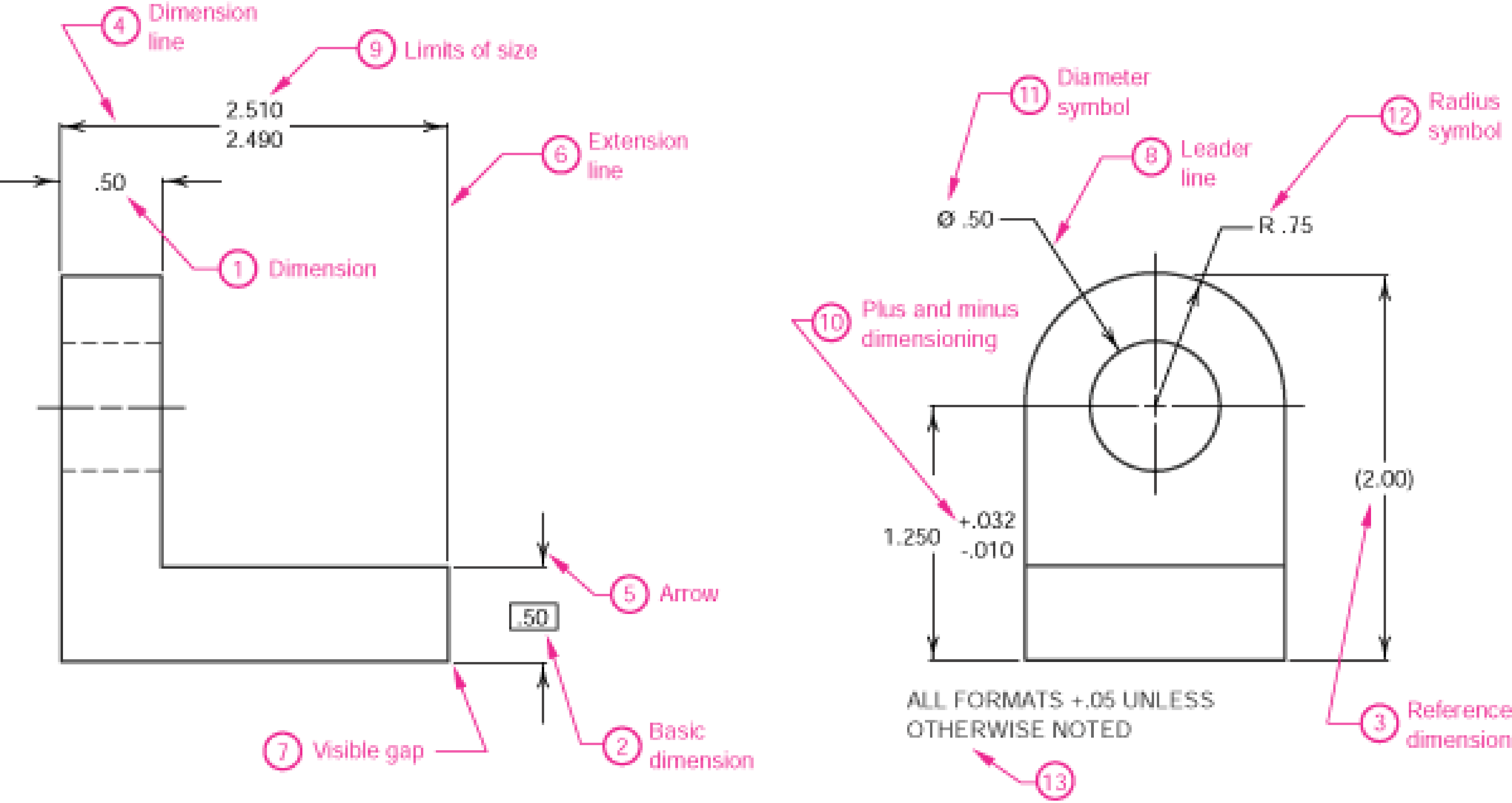

4 Dimension line
9 Limits of size
2.510
2.490
6 Extension line
.50
1 Dimension
5 Arrow
.50
7 Visible gap
2 Basic dimension
11 Diameter symbol
12 Radius symbol
8 Leader line
Ø .50
R .75
10 Plus and minus dimensioning
1.250
+.032
-.010
(2.00)
ALL FORMATS +.05 UNLESS OTHERWISE NOTED
13
3 Reference dimension

Terminology

- **Limits of size** is the largest acceptable size and the minimum acceptable size of a feature.
  - The largest acceptable size is expressed as the maximum material condition (MMC)
  - The smallest acceptable size is expressed as the least material condition (LMC).

Terminology

- **Diameter symbol** is the symbol which is placed preceding a numerical value indicating that the associated dimension shows the diameter of a circle. The symbol used is the Greek letter *phi*.
- **Radius symbol** is the symbol which is placed preceding a numerical value indicating that the associated dimension shows the radius of a circle. The radius symbol used is the capital letter *R*.


4 Dimension line
9 Limits of size
2.510
2.490
6 Extension line
.50
1 Dimension
5 Arrow
.50
7 Visible gap
2 Basic dimension
11 Diameter symbol
12 Radius symbol
8 Leader line
Ø .50
R .75
10 Plus and minus dimensioning
1.250
+.032
-.010
(2.00)
ALL FORMATS ±.05 UNLESS OTHERWISE NOTED
13
3 Reference dimension

Terminology

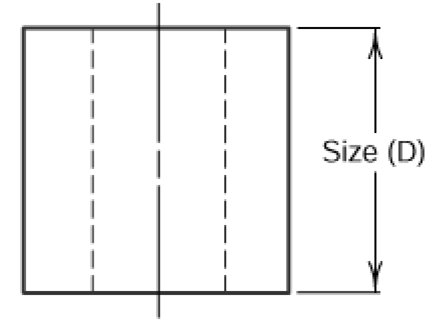
- **Datum** is the theoretically exact point used as a reference for tabular dimensioning.

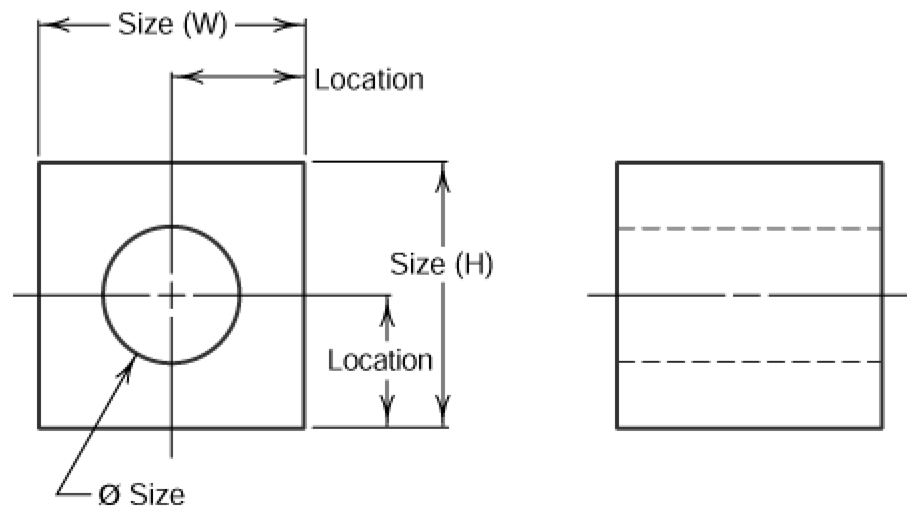
Basic Concepts

- Dimensions are used to describe the size and location of features on parts for manufacture. The basic criterion is, "What information is necessary to make the object?" Dimensions should not be excessive, either through duplication or dimensioning a feature more than one way.

Basic Concepts

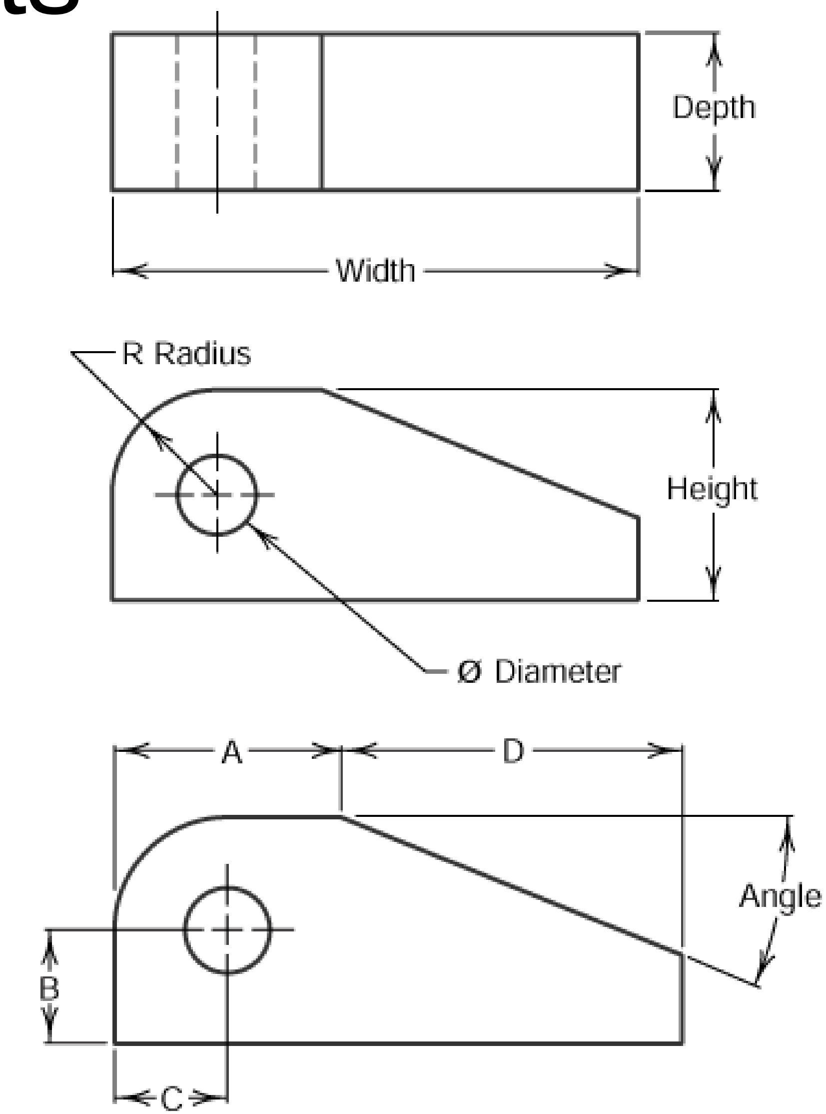
- Size dimension might be the overall width of the part or the diameter of a drilled hole.
- Location dimension might be length from the edge of the object to the center of the drilled hole.





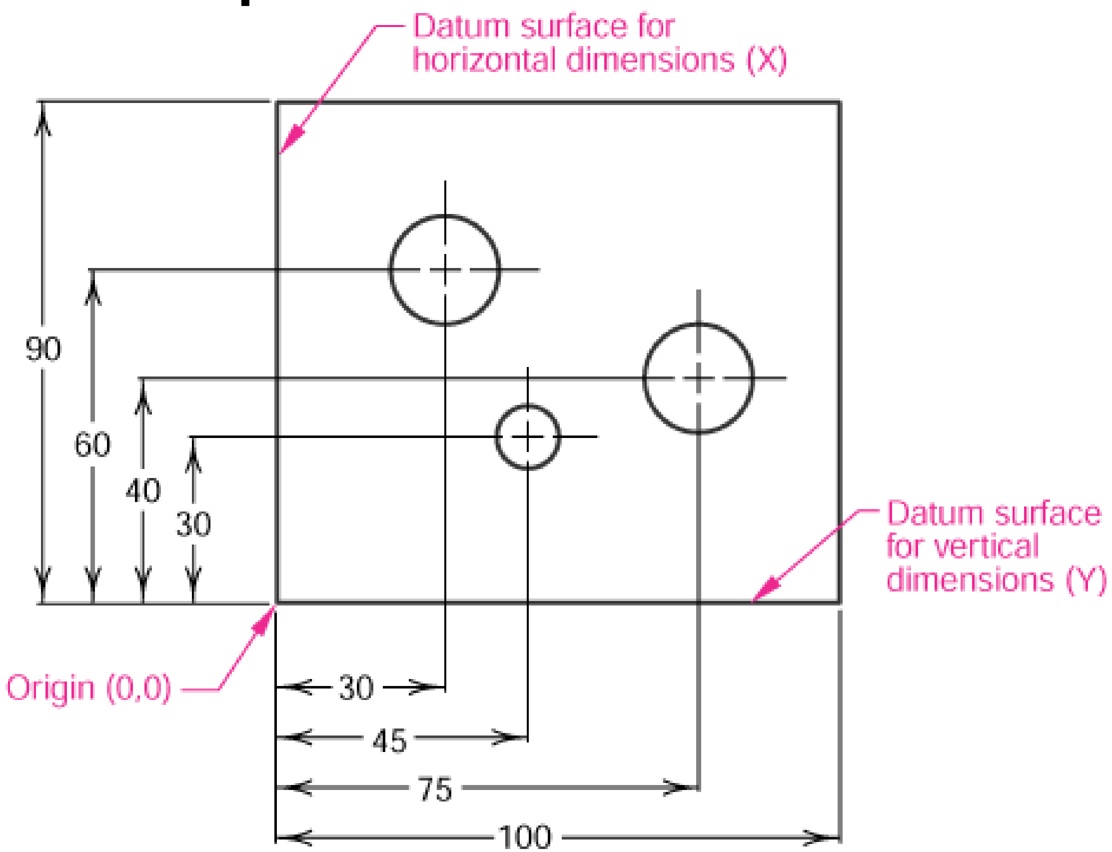
Basic Concepts

- Size dimensions
  - Horizontal
  - Vertical
  - Diameter
  - Radius
- Location and Orientation
  - Horizontal
  - Vertical
  - Angle

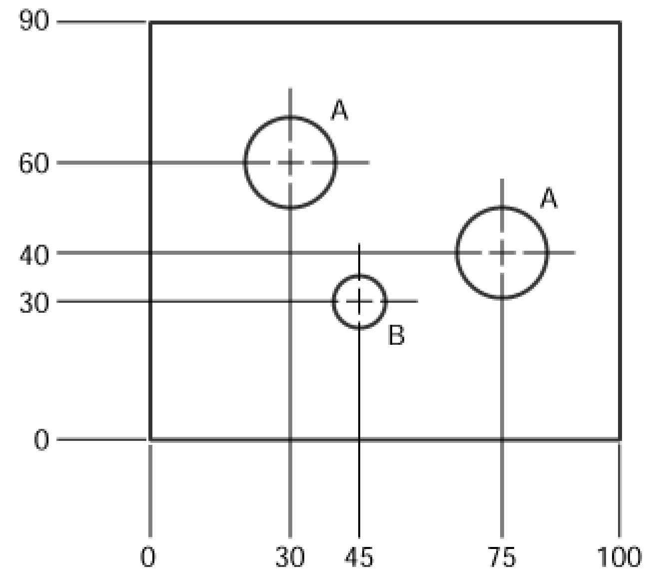


Basic Concepts

- **Rectangular coordinate dimensioning,** a base line (or datum line) is established for each coordinate direction, and all dimensions specified with respect to these baselines. This is also known as **datum dimensioning**, or **baseline dimensioning**. All dimensions are calculated as X and Y distances from an *origin point*, usually placed at the lower left corner of the part.

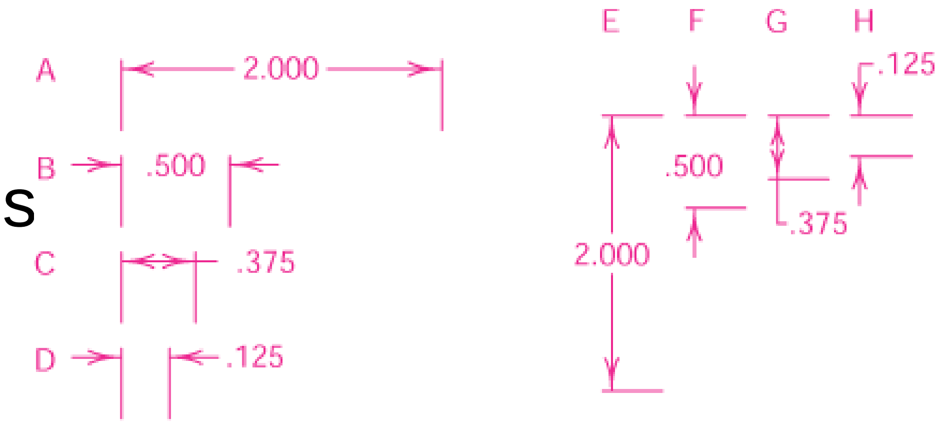


Symbol	A	B
Hole diameter	20	10

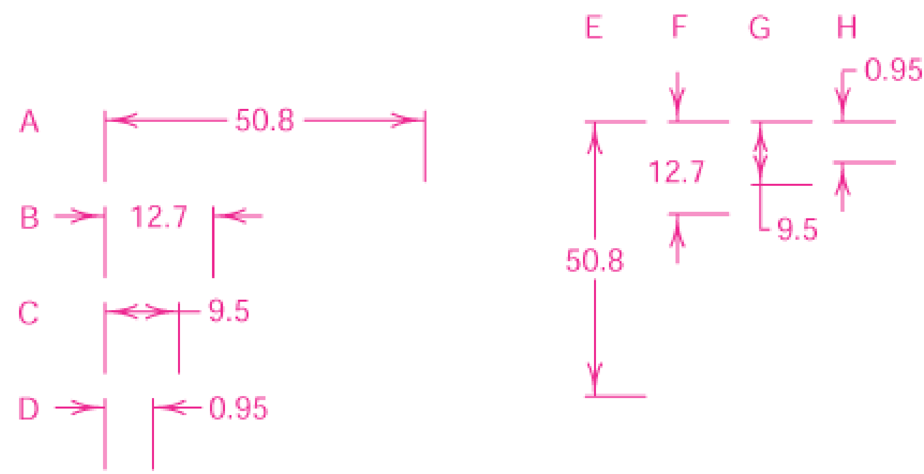


Standard Practices- Placement

- Dimension placement depends on the space available between extension lines. When space permits, dimensions and arrows are placed between the extension lines.



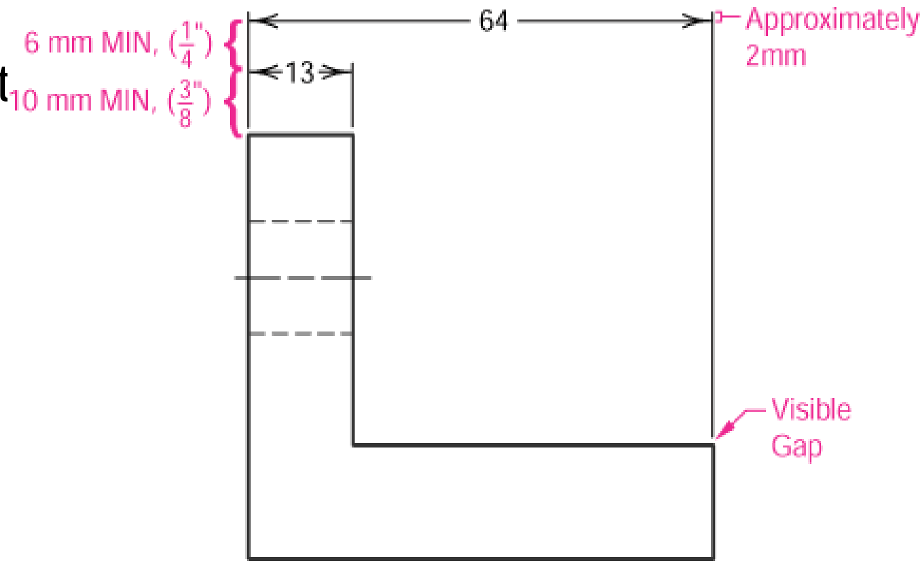
Decimal dimensioning



Millimeter dimensioning

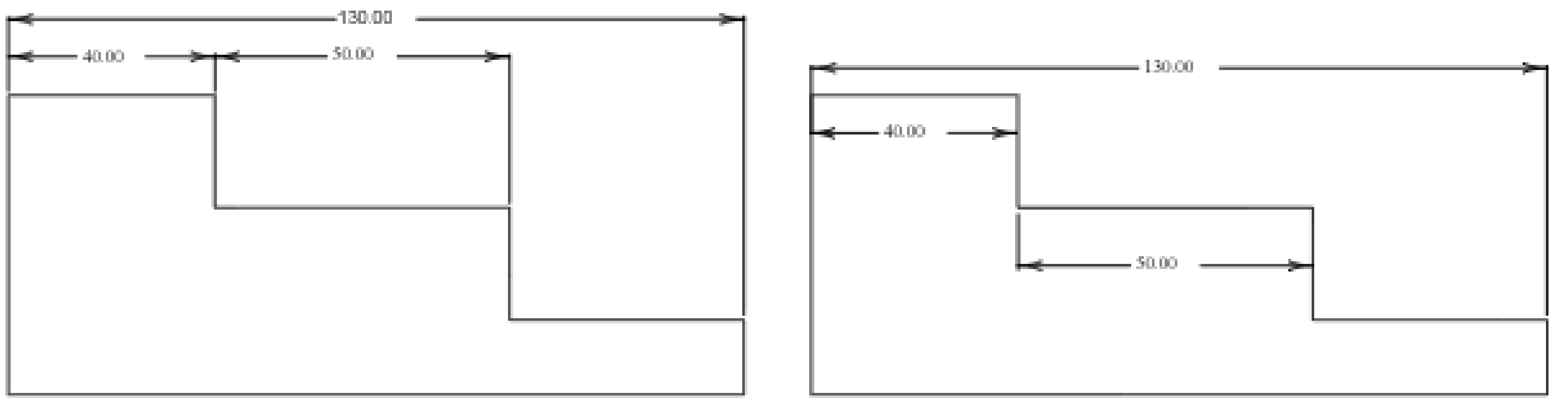
Standard Practices- Spacing

- The minimum distance from the object to the first dimension is 10mm (3/8 inch). The minimum spacing between dimensions is 6mm (1/4 inch).
- There should be a visible gap between an extension line and the feature to which it refers.
- Extension lines should extend about 1mm (1/32 inch) beyond the last dimension line.



Standard Practices- Grouping

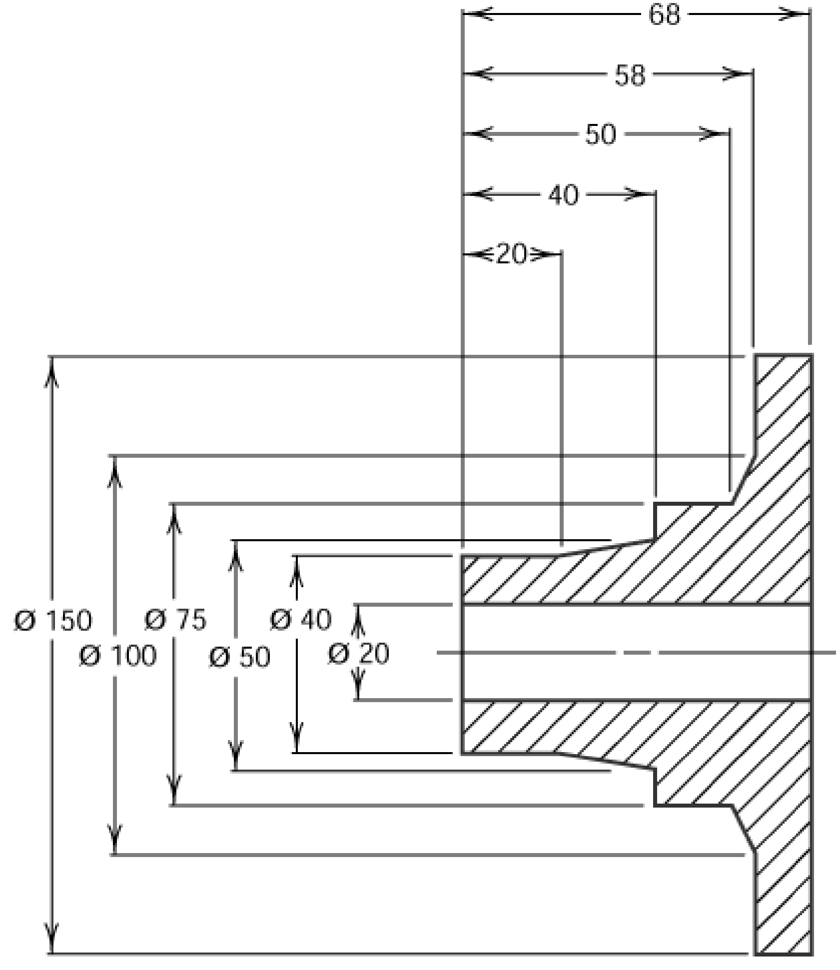
- Dimensions should be *grouped* for uniform appearance as shown.



(A) Yes (B) No!

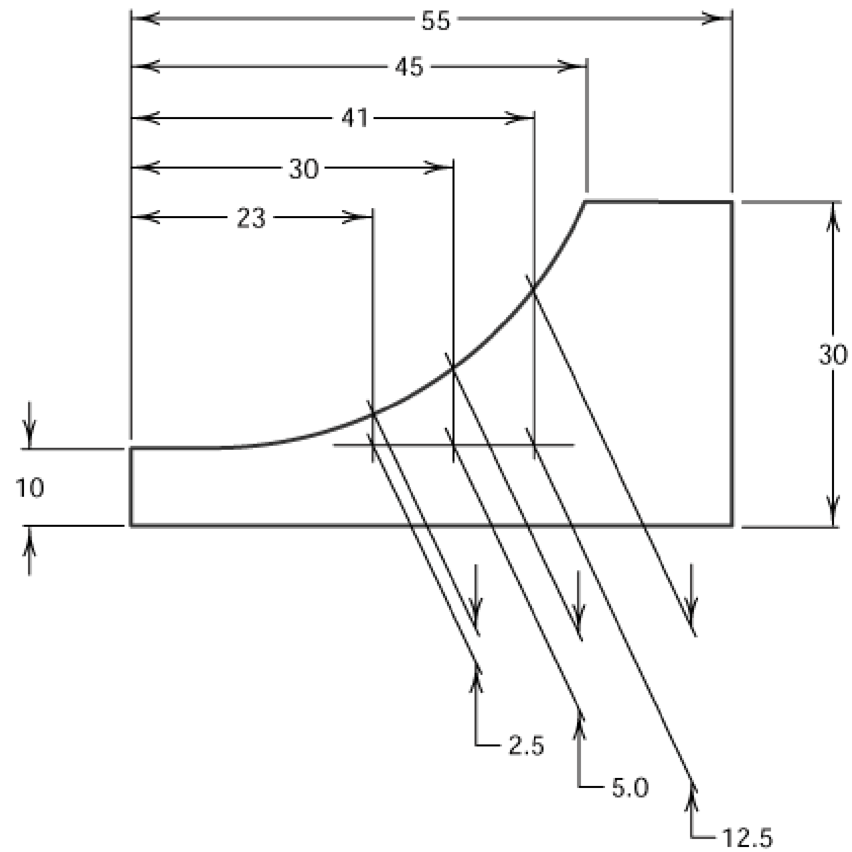
Standard Practices- Staggering

- Where there are several parallel dimensions, the values should be *staggered*.



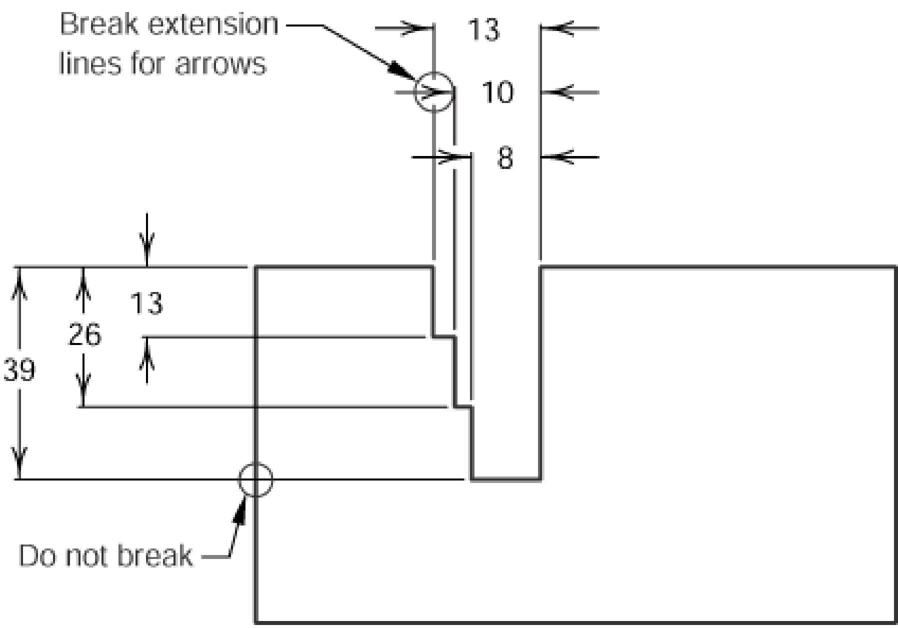
Standard Practices- Extension lines

- Extension lines are used to refer a dimension to a particular feature and are usually drawn perpendicular to the associated dimension line. Where space is limited, extension lines may be drawn at an angle.



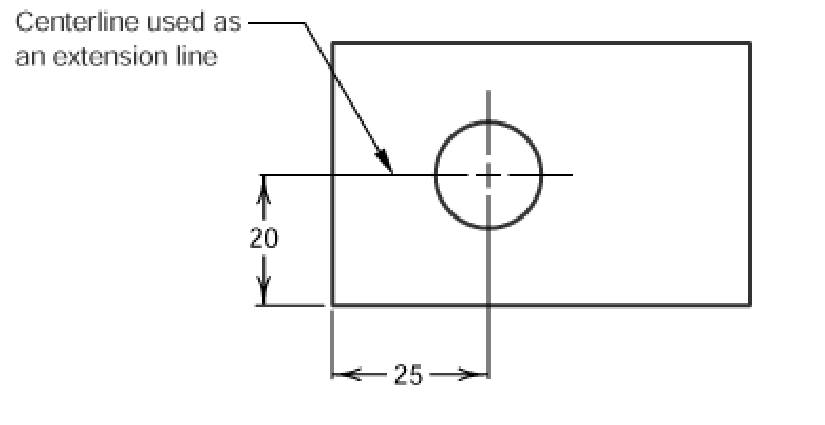
Standard Practices- Extension lines

- Extension lines should not cross dimension lines, and should avoid crossing other extension lines whenever possible.
- When extension lines cross object lines or other extension lines, they are not broken.
- When extension lines cross or are close to arrowheads, they are broken for the arrowhead.

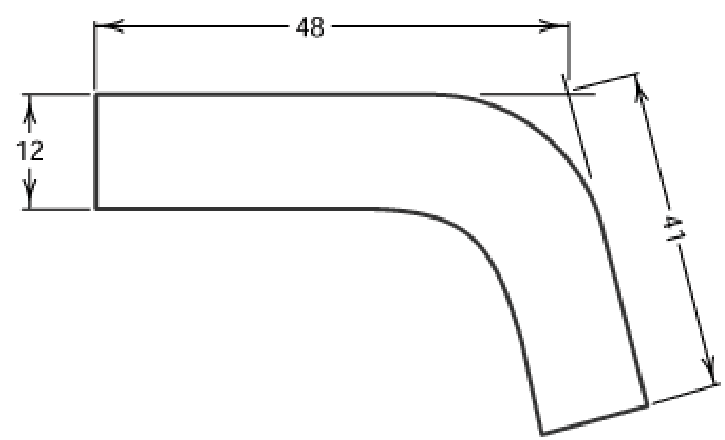


Standard Practices-Extension lines

- When the location of the center of a feature is being dimensioned, the center line of the feature is used as an extension line.
- When a point is being located by extension lines only, the extensions lines must pass through the point.



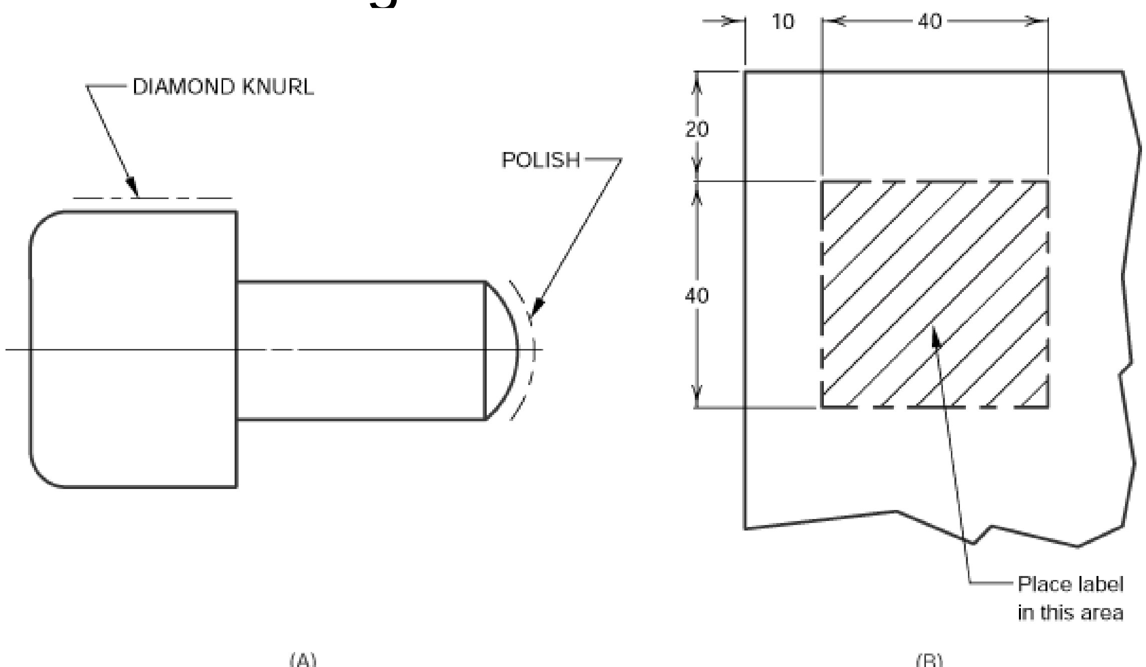
(A)



(B)

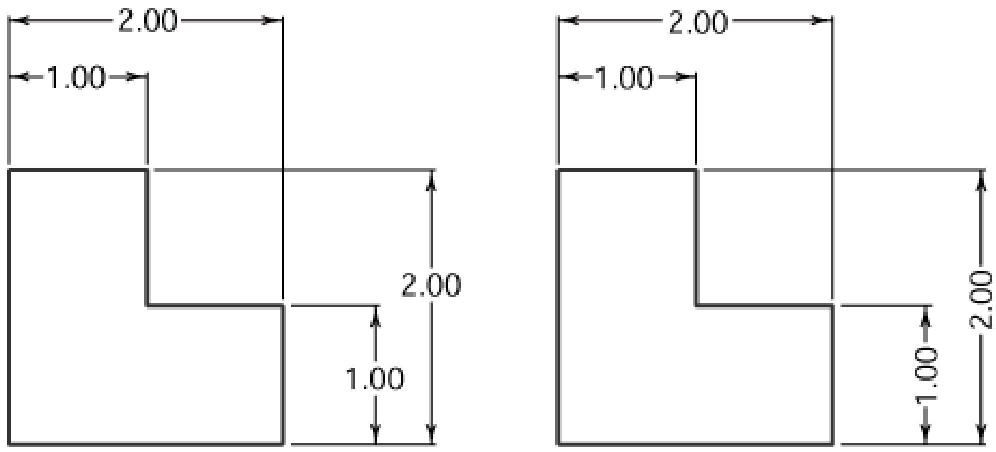
Standard Practices- Limited length or areas

- When it is necessary to define a limited length or area that is to receive additional treatment (such as the knurled portion of a shaft), the extent of the limits may be shown by a chain line. The chain line is drawn parallel to the surface being defined.



Standard Practices- Reading Direction

- All dimension and note text must be oriented to be read from the bottom of the drawing (relative to the drawing format).
- Placement of all text to be read from the bottom of the drawing is called **unidirectional dimensioning**.
- **Aligned dimensions** have text placed parallel to the dimension line with vertical dimensions read from the right of the drawing sheet.

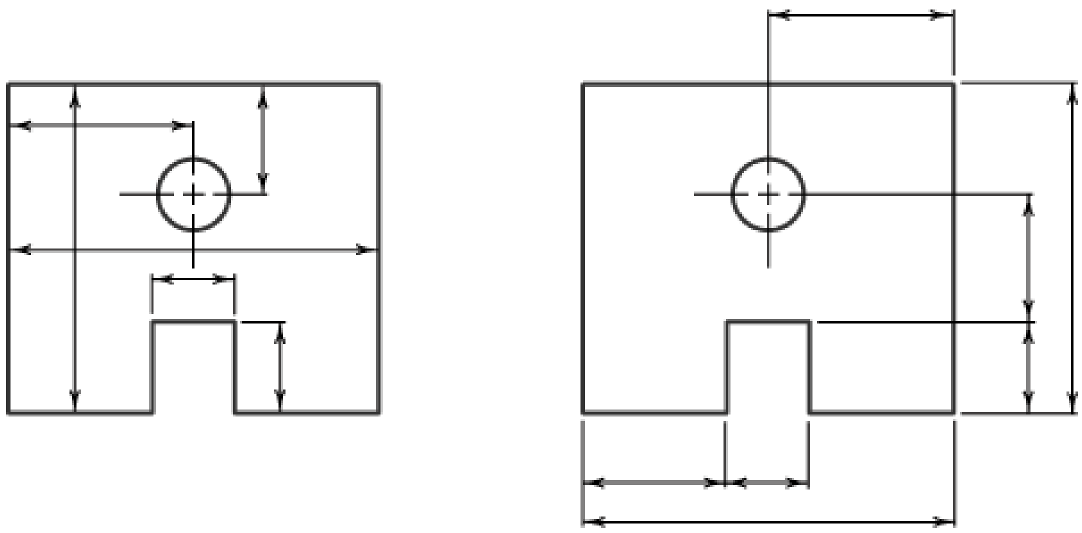


Unidirectional
Current standard

Aligned
Old standard

Standard Practices- View Dimensioning

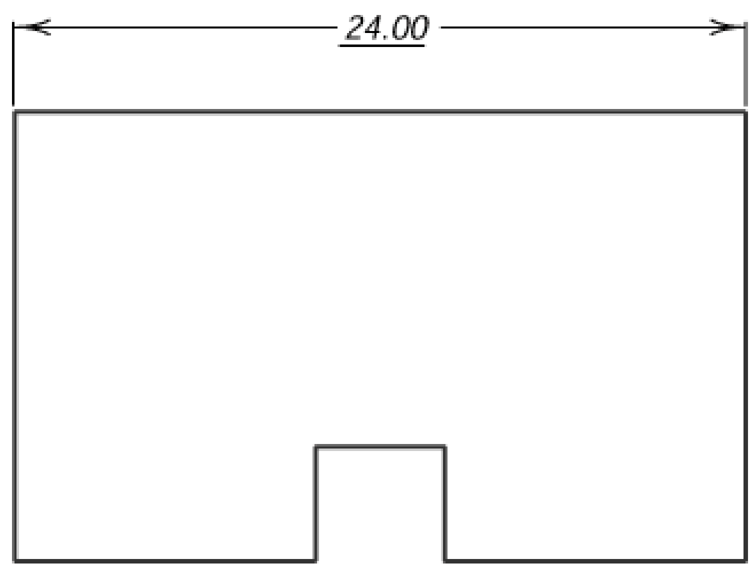
- Dimensions are to be kept outside of the boundaries of views of objects wherever practical.
- Dimensions may be place within the boundaries of objects in cases where extension or leader lines would be too long, or where clarity would be improved.



(A) No! (B) Yes

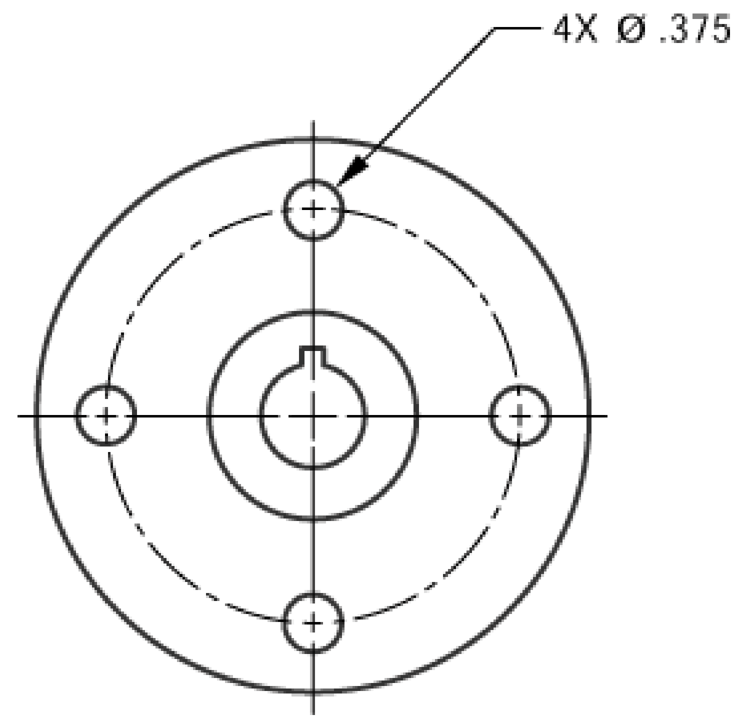
Standard Practices- Out-of-Scale Dimensions

- If it is necessary to include a dimension which is out of scale, the out of scale dimension text must be underlined.



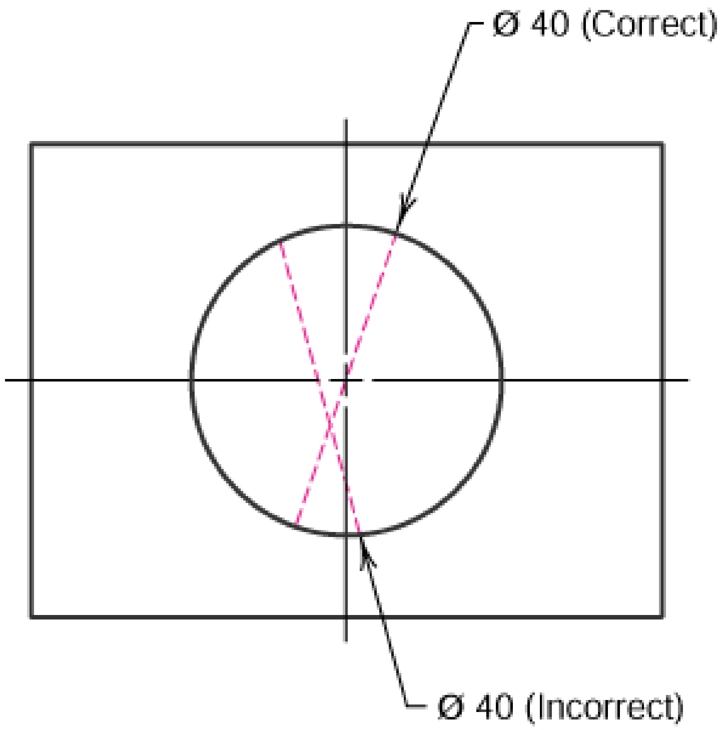
Standard Practices-Repetitive Features

- The symbol X is used to indicate the number of times a feature is to be repeated. The number of repetitions, followed by the symbol X and a space precedes the dimension text.

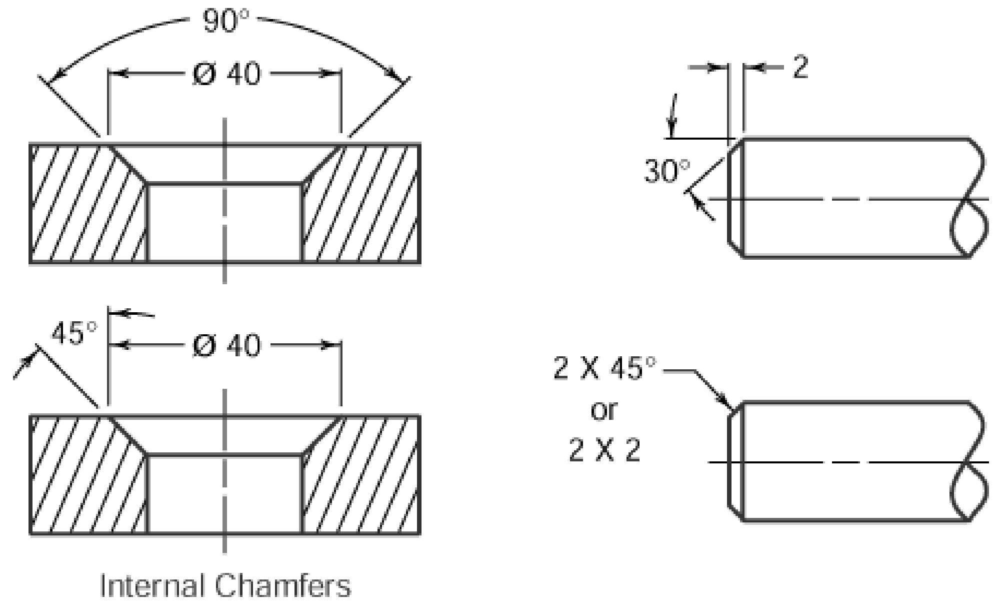


Detail Dimensions

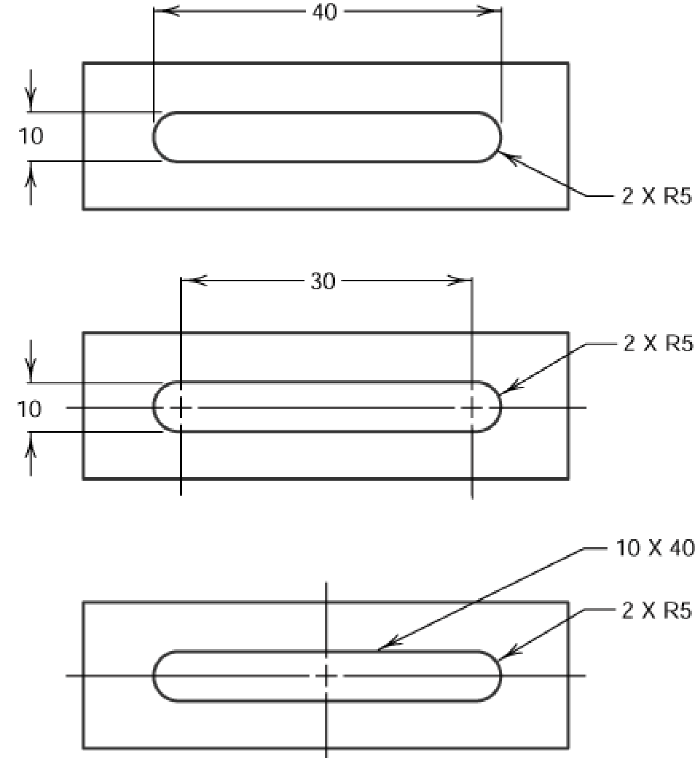
- Holes
  - Diameters must be dimensioned with the diameter symbol preceding the numerical value.
  - When holes are dimensioned with a leader line, the line must be radial. A **radial lin***e* is one that passes through the center of a circle or arc if extended.



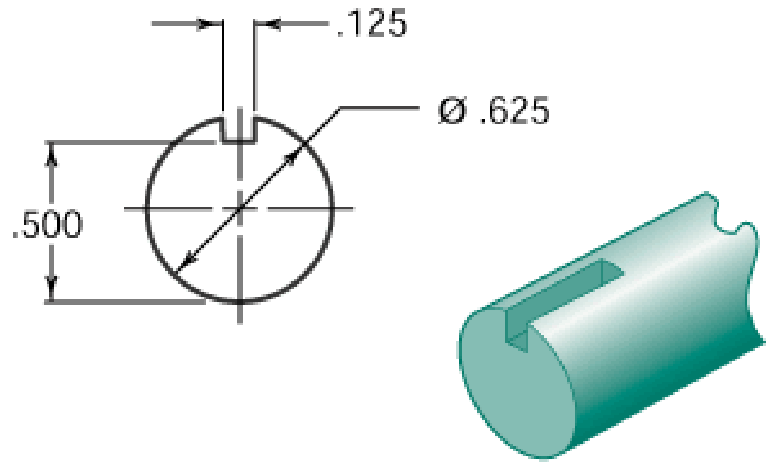
- Chamfers



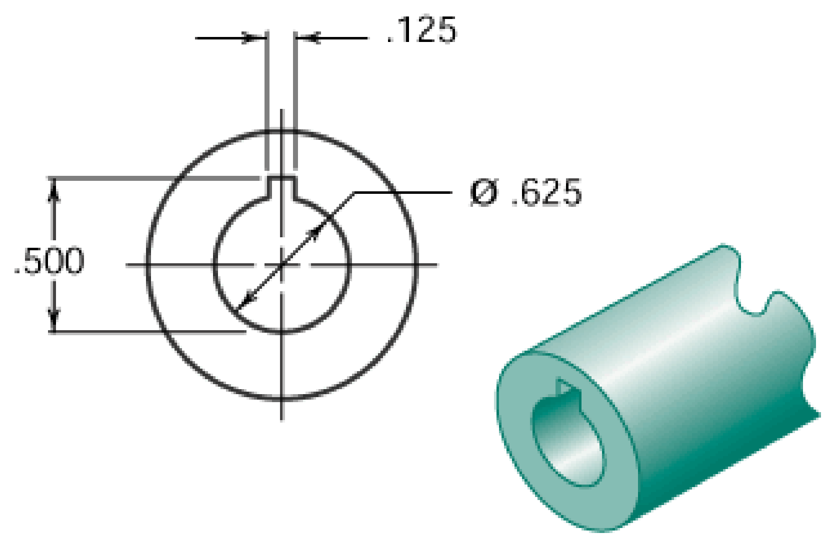
- Slotted holes



- Keyseat and Keyway

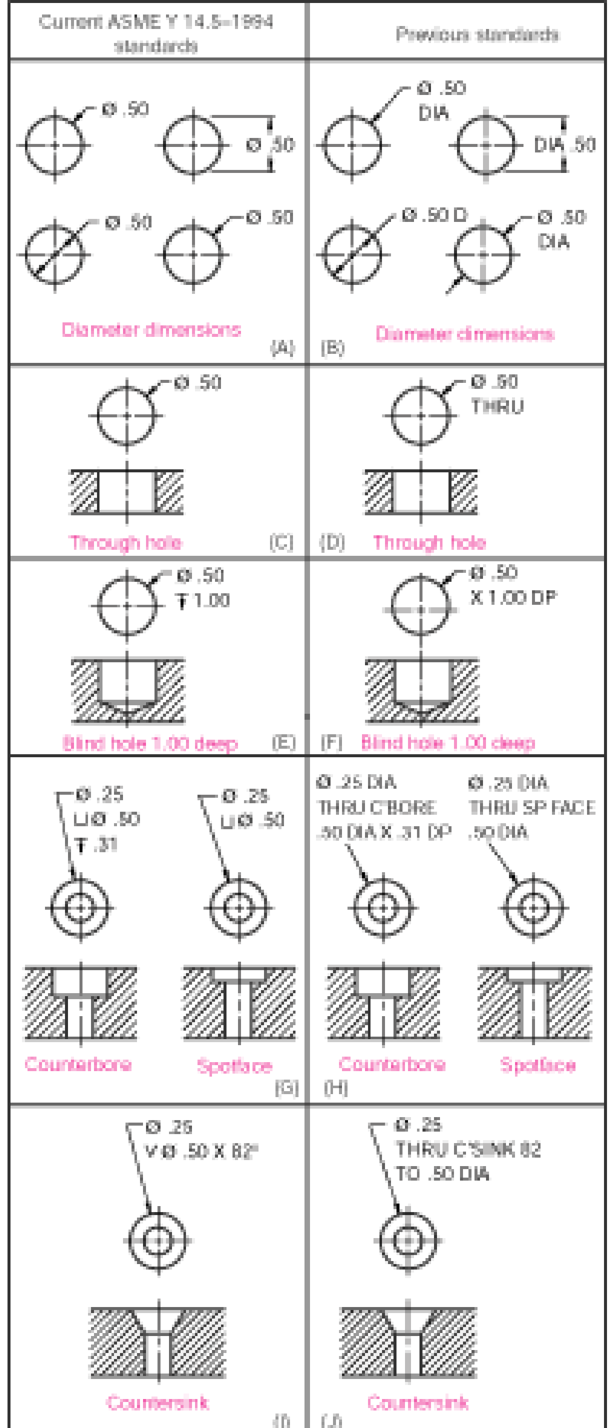


Keyseat



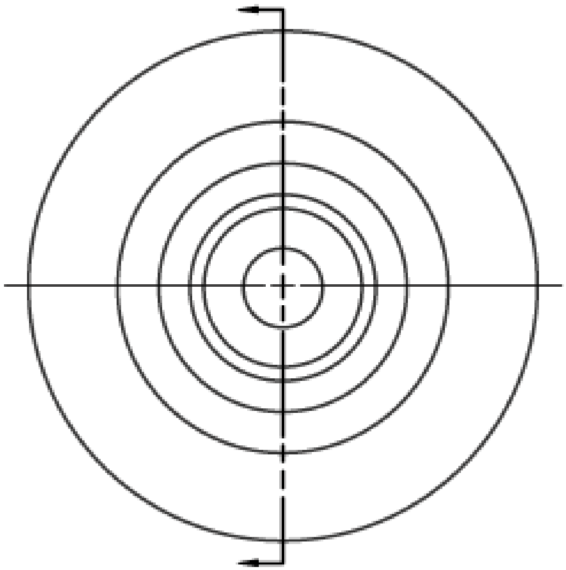
Keyway

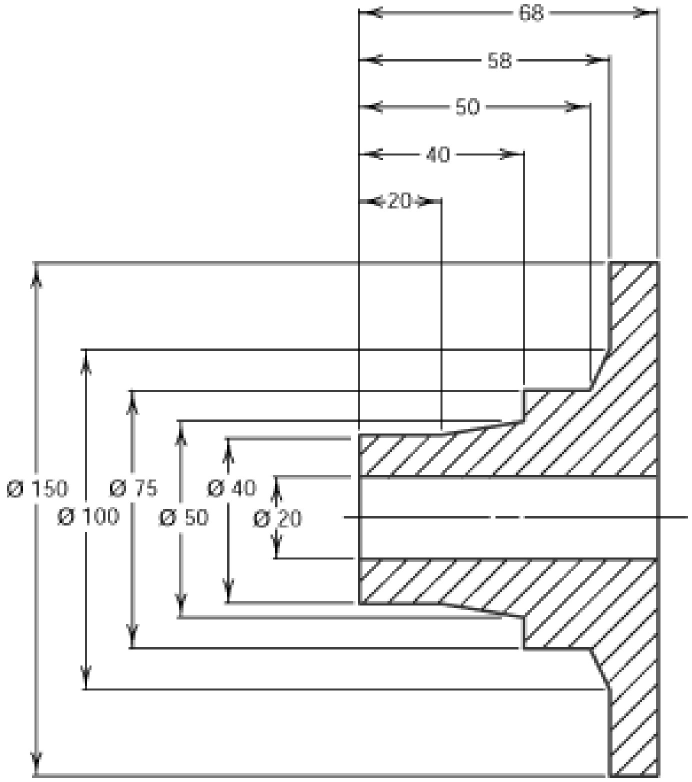
- Summary



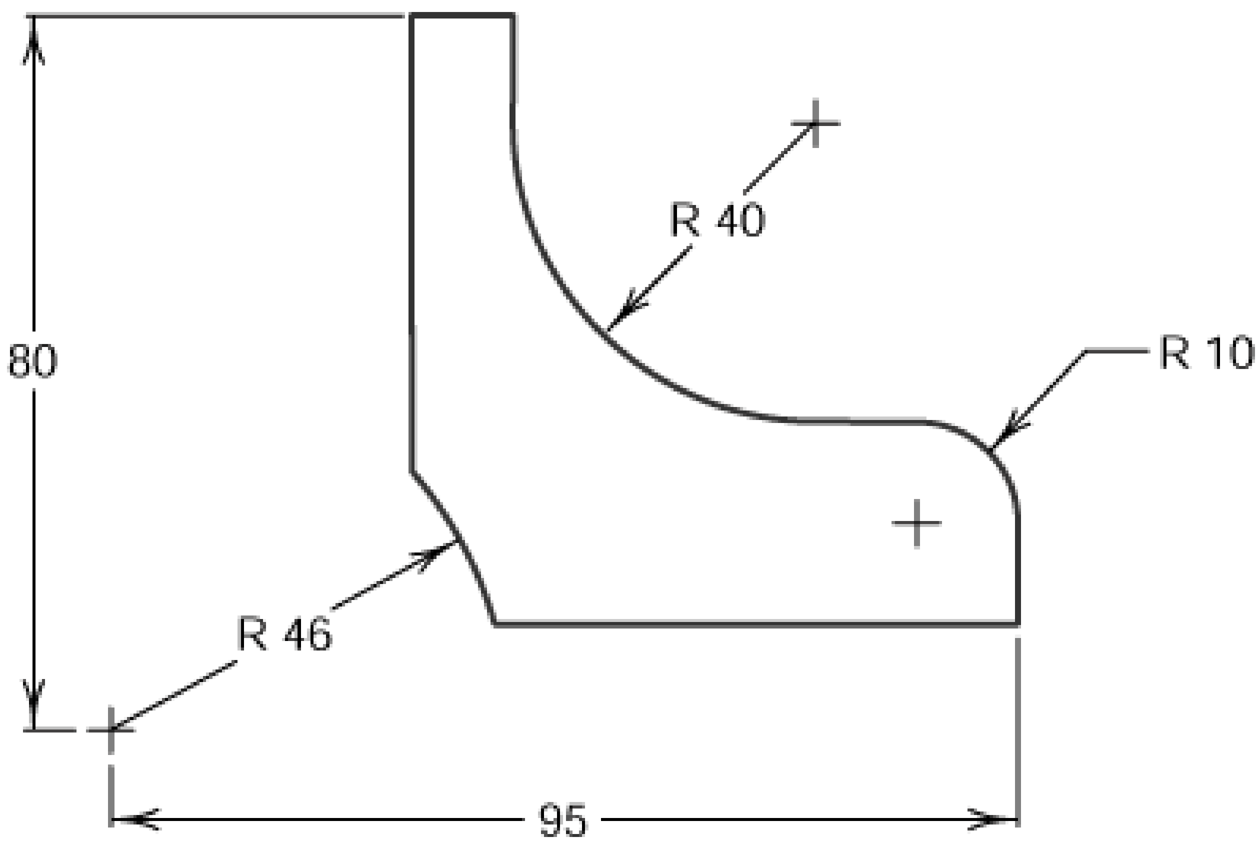
Current ASME Y 14.5–1994 standards
Previous standards
Ø .50
Ø .50
Ø .50
Ø .50
Diameter dimensions
(A)
Ø .50 DIA
DIA .50
Ø .50 D
Ø .50 DIA
(B)
Diameter dimensions
Ø .50
Through hole
(C)
Ø .50 THRU
(D)
Through hole
Ø .50
↧ 1.00
Blind hole 1.00 deep
(E)
Ø .50
X 1.00 DP
(F)
Blind hole 1.00 deep
Ø .25
⌴Ø .50
↧ .31
Ø .25
⌴Ø .50
Counterbore
Spotface
(G)
Ø .25 DIA
THRU C'BORE
.50 DIA X .31 DP
Ø .25 DIA
THRU SP FACE
.50 DIA
Counterbore
Spotface
(H)
Ø .25
⌵Ø .50 X 82°
Countersink
(I)
Ø .25
THRU C'SINK 82
TO .50 DIA
Countersink
(J)

- Concentric circles

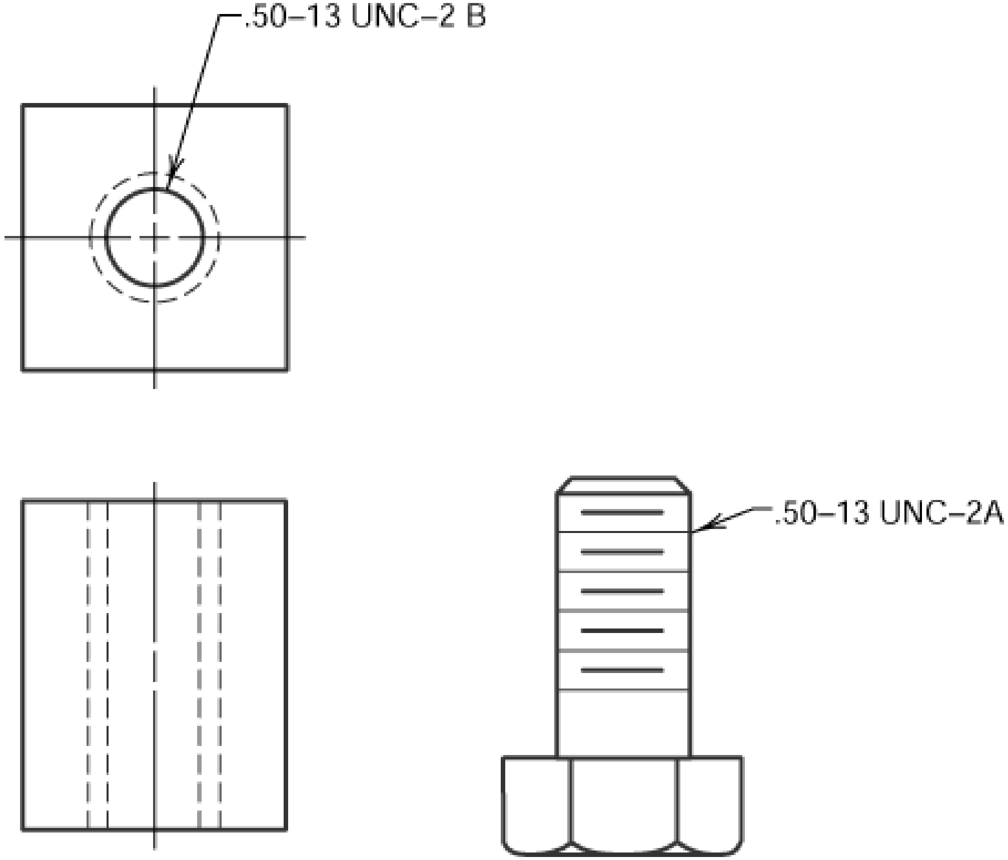




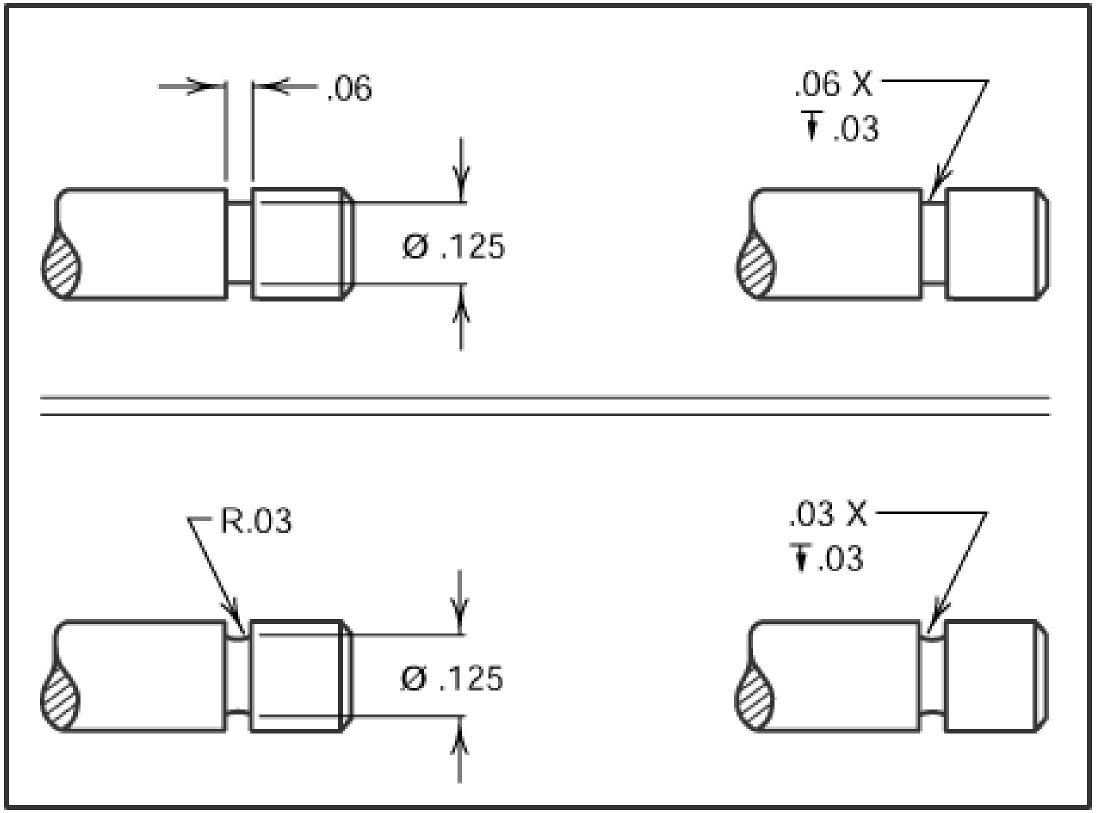
- Arcs



- Screw Threads



- Grooves



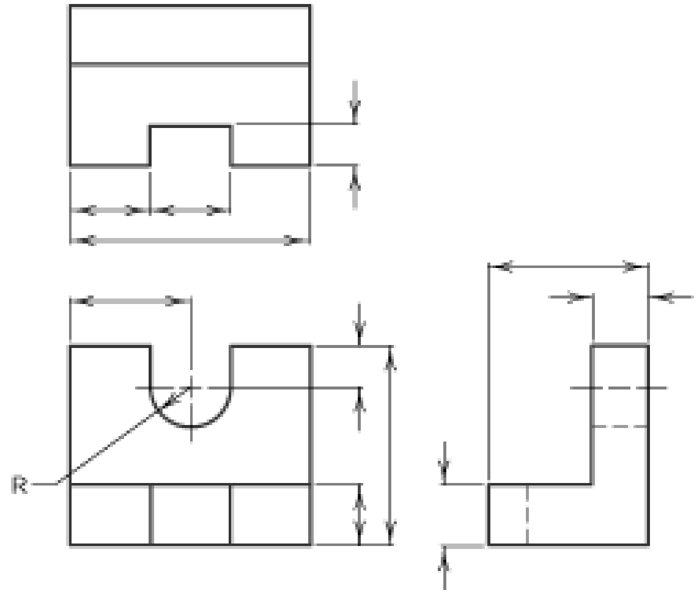
- Manufacturers' gage

Sheet-Metal Gages in Approximate Decimals of an Inch

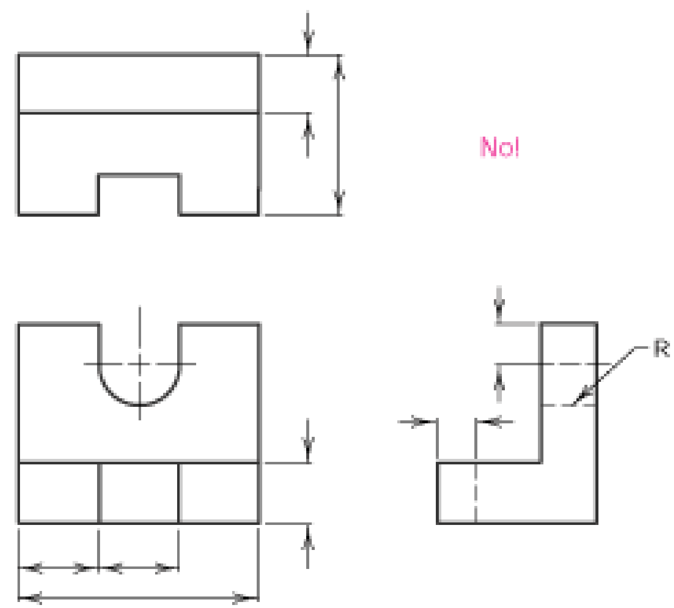
No. of Sheet-Metal Gage	Manufacturers' Standard Gage for Steel	Birmingham Gage (B.G.) for Sheets, Hoops	Galvanized Sheet Gage	Zinc Gage	No. of Sheet-Metal Gage	Manufacturers' Standard Gage for Steel	Birmingham Gage (B.G.) for Sheets, Hoops	Galvanized Sheet Gage	Zinc Gage
15/0	. . .	1.000	. . .	. . .	20	0.0359	0.0392	0.0396	0.070
14/0	. . .	0.9583	. . .	. . .	21	0.0329	0.0349	0.0366	0.080
13/0	. . .	0.9167	. . .	. . .	22	0.0299	0.03125	0.0336	0.090
12/0	. . .	0.8750	. . .	. . .	23	0.0269	0.02782	0.0306	0.100
11/0	. . .	0.8333	. . .	. . .	24	0.0239	0.02476	0.0276	0.125
10/0	. . .	0.7917	. . .	. . .	25	0.0209	0.02204	0.0247	. . .
9/0	. . .	0.7500	. . .	. . .	26	0.0179	0.01961	0.0217	. . .
8/0	. . .	0.7083	. . .	. . .	27	0.0164	0.01745	0.0202	. . .
7/0	. . .	0.6666	. . .	. . .	28	0.0149	0.01562	0.0187	. . .
6/0	. . .	0.6250	. . .	. . .	29	0.0135	0.01390	0.0172	. . .
5/0	. . .	0.5883	. . .	. . .	30	0.0120	0.01230	0.0157	. . .
4/0	. . .	0.5416	. . .	. . .	31	0.0105	0.01100	0.0142	. . .
3/0	. . .	0.5000	. . .	. . .	32	0.0097	0.00980	0.0134	. . .
2/0	. . .	0.4452	. . .	. . .	33	0.0090	0.00870	. . .	. . .
1/0	. . .	0.3964	. . .	. . .	34	0.0082	0.00770	. . .	. . .
1	. . .	0.3532	. . .	. . .	35	0.0075	0.00690	. . .	. . .
2	. . .	0.3147	. . .	. . .	36	0.0067	0.00610	. . .	. . .
3	0.2391	0.2804	. . .	0.006	37	0.0064	0.00640	. . .	. . .
4	0.2242	0.2500	. . .	0.008	38	0.0060	0.00480	. . .	. . .
5	0.2092	0.2225	. . .	0.010	39	. . .	0.00430	. . .	. . .
6	0.1943	0.1981	. . .	0.012	40	. . .	0.00386	. . .	. . .
7	0.1793	0.1764	. . .	0.014	41	. . .	0.00343	. . .	. . .
8	0.1644	0.1570	0.1681	0.016	42	. . .	0.00306	. . .	. . .
9	0.1495	0.1398	0.1532	0.018	43	. . .	0.00272	. . .	. . .
10	0.1345	0.1250	0.1382	0.020	44	. . .	0.00242	. . .	. . .
11	0.1196	0.1113	0.1233	0.024	45	. . .	0.00215	. . .	. . .
12	0.1046	0.0991	0.1084	0.028	46	. . .	0.00192	. . .	. . .
13	0.0897	0.0882	0.0934	0.032	47	. . .	0.00170	. . .	. . .
14	0.0747	0.0785	0.0785	0.036	48	. . .	0.00152	. . .	. . .
15	0.0673	0.0699	0.0710	0.040	49	. . .	0.00135	. . .	. . .
16	0.0598	0.0625	0.0635	0.045	50	. . .	0.00120	. . .	. . .
17	0.0538	0.0556	0.0575	0.050	51	. . .	0.00107	. . .	. . .
18	0.0478	0.0495	0.0516	0.055	52	. . .	0.00095	. . .	. . .
19	0.0418	0.0440	0.0456	0.060	. . .	. . .	. . .	. . .	. . .

Dimension Techniques

- Contour Dimensioning
  - contours or shapes of the object are dimensioned in their most descriptive view. For example, the radius of a arc would be dimensioned where it appears as an arc and not as a hidden feature.



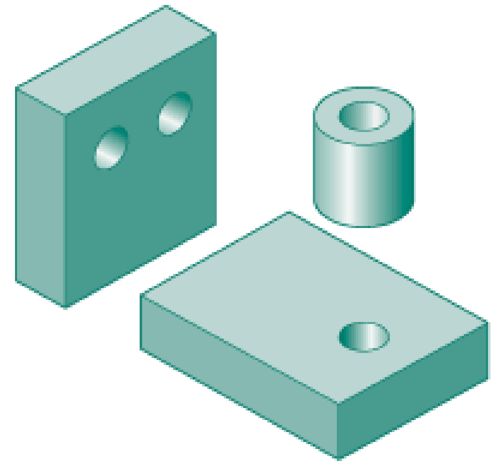
(A) Correct contour dimensioning

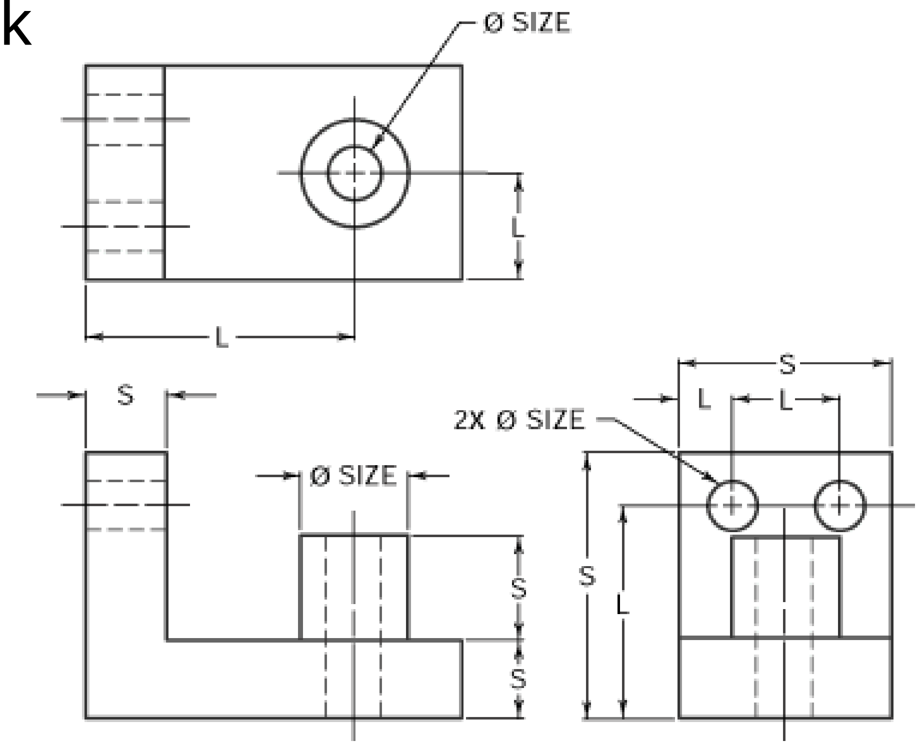


(B) Incorrect contour dimensioning

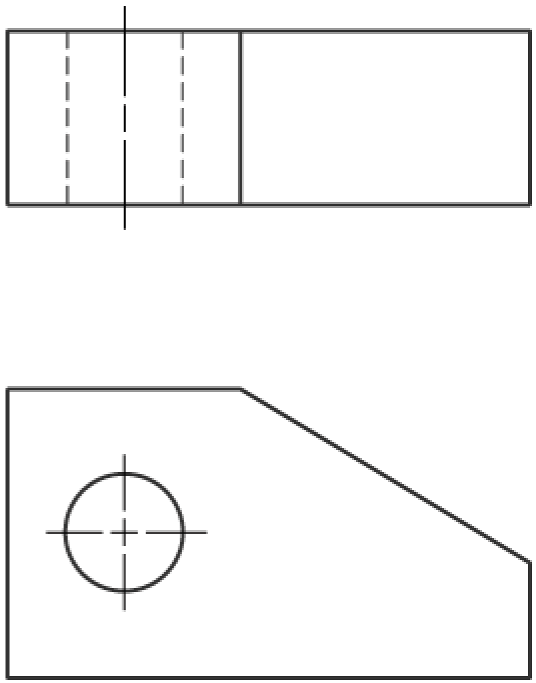
Dimension Techniques

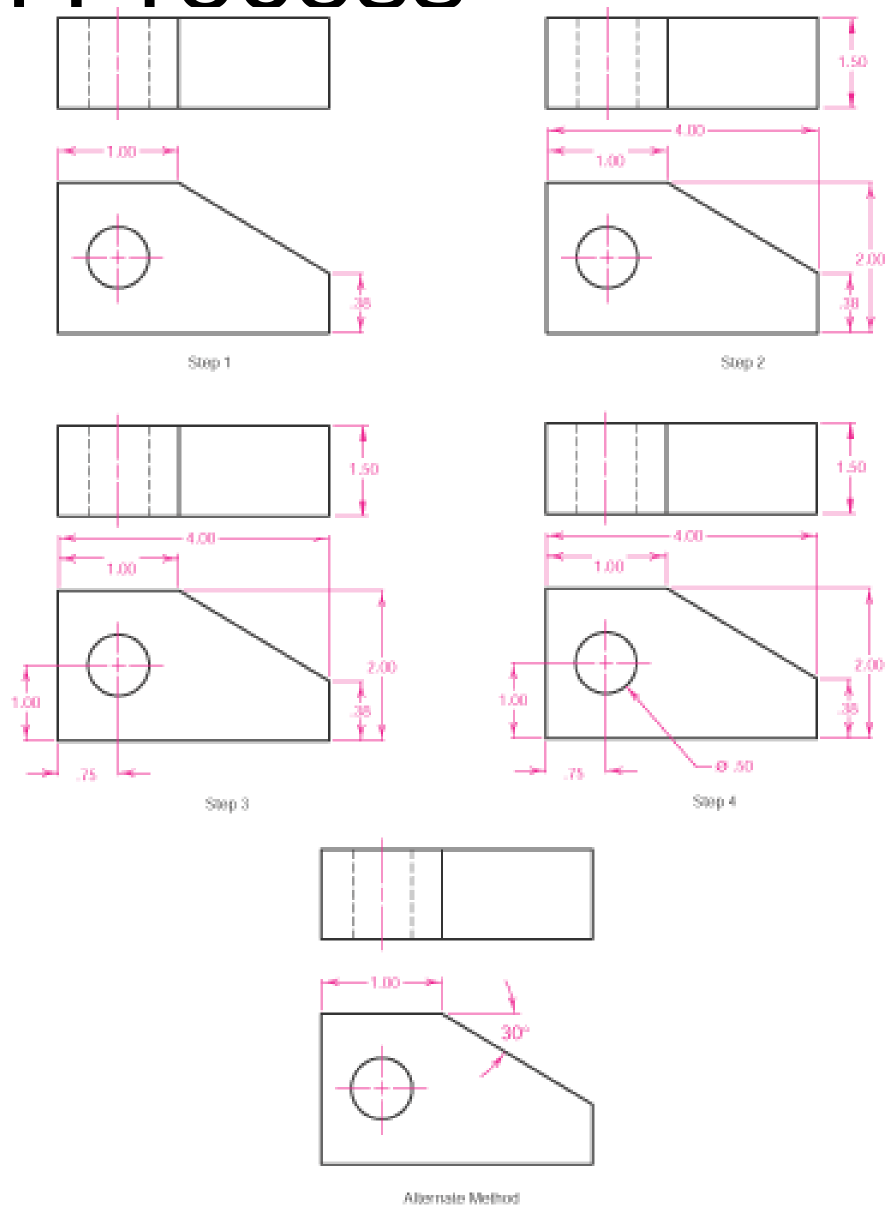
- Geometric Breakdown
  - a part is to break the part into its geometric configurations.





Dimension Process





Dimension Guidelines

- The primary guideline is that of clarity and whenever two guidelines appear to conflict, the method which most clearly communicates the size information shall prevail.
  - Every dimension must have an associated tolerance, and that tolerance must be clearly shown on the drawing.
  - Avoid over-dimensioning a part. Double dimensioning of a feature is not permitted.
  - Dimensions should be placed in the view which most clearly describes the feature being dimensioned.

Dimension Guidelines

- A minimum spacing between the object and dimensions and between dimensions must be maintained.
- A visible gap shall be placed between the end of extension lines and the feature to which they refer.
- Manufacturing methods should not be specified as part of the dimension unless no other method of manufacturing is acceptable.
- Placing dimensions within the boundaries of a view should be avoided whenever practicable.

Dimension Guidelines

- Dimensions for materials typically manufactured to gages or code numbers shall be specified by numerical values.
- Unless otherwise specified, angles shown on drawings are assumed to be 90 degrees.
- Dimensioning to hidden lines should be avoided whenever possible. Hidden lines are less clear than visible lines.
- The depth of blind, counterbored, or countersunk holes may be specified in a note along with the diameter.

Dimension Guidelines

- Diameters, radii, squares, counterbores, spotfaces, countersinks, and depth should be specified with the appropriate symbol preceding the numerical value.
- Leader lines for diameters and radii should be radial lines.

Tolerancing

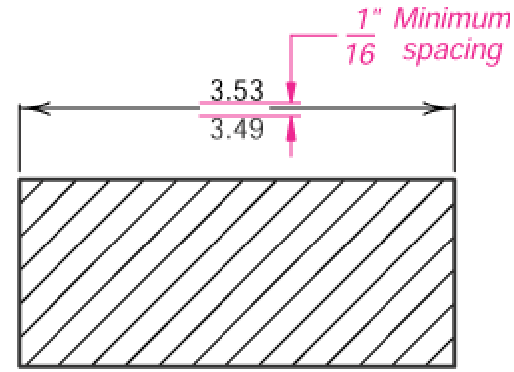
- **Tolerance** is the total amount a dimension may vary and is the difference between the **upper** (maximum) and **lower** (minimum) limits.
- Tolerances are used to control the amount of variation inherent in all manufactured parts. In particular, tolerances are assigned to mating parts in an assembly.

Tolerancing

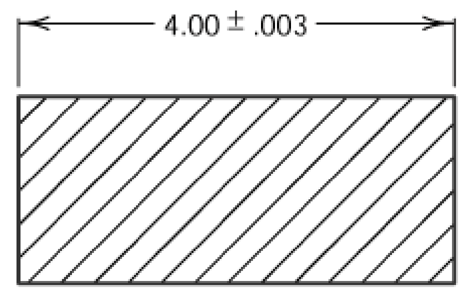
- One of the great advantages of using tolerances is that it allows for **interchangeable parts,** thus permitting the replacement of individual parts.
- Tolerances are used in production drawings to control the manufacturing process more accurately and control the variation between parts.

Tolerancing

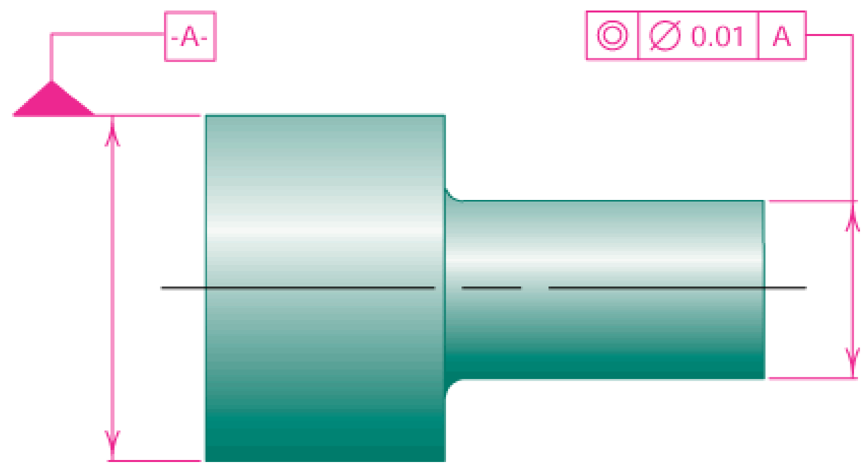
- Tolerance representation
  - Direct limits or as tolerance values applied directly to a dimension.
  - Geometric tolerances
  - Notes referring to specific condition.



(A) Direct limits

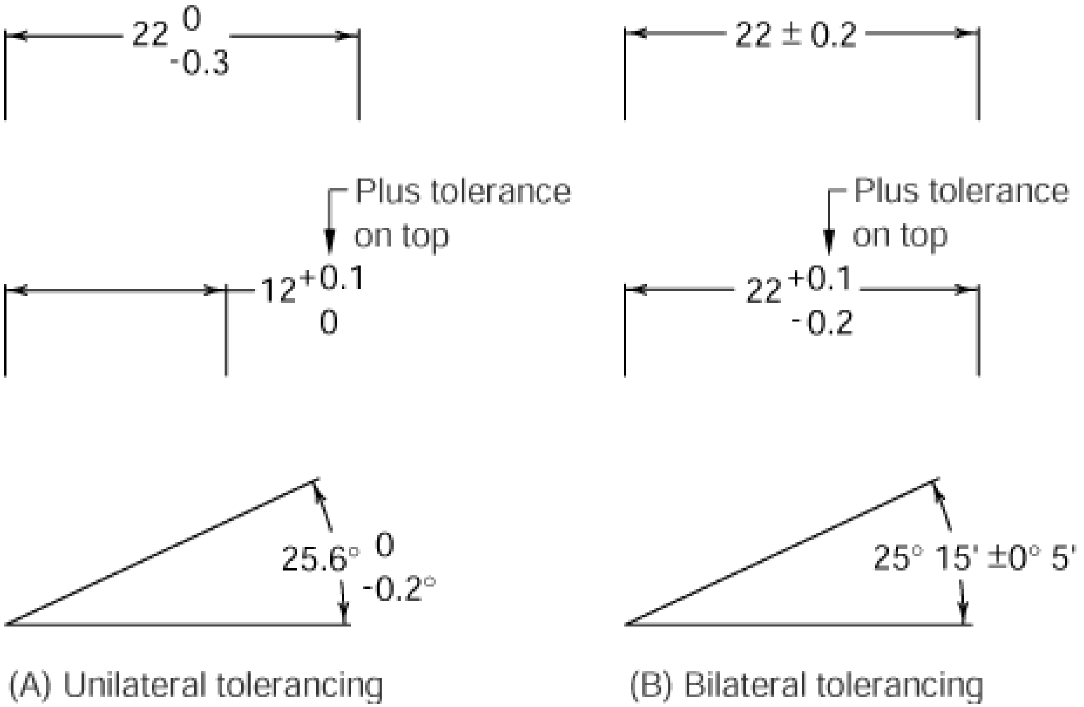


(B) Tolerance values



Tolerancing

- Tolerance representation
  - Plus/Minus



Tolerancing

- Important terms
  - **Nominal size** a dimension used to describe the general size usually expressed in common fractions.
  - **Basic size** the theoretical size used as a starting point for the application of tolerances.
  - **Actual size** the measured size of the finished part after machining.

Tolerancing

- Important terms
  - **Limits** the maximum and minimum sizes shown by the toleranced dimension.
  - **Allowance** is the minimum clearance or maximum interference between parts.
  - **Tolerance** is the total variance in a dimension which is the difference between the upper and lower limits. The tolerance of the slot in Figure 14.50 is .004" and the tolerance of the mating part is .002".

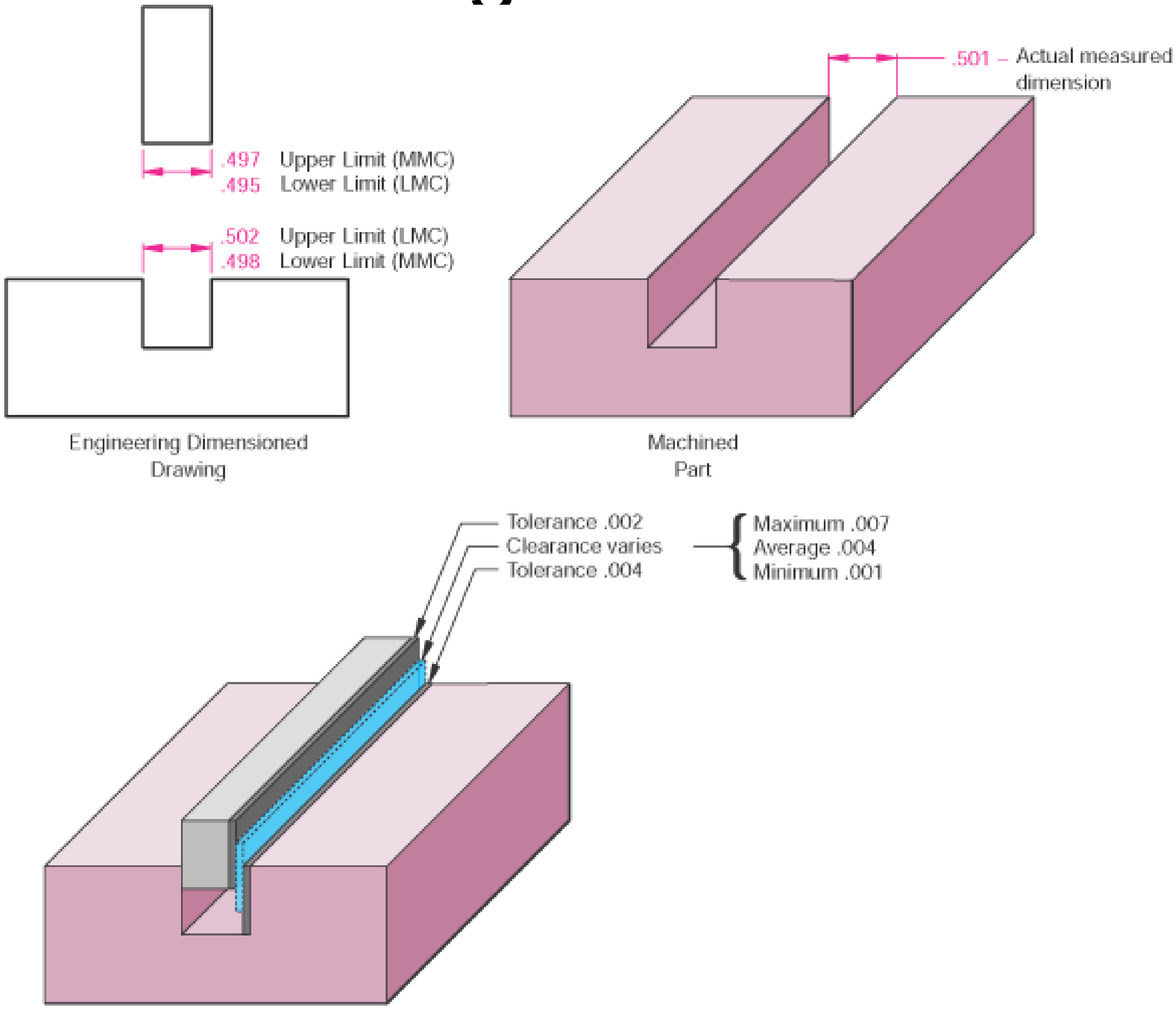
Tolerancing

- Important terms
  - **Maximum material condition (MMC)** is the condition of a part when it contains the most amount of material. The MMC of an external feature such as a shaft is the upper limit. The MMC of an internal feature such as a hole is the lower limit.

Tolerancing

- Important terms
  - **Least material condition (LMC)** is the condition of a part when it contains the least amount of material possible. The LMC of an external feature is the lower limit of the part. The LMC of an internal feature is the upper limit of the part.

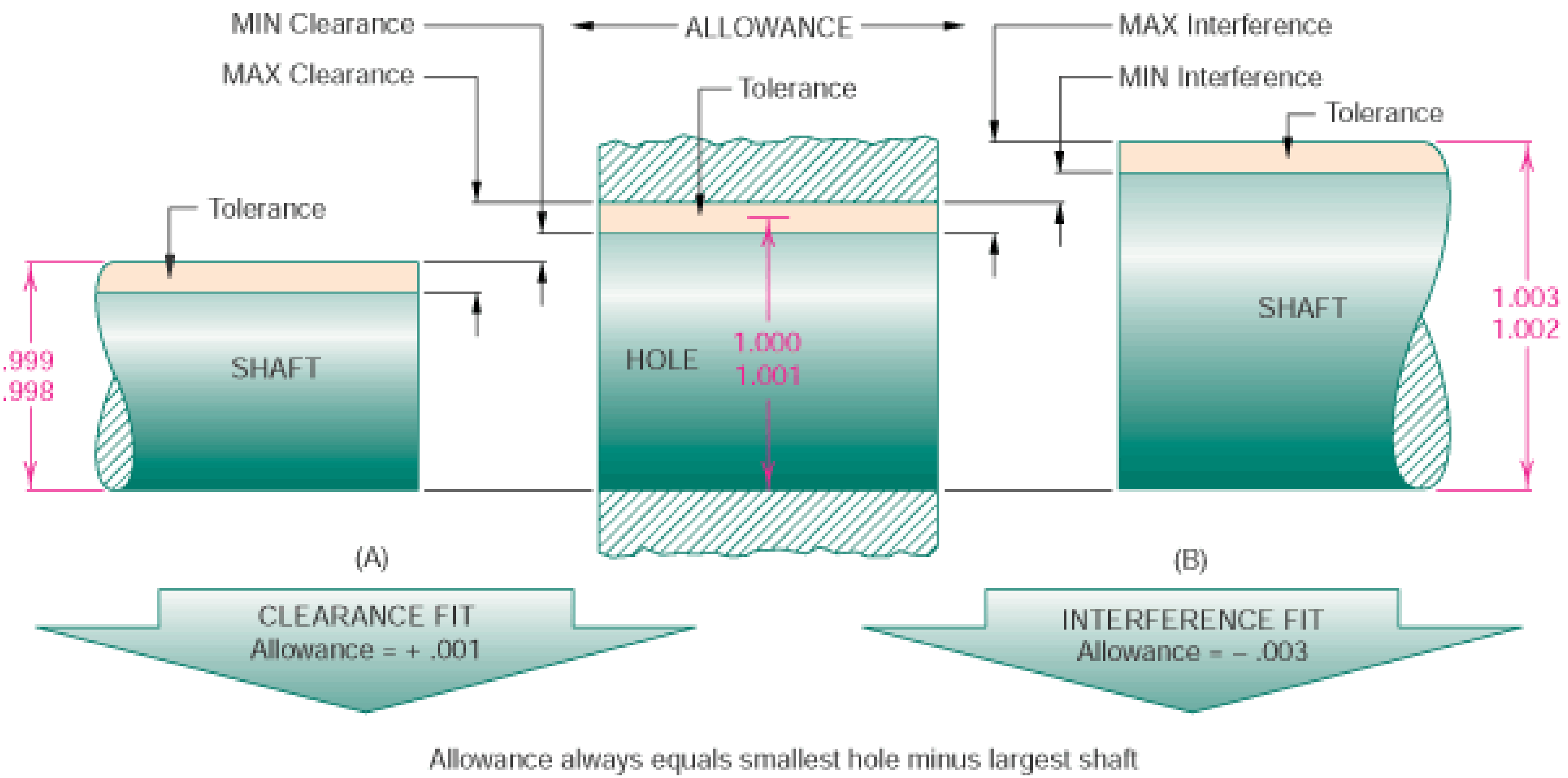
Tolerancing



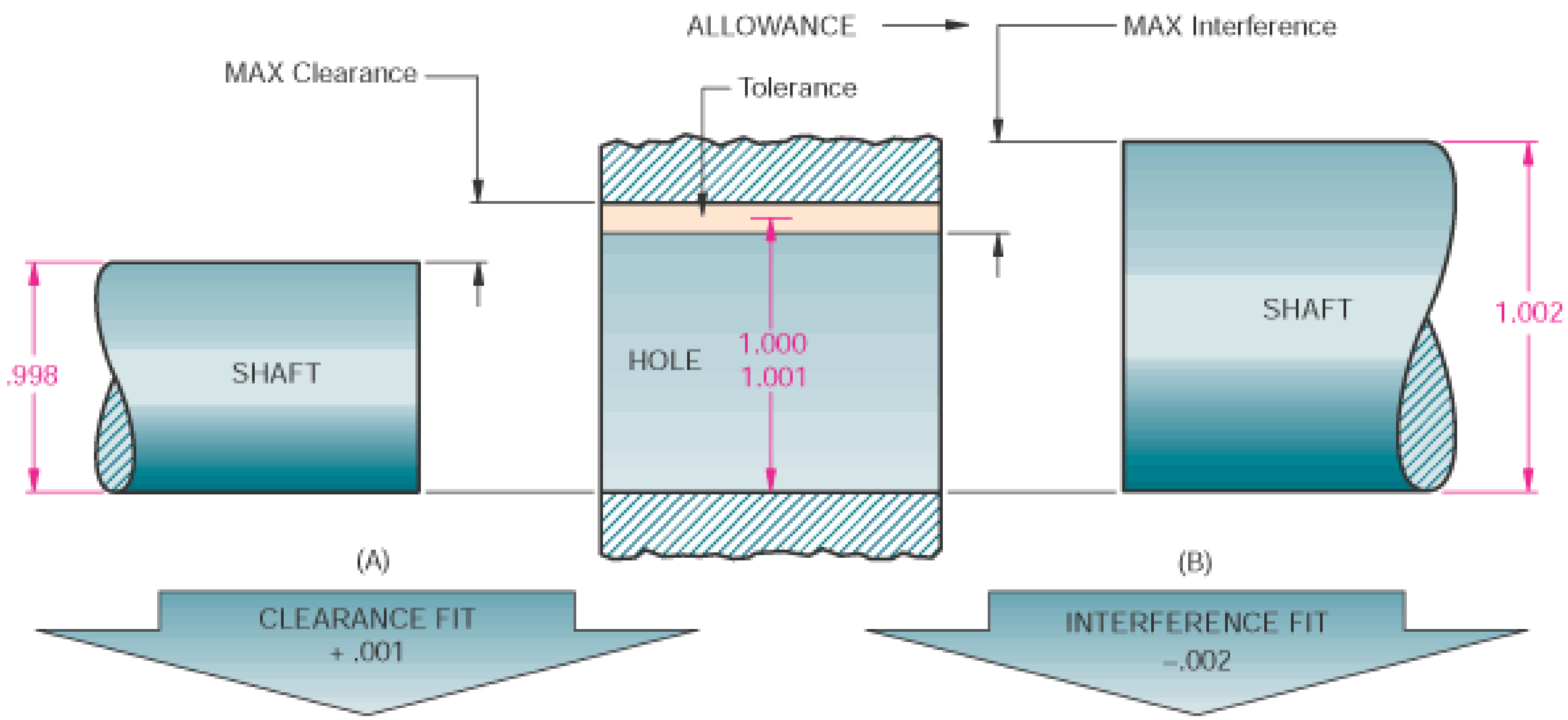
Tolerancing

- Fit types
  - **Clearance fit** occurs when two toleranced mating parts will always leave a space or clearance when assembled.
  - **Interference fit** occurs when two toleranced mating parts will always interfere when assembled.
  - **Transition fit** occurs when two toleranced mating parts will sometimes be an interference fit and sometimes be a clearance fit when assembled.

Tolerancing

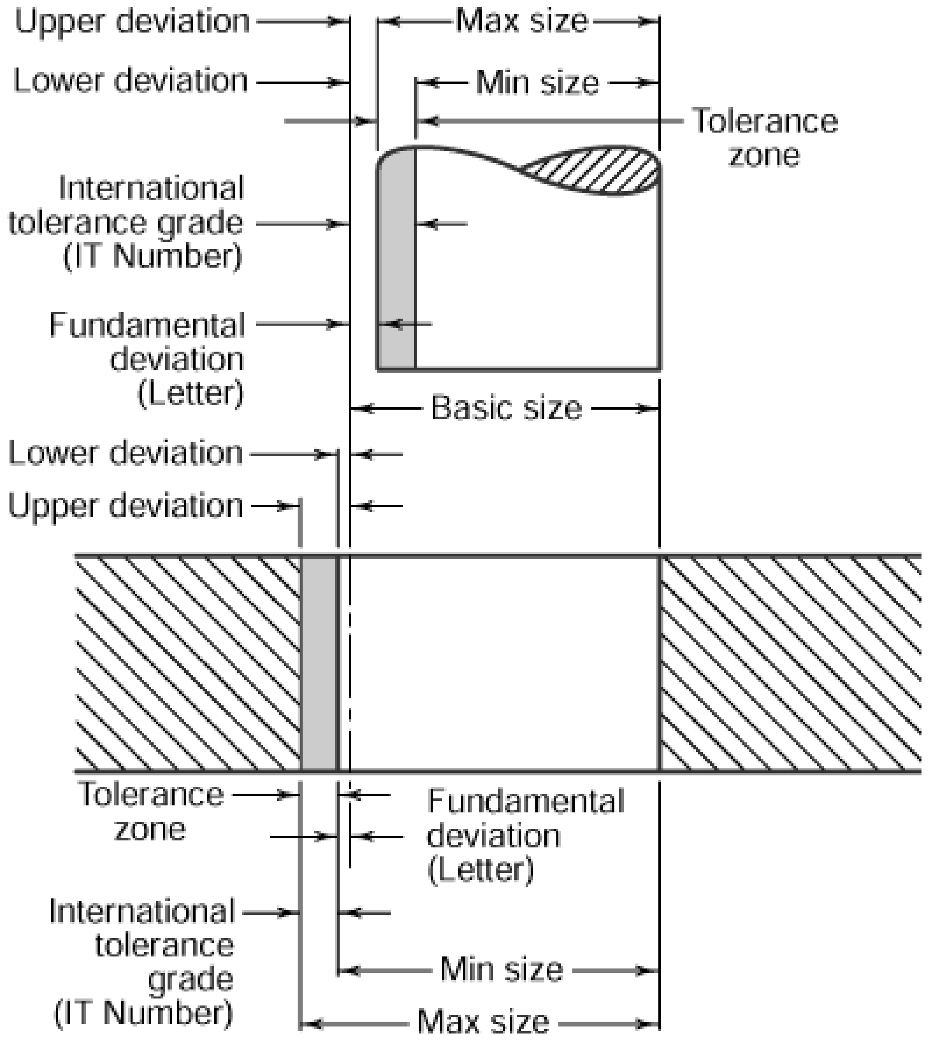


Tolerancing



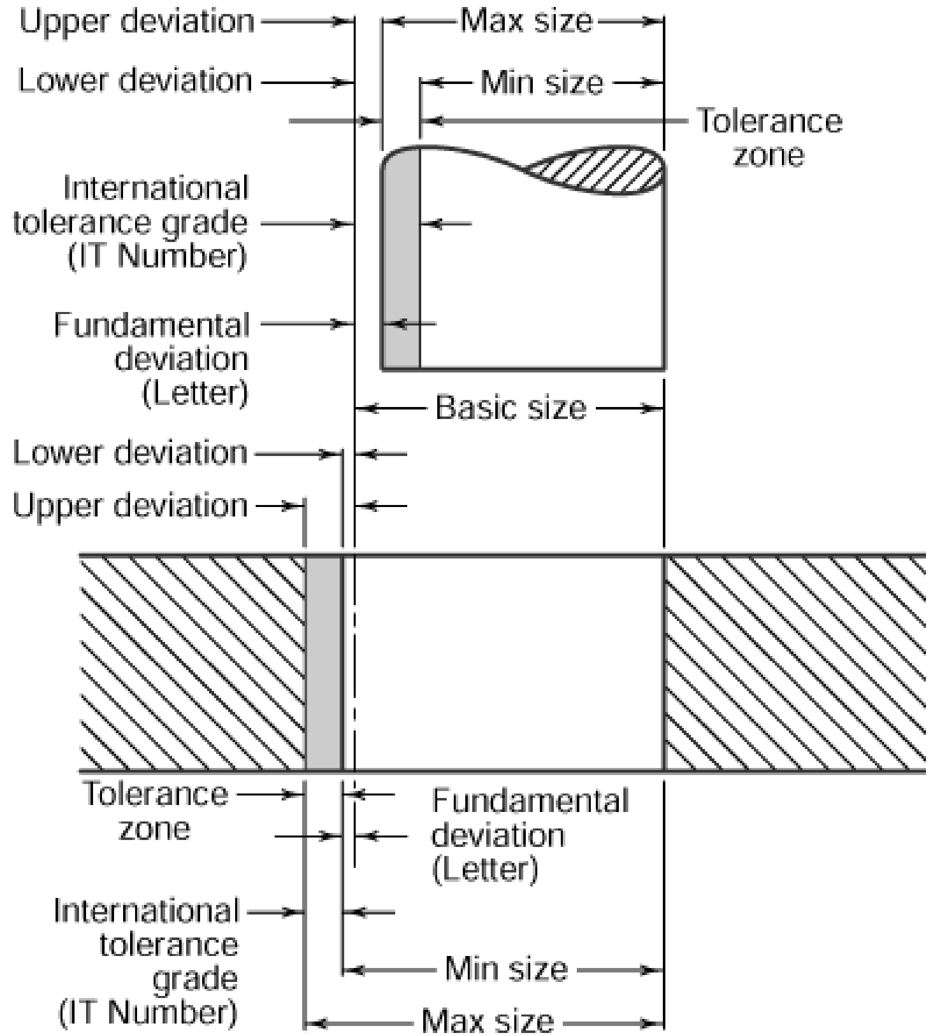
Tolerancing

- Metric Limits and Fits
  - Basic size
  - Deviation
  - Upper Deviation
  - Lower Deviation
  - Fundamental Deviation



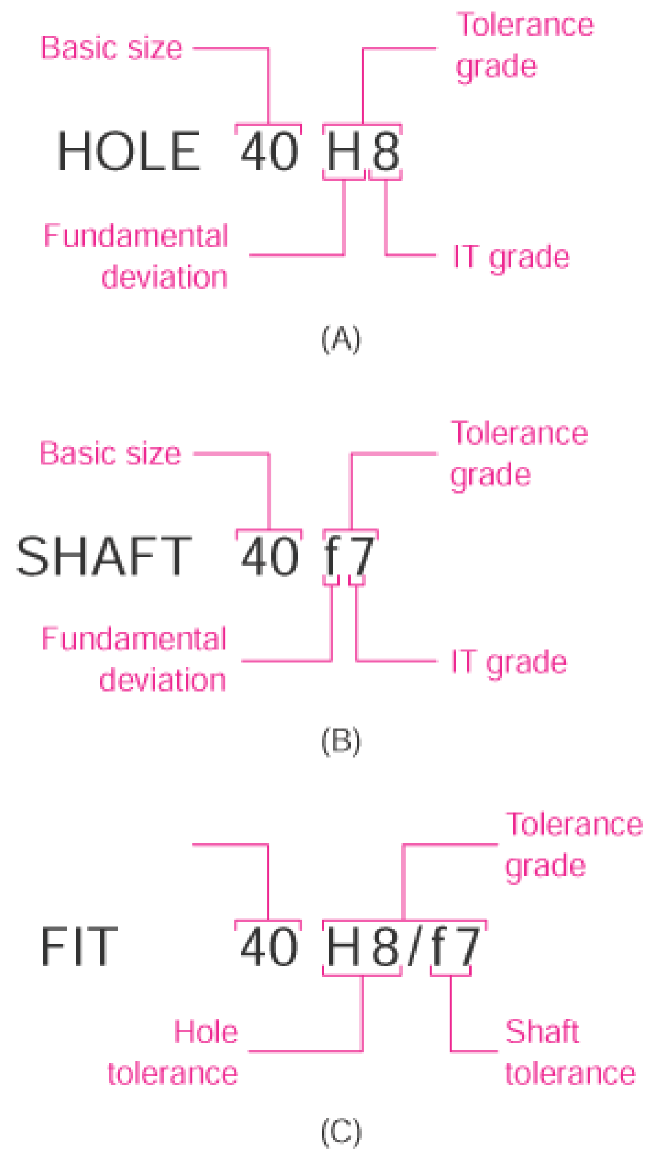
Tolerancing

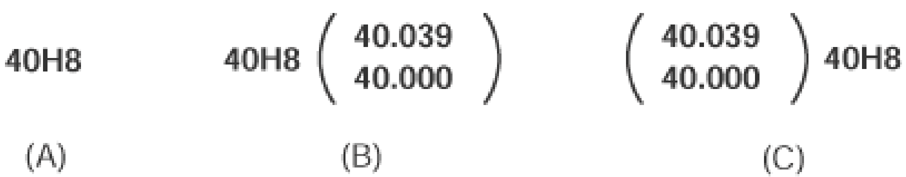
- Tolerance
- Tolerance zone
- International tolerance grade
- Hole basis
- Shaft basis



Tolerancing

- Symbols and Definitions
- Methods





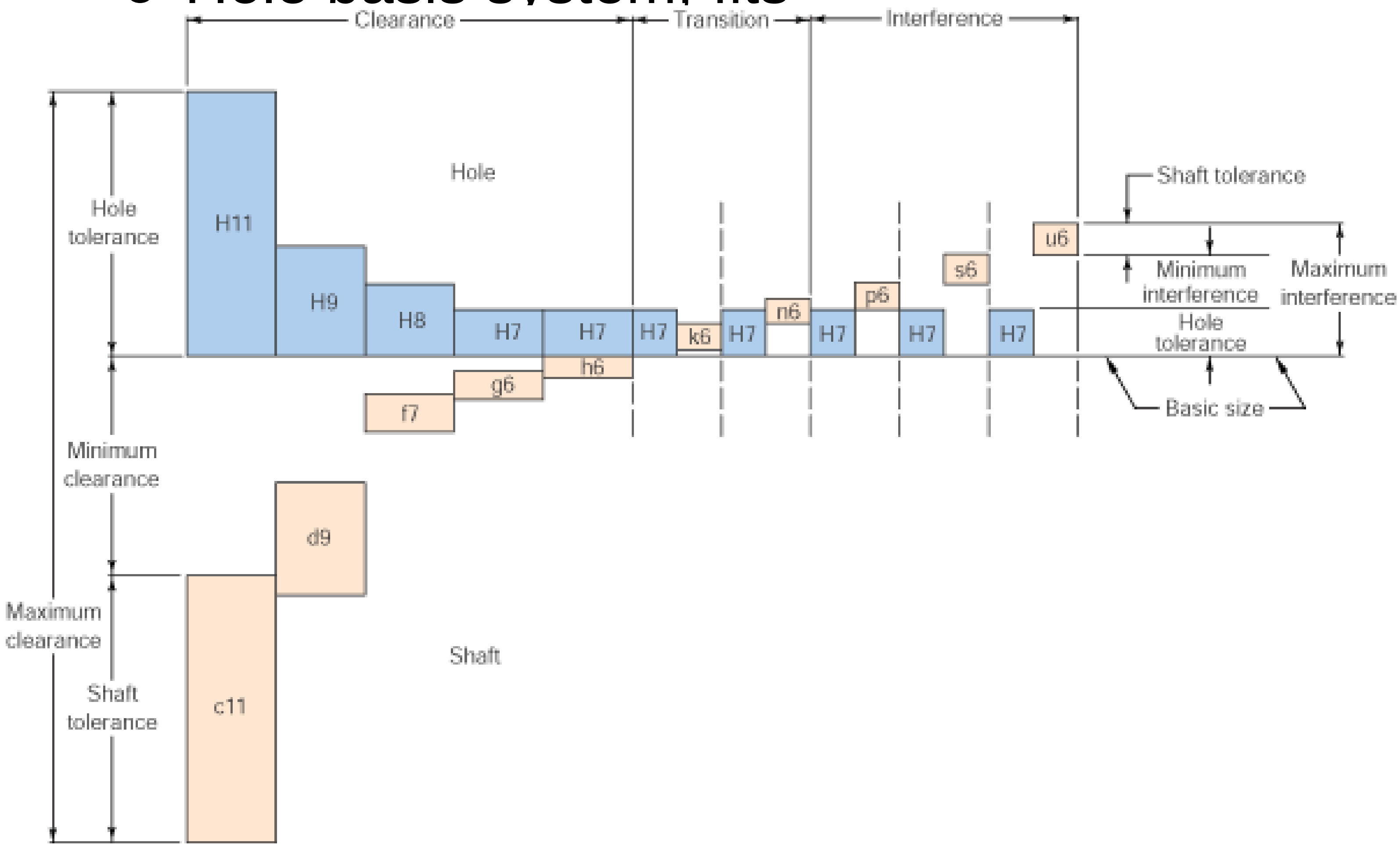
Tolerancing

- Standard Hole basis table; limits

BASIC SIZE		LOOSE RUNNING			FREE RUNNING			CLOSE RUNNING			SLIDING			LOCATIONAL CLEARANCE		
		Hole H11	Shaft c11	Fit	Hole H9	Shaft d9	Fit	Hole H8	Shaft f7	Fit	Hole H7	Shaft g6	Fit	Hole H7	Shaft h6	Fit
40	MAX	40.160	39.880	0.440	40.062	39.920	0.204	40.039	39.975	0.029	40.025	39.991	0.050	40.025	40.000	0.041
	MIN	40.000	39.720	0.120	40.000	39.858	0.060	40.000	39.950	0.025	40.000	39.975	0.009	40.000	39.984	0.000
50	MAX	50.160	49.870	0.450	50.062	49.920	0.204	50.039	49.975	0.089	50.025	49.991	0.050	50.025	50.000	0.041
	MIN	50.000	49.710	0.130	50.000	49.858	0.080	50.000	49.950	0.025	50.000	49.975	0.009	50.000	49.984	0.000
60	MAX	60.190	59.860	0.520	60.074	59.900	0.248	60.046	59.970	0.106	60.030	59.990	0.059	60.030	60.000	0.049
	MIN	60.000	59.670	0.140	60.000	59.826	0.100	60.000	59.940	0.030	60.000	59.971	0.010	60.000	59.981	0.000
80	MAX	80.190	79.550	0.530	80.074	79.900	0.248	80.046	79.970	0.106	80.030	79.990	0.059	80.030	80.000	0.049
	MIN	80.000	79.660	0.150	80.000	79.826	0.100	80.000	79.940	0.030	80.000	79.971	0.010	80.000	79.981	0.000
100	MAX	100.220	99.830	0.610	100.087	99.880	0.294	100.054	99.964	0.125	100.035	99.988	0.069	100.035	100.000	0.057
	MIN	100.000	99.610	0.170	100.000	99.793	0.120	100.000	99.929	0.036	100.000	99.966	0.012	100.000	99.978	0.000
120	MAX	120.220	119.820	0.620	120.087	119.880	0.294	120.054	119.964	0.125	120.035	119.988	0.069	120.035	120.000	0.057
	MIN	120.000	119.600	0.180	120.000	119.793	0.120	120.000	119.929	0.036	120.000	119.966	0.012	120.000	119.978	0.000
160	MAX	160.250	159.790	0.710	160.100	159.855	0.345	160.063	159.957	0.146	160.040	159.986	0.078	160.040	160.000	0.065
	MIN	160.000	159.540	0.210	160.000	159.755	0.145	160.000	159.917	0.043	160.000	159.961	0.014	160.000	159.975	0.000
200	MAX	200.290	199.760	0.820	200.115	199.830	0.400	200.072	199.950	0.168	200.046	199.985	0.040	200.046	200.000	0.075
	MIN	200.000	199.470	0.240	200.000	199.715	0.170	200.000	199.904	0.050	200.000	199.956	0.015	200.000	199.971	0.000
250	MAX	250.290	249.720	0.860	250.115	249.830	0.400	250.072	249.950	0.168	250.046	249.985	0.090	250.046	250.000	0.075
	MIN	250.000	249.430	0.280	250.000	249.715	0.170	250.000	249.904	0.050	250.000	249.956	0.015	250.000	249.971	0.000
300	MAX	300.320	299.670	0.970	300.130	299.810	0.450	300.081	299.944	0.189	300.052	299.983	0.101	300.052	300.000	0.084
	MIN	300.000	299.350	0.330	300.000	299.680	0.190	300.000	299.892	0.056	300.000	299.951	0.017	300.000	299.968	0.000
400	MAX	400.360	399.600	1.120	400.140	399.790	0.490	400.089	399.938	0.208	400.057	399.982	0.111	400.057	400.000	0.093
	MIN	400.000	399.240	0.400	400.000	399.650	0.210	400.000	399.881	0.062	400.000	399.946	0.018	400.000	399.964	0.000
500	MAX	500.400	499.520	1.280	500.155	499.770	0.540	500.097	499.932	0.228	500.063	499.980	0.123	500.063	500.000	0.103
	MIN	500.000	499.120	0.480	500.000	499.615	0.230	500.000	499.869	0.068	500.000	499.940	0.020	500.000	499.960	0.000

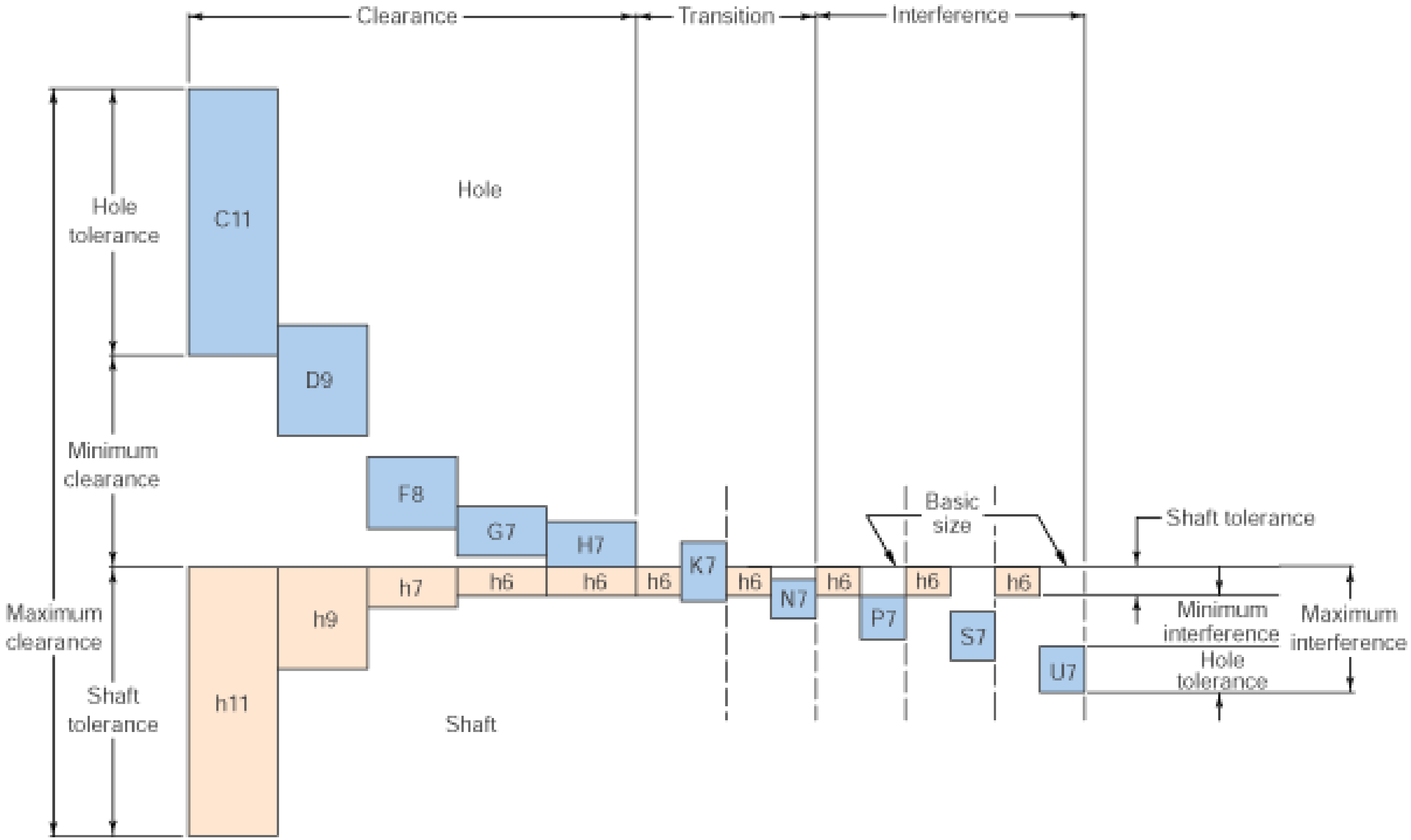
Tolerancing

- Hole basis system; fits



Tolerancing

- Shaft basis system; fits

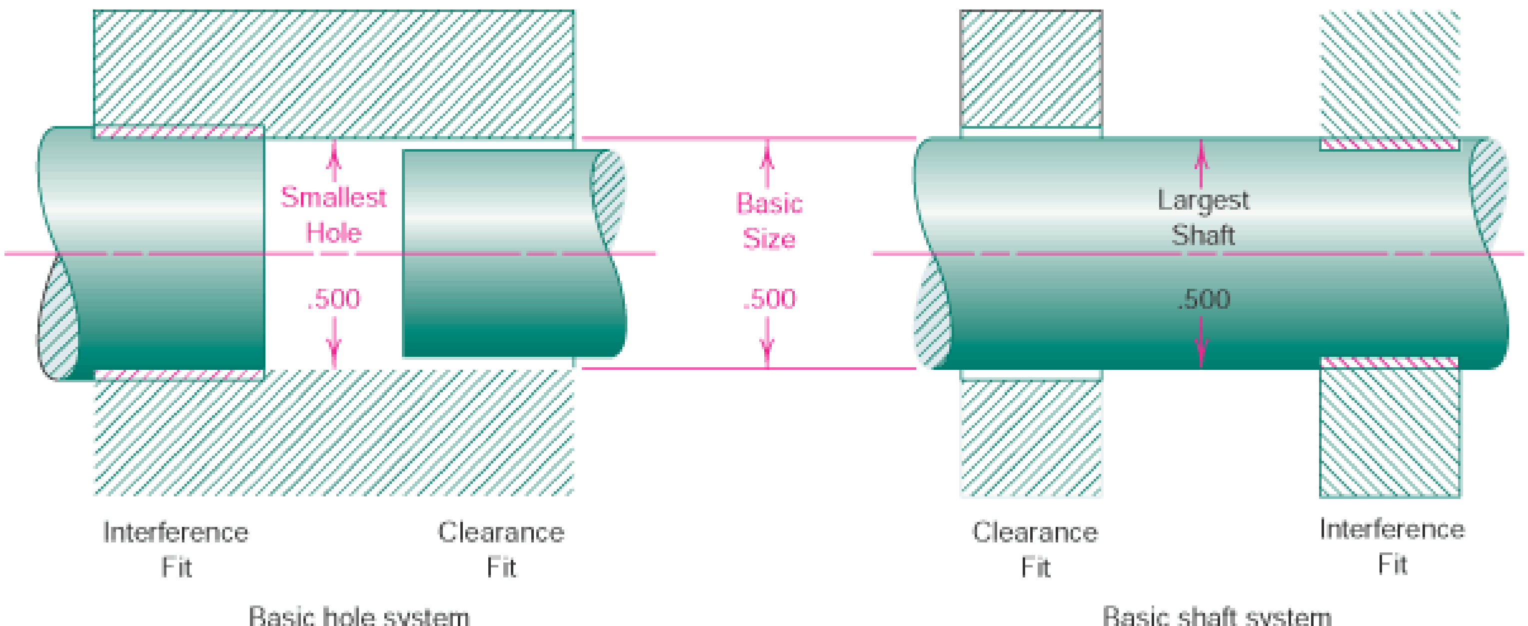


	ISO Symbol		Description	
	Hole Basis	Shaft Basis		
Clearance fits	H11/c11	C11/h11	Loose running fit for wide commercial tolerances or allowances on external members	More clearance
Clearance fits	H9/d9	D9/h9	Free running fit not for use where accuracy is essential, but good for large temperature variations, high running speeds, or heavy journal pressures	
Clearance fits	H8/f7	F8/h7	Close running fit for running on accurate machines and for accurate location at moderate speeds and journal pressures	
Clearance fits	H7/g6	G7/h6	Sliding fit not intended to run freely but to move and turn freely and locate accurately	
Transition fits	H7/h6	H7/h6	Locational clearance fit provides snug fit for locating stationary parts but can be freely assembled and disassembled	
Transition fits	H7/k6	K7/h6	Locational transition fit for accurate location; a compromise between clearance and interference	
Transition fits	H7/n6	N7/h6	Locational transition fit for more accurate location where greater interference is permissible	
Interference fits	H7/p6*	P7/h6	Locational interference fit for parts requiring rigidity and alignment with prime accuracy of location but without special bore pressure requirements	
Interference fits	H7/s6	S7/h6	Medium drive fit for ordinary steel parts or shrink fits on light sections; the tightest fit usable with cast iron	
Interference fits	H7/u6	U7/h6	Force fit suitable for parts that can be highly stressed or for shrink fits where the heavy pressing forces required are impractical	More interference

*Transition fit for basic sizes in range from 0 through 3 mm

Tolerancing

- Standard Precision Fit; English Units
  - Running and Sliding (RC)
  - Clearance Locational (LC)
  - Transition Locational (LT)
  - Interference Locational (LN)
  - Force and Shrinks (FN)



Geometric Dimensioning and Tolerancing

- GDT is a method of defining parts based on how they function, using standard ASME/ANSI symbols.

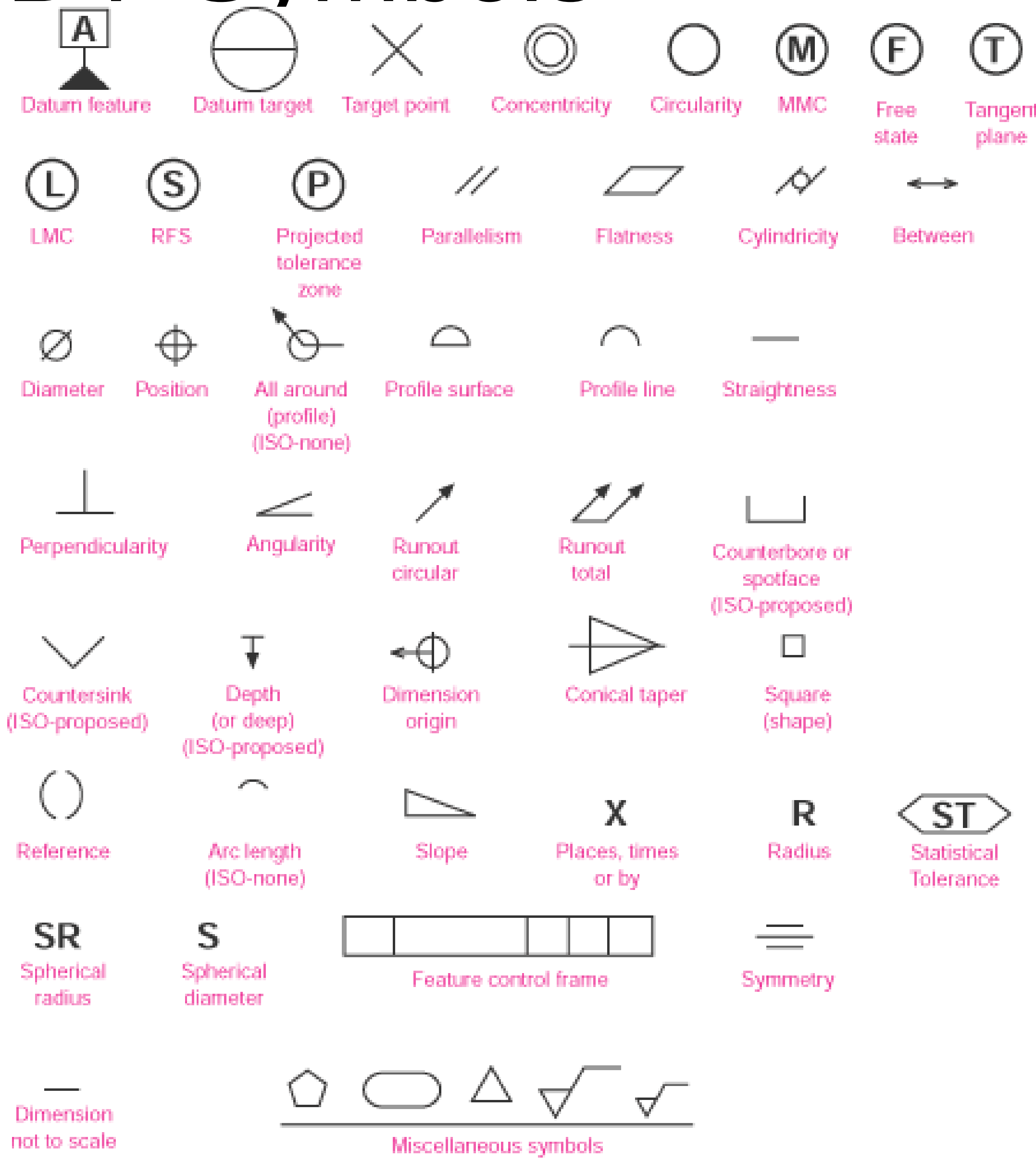
Geometric Dimensioning and Tolerancing

- Within the last 15 years there has been considerable interest in GDT, in part because of the increased popularity of **statistical process control**. This control process, when combined with GDT, helps reduce or eliminate inspection of features on the manufactured object. The flipside is that the part must be toleranced very efficiently; this is where GDT comes in.

Geometric Dimensioning and Tolerancing

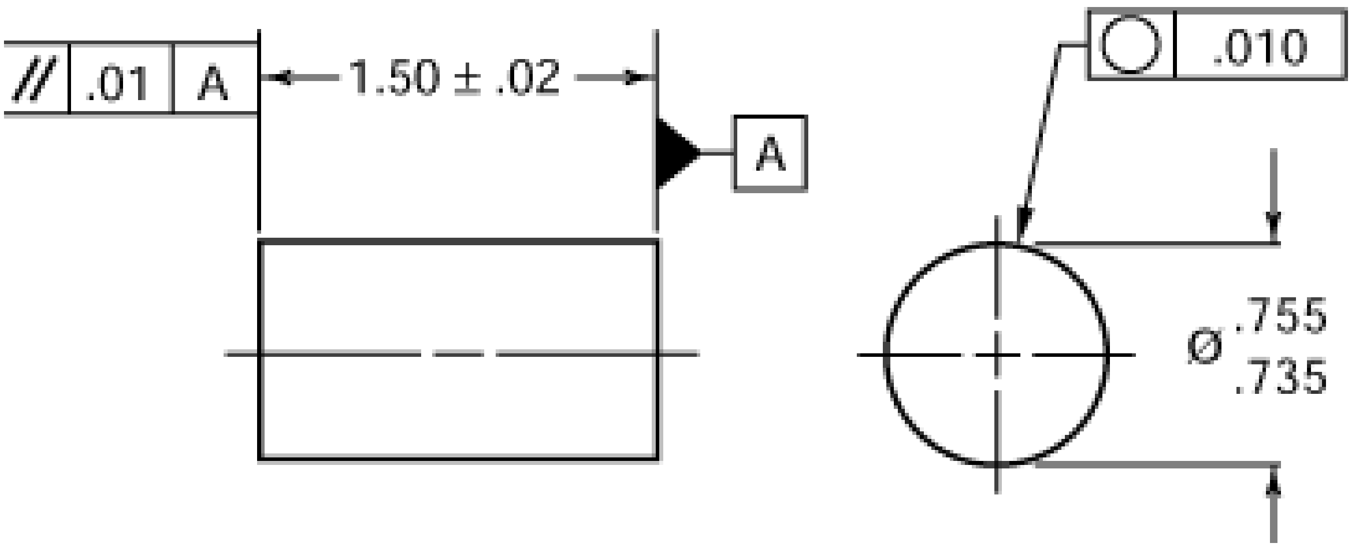
- Another reason for the increased popularity of GDT is the rise of worldwide standards, such as ISO 9000, which require universally understood and accepted methods of documentation.

GDT-Symbols


A
Datum feature
Datum target
Target point
Concentricity
Circularity
M
MMC
F
Free state
T
Tangent plane
L
LMC
S
RFS
P
Projected tolerance zone
Parallelism
Flatness
Cylindricity
Between
Diameter
Position
All around (profile) (ISO-none)
Profile surface
Profile line
Straightness
Perpendicularity
Angularity
Runout circular
Runout total
Counterbore or spotface (ISO-proposed)
Countersink (ISO-proposed)
Depth (or deep) (ISO-proposed)
Dimension origin
Conical taper
Square (shape)
Reference
Arc length (ISO-none)
Slope
X
Places, times or by
R
Radius
ST
Statistical Tolerance
SR
Spherical radius
S
Spherical diameter
Feature control frame
Symmetry
Dimension not to scale
Miscellaneous symbols

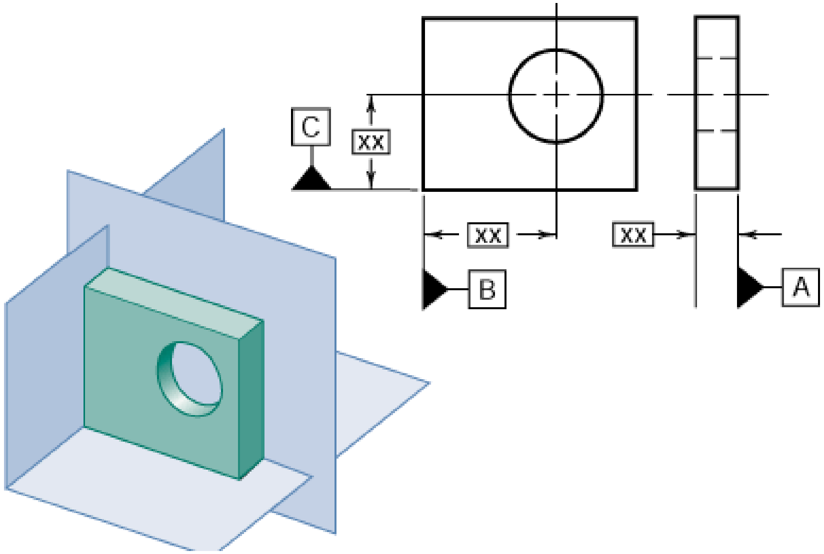
GDT

- Feature control frames



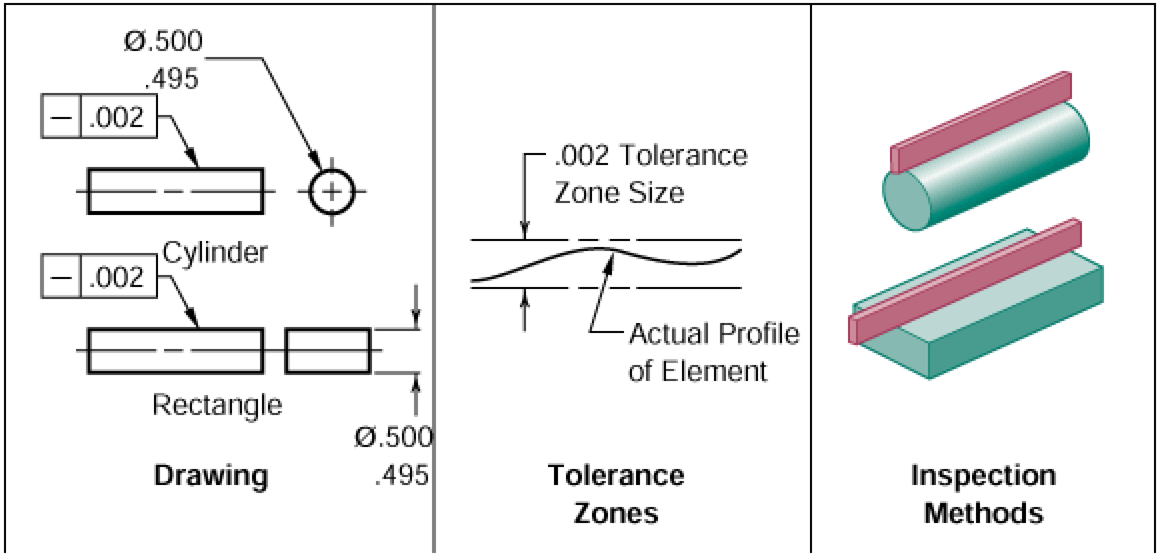
GDT

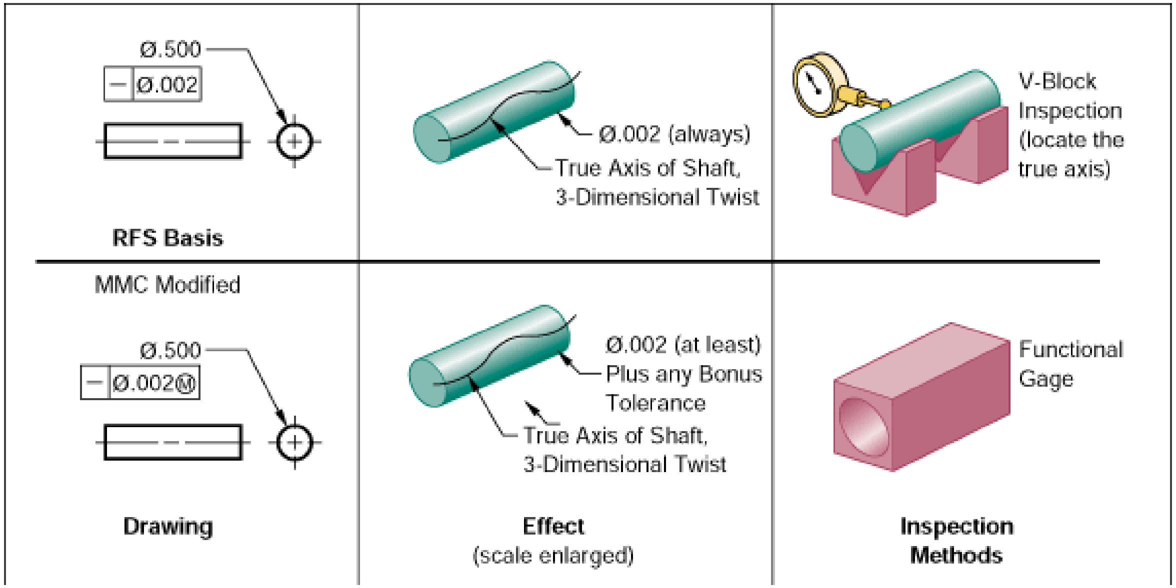
- MMC/LMC
- Datums
- Geometric Controls
  - Form
  - Orientation
  - Position



GDT

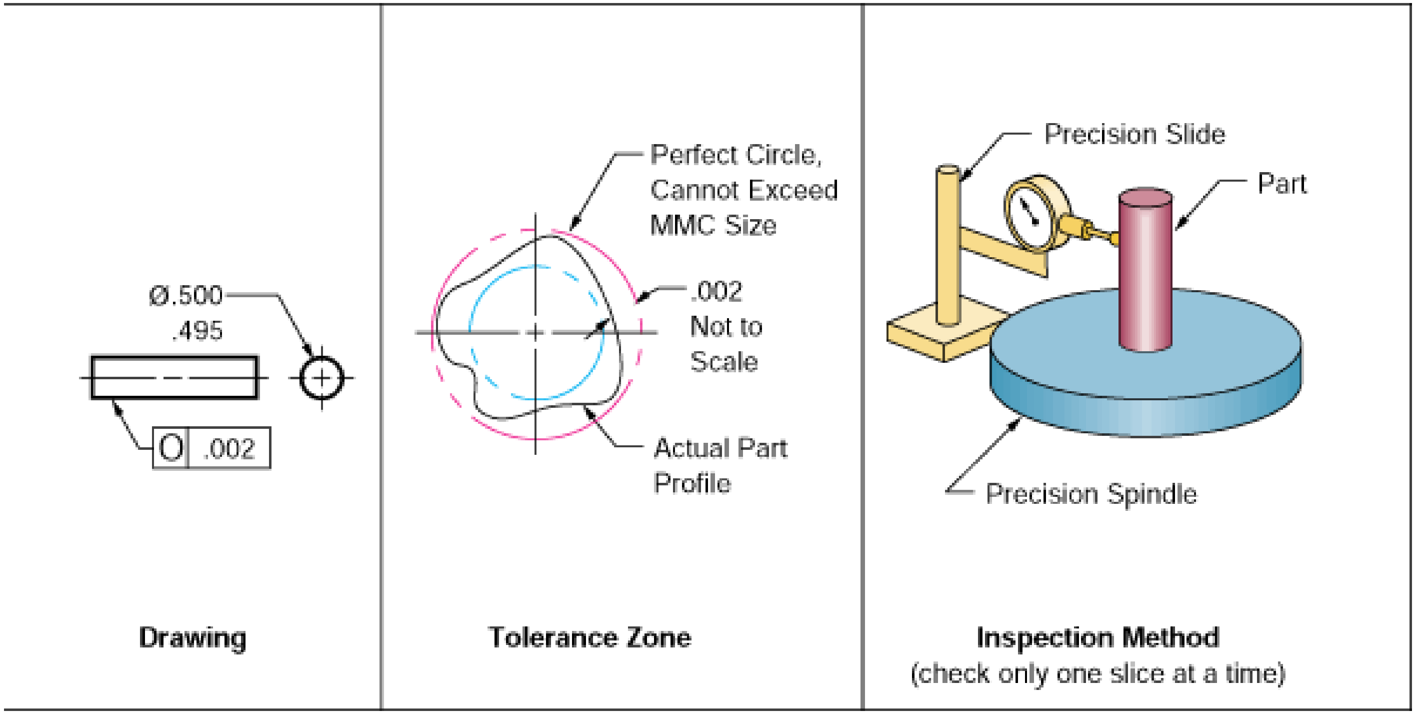
- Forms
  - Straightness
    - Line element
    - Axis





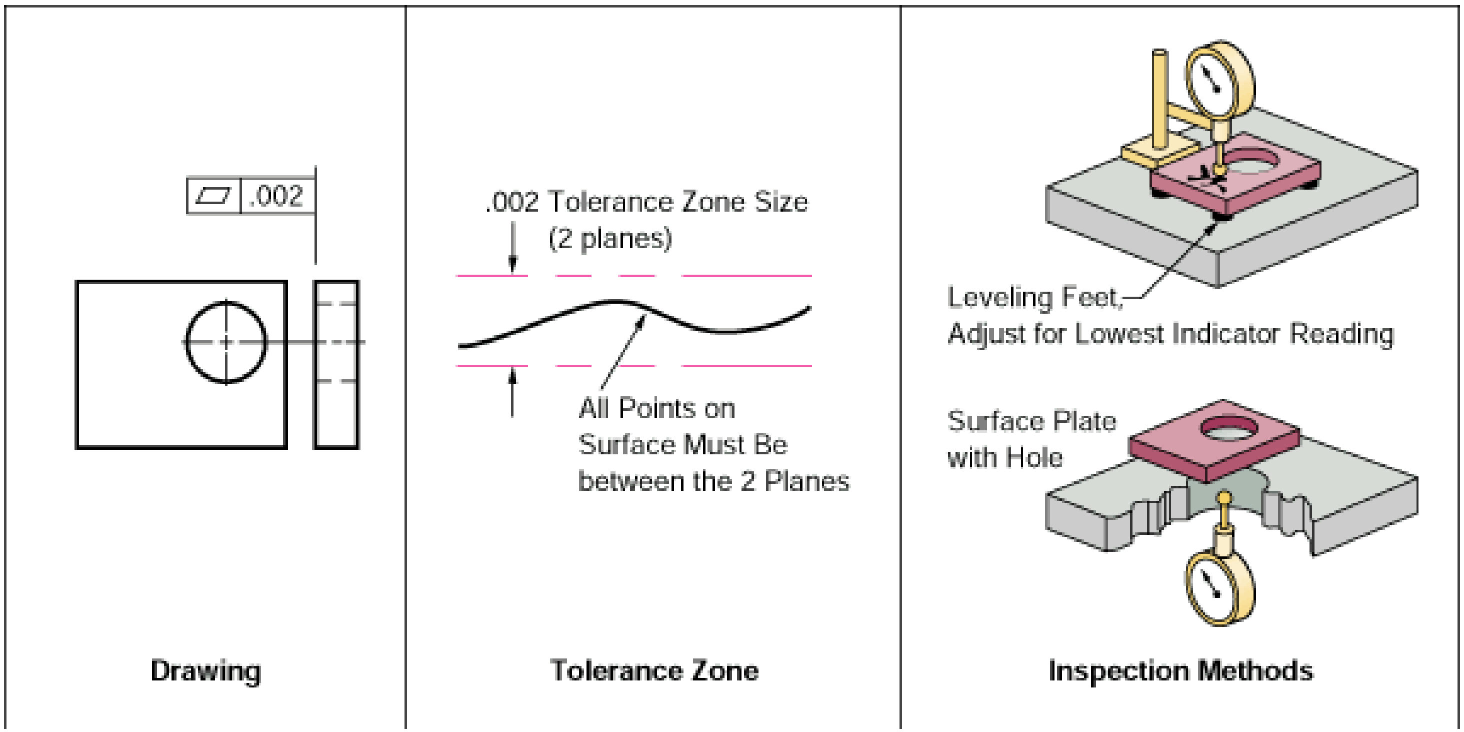
GDT

- Forms
  - Circularity



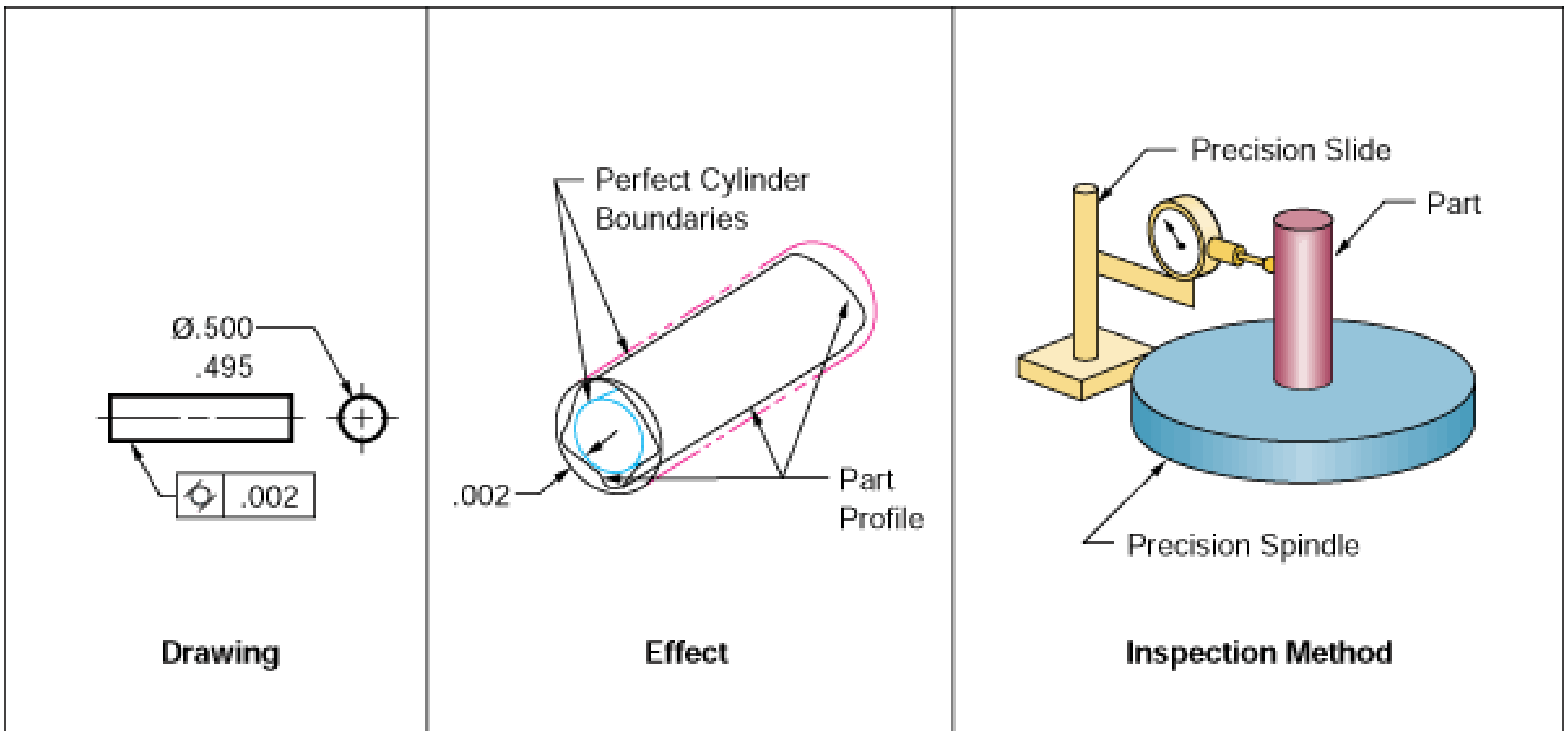
GDT

- Forms
  - Flatness



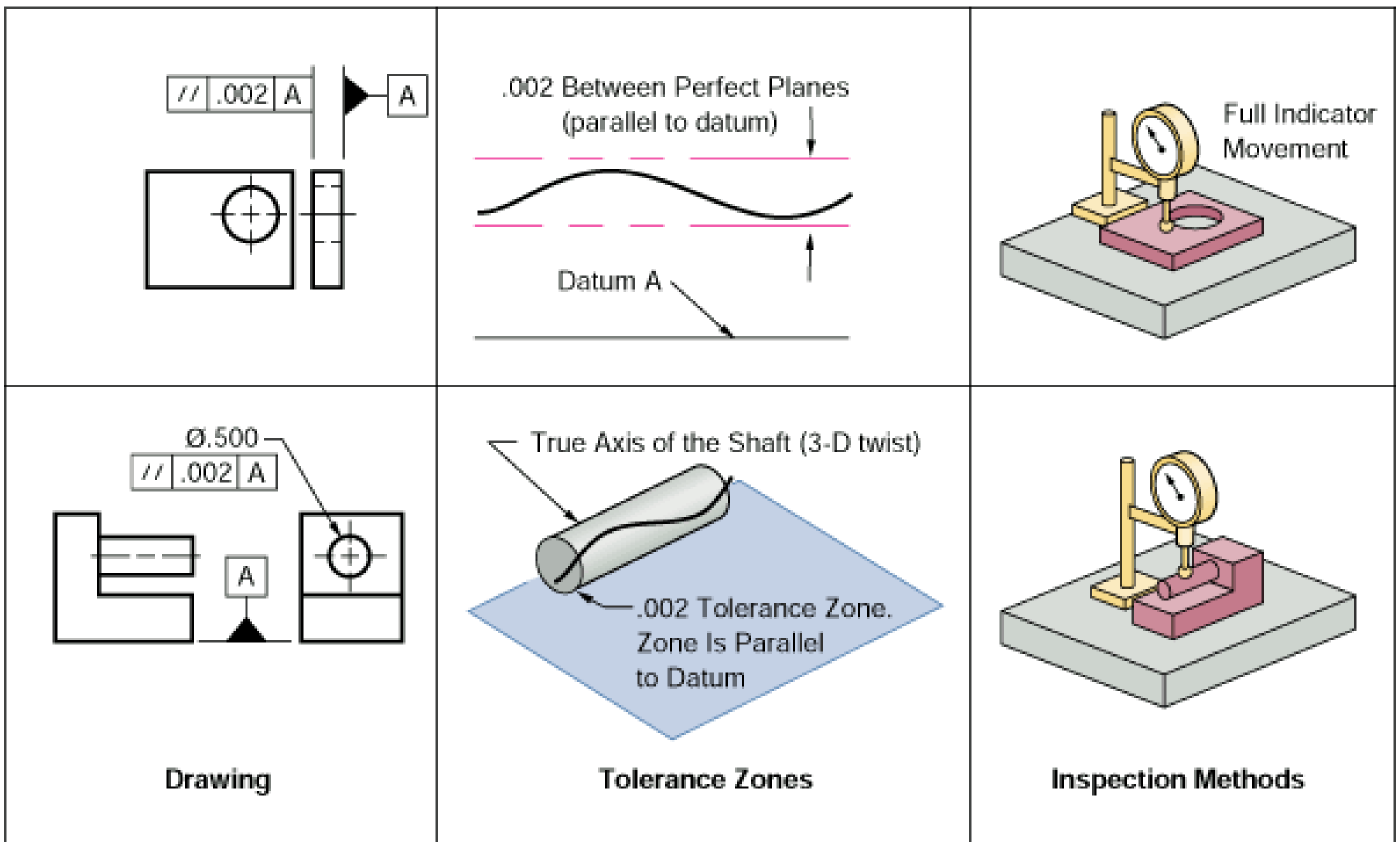
GDT

- Forms
  - Cylindricity



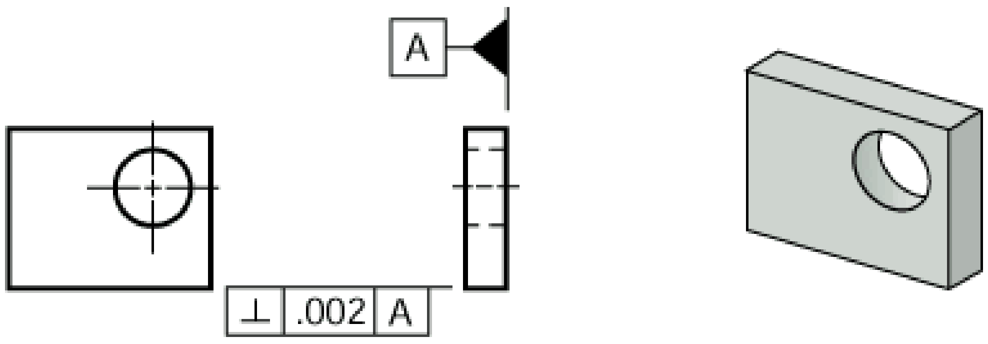
GDT

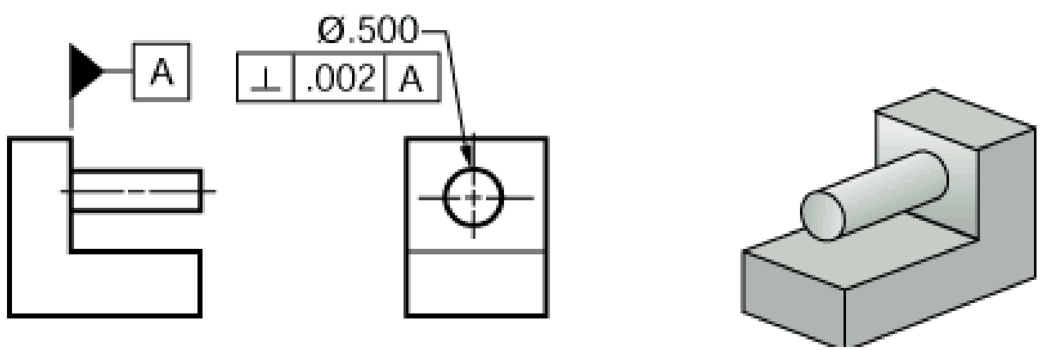
- Orientation
  - Parallelism



GDT

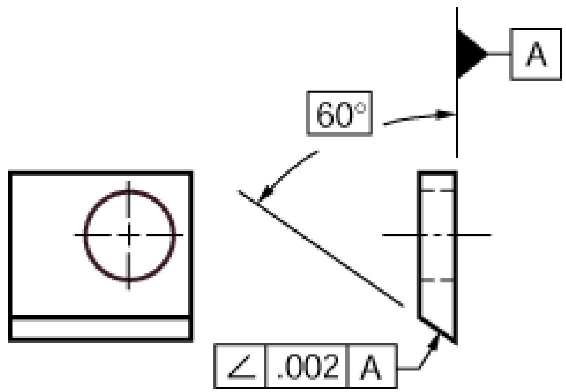
- Orientation
  - Perpendicularity

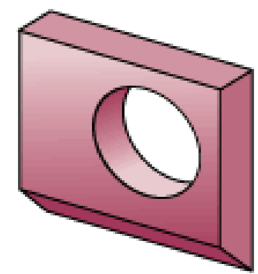


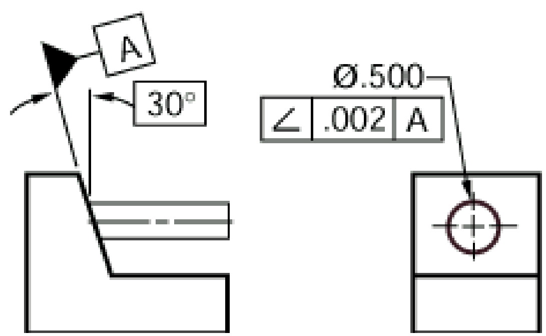


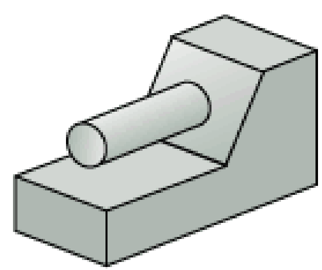
GDT

- Orientation
  - Angularity



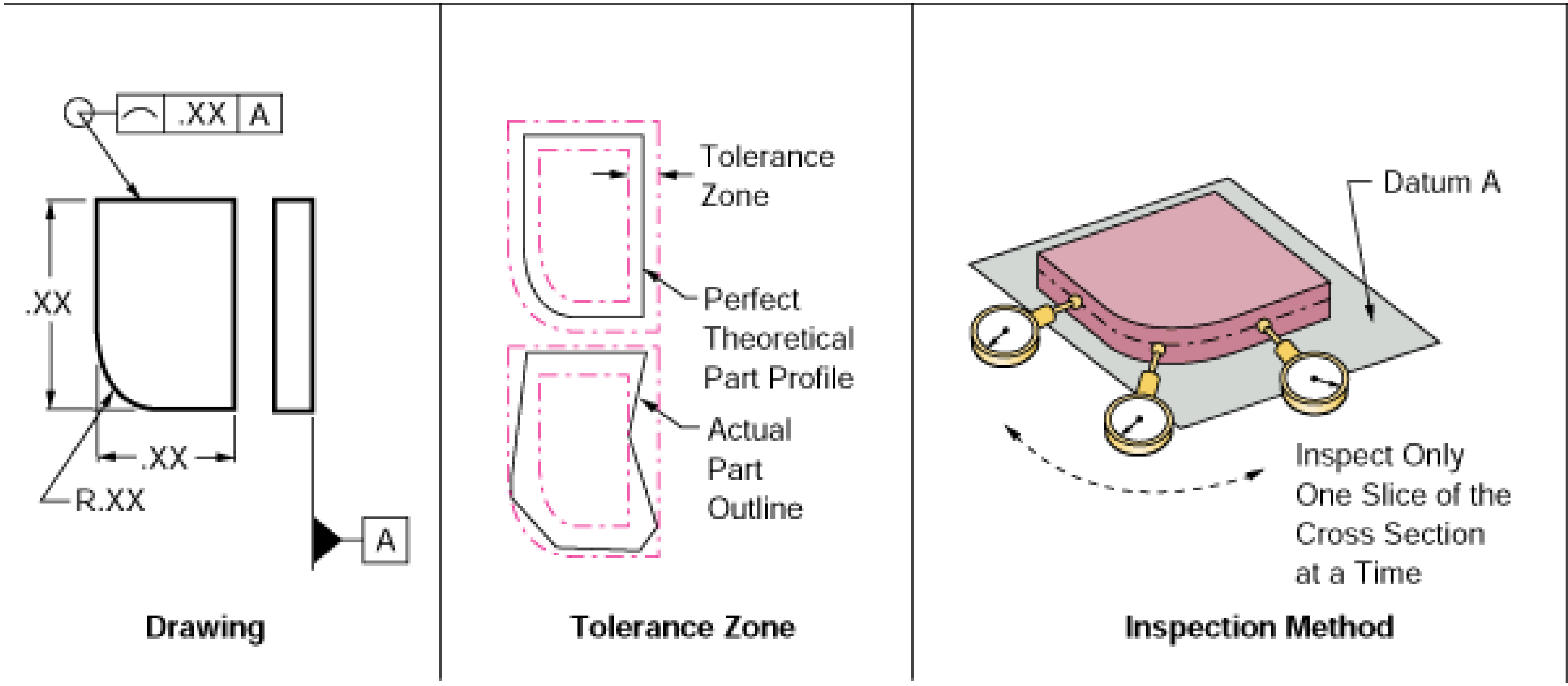






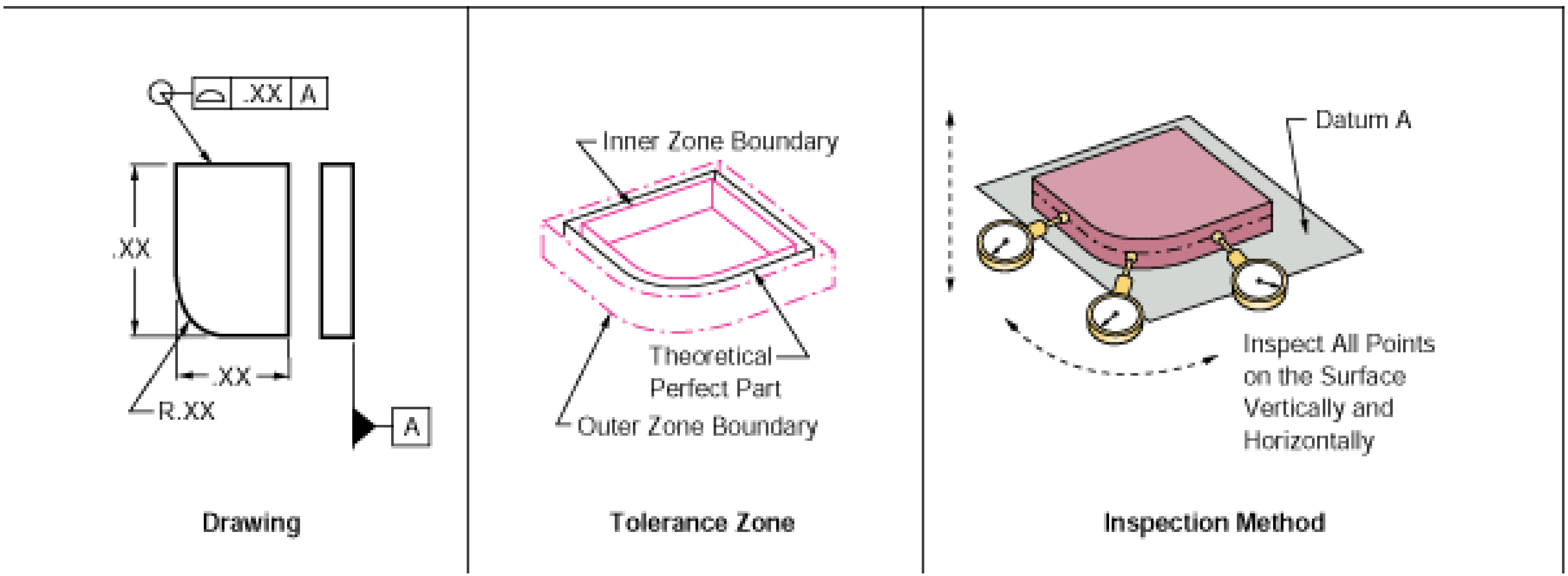
GDT

- Orientation
  - Line profile



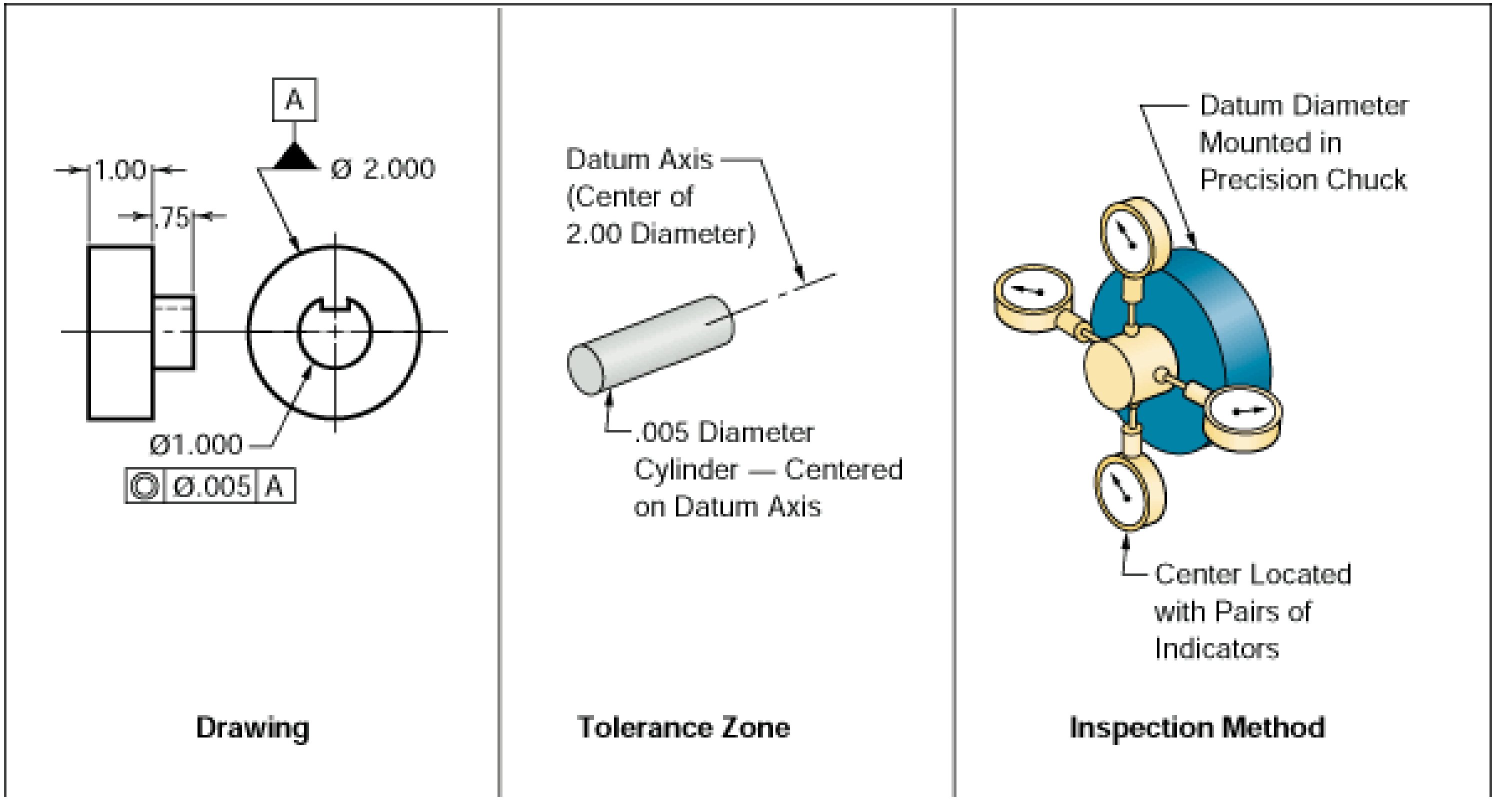
GDT

- Orientation
  - Surface profile



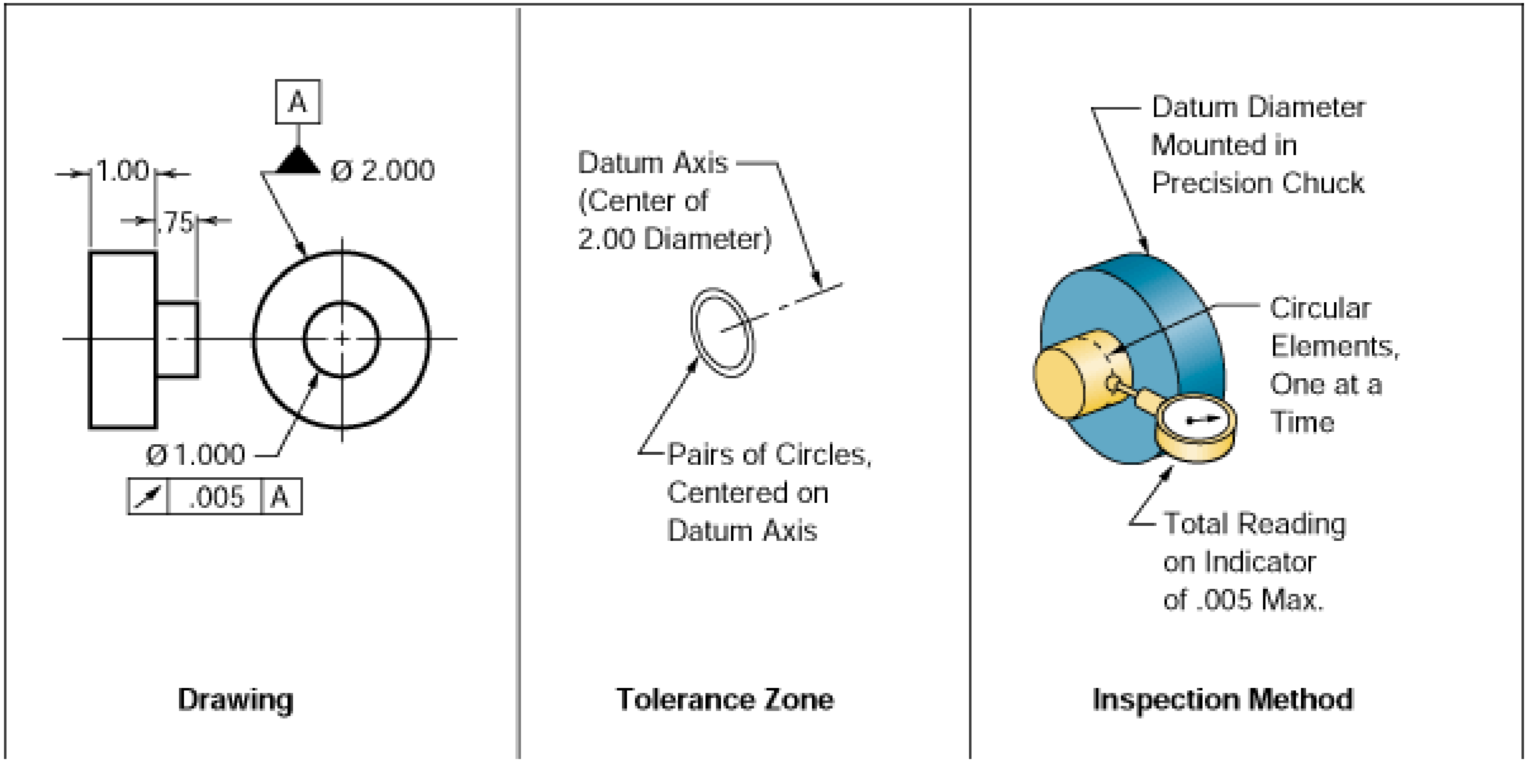
GDT

- Location
  - Concentricity



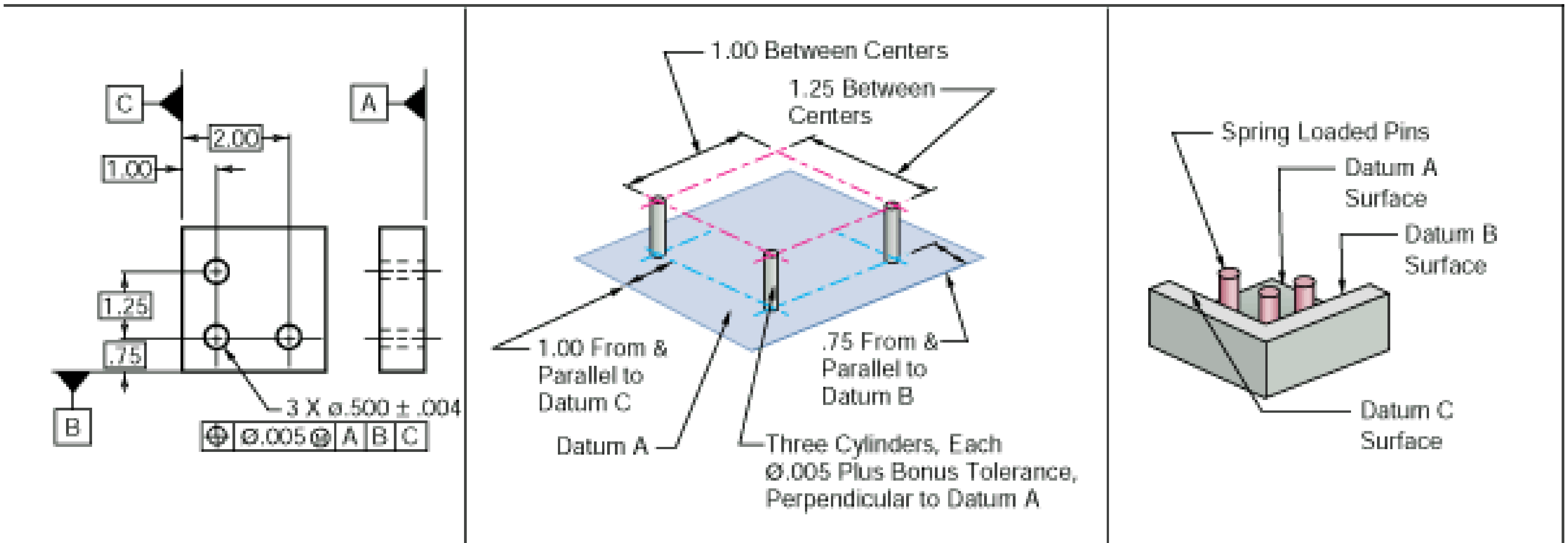
GDT

- Location
  - Runout



GDT

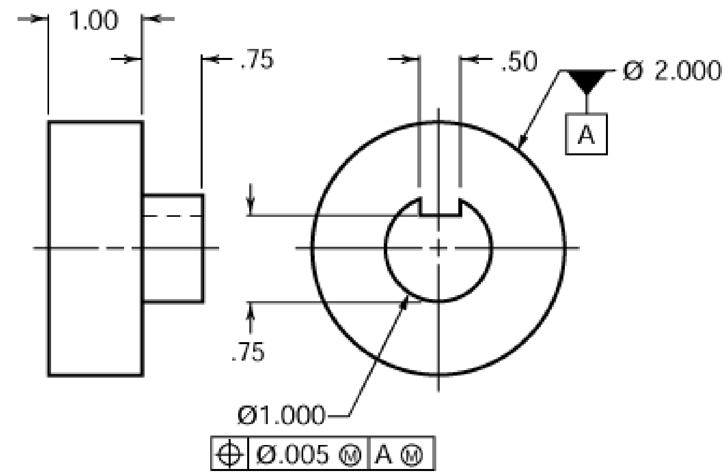
- Location
  - Position

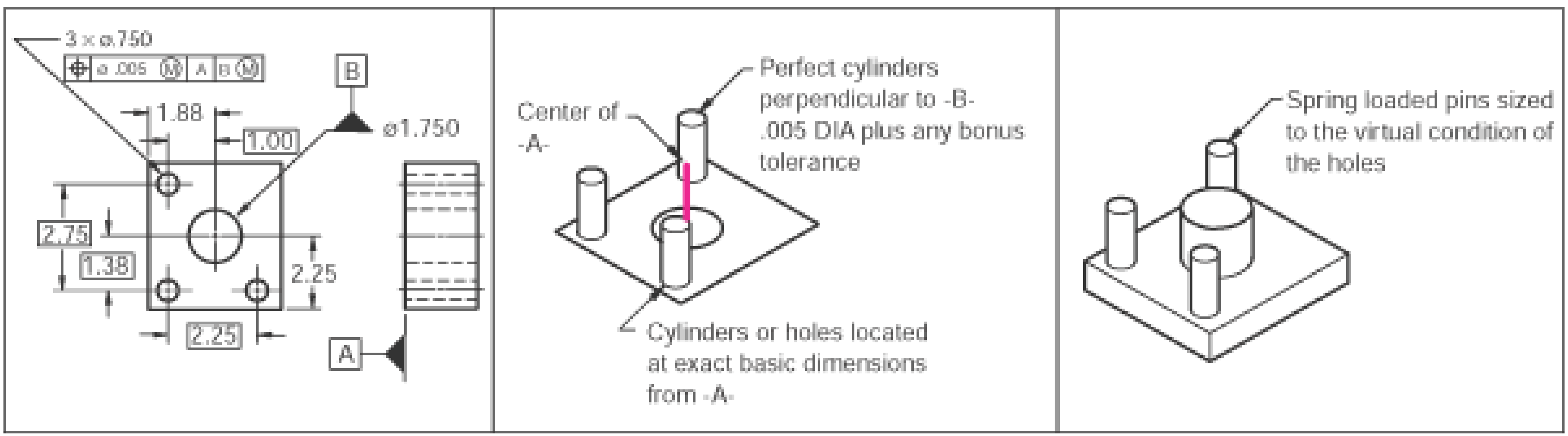


Drawing | Tolerance Zones | Functional Gage

GDT

- Location
  - Position





GDT

- Tolerance Calculation
  - *Floating fastener tolerancing* is used to confirm that loose bolts, screws or other fasteners have the standard clearance in their holes.
  - *Fixed fastener tolerancing* is measured the same as with floating fasteners except that the fastener is already fixed/located on one of the mating parts and the tolerance is now divided between the parts.

GDT

- Tolerance Calculation
  - *Hole diameter tolerancing* is used to calculate the MMC of the hole.

GDT

- Design Application
  - Five-Step
    - Isolate and define the functions of the features/part.
    - Prioritize the functions.
    - Identify the datum reference frame based on functional priorities.
    - Select the proper control(s).
    - Calculate the tolerance values.