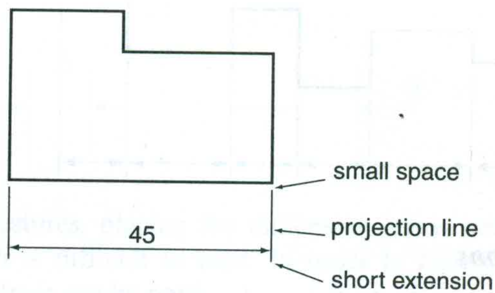
Chapter 4

Dimensioning, Line Types and Other Views

4.1 Dimensioning

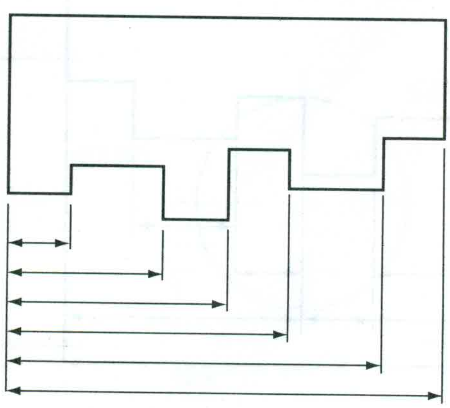
In addition to describing the shape of objects, many drawings must show dimensions, so that workers can build the structure or fabricate parts that will fit together. This is accomplished by placing the required values (measurements) along dimension lines (usually outside the outlines of the object) and by giving additional information in the form of notes which are referenced to the parts in question by angled lines called leaders[2].

An example dimension is shown below[26].

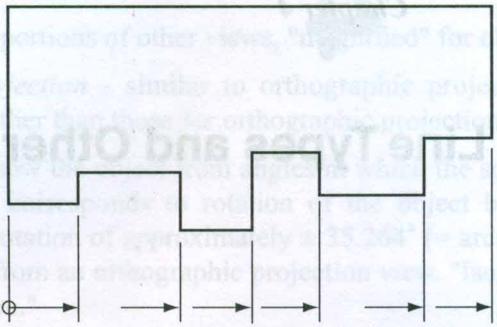


4.2 Types of Dimensioning[27]

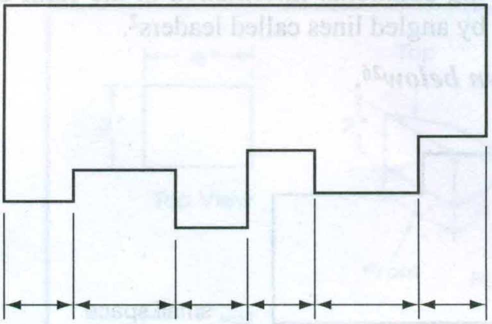
Parallel Dimensioning



Superimposed running dimension

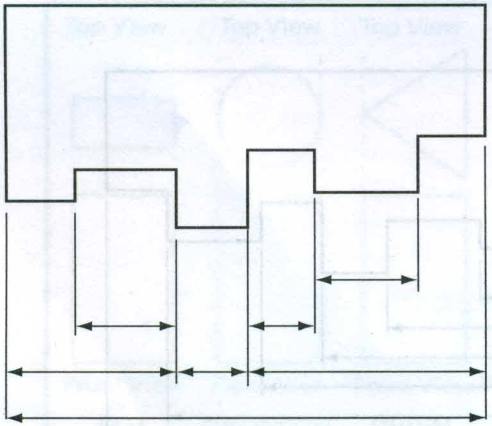


Chain Dimensioning



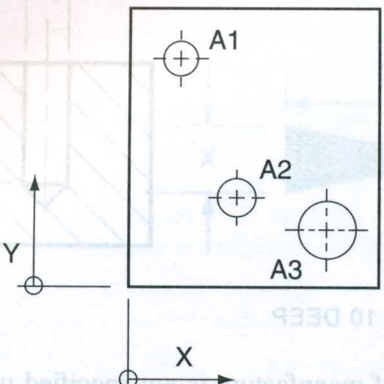
Combined Dimensions

A combined dimension uses both chain and parallel dimensioning.



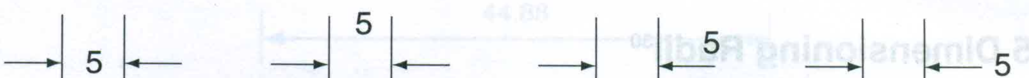
Simplified dimensioning by co-ordinates

It is also possible to simplify co-ordinate dimensions by using a table to identify features and positions.



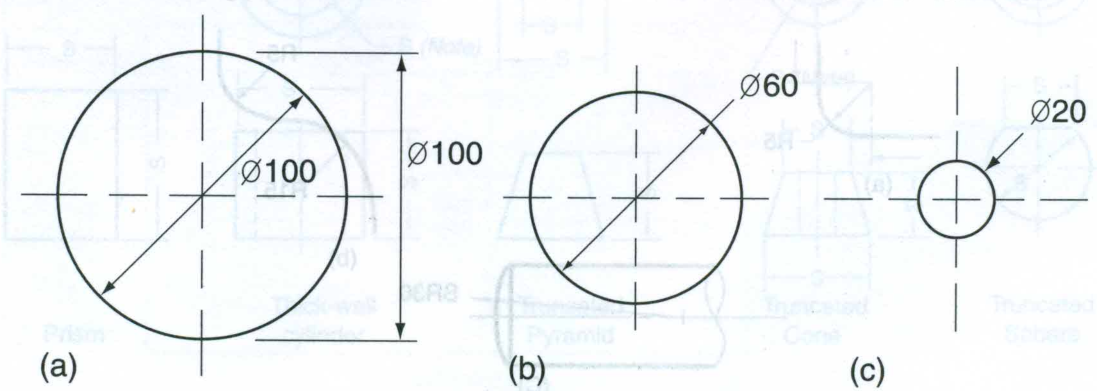
HOLE	X	Y	Q
A1	100	25	25
A2	50	40	15
A3	100	20	15

Dimensioning Small Features



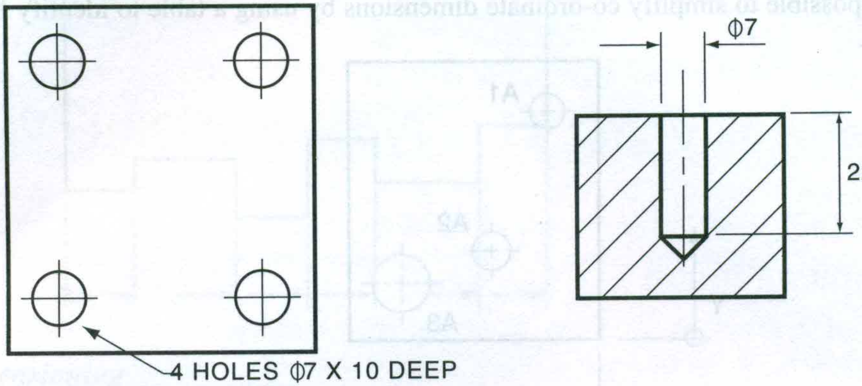
When dimensioning small features, placing the dimension arrow between projection lines may create a drawing which is difficult to read. In order to clarify dimensions on small features any of the above methods can be used.

4.3 Dimensioning Circles[28]



(a) (b) (c)

4.4 Dimensioning Holes[29]

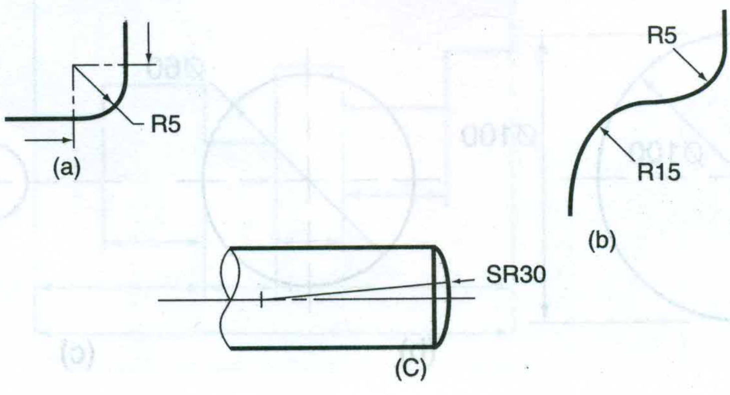


When dimensioning holes the method of manufacture is not specified unless they necessary for the function of the product. The word hole doesn't have to be added unless it is considered necessary. The depth of the hole is usually indicated if it is isn't indicated on another view. The depth of the hole refers to the depth of the cylindrical portion of the hole and not the bit of the hole caused by the tip of the drip.

4.5 Dimensioning Radii[30]

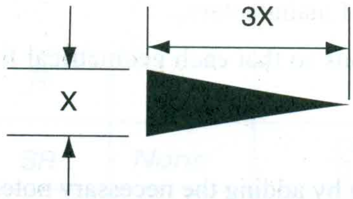
All radial dimensions are proceeded by the capital R. All dimension arrows and lines should be drawn perpendicular to the radius so that the line passes through the centre of the arc. All dimensions should only have one arrowhead which should point to the line being dimensioned. There are two methods for dimensiong radii.

(a) shows a radious dimensioned with the centre of the radius located on the drawing.
(b) shows how to dimension radii which do not need their centers locating.
(c) Spherical dimensions: The radius of a spherical surface (i.e. the top of a drawing pin) when dimensioned should have an SR before the size to indicate the type of surface.



4.6 Arrowhead

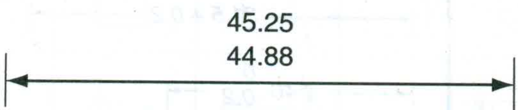
The ratio of length and width of an arrowhead should be approximately 3:1. It could be approximately 3 mm long and 1 mm wide [31].



4.7 Tolerance

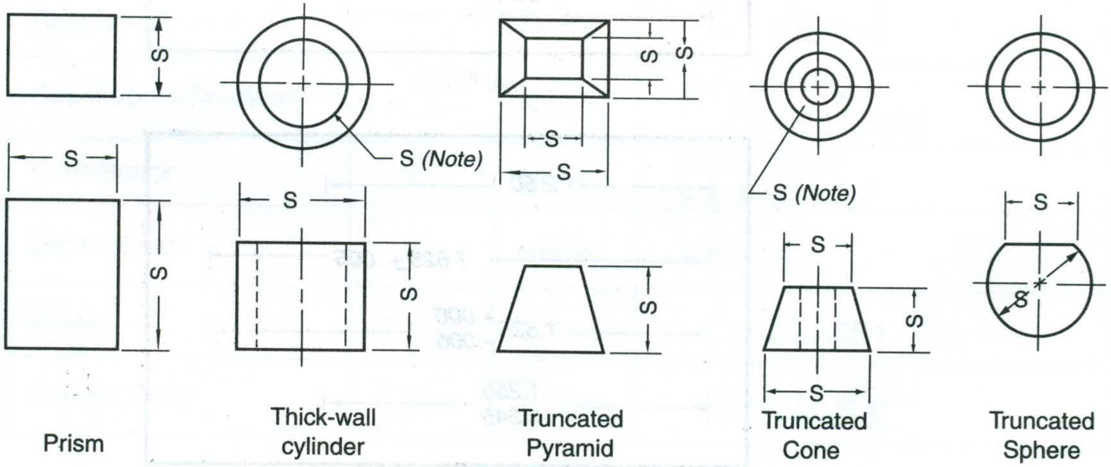
A tolerance value shows the manufacturing department the maximum permissible variation from the dimension.

The method of expressing a tolerance on a dimension as recommended by the British standards is shown below:[32]



Note the larger size limit is placed above the lower limit.

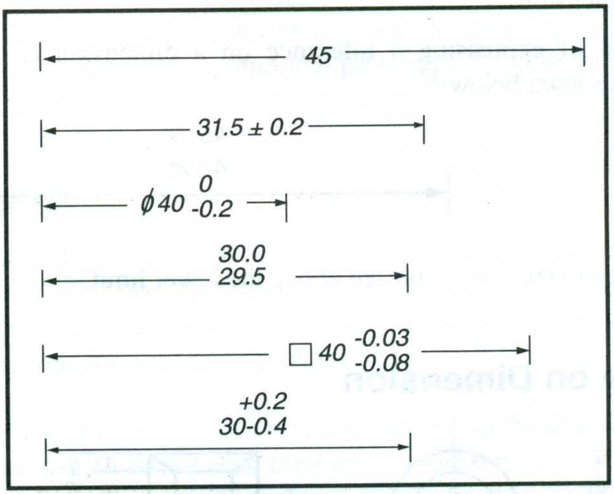
4.8 More on Dimension



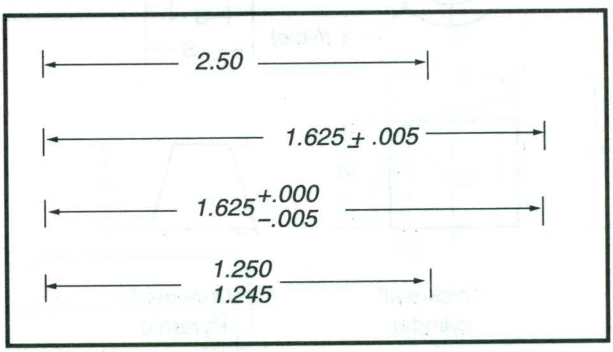
The theory of dimensioning may be applied in six steps[33], as follows:

1. Mentally divide the object into its component geometrical shapes.
2. Place the size dimensions on each form.
3. Select the locating center lines and surfaces after giving careful consideration to mating parts and to the processes of manufacture.
4. Place the location dimensions so that each geometrical form is located from a center line or finished surface.
5. Add the overall dimensions.
6. Complete the dimensioning by adding the necessary notes.

4.9 Dimension Line[34]

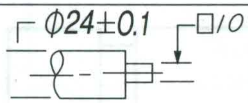
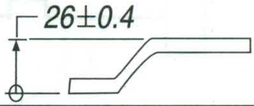


(a) Metric



(b) Decimal Inch

4.10 Drawing Symbols[35]

ANSI Y14.5M-1982 AND ISO			
SYMBOL FOR	ANSI	ISO	EXAMPLES
Radius	R	R	R1oo
Spherical Radius	SR	None	
Diameter	Ø	Ø	Ø10±0.1
Spherical Diameter	SØ	None	SØ36$^{0}_{-0.4}$
Arc Length	⌒	None	⌒40
Square (Shape)	□	□	
Dimension Origin	⌖→	None	
Dimension Not To Scale	<u>18</u>	<u>18</u>	⊢— <u>95</u> —⊣
Number of Times /Places	X	X	Space 5X Ø8
Reference Dimension	( )	None	⊢— (50) —⊣
Counterbore/Spotface	⌴	None	Ø6±0.1THRU ⌴ Ø10±0.1T3±0.1
Countersink	⌵	None	Ø6±0.1THRU ⌵Ø10±0.1X82°
Depth /Deep	↧	None	↧3±0.1
Slope	◺	◺	◺ 0.25:1
Conical Taper	⌲	⌲	⌲ 0.25:1

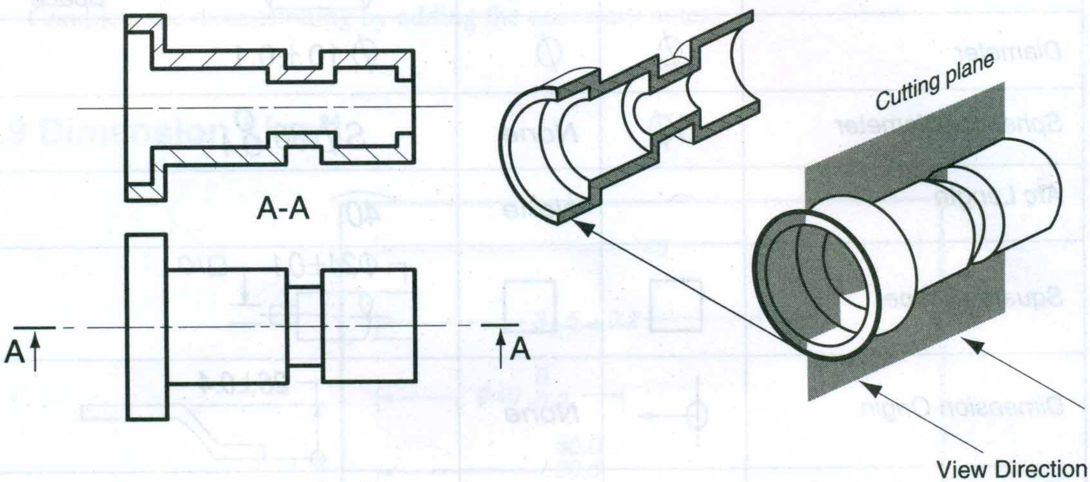
General drawing symbols.

4.11 Sectional Views[36]

Sections and sectional views are used to show hidden detail more clearly. They are created by using a cutting plane to cut the object.

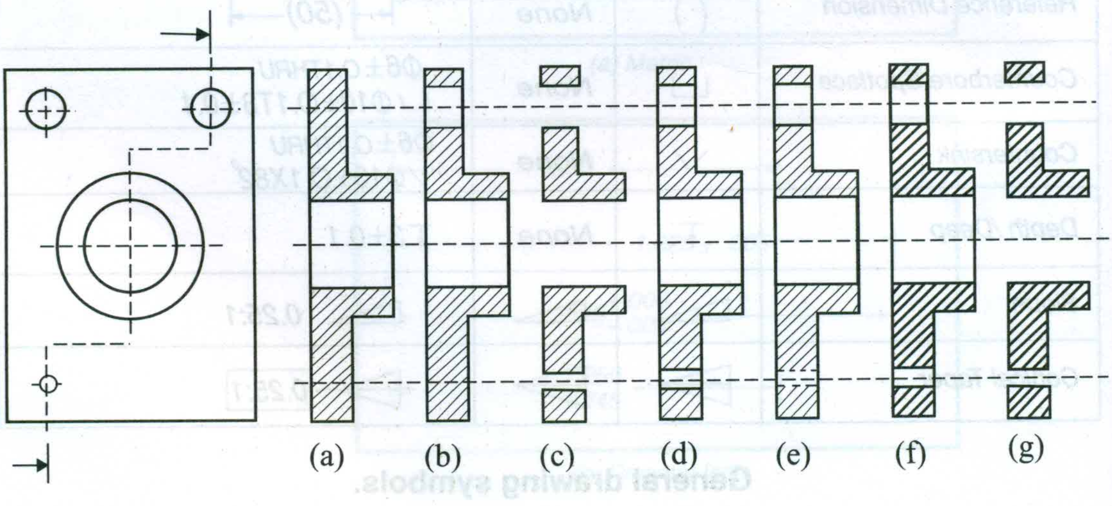
A section is a view of no thickness and shows the outline of the object at the cutting plane. Visible outlines beyond the cutting plane are not drawn.

A sectional view, displays the outline of the cutting plane and all visible outlines which can be seen beyond the cutting plane. The diagram below shows a sectional view, and how a cutting plane works.



4.12 Which Sectional View is Correct?[37]

Consider the following diagrams and find the correct sectional view.



4.13 Have You Found It?

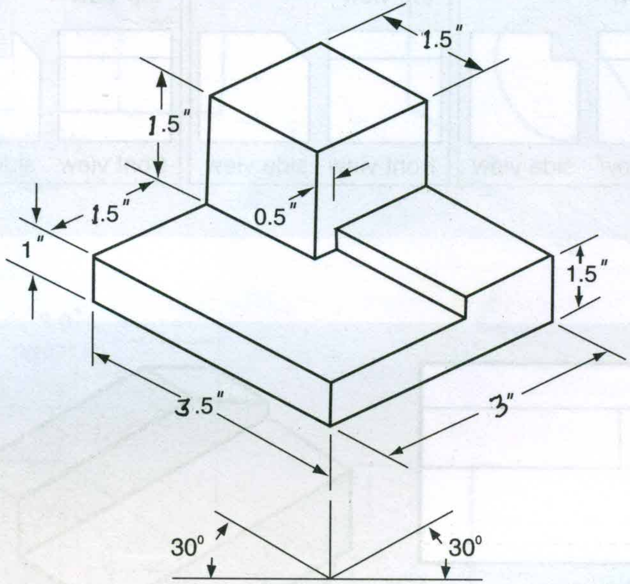
The answer may be tricky, but I wish, you try yourself as I tried myself.

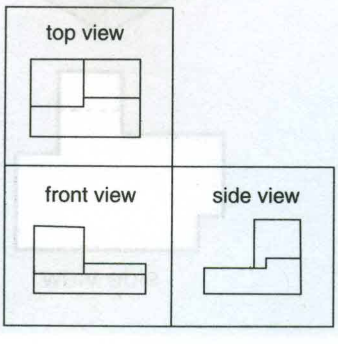
Well, may be you have the right answer. There are two figures near the answer. One in the middle (fourth from either side) and the other is the second one from right. However, the correct answer is the one in the middle not the second one from the right, since the hatching is darker. By convention, the hatching should be drawn with thin lines, lighter than the outer borders. Therefore, the one in the middle is the best choice.

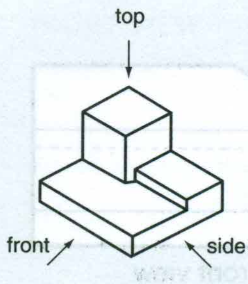
Exercises

Exercise 1:

Draw the figure with dimension. Then draw the figure with top, front and side views yourself and then check with the following. Do not cheat!

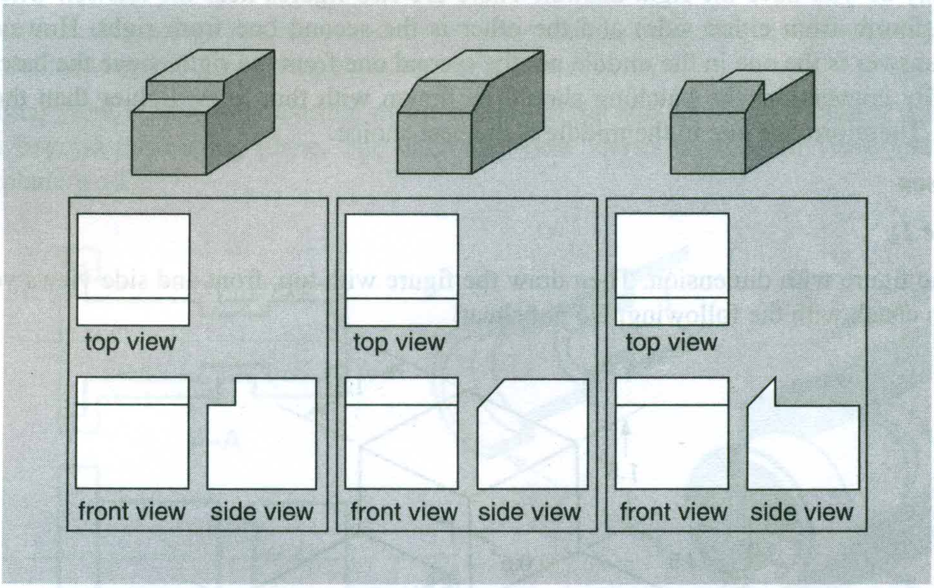


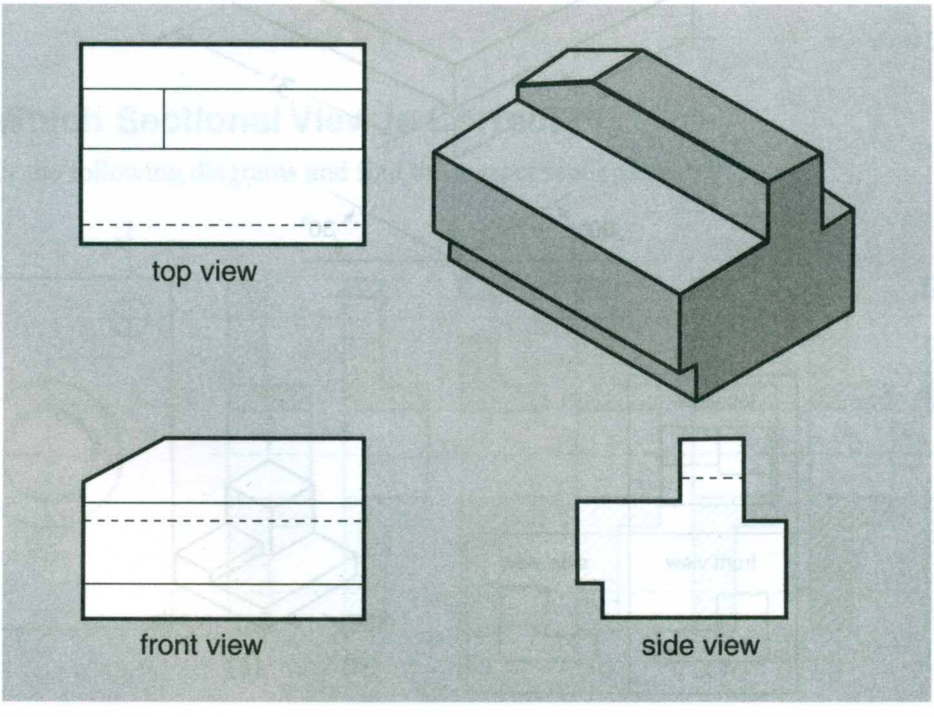




Exercise 2:

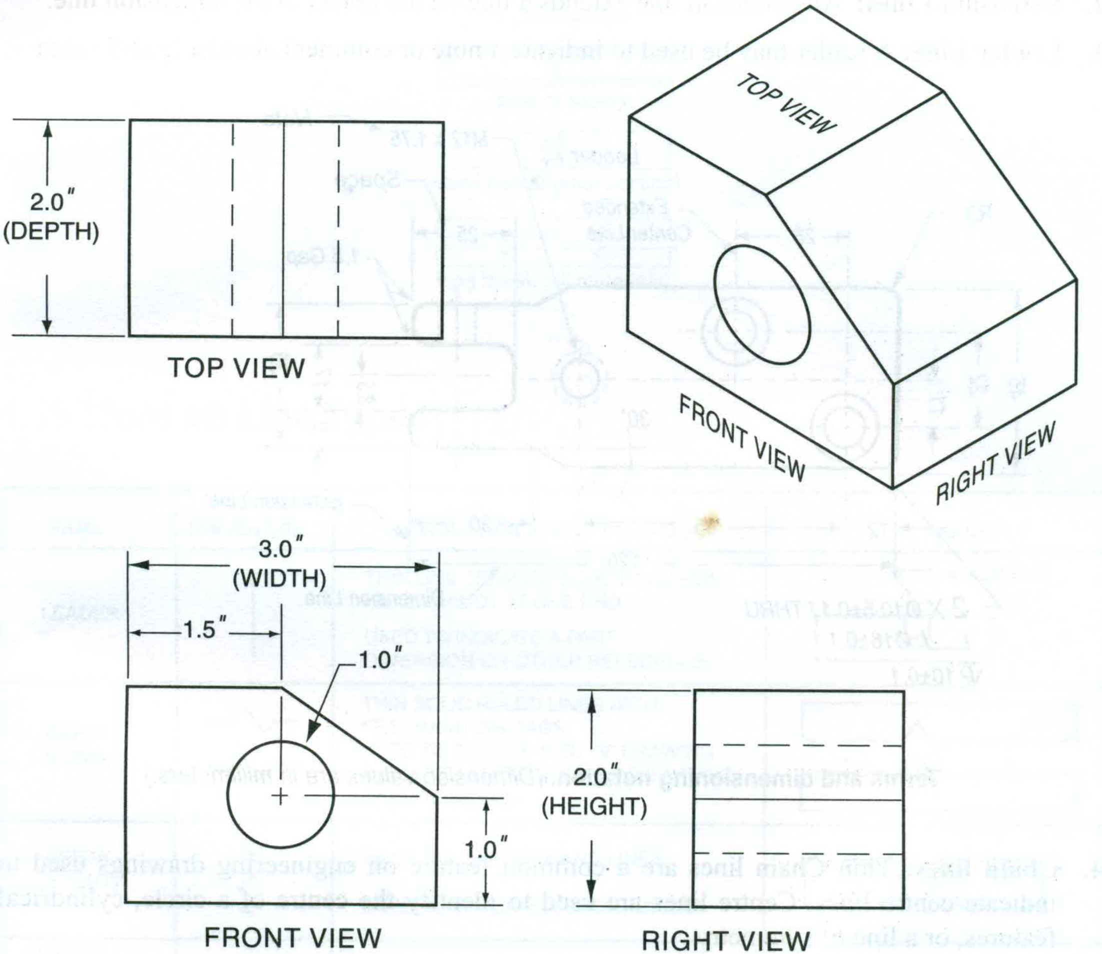
Now look at the examples below





Exercise 3 :

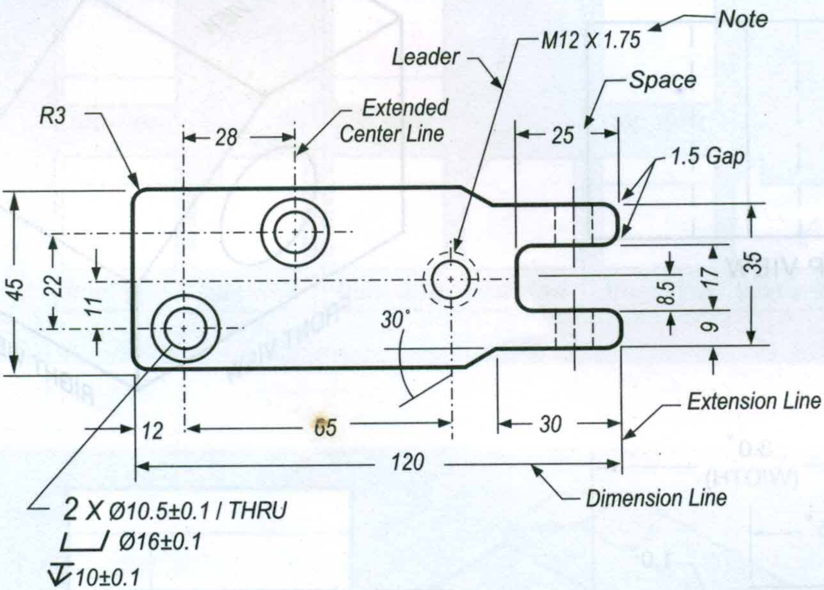
Courtesy of Art Whitton, NV, USA



4.14 Various Line Types

The following typical line types are described with the information available over the internet [38,39, 40, 41, 42]

1. **Dimension Line:** Line thickness is thin. It shows the dimension. It is sometimes broken in the middle to allow the placement of the dimension value, with arrowheads at each end.
2. **Extension Lines:** An extension line extends a line on the object to the dimension line.
3. **Leader Line:** A leader may be used to indicate a note or comment about a specific area



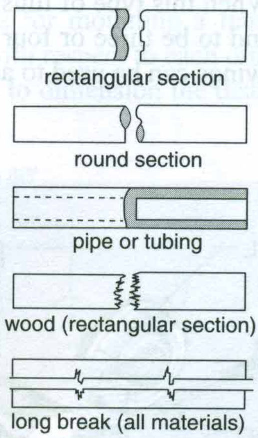
Terms and dimensioning notation. *(Dimension values are in millimeters.)*

4. **Chain lines:** Thin Chain lines are a common feature on engineering drawings used to indicate centre lines. Centre lines are used to identify the centre of a circle, cylindrical features, or a line of symmetry.

5. **Dashed lines:** Dashed lines are used to show important hidden detail for example wall thickness and holes.

4.15 Conventional Breaks

Conventional breaks and other symbols to indicate details [43]

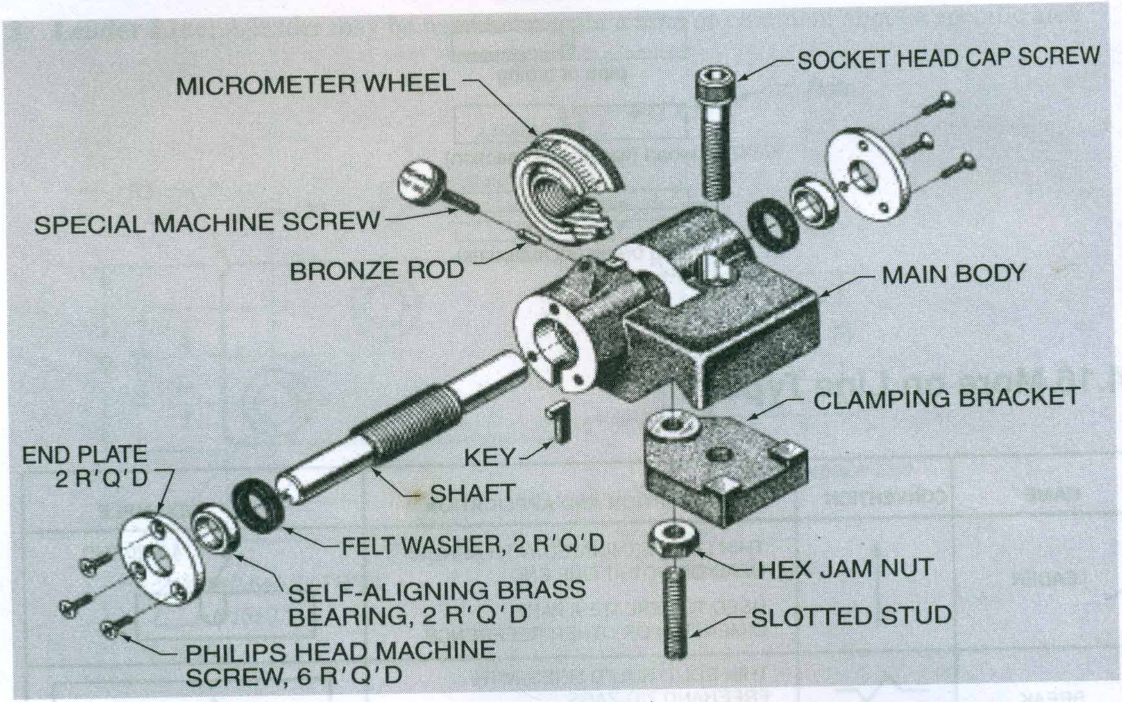


4.16 More on Line Types

NAME	CONVENTION	DESCRIPTION AND APPLICATION	EXAMPLE
LEADER		THIN LINE TERMINATED WITH ARROW HEAD OR DOT AT ONE END USED TO INDICATE A PART, DIMENSION OR OTHER REFERENCE	4 X 20 THD.
BREAK (LONG)		THIN SOLID RULED LINES WITH FREEHAND ZIG-ZAGS USED TO REDUCE SIZE OF DRAWING REQUIRED TO DELINEATE OBJECT AND REDUCE DETAIL.	
BREAK (SHORT)		THICK SOLID FREE HAND LINES USED TO INDICATE A SHORT BREAK	
CUTTING OR VIEWING PLANE, VIEWING PLANE □ OPTIONAL		THICK SOLID LINES WITH ARROWHEAD TO INDICATE DIRECTION IN WHICH SECTION OR PLANE IS VIEWED OR TAKEN	
CUTTING PLANE FOR COMPLEX OR OFFSET VIEWS		THICK SHORT DASHES USED TO SHOW OFFSET WITH ARROW HEADS TO SHOW DIRECTION VIEWED	

4.17 Exploded-View

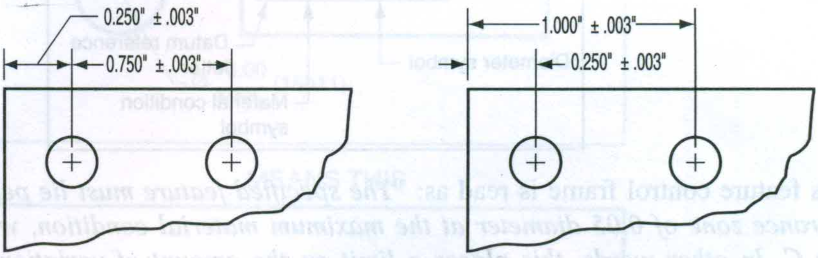
Shaded exploded-view production illustrations greatly facilitate the learning process in assembly of machines and devices. When this type of illustration is used, the initial assembly of parts into a machine has been found to be three or four times faster than if a conventional assembly drawing is used. Photo drawings can be used to achieve the same visual results[43].



4.18 A Dimensioning Example

Showing that placement should match intent

These drawings show bolts holes for mounting a flange onto a plate. When mounting the flange, the position of the holes with respect to each other is very important, or else the flange (or part) won't fit. It makes sense to dimension the distance between the holes, instead of the distances to the edge.[52]



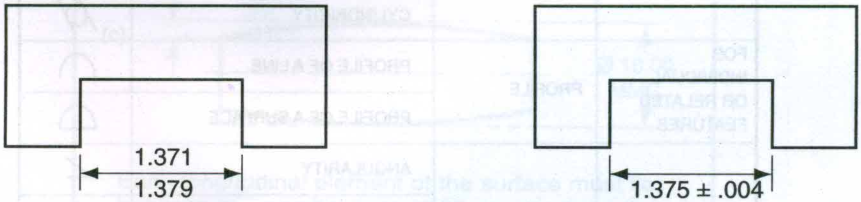
Dimension placement matches intent | Dimension placement does NOT match intent

4.19 Tolerances

There are two types of tolerances:[52]

a. Geometric Tolerances

b. Dimensional Tolerances — Limit Dimensioning / Plus & Minus Tolerancing



Limit Dimensioning | Plus & Minus Tolerancing

Geometric Tolerancing

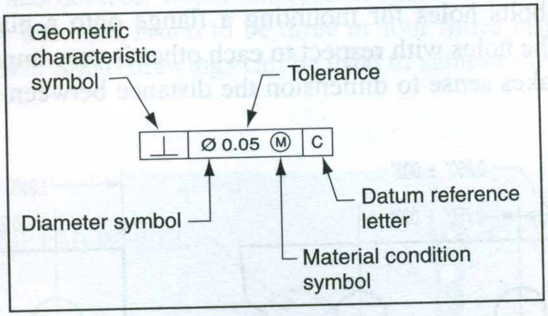
Geometric Tolerancing is used to specify the shape of features. Things like:

a. Straightness b. Flatness c. Circularity d. Cylindricity
e. Angularity f. Profiles g. Perpendicularity h. Parallelism
i. Concentricity And More...

Geometric Tolerances are shown on a drawing with a feature control frame.

4.20 The Feature Control Frame

A feature control frame gives information about geometric tolerances on the feature.



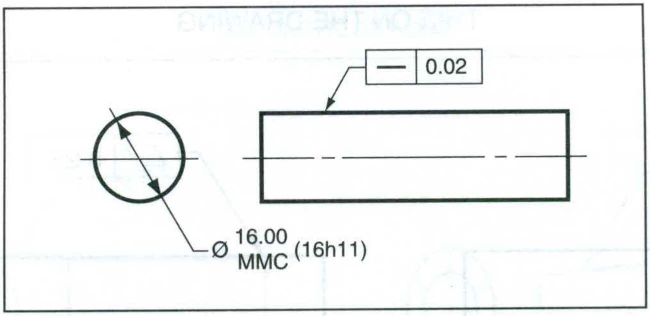
This feature control frame is read as: *"The specified feature must lie perpendicular within a tolerance zone of 0.05 diameter at the maximum material condition, with respect to datum axis C. In other words, this places a limit on the amount of variation in perpendicularity between the feature axis and the datum axis. In a drawing, this feature control frame would accompany dimensional tolerances that control the feature size and position.*

Geometric Characteristic Symbols

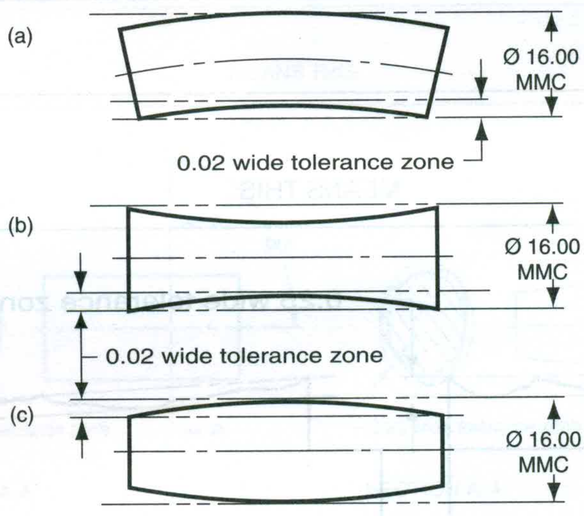
	TYPE OF TOLERANCE	CHARACTERISTIC	SYMBOL
FOR INDIVIDUAL FEATURES	FORM	STRAIGHTNESS	⏤
		FLATNESS	⏥
		CIRCULARITY (ROUNDNESS)	○
		CYLINDRICITY	⌭
FOR INDIVIDUAL OR RELATED FEATURES	PROFILE	PROFILE OF A LINE	⌒
		PROFILE OF A SURFACE	⌓
FOR RELATED FEATURES	ORIENTATION	ANGULARITY	∠
		PERPENDICULARITY	⊥
		PARALLELISM	//
	LOCATION	POSITION	⌖
		CONCENTRICITY	◎
		SYMMETRY	⌯
	RUNOUT	CIRCULAR RUNOUT	↗
		TOTAL RUNOUT	⌰

Straightness Example

THIS ON THE DRAWING



MEANS THIS

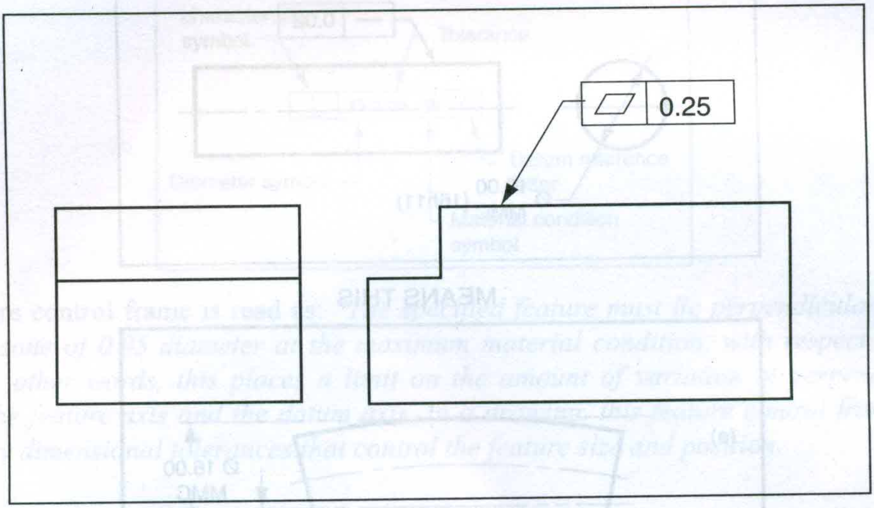


Each longitudinal element of the surface must lie between two parallel lines (0.02 apart) where the two lines and the nominal axis of the part share a common plane. The feature must be within the specified limits of size and the boundary of perfect form at MMC (16.00)

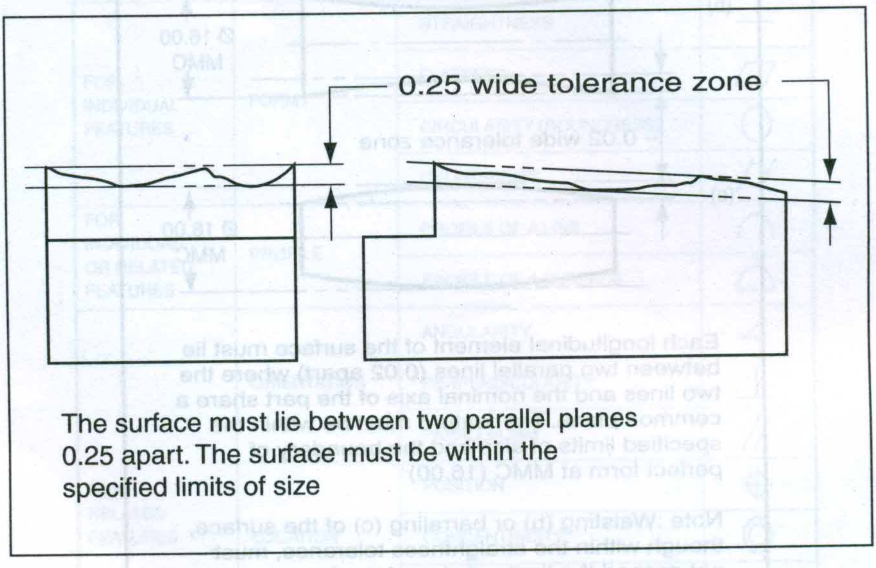
Note :Waisting (b) or barreling (c) of the surface, though within the straightness tolerance, must not exceed the limits of size of the feature.

Flatness Examples

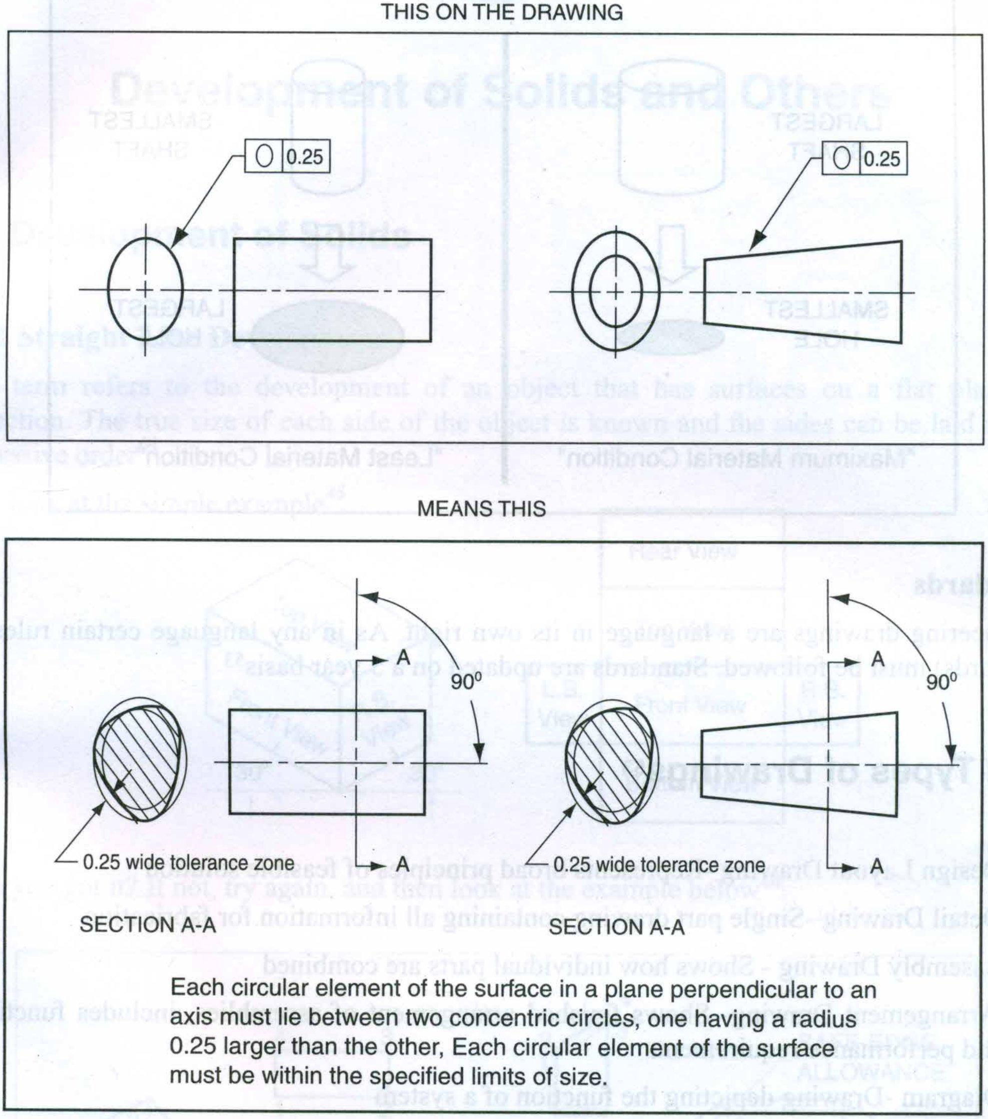
THIS ON THE DRAWING



MEANS THIS

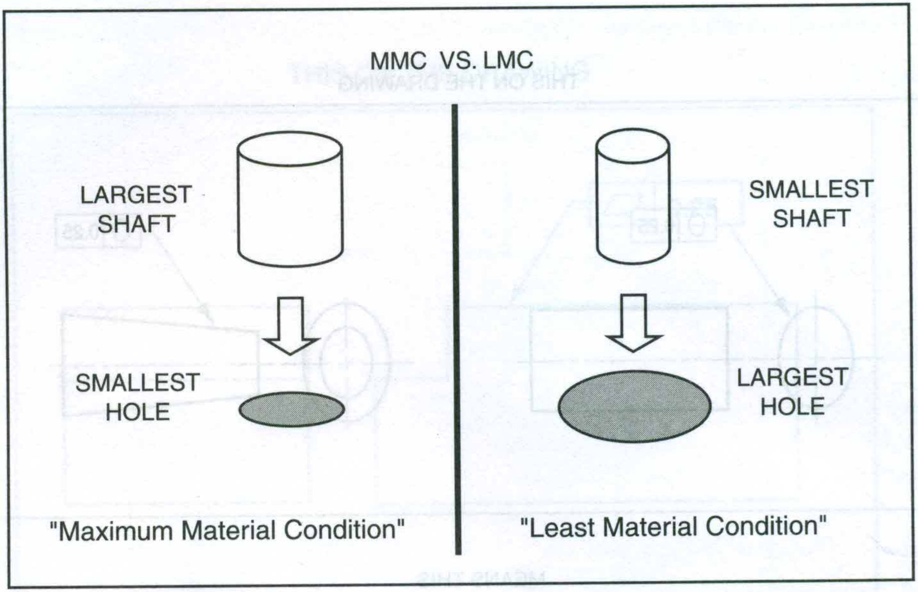


Circularity (Roundness) Example



Circular & Total Runout

Runout is specified on cylindrical parts. It is measured by placing a gage on the part, and rotating the part through 360 degrees. The total variation is recorded as the runout. Circular runout is measured at one location. Total Runout is measured along the entire specified surface.



Standards

Engineering drawings are a language in its own right. As in any language certain rules (or standards) must be followed. Standards are updated on a 5 year basis[53]

4.21 Types of Drawings[53]

a. Design Layout Drawing -Represents broad principles of feasible solution

b. Detail Drawing -Single part drawing containing all information for fabrication

c. Assembly Drawing - Shows how individual parts are combined

d. Arrangement Drawing- Shows finished arrangement of assemblies, includes functional and performance requirements

e. Diagram -Drawing depicting the function of a system