Enable | Enlighten | Enrich

KARPAGAM
ACADEMY OF HIGHER EDUCATION
(Deemed to be University)
(Established Under Section 3 of UGC Act, 1956)

COMPUTER AIDED DRAFTING AND COST ESTIMATION

15BEME8E02

KARPAGAM ACADEMY OF HIGHER EDUCATION

Faculty of Engineering

Department of Mechanical Engineering

SYLLABUS

15BEME8E02 - COMPUTER AIDED DRAFTING AND COST ESTIMATION

UNIT I STANDARDS AND CONVENTIONS

Current international standards (ISO) and Indian Standards (IS)- types of lines - principles of presentation - dimensioning - conventional representation of threaded parts, springs, and gears.

UNIT II DIMENSIONAL AND FORM TOLERANCES

Limits and fits IT system of tolerances, deviation of fit - geometric tolerance-tolerancing of form, orientation, location and runout - datums and Datum systems-Dimensioning and tolerancing of profiles

UNIT III MANUFACTURING DRAWINGS

Surface texture indication on drawing - welds symbolic representation of drawings. Given a sub-assembly/assembly to prepare manufacturing drawings of components, Sample exercises on CAD- preparation of manufacturing Drawings.

UNIT IV RE-DIMENSIONING AND TOLERANCE CHARTING

Introduction to re-dimensioning to suit manufacturing requirements-manufacturing datum-functional datum. Introduction to tolerance charting

UNIT V COST ESTIMATION

Preparation of Process chart for a given component-estimation of setting time and machining time-estimation of material cost, labour cost and overhead cost based on supplied data.

TEXT Books and References

1. Siddeshwar and Kanniah, Machine Drawing, 1st edition, Tata McGraw Hill, 2001
2. Ajeet Singh, Machine Drawing, 2nd edition, Tata McGraw Hill, 2012.
3. Gopalakrishna, K.R, Machine Drawing, Subhas Stores, 2002
4. Wade. O, Tolerance Control in design and manufacturing, Industrial Press, 1972

R. Kesavan, C. Elanchezhian, B. Vijaya Ramanath, Process Planning and Cost Estimation, 1st edition, New Age International, 2009

LESSON PLAN

Subject Name : **Computer Aided Drafting and Cost Estimation**
Subject Code : **15BEME8E02** **(Credits - 03)**
Name of the Faculty : **S. Aravind**
Designation : **Assistant Professor**
Year/Semester/Section : **IV/VIII/A**
Branch : **Mechanical Engineering**

Sl. No.	No. of Periods	Topics to be Covered	Support Materials
UNIT – I : STANDARDS AND CONVENTIONS			
1.	1	Current international standards (ISO)	W [1]
2.	1	Current Indian Standards (IS)	W [2]
3.	1	Types of lines as per BIS	T[1] – pp. 44 - 48
4.	1	Principles of presentation and Types of Projections	T[1] – pp. 86 - 131
5.	1	Principles of dimensioning	T[1] – pp. 177 - 195
6.	1	**Tutorial 1** (Sketching Assignment).	T[1] – pp. 125 - 131
7.	1	Conventional representation of threaded parts	T[1] – pp. 270 - 295
8.	1	Conventional representation of springs	T[1] – pp. 313 - 315
9.	1	Conventional representation of gears	T[1] – pp. 730 - 745
10.	1	**Tutorial 2** (Representation Assignment).	T[1] – pp. 294 - 298
11.	1	**Discussion on Competitive Examination related Questions / University previous year questions**	GATE Material ESE Question Bank
Total No. of Hours Planned for Unit - I			**11**

Sl. No.	No. of Periods	Topics to be Covered	Support Materials
UNIT – II : DIMENSIONAL AND FORM TOLERANCES			
12.	1	Limits, fits and Tolerances	T[1] – pp. 195 - 205
13.	1	IT system of tolerances	T[1] – pp. 195 - 205
14.	1	Deviation of fit	T[1] – pp. 195 - 205
15.	1	**Tutorial 3** (Problems from Limits, fits and Tolerances).	T[1] – pp. 217 - 234
16.	1	Geometric tolerance	T[1] – pp. 510 - 553

Sl. No.	No. of Periods	Topics to be Covered	Support Materials
17.	1	Tolerancing of form and runout	T[1] – pp. 522 - 527
18.	1	Tolerancing of orientation	T[1] – pp. 535 – 548
19.	1	Tolerancing of location	T[1] – pp. 549 - 558
20.	1	Datums and Datum systems	T[1] – pp. 529 - 532
21.	1	Dimensioning and tolerancing of profiles	T[1] – pp. 510 - 553
22.	1	**Tutorial 4** (Problems from Geometric tolerance)	T[1] – pp. 600 - 627
23.	1	**Discussion on Competitive Examination related Questions / University previous year questions**	GATE Material ESE Question Bank
Total No. of Hours Planned for Unit - II			**12**

Sl. No.	No. of Periods	Topics to be Covered	Support Materials
UNIT – III : MANUFACTURING DRAWINGS			
24.	1	Surface texture representation	T[1] – pp. 208 - 216
25.	1	Surface texture indication on drawing	T[1] – pp. 211 - 216
26.	1	Welds symbolic representation of drawings	T[1] – pp. 323 -325
27.	1	Welds symbolic indication on drawing	T[1] – pp. 323 -325
28.	1	**Tutorial 5** (Preparation of Working Drawing).	T[1] – pp. 233 -235
29.	1	Prepare manufacturing drawings of components	T[1] – pp. 398 - 416
30.	1	Preparation of manufacturing Drawings	T[1] – pp. 398 - 416
31.	1	**Tutorial 6** (Sample exercises on CAD)	T[1] – pp. 400 - 416
32.	1	**Discussion on Competitive Examination related Questions / University previous year questions**	GATE Material ESE Question Bank
Total No. of Hours Planned for Unit - III			**9**

Sl. No.	No. of Periods	Topics to be Covered	Support Materials
UNIT – IV : RE-DIMENSIONING AND TOLERANCE CHARTING			
33.	1	Design features to facilitate machining and Datum features - Functional and Manufacturing	T[2]- pp. 24-35 T[2]- pp. 37-55
34.	1	Solving problems from Functional and Manufacturing datum features.	R[1]- pp. 219-255

Sl. No.	No. of Periods	Topics to be Covered	Support Materials
35.	1	Component design-machining considerations and Redesign for manufacture with examples.	T[2]- pp. 85-108
36.	1	**Tutorial 7** (problems from Functional and Manufacturing datum features)	T[2]- pp. 85-114
37.	1	Tolerance Charting Technique: Operation sequence for typical shaft type of components and Preparation of process drawings for different operations.	R[2]- pp. 12-16
38.	1	Tolerance worksheets and centrality analysis with examples. Tolerance charting technique procedure.	R[2]- pp. 17-22 R[3]- pp. 294-296
39.	1	**Tutorial 8** (Problems on Tolerance charting technique)	R[2]- pp. 17-28
40.	1	**Discussion on Competitive Examination related Questions / University previous year questions**	GATE Material ESE Question Bank
Total No. of Hours Planned for Unit - IV			**8**

Sl. No.	No. of Periods	Topics to be Covered	Support Materials
UNIT – V : COST ESTIMATION			
41.	1	Introduction to Process Chart	T[3] – pp. 12 - 16
42.	1	Preparation of Process chart for a given component	T[3] – pp. 20 - 40
43.	1	**Tutorial 9** (Preparation of Process chart)	T[3] – pp. 12 - 40
44.	1	Estimation of setting time and machining time	T[3] – pp. 95 - 101
45.	1	Estimation of material cost	T[3] – pp. 95 - 101
46.	1	Estimation of labor cost and overhead cost	T[3] – pp. 114 - 118
47.	1	**Tutorial 10** (Problems from Cost Estimation)	T[3] – pp. 120 - 125
48.	1	**Discussion on Competitive Examination related Questions / University previous year questions**	GATE Material ESE Question Bank
Total No. of Hours Planned for Unit - V			**8**

TOTAL PERIODS : 48

TEXT BOOKS

T [1] – Cencil Jensen, 2012, Engineering Drawing & Design, Tata McGraw Hill, New Delhi.
T [2] – Harry Peck, 1983, Designing for Manufacture, Pitman Publications, London.
T [3] – R. Kesavan, 2004, Process, Planning And Cost Estimation, New Age International.

REFERENCES

R [1] - Geoffrey Boothroyd, June 2005, Assembly Automation and product Design (Second Edition) – Taylor & Francis
R [2] - Spotts M F, 1983, Dimensioning and Tolerance for Quantity Production, Prentice Hall Inc., New Jersey, USA.
R [3] - Oliver R Wade, 1967, Tolerance Control in Design and Manufacturing, Industrial Press Inc., New York.
R [4] - Study materials on Design for Manufacturing and Assembly – PSG Tech

WEBSITES

W [1] - https://www.iso.org/
W [2] - http://www.bis.gov.in/
W [3] - http://nptel.ac.in/courses/112101005/

JOURNALS

J [1] - Bogue, R., 2012. Design for manufacture and assembly: background, capabilities and applications. Assembly Automation, 32(2), pp.112-118.

J [2] – Ijomah, W.L., McMahon, C.A., Hammond, G.P. and Newman, S.T., 2007. Development of design for remanufacturing guidelines to support sustainable manufacturing. Robotics and Computer-Integrated Manufacturing, 23(6), pp.712-719.

UNIT	Total No. of Periods Planned	Lecture Periods	Tutorial Periods
I	11	09	02
II	12	10	02
III	09	07	02
IV	08	06	02
V	08	06	02
TOTAL	48	38	10

I. CONTINUOUS INTERNAL ASSESSMENT : 40 Marks

(Internal Assessment Tests: 30, Attendance: 5, Assignment/Seminar: 5)

II. END SEMESTER EXAMINATION : 60 Marks

TOTAL : 100 Marks

UNIT I

STANDARDS AND CONVENTIONS

Current international standards (ISO) and Indian Standards (IS)- types of lines - principles of presentation - dimensioning - conventional representation of threaded parts, springs, and gears.

INTRODUCTION:

ISO 128 is an international standard (ISO), about the general principles of presentation in technical drawings, specifically the graphical representation of objects on technical drawings.

Since 2003 the ISO 128 standard contains twelve parts, which were initiated between 1996 and 2003. It starts with a summary of the general rules for the execution and structure of technical drawings. Further it describes basic conventions for lines, views, cuts and sections, and different types of engineering drawings, such as those for mechanical engineering, architecture, civil engineering, and shipbuilding. It is applicable to both manual and computer-based drawings, but it is not applicable to three-dimensional CAD models.

The ISO 128 replaced the previous DIN 6 standard for drawings, projections and views, which was first published in 1922 and updated in 1950 and 1968. ISO 128 itself was first published in 1982, contained 15 pages and "specified the general principles of presentation to be applied to technical drawings following the orthographic projection methods".[2] Several parts of this standard have been updated individually. The last parts and the standard as a whole were withdrawn by the ISO in 2001.

A thirteenth part was added in 2013.

THE 15 PARTS OF THE ISO 128 STANDARD ARE:

- ISO 128-1:2003 Technical drawings—General principles of presentation—Part 1: Introduction and index
- ISO 128-15:2013 Technical product documentation (TPD)—General principles of presentation—Part 15: Presentation of shipbuilding drawings
- ISO 128-20:1996 Technical drawings—General principles of presentation—Part 20: Basic conventions for lines
- ISO 128-21:1997 Technical drawings—General principles of presentation—Part 21: Preparation of lines by CAD systems

- ISO 128-22:1999 Technical drawings—General principles of presentation—Part 22: Basic conventions and applications for leader lines and reference lines
- ISO 128-23:1999 Technical drawings—General principles of presentation—Part 23: Lines on construction drawings
- ISO 128-24:2014 Technical drawings—General principles of presentation—Part 24: Lines on mechanical engineering drawings
- ISO 128-25:1999 Technical drawings—General principles of presentation—Part 25: Lines on shipbuilding drawings
- ISO 128-30:2001 Technical drawings—General principles of presentation—Part 30: Basic conventions for views
- ISO 128-34:2001 Technical drawings—General principles of presentation—Part 34: Views on mechanical engineering drawings
- ISO 128-40:2001 Technical drawings—General principles of presentation—Part 40: Basic conventions for cuts and sections
- ISO 128-43:2015 Technical product documentation (TPD)—General principles of presentation—Part 43: Projection methods in building drawings
- ISO 128-44:2001 Technical drawings—General principles of presentation—Part 44: Sections on mechanical engineering drawings
- ISO 128-50:2001 Technical drawings—General principles of presentation—Part 50: Basic conventions for representing areas on cuts and sections
- ISO/TS 128-71:2010 Technical product documentation (TPD)—General principles of presentation—Part 71: Simplified representation for mechanical engineering drawings

OTHER ISO STANDARDS RELATED TO TECHNICAL DRAWING

- ISO 129 Technical drawings—Indication of dimensions and tolerances
- ISO 216 paper sizes, e.g. the A4 paper size
- ISO 406:1987 Technical drawings—Tolerancing of linear and angular dimensions
- ISO 1660:1987 Technical drawings—Dimensioning and tolerancing of profiles
- ISO 2203:1973 Technical drawings—Conventional representation of gears
- ISO 3040:1990 Technical drawings—Dimensioning and tolerancing -- Cones

- ISO 3098/1:1974 Technical Drawing - Lettering - Part I: Currently Used Characters
- ISO 4172:1991 Technical drawings -- Construction drawings—Drawings for the assembly of prefabricated structures
- ISO 5261:1995 Technical drawings—Simplified representation of bars and profile sections
- ISO 5455:1979 Technical drawings—Scales
- ISO 5456 Technical drawings -- Projection methods
- ISO 5456-1:1996 Technical drawings—Projection methods—Part 1: Synopsis
- ISO 5456-2:1996 Technical drawings—Projection methods—Part 2: Orthographic representations
- ISO 5456-3:1996 Technical drawings—Projection methods—Part 3: Axonometric representations
- ISO 5456-4:1996 Technical drawings—Projection methods—Part 4: Central projection
- ISO 5457:1999 Technical product documentation—Sizes and layout of drawing sheets
- ISO 5459:1981 Technical drawings -- Geometrical tolerancing—Datums and datum-systems for geometrical tolerances
- ISO 5845-1:1995 Technical drawings—Simplified representation of the assembly of parts with fasteners—Part 1: General principles
- ISO 6410-1:1993 Technical drawings—Screw threads and threaded parts—Part 1: General conventions
- ISO 6411:1982 Technical drawings—Simplified representation of centre holes
- ISO 6412-1:1989 Technical drawings—Simplified representation of pipelines—Part 1: General rules and orthogonal representation
- ISO 6413:1988 Technical drawings—Representation of splines and serrations
- ISO 6414:1982 Technical drawings for glassware
- ISO 6428:1982 Technical drawings—Requirements for microcopying
- ISO 6433:1981 Technical drawings -- Item references
- ISO 7200:1984 Technical drawings — Title blocks
- ISO 7083:1983 Technical drawings—Symbols for geometrical tolerancing—Proportions and dimensions
- ISO 7437:1990 Technical drawings -- Construction drawings—General rules for execution of production drawings for prefabricated structural components
- ISO 7518:1983 Technical drawings -- Construction drawings—Simplified representation of demolition and rebuilding

- ISO 7519:1991 Technical drawings -- Construction drawings—General principles of presentation for general arrangement and assembly drawings
- ISO 8015:1985 Technical drawings—Fundamental tolerancing principle
- ISO 8048:1984 Technical drawings -- Construction drawings—Representation of views, sections and cuts
- ISO 8560:1986 Technical drawings -- Construction drawings—Representation of modular sizes, lines and grids
- ISO 8560:1986 Technical drawings—Construction drawings—Representation of modular sizes, lines and grids
- ISO 8826-1:1989 Technical drawings—Rolling bearings—Part 1: General simplified representation
- ISO 8826-2:1994 Technical drawings—Rolling bearings—Part 2: Detailed simplified representation
- ISO 9222-1:1989 Technical drawings—Seals for dynamic application—Part 1: General simplified representation
- ISO 9222-2:1989 Technical drawings—Seals for dynamic application—Part 2: Detailed simplified representation
- ISO 9958-1:1992 Draughting media for technical drawings—Draughting film with polyester base—Part 1: Requirements and marking
- ISO 9961:1992 Draughting media for technical drawings—Natural tracing paper
- ISO 10209-1:1992 Technical product documentation—Vocabulary—Part 1: Terms relating to technical drawings: general and types of drawings
- ISO 10578:1992 Technical drawings—Tolerancing of orientation and location—Projected tolerance zone
- ISO 10579:1993 Technical drawings—Dimensioning and tolerancing—Non-rigid parts
- ISO 13567 is an international Computer-aided design (CAD) layer standard.
- ISO 13715:2000 Technical drawings—Edges of undefined shape—Vocabulary and indications
- ISO 15786:2008 Technical drawings—Simplified representation and dimensioning of holes

TYPES OF LINES

Lines is one important aspect of technical drawing. Lines are always used to construct meaningful drawings. Various types of lines are used to construct drawing, each line used in some specific sense. Lines are drawn following standard conventions mentioned in BIS (SP46:2003). A line may be curved, straight, continuous, segmented. It may be drawn as thin or thick. A few basic types of lines widely used in drawings are shown in Table 1.

Table 1. Types of letters used in engineering drawing.

Illustration	Application
Thick	Outlines, visible edges, surface boundaries of objects, margin lines
Continuous thin	Dimension lines, extension lines, section lines leader or pointer lines, construction lines, boarder lines
Continuous thin wavy	Short break lines or irregular boundary lines – drawn freehand
Continuous thin with zig-zag	Long break lines
Short dashes, gap 1, length 3 mm	Invisible or interior surfaces
Short dashes	Center lines, locus lines Alternate long and short dashes in a proportion of 6:1,
Long chain thick at end and thin elsewhere	Cutting plane lines

Line Strokes

Line strokes refer to the directions of drawing straight and curved lines. The standards for lines is given in BIS : SP-46, 2003 Vertical and inclined lines are drawn from top to bottom, horizontal lines are drawn from left to right. Curved lines are drawn from left to right or top to bottom. The direction of strokes are illustrated in figure 1.

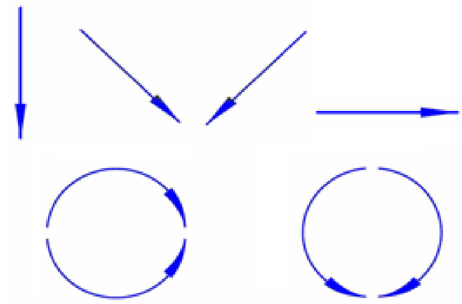


Figure 1. The line strokes for drawing straight and curved lines.

Conventions used in lines

- International systems of units (SI) – which is based on the meter.
- Millimeter (mm) - The common SI unit of measure on engineering drawing.
- Individual identification of linear units is not required if all dimensions on a drawing are in the same unit (mm).
- The drawing should contain a note: ALL DIMENSIONS ARE IN MM. (Bottom left corner outside the title box)

Typical figures showing various lines used in the construction of engineering drawing is shown in figure 2.

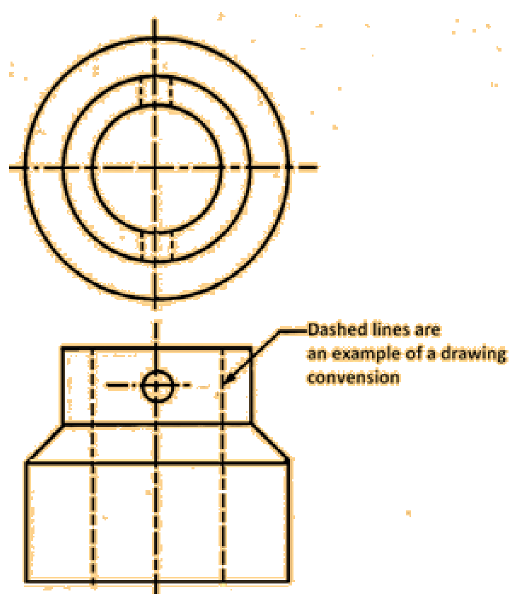
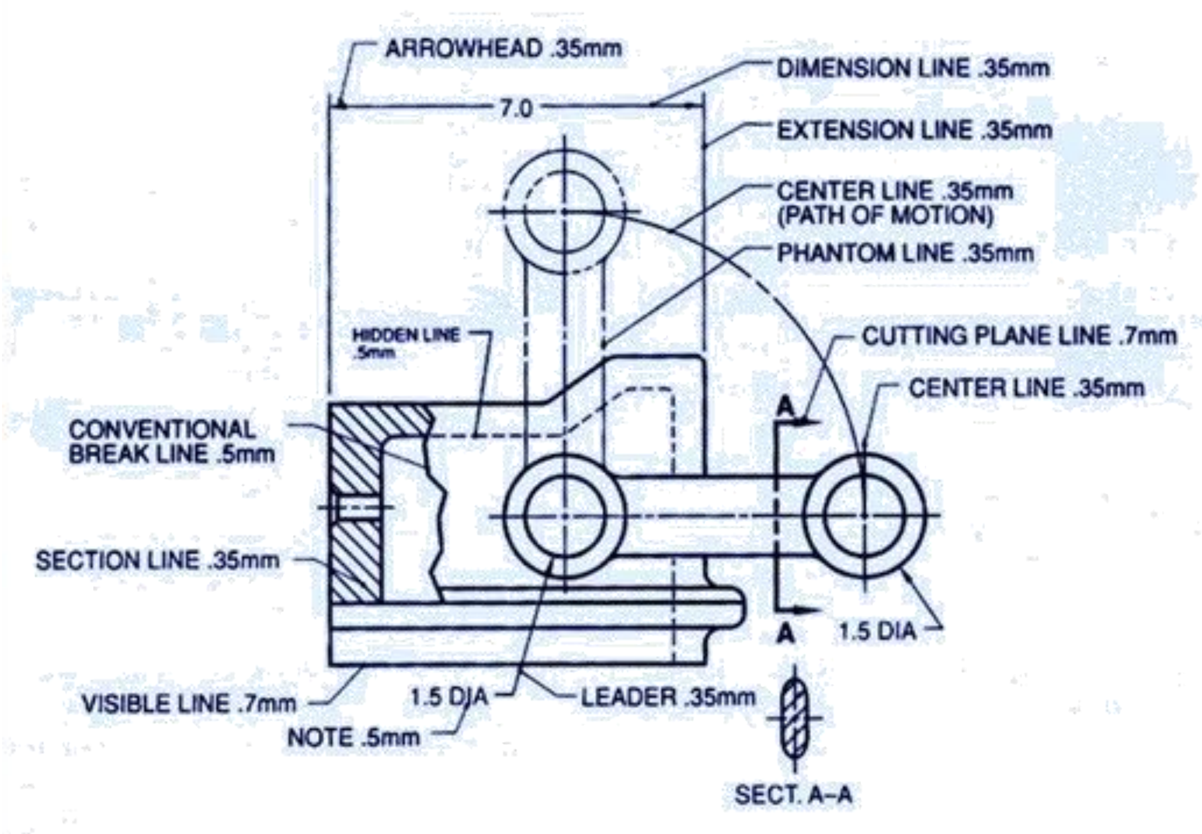


Figure 2 Typical figure showing various lines used engineering drawing

A typical use of various lines in an engineering drawing is shown in figure below:



Dimensioning

The size and other details of the object essential for its construction and function, using lines, numerals, symbols, notes, etc are required to be indicated in a drawing by proper dimensioning. These dimensions indicated should be those that are essential for the production, inspection and functioning of the object and should be mistaken as those that are required to make the drawing of an object. The dimensions are written either above the dimension lines or inserted at the middle by breaking the dimension lines.

Normally two types of dimensioning system exist. i.e. Aligned system and the unidirectional system.These are shown in figure 3.

In the aligned system the dimensions are placed perpendicular to the dimension line in such a way that it may be read from bottom edge or right hand edge of the drawing sheet. The horizontal and inclined dimension can be read from the bottom where as all the vertical dimensions can be read from the right hand side of the drawing sheet.

In the unidirectional system, the dimensions are so oriented such that they can be read from the bottom of the drawing.

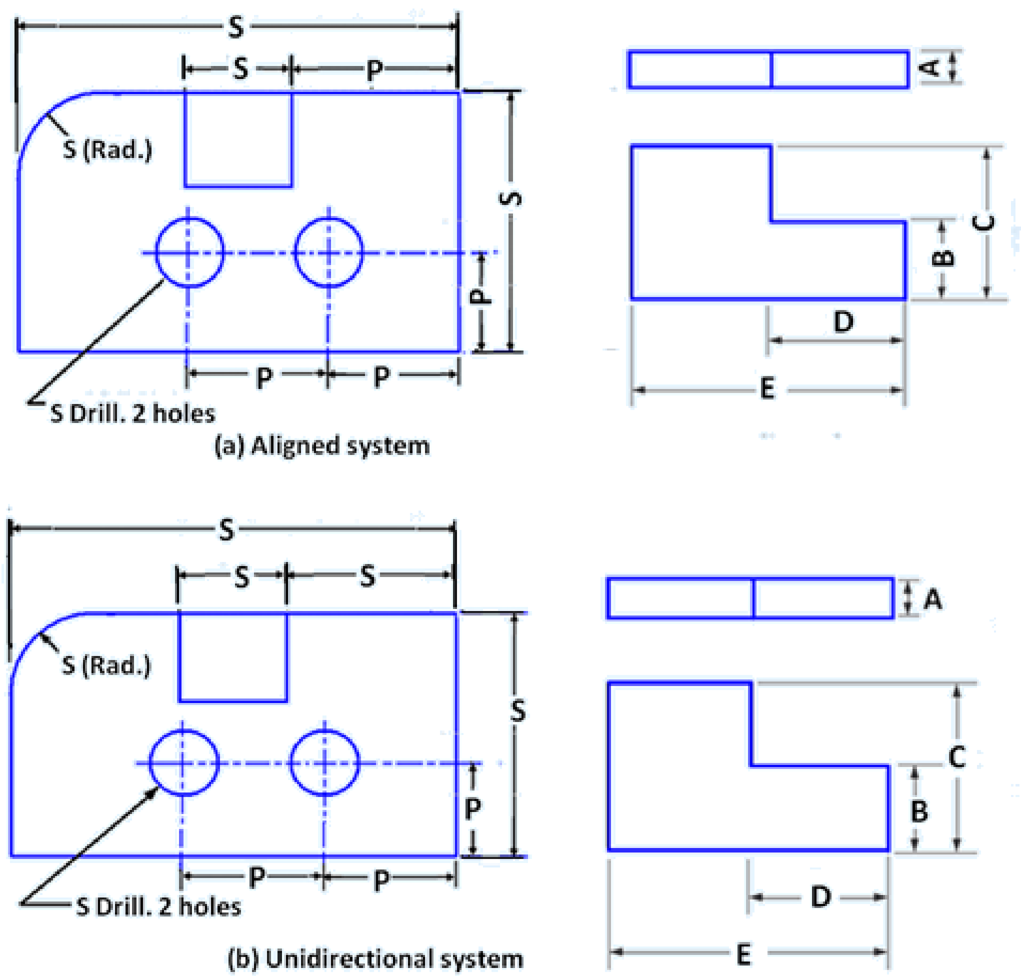


Figure 3. The aligned system and unidirectional system of dimensioning.

Rules to be followed for dimensioning. Refer figure 4.

- Each feature is dimensioned and positioned only once.
- Each feature is dimensioned and positioned where its shape shows.
- Size dimensions – give the size of the component.

- Every solid has three dimensions, each of the geometric shapes making up the object must have its height, width, and depth indicated in the dimensioning.

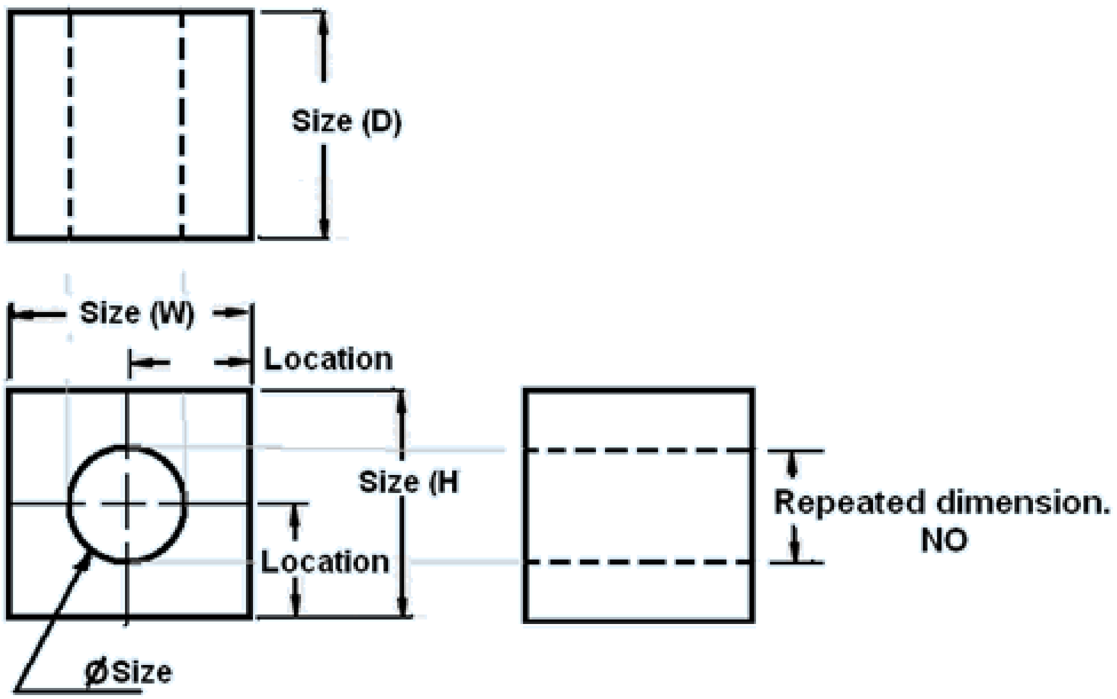


Figure 4. typical dimension lines

Dimensioning consists of the following:

- A thin, solid line that shows the extent and direction of a dimension. Dimension lines are broken for insertion of the dimension numbers
- Should be placed at least 10 mm away from the outline and all
- other parallel dimensions should be at least 6 mm apart, or more, if space permits

The important elements of dimensioning consists of extension lines, leader line, arrows and dimensions.

Extension line – a thin, solid line perpendicular to a dimension line, indicating which feature is associated with the dimension. There should be a visible gap of 1.5 mm between the feature's corners and the end of the extension line.Figure 5 shows extension lines.
Leader line - A thin, solid line used to indicate the feature with which a dimension, note, or symbol is associated. Generally this is a straight line drawn at an angle that is neither horizontal nor vertical. Leader line

is terminated with an arrow touching the part or detail. On the end opposite the arrow, the leader line will have a short, horizontal shoulder. Text is extended from this shoulder such that the text height is centered with the shoulder line

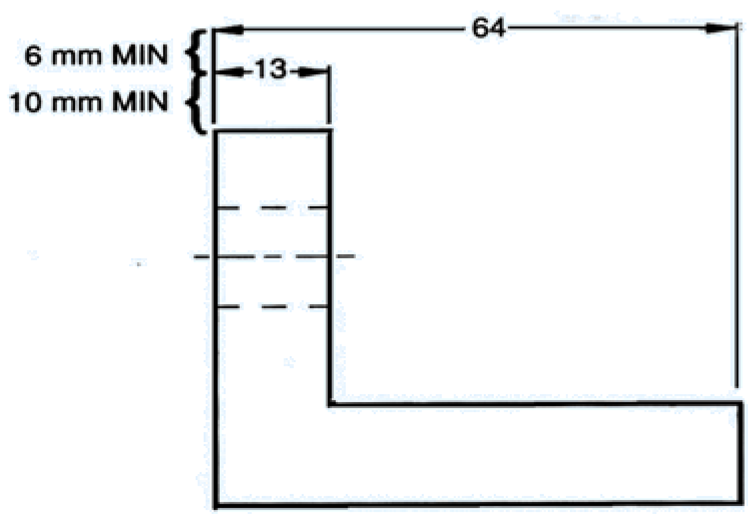


Figure 5. showing extension lines

Arrows – 3 mm wide and should be 1/3rd as wide as they are long - symbols placed at the end of dimension lines to show the limits of the dimension. Arrows are uniform in size and style, regardless of the size of the drawing.Various types of arrows used for dimensioning is shown in figure 6.

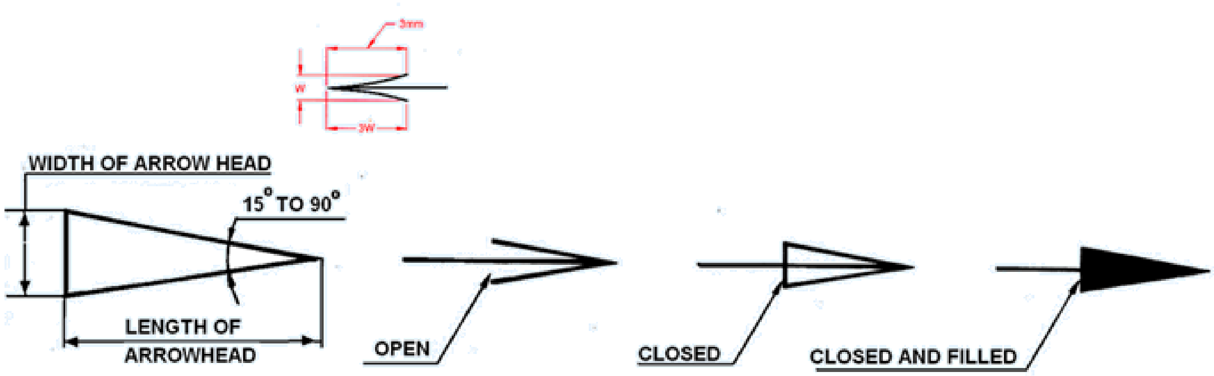


Figure 6.Various types of arrows used for dimensioning

The specification of dimension lines are shown in figure 7.

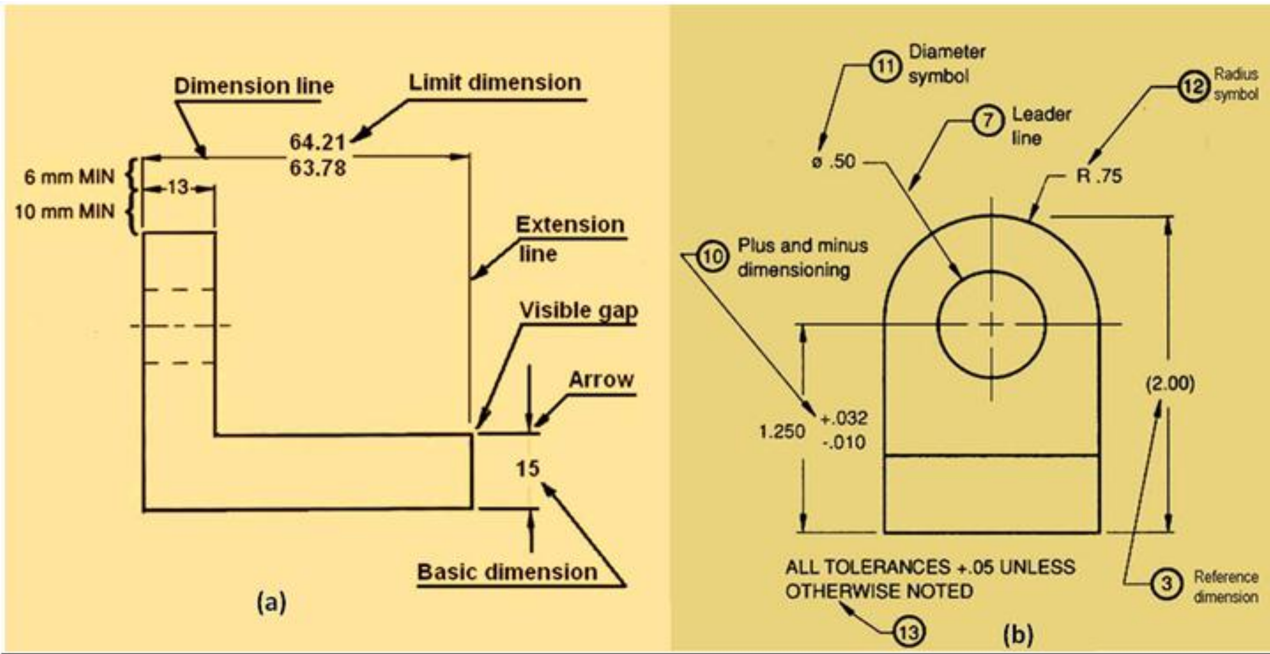


Figure 7 showing the specification of dimension lines.

Dimensioning of angles: The normal convention for dimensioning of angles are illustrated in figure 8.

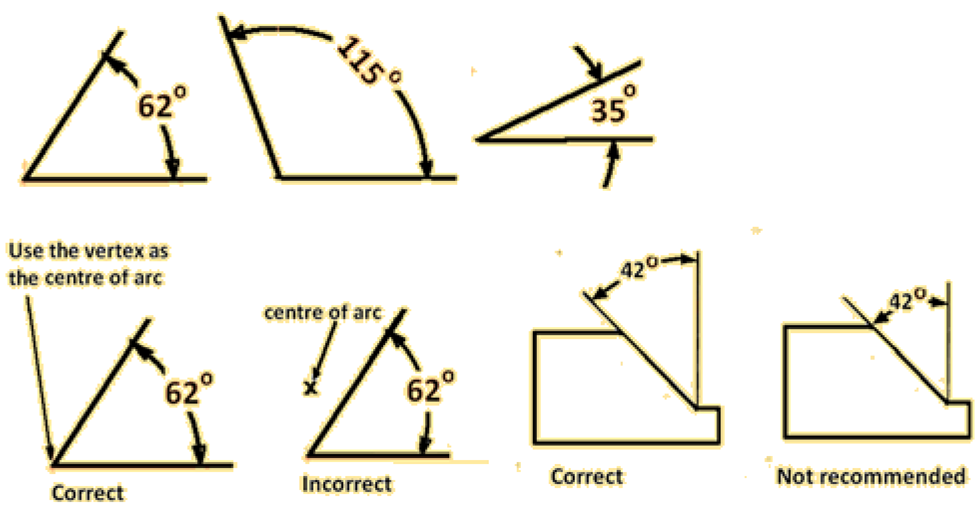
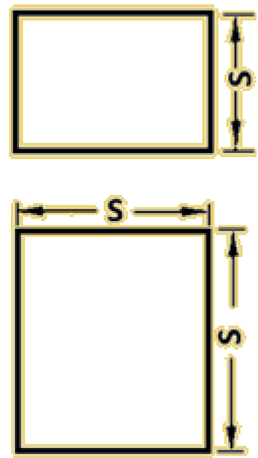


Figure 8 conventions used for dimensioning angles.

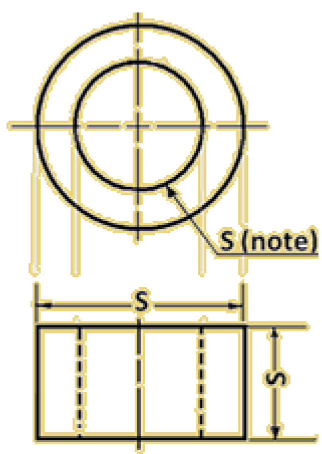
Few examples during dimensioning of solids are shown below:

- Prism – This is the most common shape and requires three dimensions. Two dimensions shown on the principal view and the third dimension on the other view.



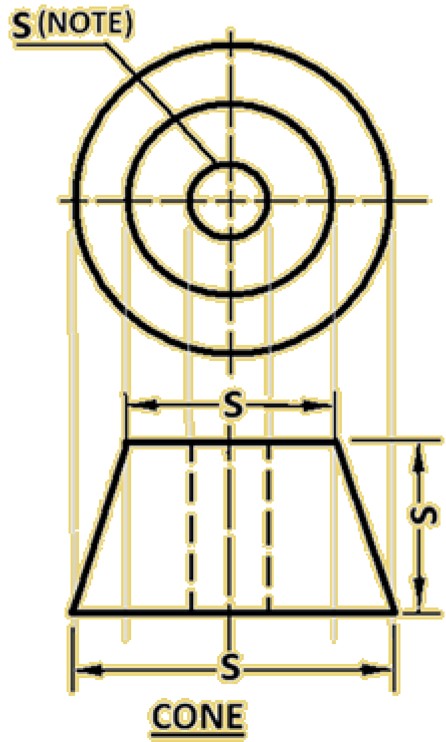
PRISM

- Cylinder – Cylinder is the second most common shape. It requires two dimensions: diameter and length, both shown preferably on the rectangular view.

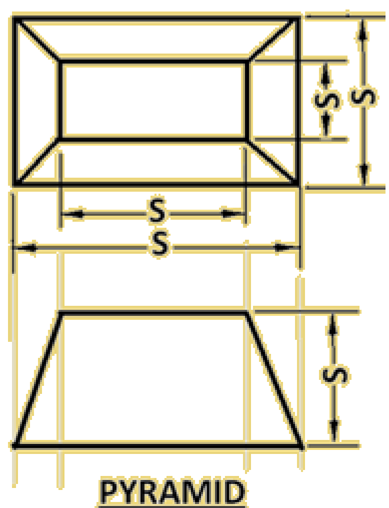


Cylinder

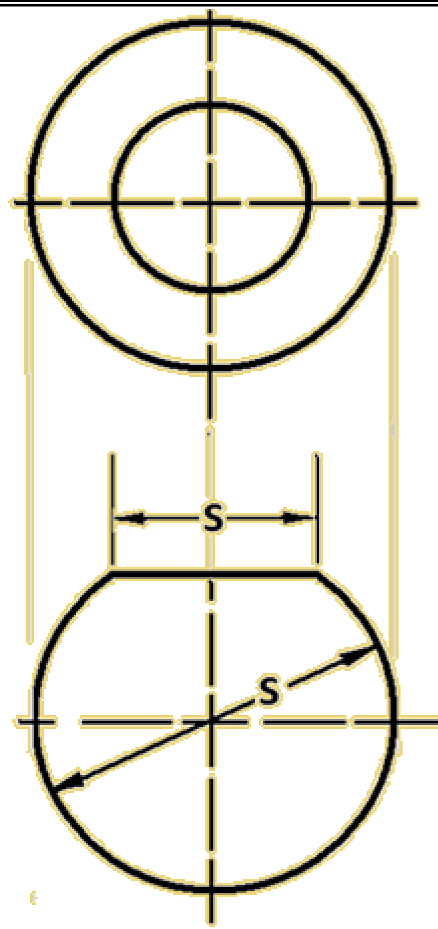
- Cone – requires two dimensions – diameter of the base and altitude on the same view and length. Both shown on the rectangular view is preferred.



- Right pyramids – requires three dimensions – dimensions of the base and altitude.



- Spheres – requires only one dimension. i.e. diameter. However in case of extra features, those dimensions are required to be provided.



RULES OF DIMENSIONING

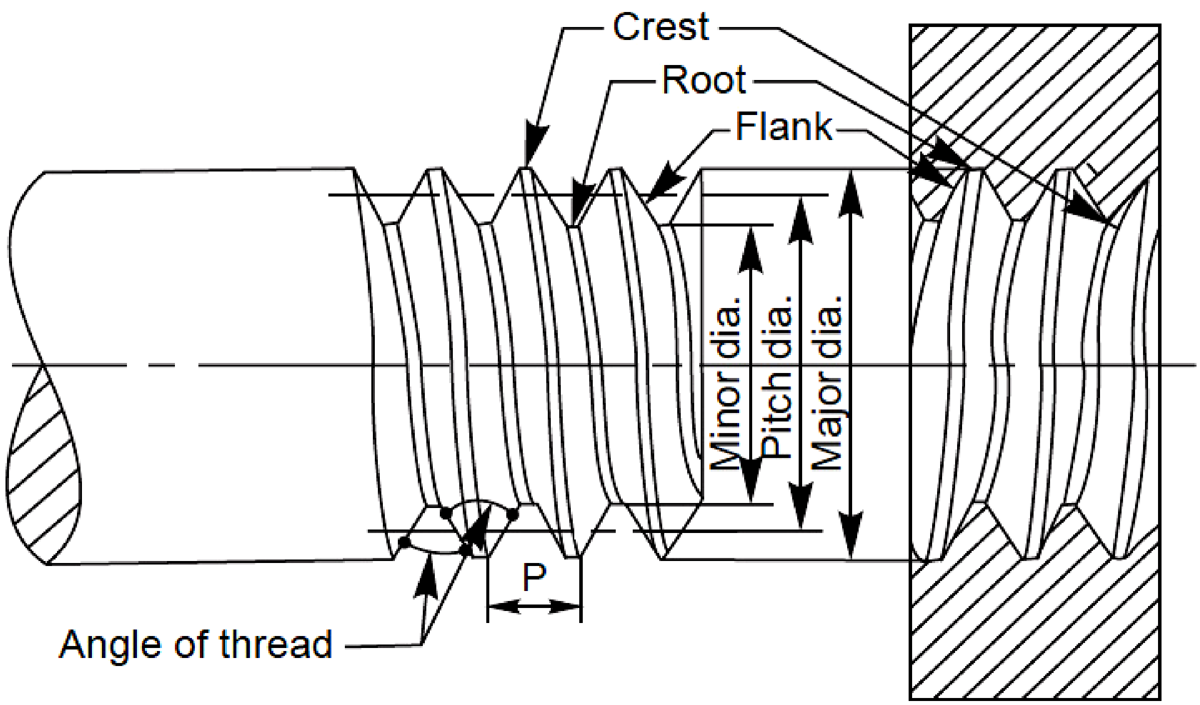
1. Between any two extension lines, there must be one and only one dimension line bearing one dimension.
2. As far as possible, all the dimensions should be placed outside the views. Inside dimensions are preferred only if they are clearer and more easily readable.
3. All the dimensions on a drawing must be shown using either Aligned System or Unidirectional System. In no case should, the two systems be mixed on the same drawing.
4. The same unit of length should be used for all the dimensions on a drawing. The unit should not be written after each dimension, but a note mentioning the unit should be placed below the drawing.
5. Dimension lines should not cross each other. Dimension lines should also not cross any other lines of the object.
6. All dimensions must be given.

7. Each dimension should be given only once. No dimension should be redundant.
8. Do not use an outline or a centre line as a dimension line. A centre line may be extended to serve as an extension line.
9. Avoid dimensioning hidden lines.
10. For dimensions in series, adopt any one of the following ways.
    i. Chain dimensioning (Continuous dimensioning) All the dimensions are aligned in such a way that an arrowhead of one dimension touches tip-to-tip the arrowhead of the adjacent dimension. The overall dimension is placed outside the other smaller dimensions.
    ii. Parallel dimensioning (Progressive dimensioning) All the dimensions are shown from a common reference line. Obviously, all these dimensions share a common extension line. This method is adopted when dimensions have to be established from a particular datum surface
    iii. Combined dimensioning. When both the methods, i.e., chain dimensioning and parallel dimensioning are used on the same drawing, the method of dimensioning is called combined dimensioning.

Conventional representation of threaded parts

The true projection of a threaded portion of a part consists of a series of helices and it takes considerable time to draw them. Hence it is the usual practice to follow some conventional methods to represent screw threads. Screw thread nomenclature figure below shows the true projection of a screw thread, whereas the conventional representation of external and internal threads as recommended by BIS is shown in Conventional representation of threads figure below.

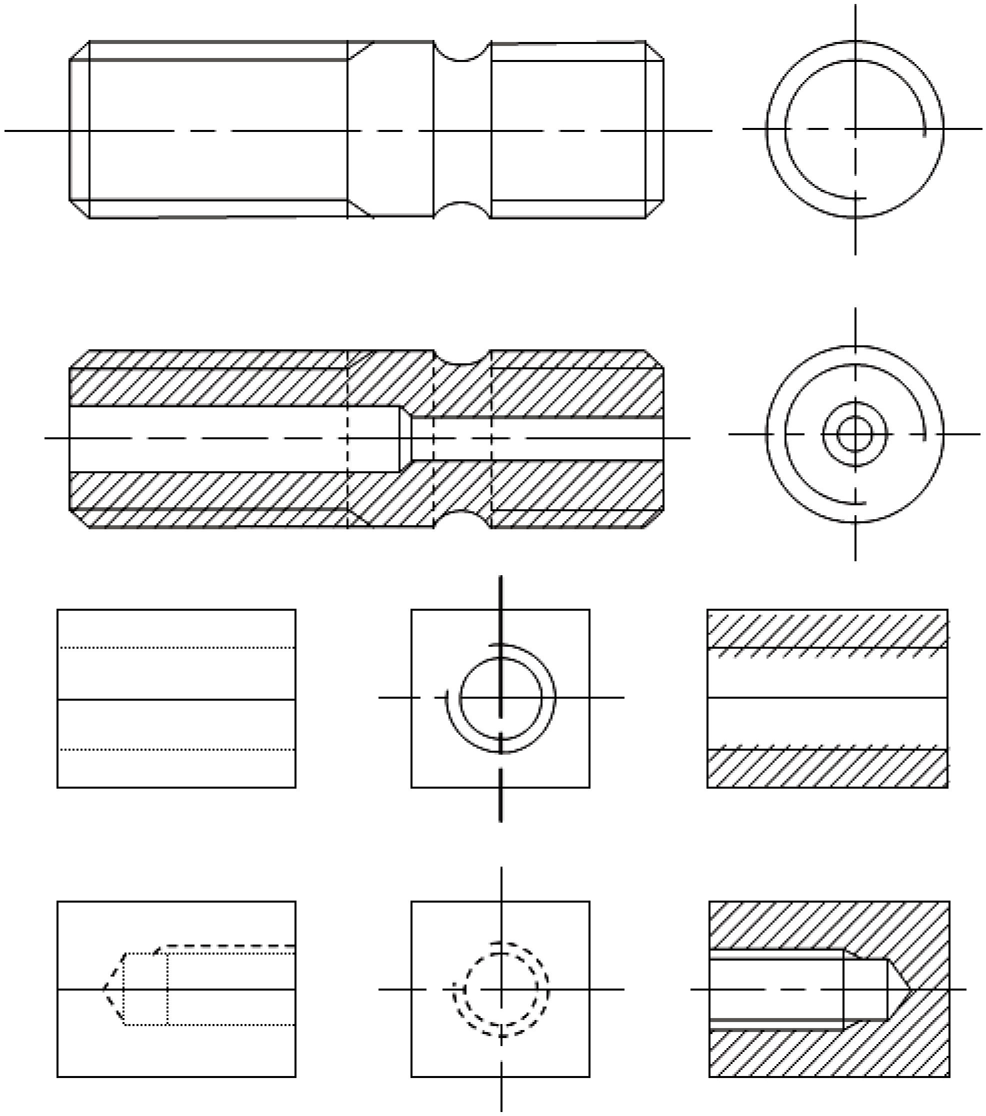
It may be noted from Conventional representation of threads figure below, that the crests of threads are indicated by a continuous thick line and the roots, by a continuous thin line. For hidden screw threads, the crests and roots are indicated by dotted lines. For threaded parts in section, hatching should be extended to the line defining the crest of the thread. In the view from side, the threaded roots are represented by a portion of a circle, drawn with a continuous thin line, of length approximately three-quarters of the circumference.



Screw thread nomenclature

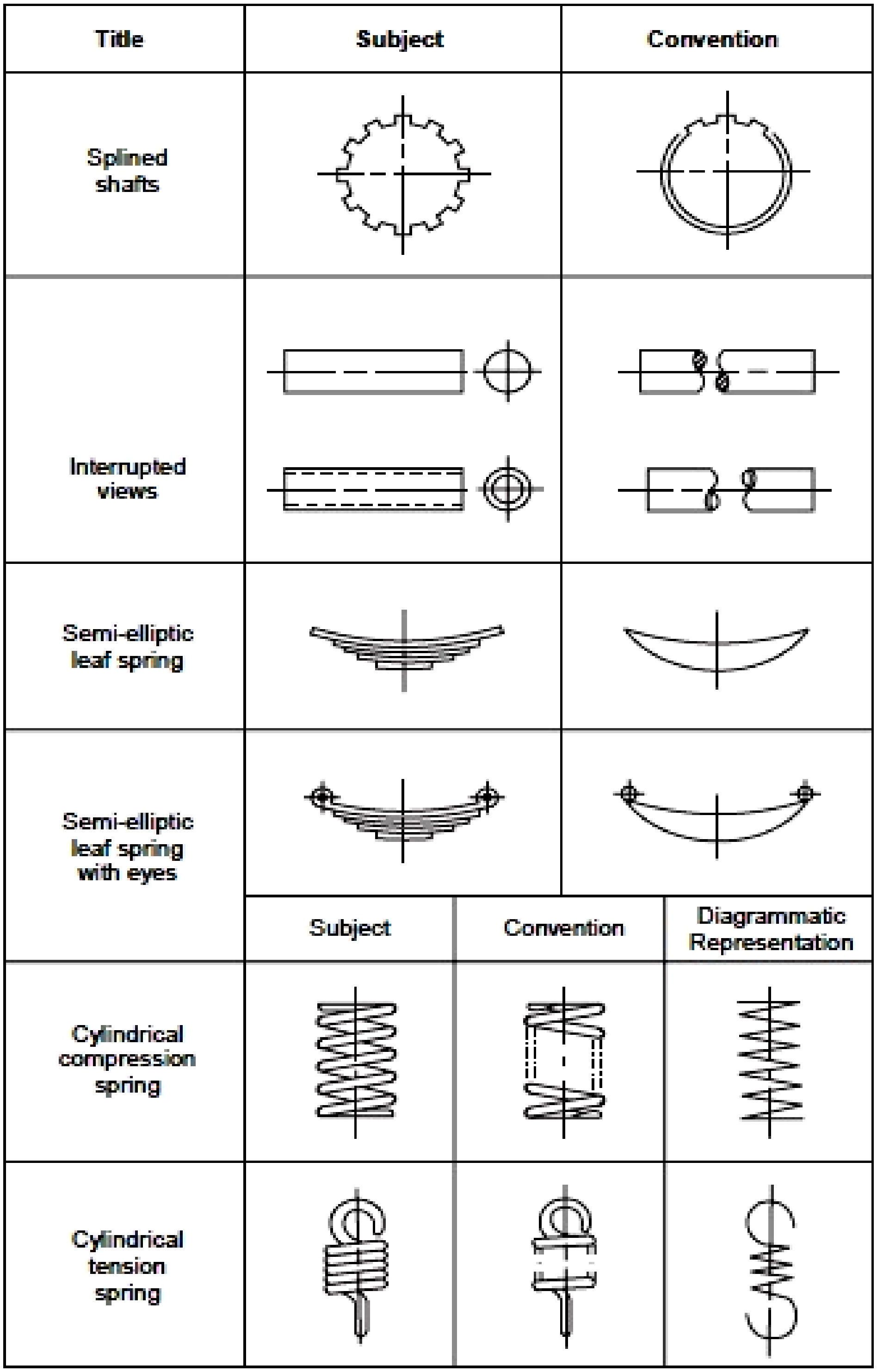
The limit of useful length of screw threads is represented by a continuous thick line or a dotted line, depending on its visibility. The length upto which the incomplete threads are formed beyond the useful limit, is known as a run-out. It is represented by two inclined lines.

The simplified representation, though it saves time, is not an effective method to convey thread forms. The schematic representation, used for the purpose is shown in Schematic representation of threaded parts–V-threads figure below. In practice, the schematic representation is followed for only visible threads, *i.e.*, for external threads and internal threads in section. From the below figure, it may be observed that the crest diameters, both in external and internal threads, are drawn by thick lines. Further, the crests are represented by thin lines, extending upto the major diameter and the roots by thick lines, extending upto the minor diameter, these lines being drawn inclined with a slope equal to half the pitch.



Conventional representation of threads

Conventional Representation of Springs



(b)

Conventional representation of gears

Contours and Edges

Represent the contours and the edges of each gear (see Figures 1,2 and 3), as if they were, - in an unsectioned view, a solid gear bounded by the tip surface; - in an axial section, a spur gear having two diametrically opposed teeth, represented unsectioned, even in the case of a gear that does not have spur teeth or that has an odd number of teeth.

Pitch Surface

Draw the pitch surface with a thin, long chain line, even in concealed portions and sectional views, and represent it, - in a projection normal to the axis, by its pitch circle (external pitch circle in the case of a bevel gear and the median pitch circle in the case of a worm wheel) (see Figures 1,2 and 3); - in a projection parallel to the axis, by its apparent contour, extending the line beyond the gear contour on each side (see Figures 1, 2 and 3).

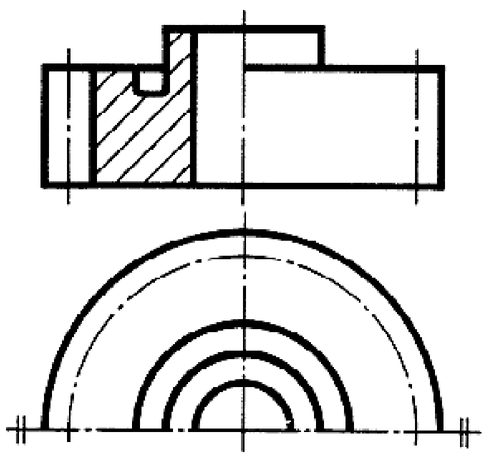


FIGURE 1

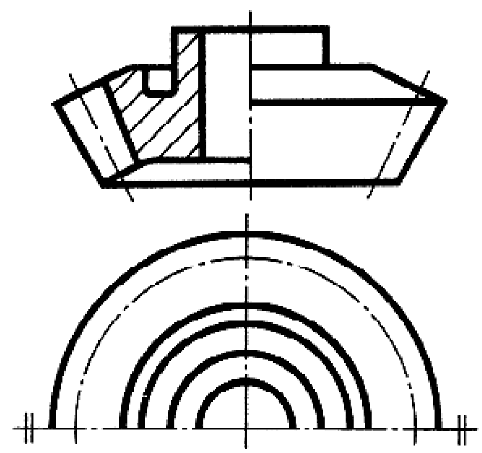


FIGURE 2

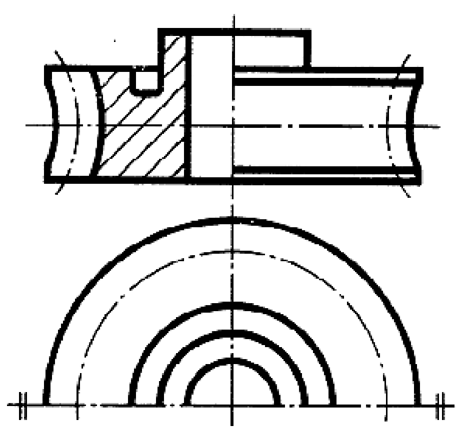


FIGURE 3

Root surface

As a general rule, do not represent the root surface except in sectional views.

However, if it seems helpful to show it also on unsectioned views, always draw it, in this case, as a thin continuous line (see Figures 4, 5 and 6).

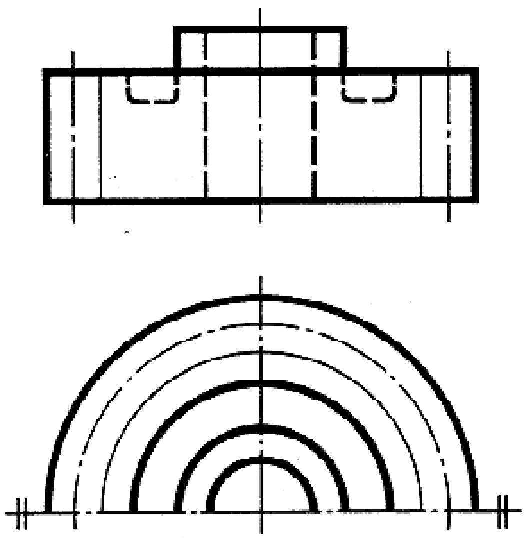


FIGURE 4

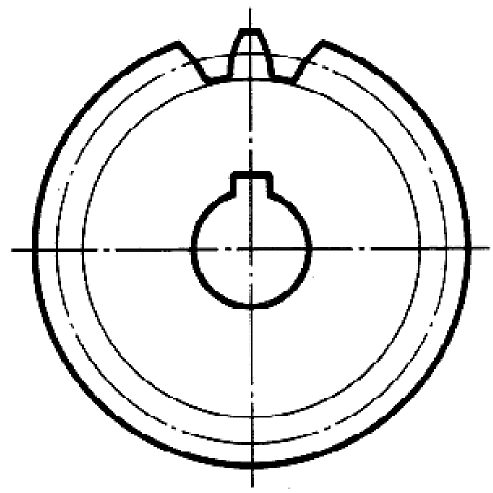


FIGURE 6

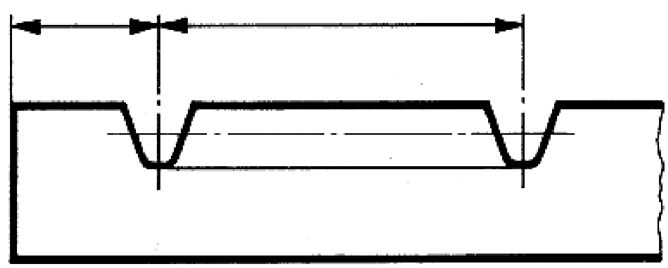


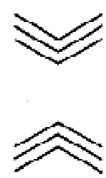
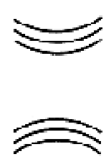
FIGURE 5

Teeth

Specify the teeth profile either by reference to a standard or by a drawing to a suitable scale. If it is essential to show one or two teeth on the drawing itself (either to define the ends of a toothed portion or rack, or in order to specify the position of the teeth in relation to a given axial plane), draw them as thick continuous lines (see Figures

5 and 6).It is necessary to indicate the direction of the teeth of a gear or rack on the view of the tooth surface in a projection parallel to the gear axes, three thin continuous lines of the corresponding form and direction should be shown (see Table and Figure 7).

TABLE

Tooth system	Symbol
Helical to the right	
Helical to the left	
Double helical	
Spiral	

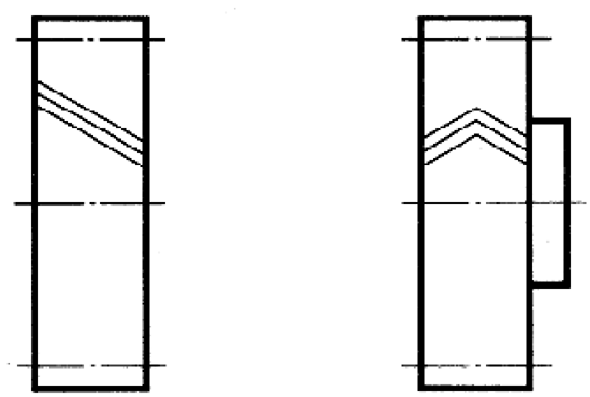


FIGURE 7

UNIT I – Multiple Choice Questions and Answers

S.No	Question	Option a	Option b	Option c	Option d
1	The straight lines which are drawn from various points on the contour of an object to meet a plane are called as __________	connecting lines	projectors	perpendicular lines	hidden lines
2	In orthographic projection an object is represented by two or three views on different planes which ___________________	gives views from different angles from different directions	are mutually perpendicular projection planes	are parallel along one direction but at different cross-section	are obtained by taking prints from 2 or 3 sides of object
3	The Top view of an object is shown on which plane?	Profile plane	Vertical plane	Horizontal plane	Parallel plane
4	A double –threaded screw has pitch of screw 2 mm. How much the screw advances if it is made 3 revolutions?	5	6	12	10
5	When the projectors are parallel to each other and also perpendicular to the plane, the projection is called ______________________________	perspective projection	oblique projection	isometric projection	orthographic projection
6	To represent the object on paper by orthographic projection the horizontal plane (H.P) should be placed in which way?	The H.P is turned in clockwise direction up to 90 degrees	The H.P is turned in anti-clockwise direction up to 90 degrees	H.P plane is placed to left side of vertical plane parallel to it	H.P plane is placed to right side of vertical plane parallel to it
7	The side view of an object is shown on which plane?	Profile plane	Vertical plane	Horizontal plane	Parallel plane
8	A triple –threaded screw is made 4 revolutions. What is the pitch of screw if the screw advances to 6 cm?	24 mm	5 mm	1 cm	5 cm
9	In the Oblique projection an object is represented by how many views?	One view	Two views	Three views	Four views
10	The hidden parts inside or back side of object while represented in orthographic projection are represented by which line?	Continuous thick line	Continuous thin line	Dashed thin line	Long-break line
11	For a Double-threaded screw, Pitch of the helix = lead = _______ the pitch of the screw.	four times	thrice	twice	one time
12	A double–threaded screw is made _____ revolutions. The pitch of screw is 6 mm and the screw advanced to 6 cm	6	5	7	4

13	The object we see in our surrounding usually without drawing came under which projection?	Perspective projection	Oblique projection	Isometric projection	Orthographic projection
14	What is additional 3rd view on orthographic projection in general for simple objects?	Front view	Top view	Side view	View at 45 degrees perpendicular to horizontal plane
15	When a double –threaded screw is made to turn 120 degrees about axis. How much the screw advances through axis?	1⁄3 of pitch of helix	1⁄3 of pitch of screw	1⁄4 of pitch of helix	The advancement is equal to pitch of helix
16	A multiple-threaded screw has pitch of screw 4mm and if the screw is made to 5 revolutions the screw will advances to 40 mm. What type of screw is it?	Single-threaded screw	Double threaded screw	Triple-threaded screw	Four –threaded screw
17	In orthographic projection each projection view represents how many dimensions of an object?	1	2	3	0
18	The front view of an object is shown on which plane?	Profile plane	Vertical plane	Horizontal plane	Parallel plane
19	A triple-threaded screw advances ________ times of its pitch of screw for one complete rotation	6	2	3	4
20	For a triple threaded screw the pitch of screw is 5 mm. The lead (pitch of helix) is_______	15	8	10	30

UNIT II

DIMENSIONAL AND FORM TOLERANCES

Limits and fits IT system of tolerances, deviation of fit - geometric tolerance-tolerancing of form, orientation, location and runout - datums and Datum systems-Dimensioning and tolerancing of profiles

1.1 Limits Fits and Tolerance

Two extreme permissible sizes of a part between which the actual size is contained are called limits. The relationship existing between two parts which are to be assembled with respect to the difference on their sizes before assembly is called a fit. Tolerance is defined as the total permissible variation of a size. It is the difference between maximum limit and minimum limit of size.

1.2 Fits

When two parts are to be assembled the relation resulting from the difference between their sizes before assembly is called a fit. The fit signifies the range of tightness or looseness which may result from the application of a specific combination of allowances and tolerances in the design of mating parts.

1.2.1 Types of Fits

The three types of fits are shown in Fig. 1.1 The disposition of tolerance zones for the three classes of fit are shown in Fig. 1.2.

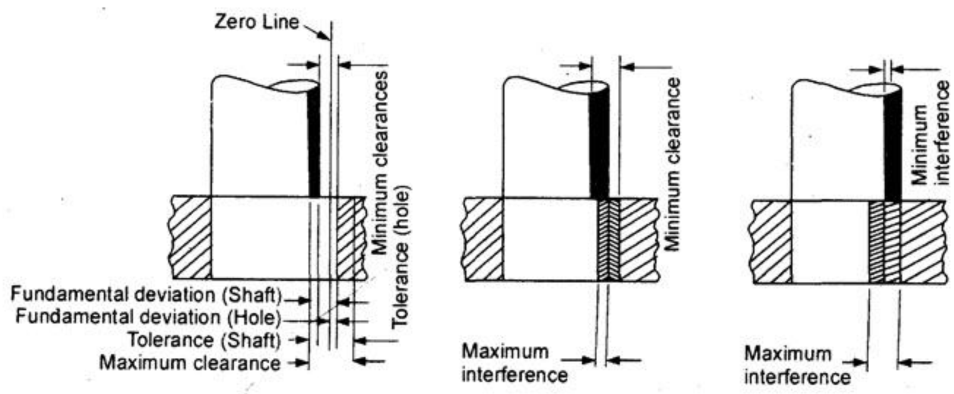


Fig. 1.1 Types of fits

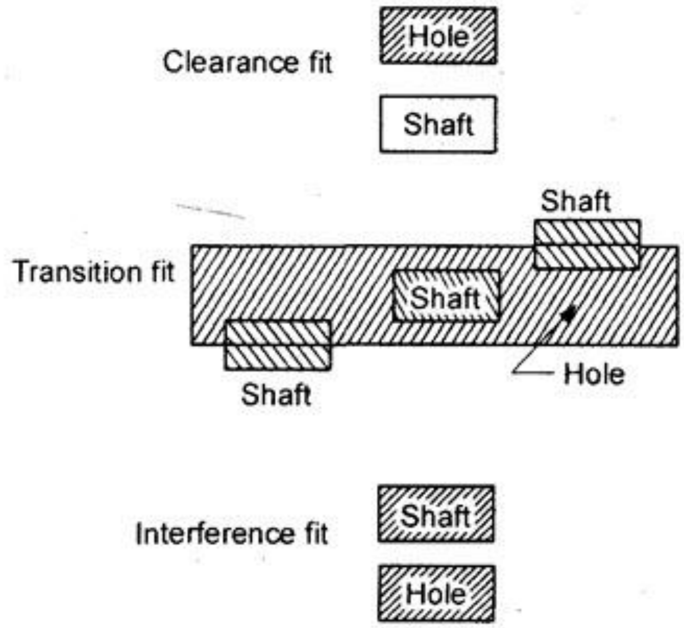


Fig. 1.2 Disposition of tolerance zones for the three classes of fit

There are three general types of fit between the mating parts

1. Clearance fit: A clearance fit is one having limits of size so prescribed that a clearance always results when mating parts are assembled.

2. Interference fit: An interference fit is one having limits of size so prescribed that an interference always results when mating parts are assembled.

3. Transition fit: A transition fit is one having limits of size so prescribed that either a clearance or interference may always result when mating parts are assembled.

1.3 Terminology

The terminology used in fits and tolerances is shown in Fig. 1.3. The important terms are

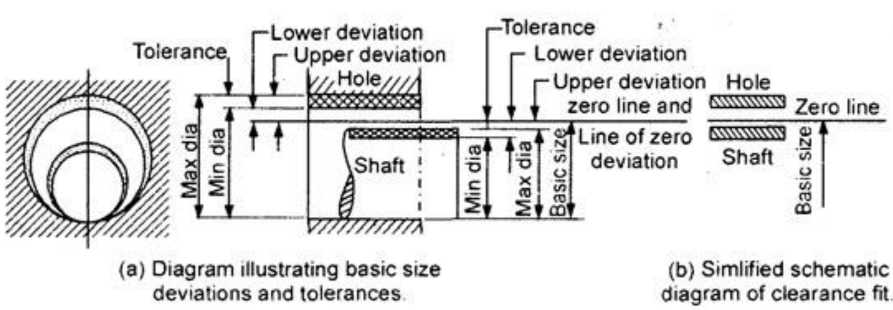


Fig. 1.3 Terminology for fits and tolerances

Basic size: It is the exact theoretical size arrived at by design. It is also called nominal size.

Actual size: The size of a part as may be found by measurement.

Maximum limit of size: The greater of the two limits of size.

Minimum limit of size: The smaller of the two limits of size.

Allowance: It is an intentional difference between maximum material limits of mating parts. It is a minimum clearance or maximum interference between mating parts.

Deviation: The algebraic difference between a size (actual, maximum, etc.) and the corresponding basic size.

Actual deviation: The algebraic difference between the actual size and the corresponding basic size.

Upper deviation: The algebraic difference between the maximum limit of size and the corresponding basic size.

Upper deviation of hole = ES (& art Superior)

Upper deviation of shaft es

Lower deviation: The algebraic difference between the minimum limit of size and the corresponding basic size.

Lower deviation of hole = El (Ecart Inferior)

Lower deviation of shaft = ei

Upper deviation Lower deviation + Tolerance

Zero line: It is the line of zero deviation and represents the basic size.

Tolerance zone: It is the zone bounded by the two limits of size of the parts and defined by its magnitude, i.e. tolerance and by its position in relation to the zero line.

Fundamental deviation: That one of the two deviations which is conveniently chosen to define the position of the tolerance zone in relation to zero line, as shown in fig. 1.4.

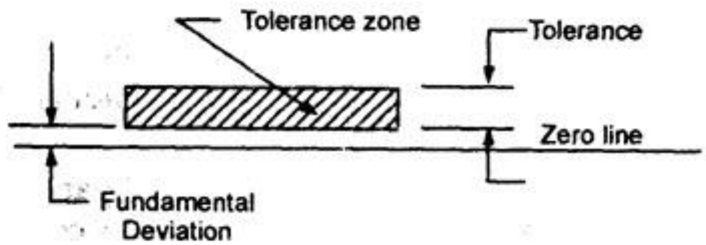


Fig. 1.4 Disposition of fundamental deviation and tolerance zone with respect to the zero line

Basic shaft: A shaft whose upper deviation is zero.

Basic hole: A hole whose, lower deviation of zero.

Clearance: It is the positive difference between the hole size and the shaft size.

Maximum clearance: The positive difference between the maximum size of a hole and the minimum size of a shaft.

Minimum clearance: The positive difference between the minimum size of a hole and the maximum size of a shaft.

1.4 Standard Tolerances

There are 18 standard grades of tolerances as specified by BIS with designations ITOI, ITO and IT to IT 16.

$$\text{Standard tolerance unit, } i = 0.45\ D^{1/3} + 0.001\ D$$

Where i = standard tolerance unit in microns

D = diameter in mm

The standard tolerances for the various grades are given in Table 1.1 and tolerance grades for various manufacturing processes in Table 1.2

Table 1.1 Standard tolerances.

Grade	IT5	IT6	IT7	IT8	IT9	IT10	IT11	IT12	IT13	IT14	IT15	IT16
Value	7i	10i	16i	25i	40i	64i	100i	106i	200i	400i	640i	1000i

Table 1.2 Tolerance grade in various manufacturing processes.

Tolerance grade	Manufacturing process that can produce
16	Sand casting : flame cutting
15	Stamping
14	Die casting or moulding ; rubber moulding
13	Press work, tube rolling
12	Light press work ; tube drawing
11	Drilling, rough turning, boring, precision tube drawing
10	Milling, slotting, planing, metal rolling, or extrusion.
9	Worn capstan or automatic ; horizontal or vertical boring
8	Centre lathe turning and boring, reaming, capstan or automatic in good condition.
7	High quality turning, broaching, honing
6	Grinding or fine honing
5	Machine lapping, diamond or fine boring, fine grinding.

1.5 Hole Basis and Shaft Basis for Fits

1. Hole basis system: In this system, the different clearances and interferences are obtained in associating various shafts with a single hole, whose lower deviation is zero.

2. Shaft basis system: In this system, the different clearances and interferences are obtained in associating various holes with a single shaft, whose upper deviation is zero.

1.6 Selection of Fits

Hole basis system is the most commonly used system because due to the fixed character of hole production tools, it is difficult to produce holes with odd sizes. Commonly used types of fits are given in Table 1.3. Shafts ‘a’ to ‘h’ produce clearance fit, ‘j’ to ‘n’ transition fit, and ‘p’ onwards interference fit with hole.

Table 1.3 commonly used fits

Type of fit	Class of shaft	With holes			Remarks
		H6	H7	H8	
Clearance	d		d8	d8	Loose running fit used for plummer block bearings, loose pullyes, etc.
	e	e7	e8	e8–e9	Easy running fit used for properly lubricated bearings. Finer grades are used for heavily loaded bearings of turbogenerators, electric motors, etc.
	f	f6	f7	f8	Normal running fit used for normal grease or oil lubricated bearings where temperature changes are not too much. This fit may be used for bearings of small electric motors, pumps, or bearings of gear box shaft, etc.
	g	g5	g6	g7	It is close running fit or sliding fit or spigot and location fit.
	h	h5	h6	h7–h8	It is precision sliding fit or fine spigot or location fit.

Type of fit	Class of shaft	With holes			Remarks
		H6	H7	H8	
Transition	j	j5	j6	j7	It is very accurate location fit giving easy assembly and dismentling. It is used in case like coupling spigots and recesses.
	k	k5	k6	k7	It is light keying fit.
	m	m5	m6	m7	It is medium kelvin fit.
	n	n5	n6	n7	It is heavy keying fit for tight assembly of mating surfaces.
Interference	p	p5	p6		It is light press fit with easy dismantling for non-ferrous parts and is standard fit with easy dismantling for assembly of ferrous and non-ferrous parts.
	r	r5	r6		It is light drive fit or non-ferrous parts and medium drive fit for ferrous parts assembly.
	s	s5	s6	s7	It is heavy drive fit for ferrous parts giving permanent or semi-permanent assembly but it is standard press fit for non-ferrous parts. It is used for pressing collars on to shafts, valve seatings etc.
	t	t5	t6	t7	It is force fit on ferrous parts for permanent assembly.

1.7 Dimensioning of Tolerances -Rules

1. The upper deviation should be written above the lower deviation value irrespective of whether it is a shaft or a hole (Fig. 1.5 (a)).

2. Both deviations are expressed to the same number of decimal places, except in the cases where the deviation in one direction is nil (Fig. 1.5 (b)).

3. Tolerances should be applied either to individual dimensions or by a general note, assigning uniform or graded tolerances (Fig. 1.5 (c)).

4. The use of general tolerance not greatly simplifies the drawing and saves much labour in its preparation. On the drawing, the limits on a dimension can be specified in two ways, i.e. (i) unilateral, and (n) bilateral. In unilateral tolerance system, the variation in size is permitted in one direction

1.8 Limit Gauges

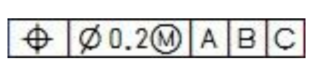
Two sets of limit gauges are necessary for checking the size of various parts. There are two gauges: Go limit gauge, and Not Go limit gauge.

1. Go Limit: The Go limit applied to that of the two limits of size corresponds to the maximum material condition, i.e. (1) an upper limit of a shaft, and (ii) the lower limit of a hole. This is checked by the Go gauge.

2. Not Go Limit: The Not Go limit applied to that of the two limits of size corresponds to the minimum material condition, i.e. (1) lower limit of a shaft, and (ii) the upper limit of a hole. This is checked by the Not Go gauge.

Geometrical Tolerancing

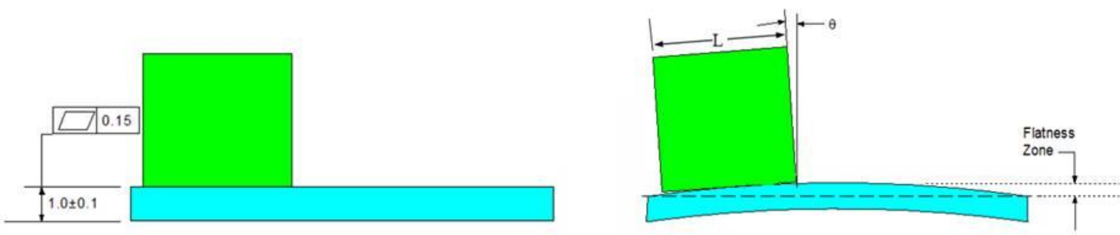
Geometric Tolerancing is the art of applying GD&T. Geometric Tolerancing differs from GD&T which is a mechanical engineering language, GD&T, or Geometric Dimensioning and Tolerancing, represents a way to define the size, location, orientation, and form of a part feature. The definition includes a symbol to communicate design intent, a tolerance that specifies the permissible variation, and coordinate system, i.e. datum reference frame, for inspection and manufacturing purposes. Using GD&T, either manually or with GD&T software, the design engineer can precise communicate a great deal of information that is useful throughout the product development process. An example GD&T callout is shown below.



Geometric tolerancing is governed by two standards. One standard is ASME Y14.5 and the other is an international standard, ISO 1101:2004. While both standards have a great deal in common, some differences exist and should be understood when applying to drawings to ensure accuracy in communicating design intent.

Using geometric tolerancing imparts a well defined control for the variation of a part feature. Functional features of a part, those features that are used for assembly, manufacturing, or product function, are critical to the quality of an assembly. When a functional feature on one part varies, the corresponding mating parts must also vary. Mating conditions between parts dictate how the variation will propagate through the assembly. This variation is propagated through the assembly.

For example, in this case the surface deformation causes an orientation error on the mating part. The tolerance applied to a feature of size controls bending, twisting, or warping of the plate, as shown in the figure below.



Too much variation at a critical feature location can cause problems when assembling the parts or can cause the assembly not to perform as intended. This will result in scrap, rework, field failures, loss of productivity, and warranty and liability costs. Therefore, it is critical that tolerance analysis, or assembly variation analysis, be conducted as a routine part of the product development process.

To conduct a tolerance analysis that includes geometric tolerancing, the GD&T callout must be defined in terms of the implied translational and/or rotational variation. In the example above, the tolerance assigned to the rotational variable, (θ), of the green block could be expressed by the following expression:

$$T_{\theta} = \tan^{-1}(BW)/L)],$$

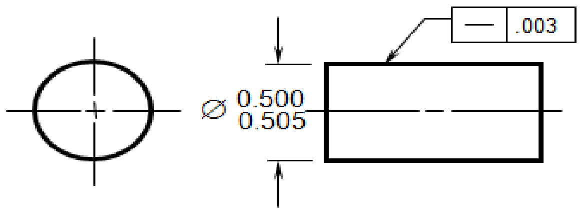
Where –

BW = the tolerance bandwidth

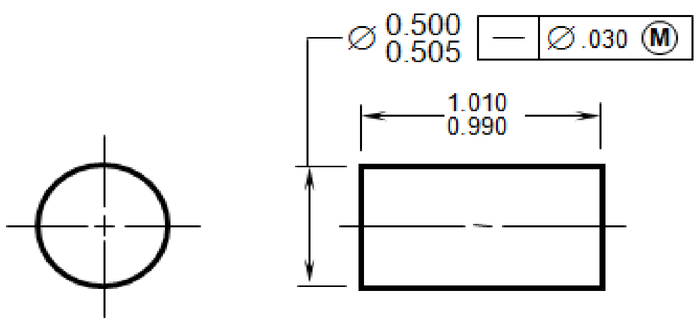
L = the length of contact of mating parts

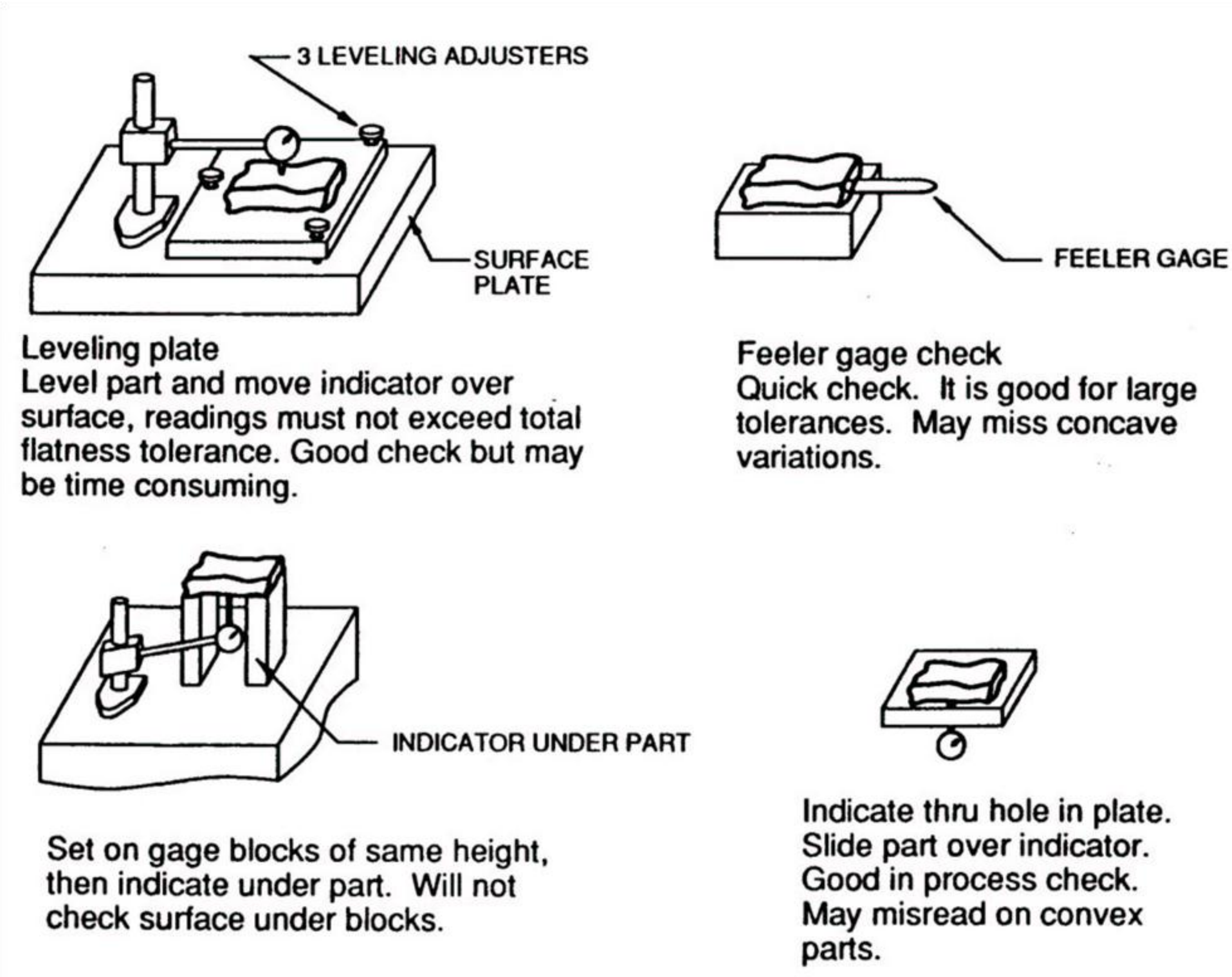
By mathematically quantifying variation and understanding how part features cause shifting in assemblies, it is possible to perform not only one-dimensional (1D) tolerance stack-ups, but also two-dimensional (2D) and three-dimensional (3D) statistical tolerance analyses. And, through direct integration with the CAD model, the process of creating tolerance stack-ups has never been easier.

Straightness applied to the surface of a diameter: The straightness of the feature must be within .003 tolerance zone.



Straightness of an Axis at MMC: The derived median line straightness of the feature must be within a diametric zone of .030 at MMC.



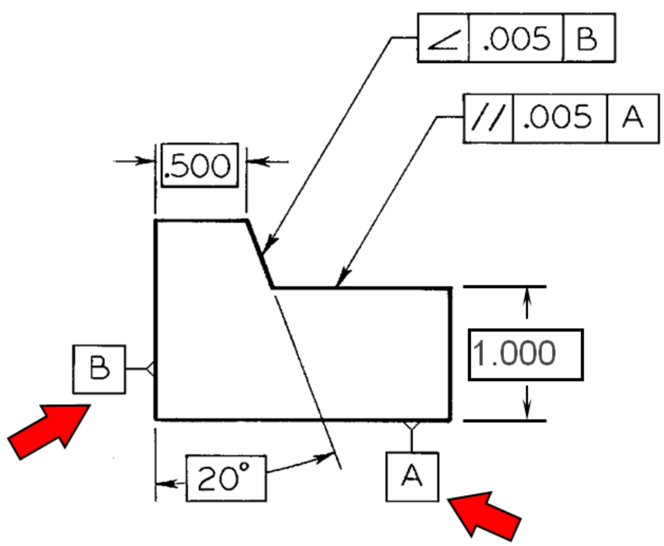


Features that Require Datum Reference

- **Orientation**
  - ⊥ – Perpendicularity
  - ∠ – Angularity
  - // – Parallelism
- **Runout**
  - ↗ – Circular Runout
  - ⇗ – Total Runout
- **Location**
  - ⌖ – Position
  - ◎ – Concentricity
  - ⌯ – Symmetry

Datum

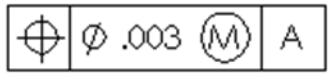
Datums are features (points, axis, and planes) on the object that are used as reference surfaces from which other measurements are made. Used in designing, tooling, manufacturing, inspecting, and assembling components and sub-assemblies. As you know, not every GD&T feature requires a datum, i.e., Flat



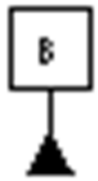
Features are identified with respect to a datum.

- Always start with the letter A
- Do not use letters I, O, or Q

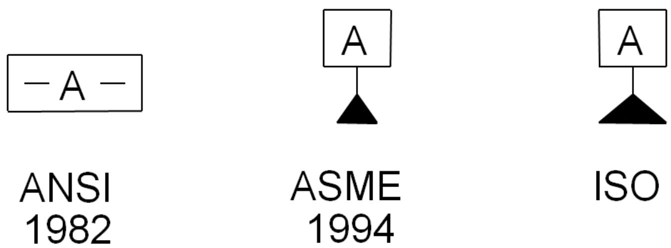
- May use double letters AA, BB, etc.
- This information is located in the feature control frame.



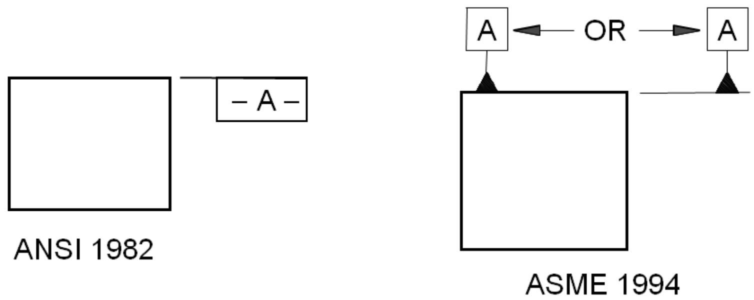
- Datums on a drawing of a part are represented using the symbol shown below.

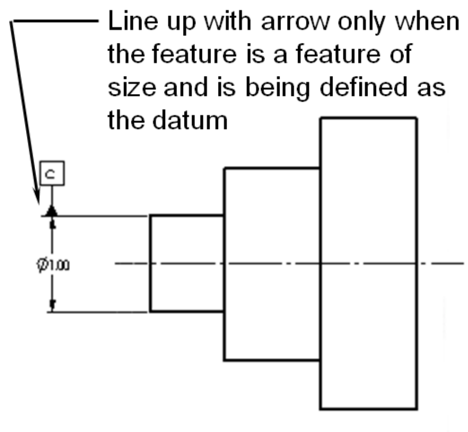


The datum feature symbol identifies a surface or feature of size as a datum.

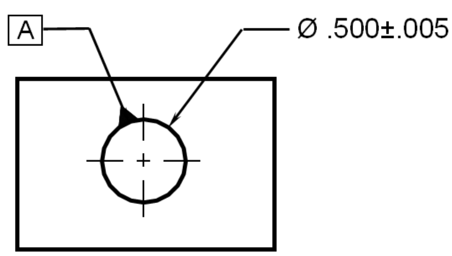


Datums are generally placed on a feature, a centerline, or a plane depending on how dimensions need to be referenced.

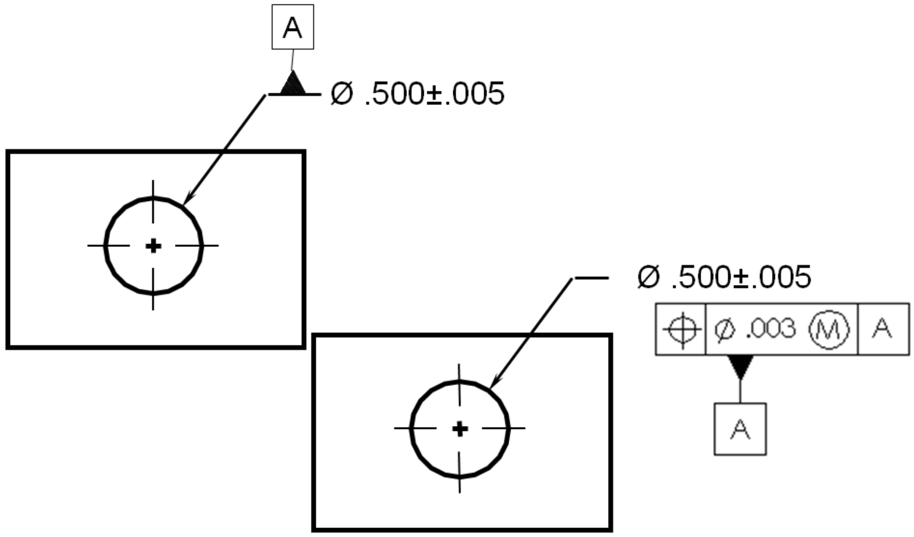




Feature sizes, such as holes

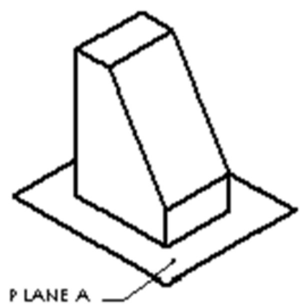


Sometimes a feature has a GD&T and is also a datum

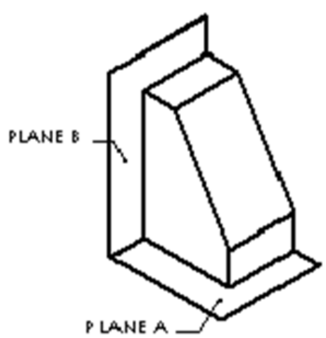


Datums must be perpendicular to each other

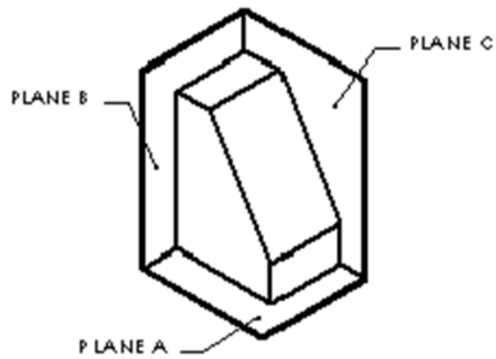
Primary



Secondary

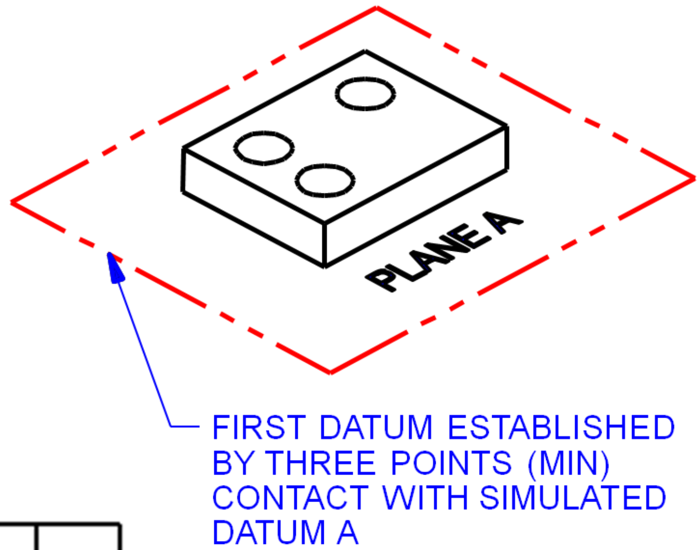


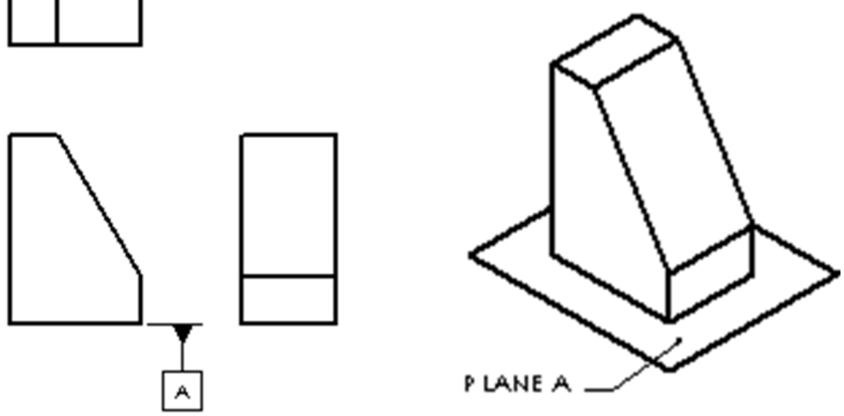
Tertiary Datum



A primary datum is selected to provide functional relationships, accessibility, and repeatability.

- Functional Relationships
  - A standardization of size is desired in the manufacturing of a part.
  - Consideration of how parts are orientated to each other is very important.
  - For example, legos are made in a standard size in order to lock into place. A primary datum is chosen to reference the location of the mating features.
  - Accessibility
  - Does anything, such as, shafts, get in the way?
- Repeatability
  - For example, castings, sheet metal, etc.
  - The primary datum chosen must insure precise measurements. The surface established must produce consistent
  - Measurements when producing many identical parts to meet requirements specified.
- Restricts 6 degrees of freedom

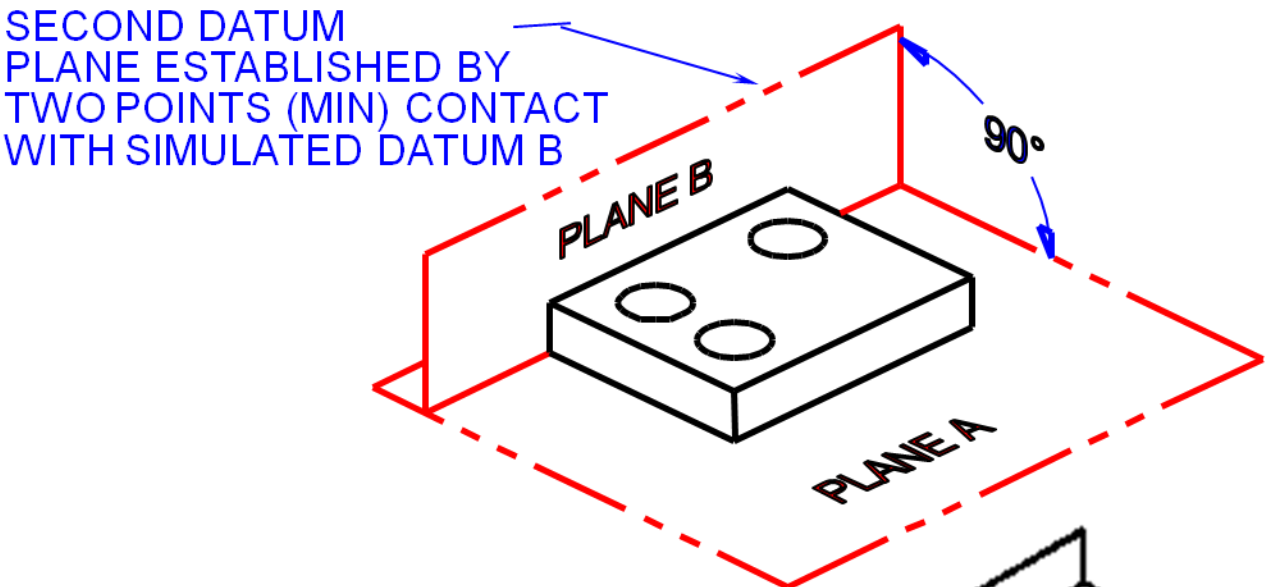


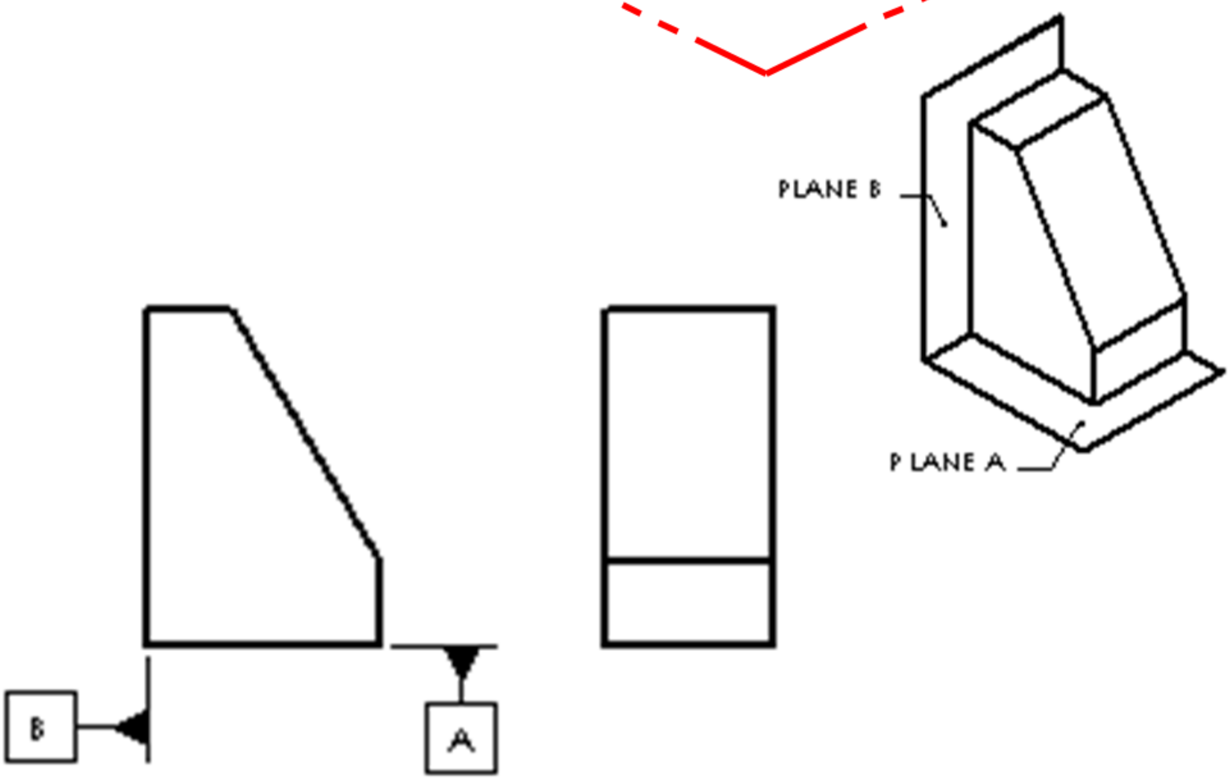


Secondary & Tertiary Datums

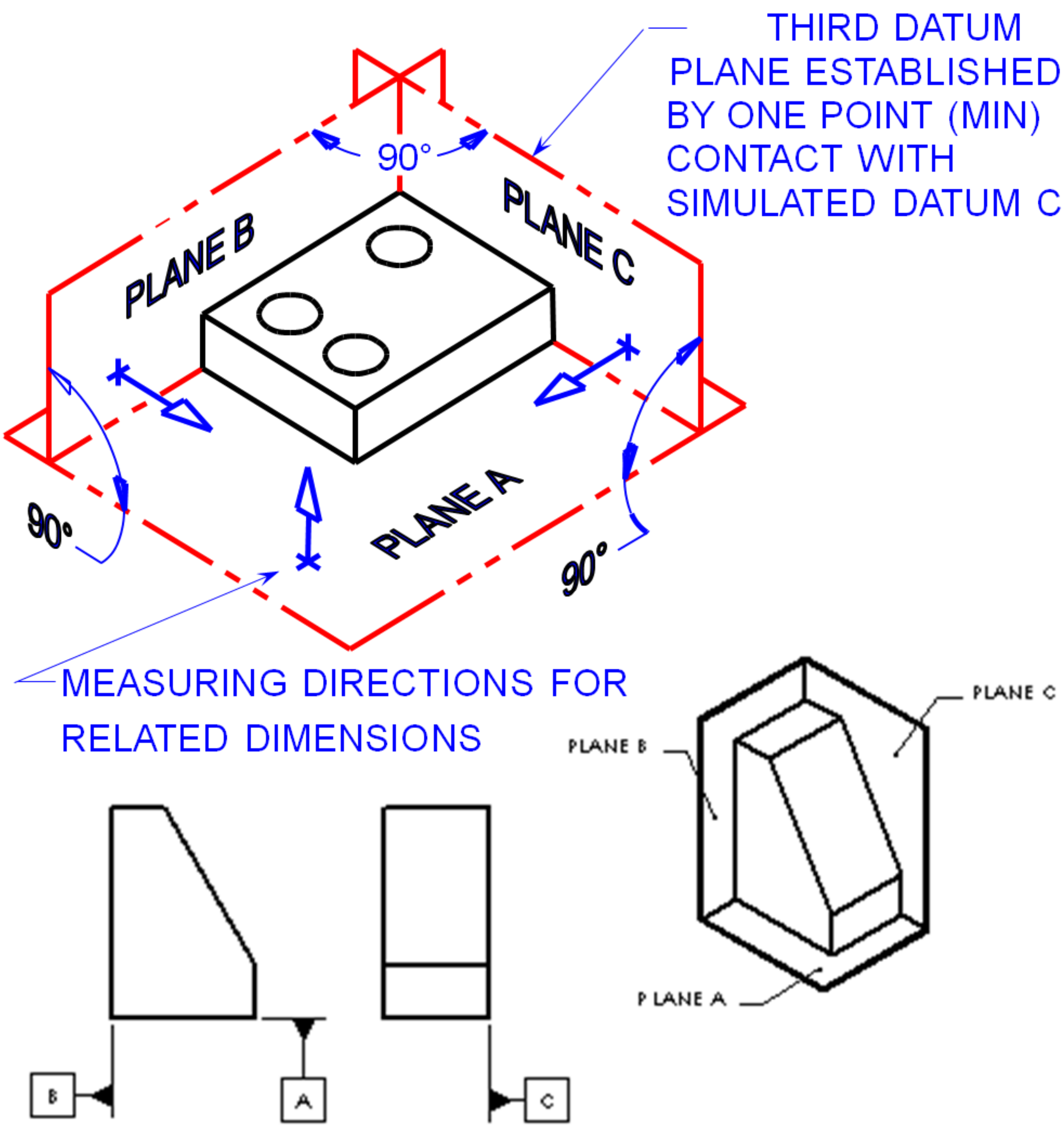
- All dimension may not be capable to reference from the primary datum to ensure functional relationships, accessibility, and repeatability.
  - Secondary Datum
    - Secondary datums are produced perpendicular to the primary datum so measurements can be referenced from them.
  - Tertiary Datum
    - This datum is always perpendicular to both the primary and secondary datums ensuring a fixed position from three related parts.

Restricts 10 degrees of freedom





Restricts 12 degrees of freedom



Orientation Tolerances

⊥ –Perpendicularity

∠ –Angularity

// –Parallelism

- Controls the orientation of individual features
- Datums are required
- Shape of tolerance zone: 2 parallel lines, 2 parallel planes, and cylindrical

UNIT – III Multiple Choice Questions and Answers

S. No	Question	Option a	Option b	Option c	Option d	Answer
1	Circularity applied for ____________plane of a diametrical cross section.	single	double	whole	All of these	single
2	____________ fit may result either in clearance or interference condition?	Clearance fit	Transition fit	Interference fit	All of these	Transition fit
3	_______________is a dimensioning system where a part feature is located (or define by means of a rectangular dimension with the given tolerance.	Geometric Tolerance	Coordinate Tolerance	Production Tolerance	All of these	Coordinate Tolerance
4	The cylindricity tolerance zone is the annular space between two coaxial cylinders and its value is the ______________distance between them.	angular	shortest	radial	none of these	radial
5	Cylindricity applied for ___________ surface of cylindrical objects.	single	double	whole	All of these	whole
6	The limits of size are so specified that a clearance or surface contact may result when mating parts are assembled?	Clearance fit	Transition fit	Interference fit	Line fit	Line fit
7	_____________ is the shortest distance between two points whose tolerance value is the specified distance between two parallel straight lines.	Straightness	Flatness	Profile of a Line	Profile of a Surface	Straightness
8	________________ is the relationship between two cylinders, which have the same axis or common centre	Concentricity	Flatness	Circularity	Cylindricity	Concentricity
9	What are the different types of Fits?	Clearance fit	Transition fit	Interference fit	All of these	All of these
10	In an interference fit the allowance is always____________?	positive	negative	none of the above	All of these	negative
11	______________ tolerance controls the deviation of the surface from the true plane and is the space between the two parallel planes.	Straightness	Flatness	Profile of a Line	Profile of a Surface	Flatness
12	Concentricity tolerance is the deviation of the axis from the __________ position.	TRUE	deviated	common	All of these	TRUE

13	When the internal member is larger than the external member in case of hole and shaft system then there is always a __________?	Clearance fit	Transition fit	Interference fit	All of these	Clearance fit
14	_____________ diameter of hole is taken as the basic size in case of basic hole system?	Maximum	Minimum	Maximum Allowance	Minimum Allowance	Minimum
15	_________________ is the condition where the feature is a continuous curved surface, any point on the surface is at a constant distance from the centre or axis.	Straightness	Flatness	Circularity	Cylindricity	Circularity
16	________________ is the condition where a line, plane or surface lies at 90 degrees to another	Straightness	Datum	Flatness	Squareness	Squareness
17	______________ is an internal member fits in an external member?	Clearance fit	Transition fit	Interference fit	All of these	Clearance fit
18	______________ diameter of shaft is taken as the basic size in case of basic shaft system?	Maximum	Minimum	Maximum Allowance	Minimum Allowance	Maximum
19	______________ is a combination of parallelism, straightness and roundness, applied to the surface of a cylinder	Straightness	Flatness	Circularity	Cylindricity	Cylindricity
20	_________________ is the space between the two parallel lines or surfaces.	Straightness	Datum	Flatness	Squareness	Squareness

UNIT III

MANUFACTURING DRAWINGS

Surface texture indication on drawing - welds symbolic representation of drawings. Given a sub-assembly/assembly to prepare manufacturing drawings of components, Sample exercises on CAD- preparation of manufacturing Drawings.

Indication of Surface Roughness

The roughness values Ra (urn) are given in Table 1.4

Table 1.4 Surface roughness values, Ra (a m)

Roughness value, R_a	50	25	12.5	6.3	3.2	1.6	0.8	0.4	0.2	0.10	0.050	0.025
Roughness grade symbols	N12	N11	N10	N9	N8	N7	N6	N5	N4	N3	N2	N1

The value defining the roughness value Ra in micron and roughness grade symbols are given on production drawings as shown in Fig. 1.9.

Fig. 1.9 Indication of surface roughness in micrometers or roughness grade symbols

When it is necessary to specify the maximum and minimum limits of the surface roughness, both the values or grades should be given as shown in Fig. .10.

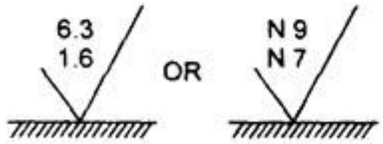


Fig. 1.10 Invocation of the maximum and minimum limits of surface roughness.

2. If it is necessary to indicate the sampling length, it is shown adjacent to the symbol (Fig. 1.11 (a))

3. If it is necessary to control direction of lay or the direction of the predominant surface patterns, it is indicated by a corresponding symbol added to the surface roughness symbol (Fig. 1.11 (b))

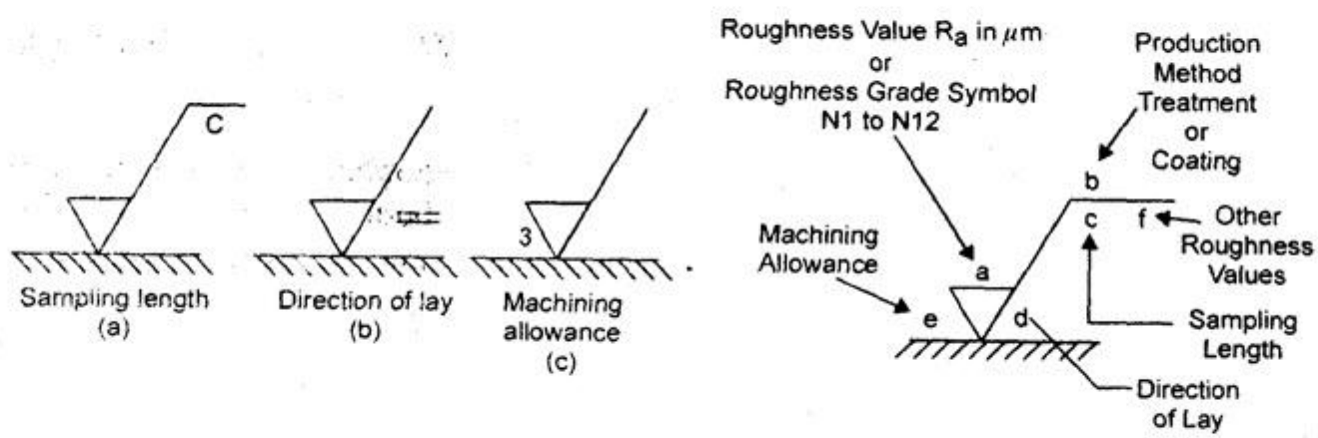


Fig. 1.11

4. Whenever, it necessary to specify the value of machining allowance, it is indicated in the left of the symbol (Fig. 1.11 (c)). This value is generally expressed in millimetres.

Thus, combining the above points, we can establish that the specification of surface

Roughness should be placed relative to the symbol as shown in Fig. 1.11 (d)

Where, a = Roughness value Ra in micrometers or Roughness grade symbol NI to N12

b = Production method, treatment or coating to be used

c = Sampling length

d = Direction of lay

e = Manufacturing allowance -

f = Other roughness value in bracket

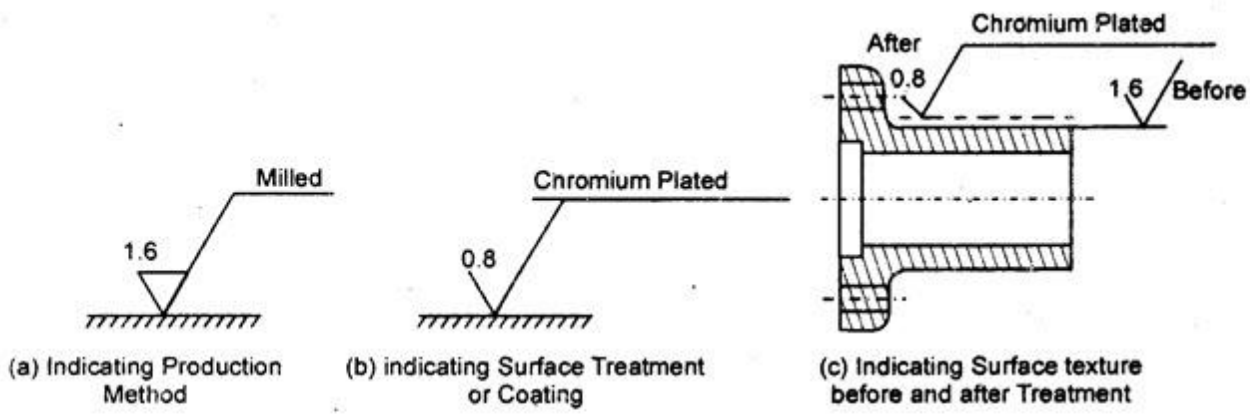


Fig. 1.12 Use of notes with surface texture symbol

5. If it is necessary to define surface roughness both before and after treatment should be explained in a suitable note or in accordance with Fig. 1.12.

Standards

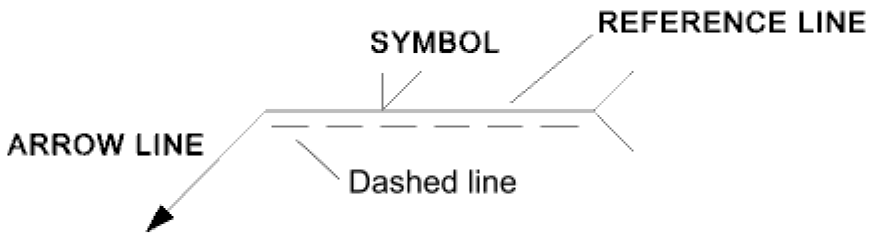
The British Standard for weld symbols is BS EN 22553. When identification of the weld process is required as part of the weld symbol the relevant weld process code is listed in BS EN ISO 4063.

Basic Weld Symbol

The weld symbol always includes

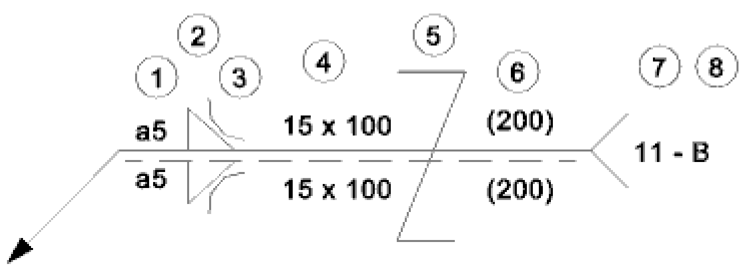
1. An arrow line

2. A reference line
3. A dashed line
4. A symbol



Note: Weld symbols on the full reference line relates to welds on the near side of the plate being welded. Weld symbols on the dashed line relates to weld on the far side of the plate. If the welds are symmetrical on both sides of the plate the dashed line is omitted. If the dashed line is above the full line then the symbol for the nearside weld is drawn below the reference line and the symbol for the farside weld is above the dashed line. For example see sketch below Supplemetary symbols below.

More Detailed Symbolic Representation of Weld

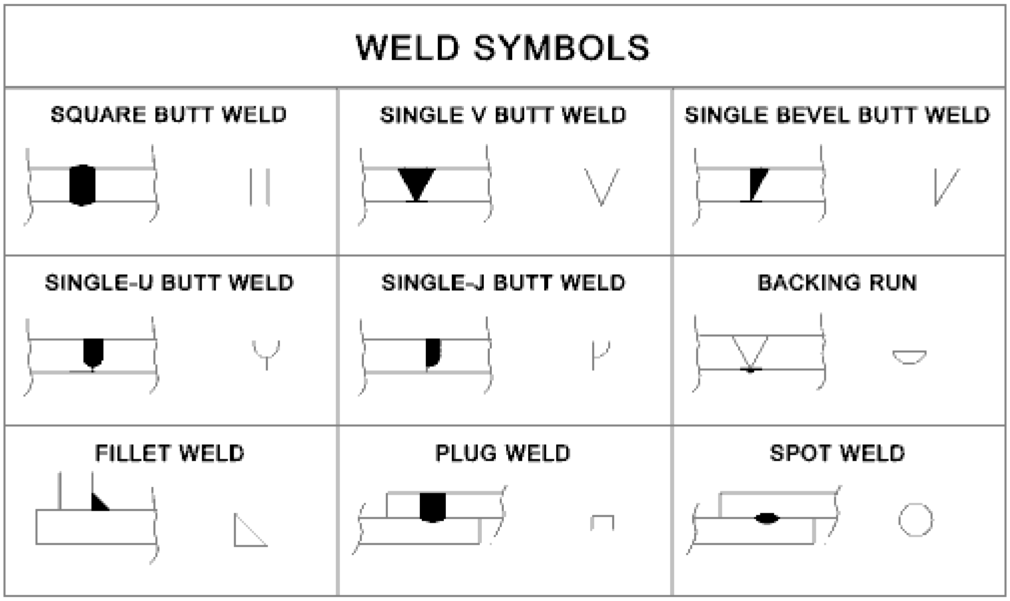


Information above reference line identifies weld on same side as symbolic representation
Information below reference line identifies weld on opposite side to symbolic representation.

1) Dimension referring to cross section of weld
2) Weld Symbol
3) Supplementary symbol
4) Number of weld elements x length of weld element
5) Symbol for staggered intermittent weld
6) Distance between weld elements
7) Welding process reference
8) Welding class

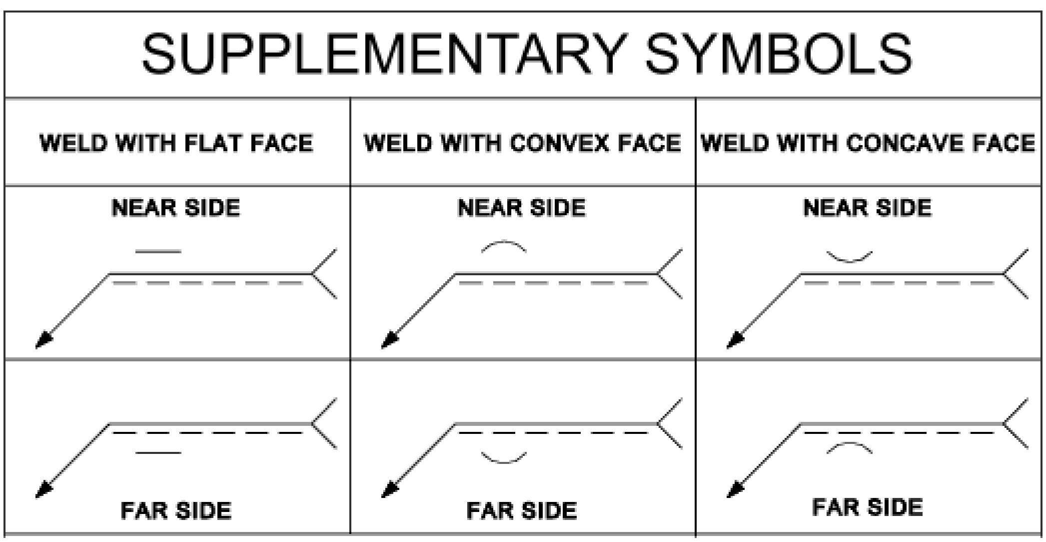
Welding.....Weld process numbers.

Table of Weld Symbols



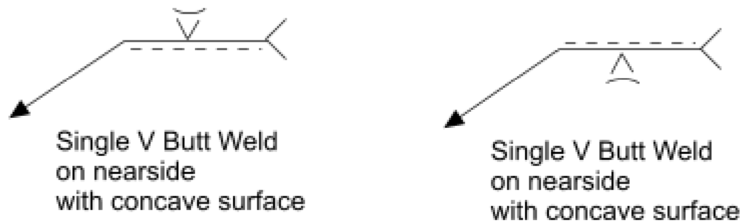
SupplementarySymbols

The weld symbols below are used in addition to the primary weld symbols as shown above. They are not used on their own.

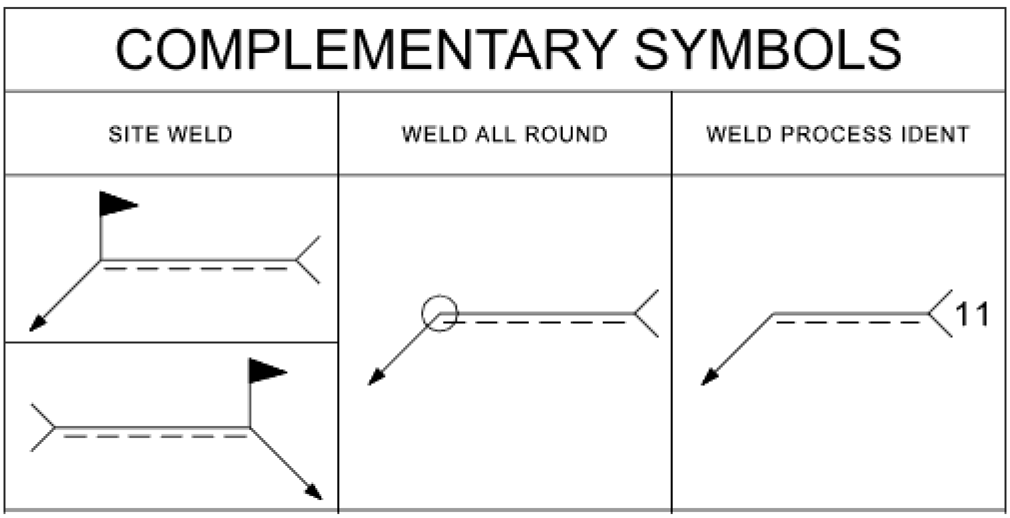


Below is an example of the application of one of these symbol illustrating the identification of the location of the weld relative to the symbol.

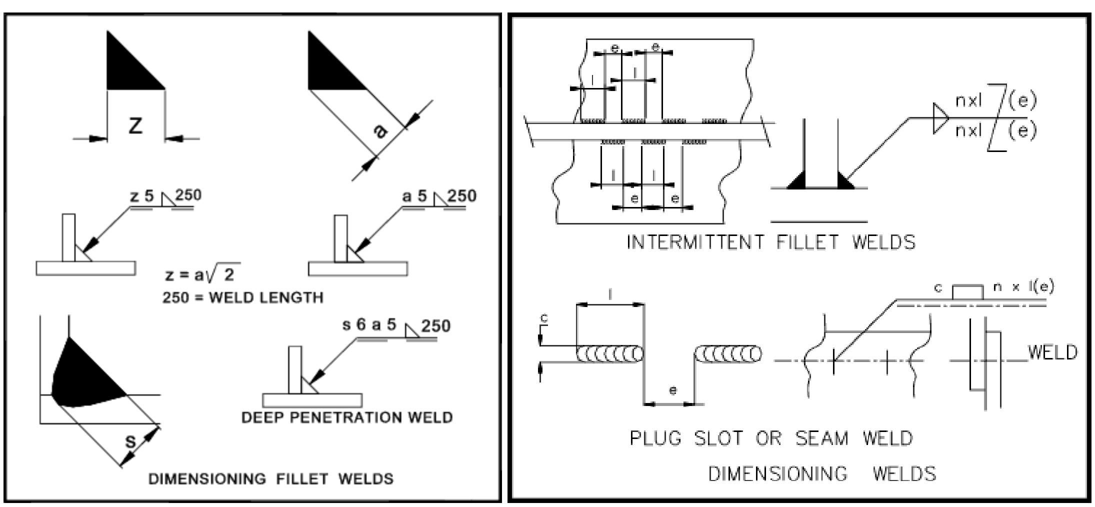
Note: Both of the weld representations below are acceptable



Complementary Indication



Dimensioning Welds



Producing Drawing

A component or part drawing is termed as a production drawing, if it facilities its manufacture. It is an authorized document to produce the component in the shop floor. It furnishes all dimensions, limits and special finishing processes such as heat treatment, grinding, etc., in addition to the material used. It should also mention the number of parts that are required for making of the assembled unit, of which the part is a member.

Production drawing of a component should also indicate the sub or main assembly where it will be assembled. It is necessary to prepare the production drawing of each component on a separate sheet, since a craftsman will ordinarily make one component at a time. However, in some cases, the drawings of related components may also appear on the same sheet. Figure 1.2 shows the production drawing of a jig bush.

Need for a production drawing

The graphic representation of a product, starts at the transformation stage of ideas into a drawing by a design engineer. A production drawing is a complete working drawing, representing all the details of the product, regarding size, shape, material, process, tools and equipment. The craftsman is completely guided by the production drawing, during the manufacture of the product. Hence, any mistake in a production drawing will result in loss of time, money and decreased productivity. Further, it is a legal document while going for subcontracting of works. Hence, a production drawing should be prepared without any scope for more than one interpretation.

The design engineer uses orthographic or pictorial views to record his ideas, free hand. These are called working sketches. These sketches are used for both the component and assembly drawings. The working drawings are sent to the shop, in the form of blue prints, ammonia prints or other similar forms of reproduction. Therefore, the drawings must be made as tracings.

Elements of production drawings

The basic elements of production drawings include

- Size and shape of component
- Format of drawing sheet
- Process sheet
- Projection method
- Limits, fits, and tolerances of size, form, and position
- Production method
- Indication of surface roughness and other heat treatments
- Material specification and Shape such as Castings, Forgings, Plates, Rounds, etc.
- Conventions used to represent certain machine components

- Inspection and Testing Methods
- Specification of Standard Components

Basic principles of dimensioning in production drawings

The basic principles of dimensioning in production drawings include the following:

- The drawing module should dimension each feature only once.
- The drawing should show no more dimensions than necessary.
- Place dimensions outside the drawing view as far as possible.
- Represent dimensions by visible outlines rather than by hidden lines.
- Avoid dimensioning the center line, except when it passes through the center hole.
- Avoid intersecting projection or dimension lines.
- If the space for dimensioning is insufficient, you may reverse arrow heads and replace adjacent arrow heads with dots.

Principles of production drawings

Production drawings are to be prepared on standard size drawing sheets and or prints. The correct size of sheet and size of object can be visualized not only from the views graphic views of but also from the various types of lines used, dimensions, notes, scales, etc. which enable everyone concerned - so long as they are fully conversant with the conventions used - to have a clear and unambiguous understanding of tasks, products and process. The ISO 128 international standards describe drawing conventions is some detail, including views, lines, cuts and sections but these are not in universal use (in India these standards are set by the Bureau of Indian Standards).

Drawing sheets

In production drawing standard size sheets are generally used to save paper and facilitate convenient storage of drawings. In specifications of sheets their size, the size of the title block and its position, the thickness of borders and frames etc., must be considered.

Sheet size

The basic principles to be followed in the sizes of drawing sheets are:

X:Y=1:1.414

X:Y=1, where X and Y are the sheet width and length.

For the reference size (A4), with a surface area of 1 sq meter, X=210mm and Y=297mm.

Title block

The title block, containing the identification of the drawing, should lie within the drawing space at the bottom right hand corner. The direction of viewing of the title block should correspond in general with that of the drawing. The block can have a maximum length of 180 mm.

Drawing Sheet Sizes

Drawing paper and cloth are available in rolls of various widths and in standard trimmed sizes. Most of the draughting rooms use standard sheets, printed with border and title block. There are five standard sizes for drawing sheets (First choice), specified by Bureau of Indian Standards (BIS) SP: 46-1988, as given below. The standard sizes help save paper and are also convenient for storing.

Designation	Dimension (mm)
A0	841 X 1189
A1	594 X 841
A2	420 X 594
A3	297 X 420
A4	210 X 297

Drawing sheets may be used with their longer sides positioned horizontally or vertically. The original drawing should be made on the smallest sheet, permitting the necessary clarity and resolution.

Drawing sheet layout

The layout of a drawing sheet should, by the clarity and neatness of its appearance, facilities the reading of the drawing. It should also facilitate essential references to be located easily. Borders, enclosed by the edges of the trimmed sheet and the frame, limiting the drawing space shall be provided with all the sheet sizes. It is recommended that these borders have a minimum width of 20 mm for the sizes A0 and A1 and a minimum width of 10 mm for other sizes. A file margin for taking perforations may be provided on the edges, far left of the title block. It should have a minimum width of 20 mm.

Four centering marks shall be provided in order to facilitate positioning of the drawing, when reproduced or microfilmed. Two orientation marks may be provided to indicate the orientation of the drawing sheet on the drawing board. It is recommended to provide on all drawings, a figureless metric reference graduation, with minimum length of 100 mm and divided into 10 equal parts. The metric reference graduation shall preferably be disposed symmetrically about the centring mark, near the frame in the border, with a minimum width of 5 mm.

The provision of the grid reference system (zoning) is recommended for all sizes, in order to permit easy location on the drawing, of details, additions, modifications, etc. The number of divisions should be even and be chosen in relation to the complexity of the drawing. However, the length of any side of the rectangle shall be from 25 to 75 mm. The rectangles of the grid should be referenced by means of capital letters along one edge and numerals along the other.

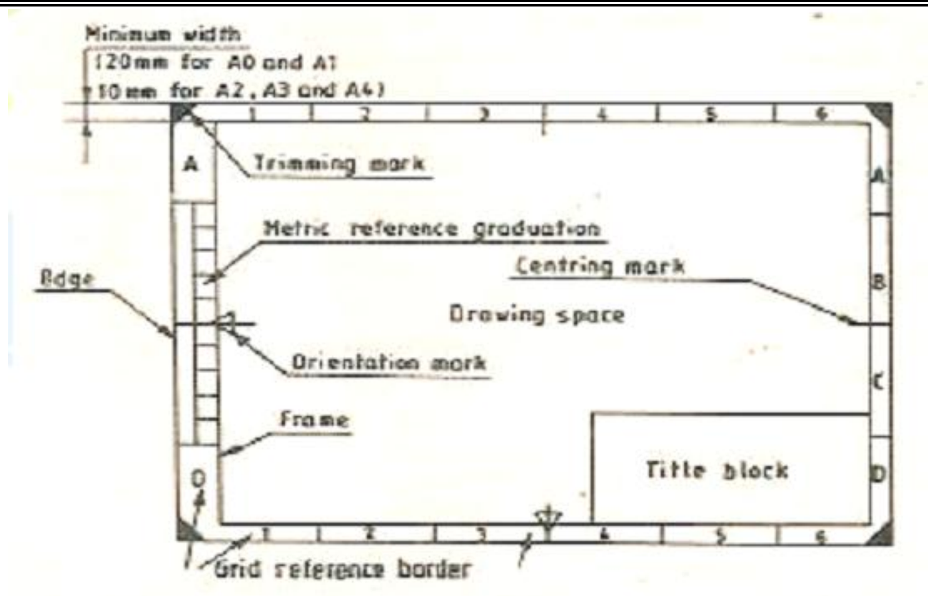
The numbering direction may start at the sheet corner, opposite to the title block and be repeated on the opposite sides. The trimming marks may be provided in the borders, at the four comers of the sheet in order to facilitate trimming. These marks may be in the form of right angled isosceles triangles.

The pre-printed drawing sheets when used, should include the following features:

1. Title block,
2. Frame for limiting the drawing space,
3. Centring marks, and
4. Optional features:

i) metric reference graduation, ii) grid reference system, and iii) trimming marks.

Fig. 2 represents a typical layout of a drawing sheet



Title block

The drawing sheet layout must also provide a title block, which should be located at the bottom right hand corner of the sheet; both for sheets positioned horizontally or vertically, with a maximum length of 170 mm. This should provide the following basic information

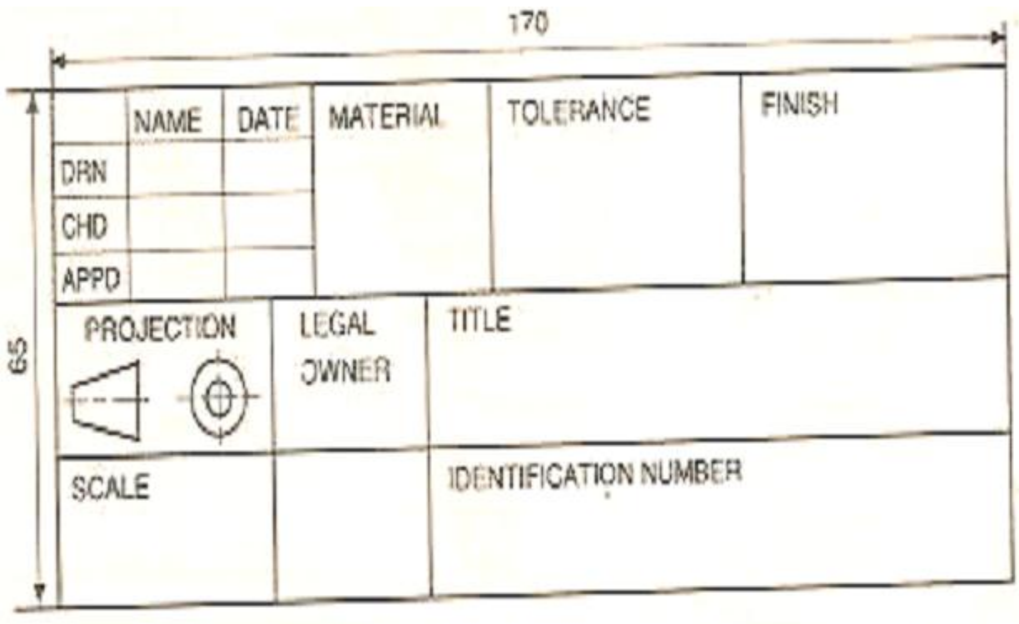
1. Title of the drawing,
2. Sheet number,
3. Scale (s),
4. Symbol, denoting the method of projection,
5. Name of the firm, and
6. Initials of the staff designed, drawn, checked and approved.

The direction of viewing the title block should correspond in general, with that of the drawing. A typical layout of the title block is shown in Fig. 3. However, the heading inside the title block may be arranged as per the convenience, within the overall size specified.

A production drawing may include the following additional information, located either in the drawing sheet or in the title block:

1. Job order number,

2. Surface treatment, roughness, etc.,
3. Key to machining and other symbols,
4. A general note on tolerance on dimensions, not individually toleranced,
5. Reference to tools, gauges, jigs and fixtures,
6. Parts list, and
7. Alternations and revisions



UNIT – III Multiple Choice Questions and Answers

S.No	Question	Option a	Option b	Option c	Option d	Answer
1	________________ is the condition where two lines or surfaces are separated by a uniform distance.	Straightness	Parallelism	Flatness	Squareness	Parallelism
2	Each dimension shall have a _____________ except those dimensions specifically identified as reference, maximum, minimum, or stock.	tolerance	accuracy	error	precision	tolerance
3	Welding drawings are a special type, of this kind of drawing	Symbol	Perspective	Assembly	Isometric	Assembly
4	Which of the following is one of the basic types of welded joints?	T-joint	Rear joint	Angle joint	Groove joint	T-joint
5	____________________ tolerances control the parallelism between the two lines or surfaces and the tolerance zone is the distance between them.	Straightness	Parallelism	Flatness	Squareness	Parallelism
6	Features toleranced with GD&T reflect the _______________between mating parts.	Actual relationship	Comparison	Actual dimension	Pentagon	Actual relationship
7	This symbol indicates a surface to be built up	Groove weld	Surface weld	Corner joint	Lap joint	Surface weld
8	These weld symbols have no arrow-side or other-side significance	Projection or seam weld	Back or backing weld	Surface or groove weld	Flash and upset weld	Flash and upset weld
9	_________________defines the position between two lines or surfaces which neither are nor parallel or perpendicular to each other.	Straightness	Parallelism	Flatness	Angularity	Angularity

10	Plus or minus tolerancing generates a _________ -shaped tolerance zone.	square	rectangle	circle	co-ordinate dimensions	rectangle
11	A projection weld is a type of	Resistance weld	Arc weld	Gas weld	Fillet weld	Resistance weld
12	A back or backing weld is a type of	Groove weld	Resistance weld	Arc and gas weld	Upset weld	Arc and gas weld
13	_________________ tolerance controls the position between a feature and a datum or from another feature..	Form	Orientation	Position	Datum	Position
14	What are the grades of hole used for the manufacturing process "BORING & REAMING?"	H6	H7	H8	H9	H9
15	The basic element of the welding symbol is the	Bent arrow	Shoulder	Note	Upset	Bent arrow
16	This type of weld has equal legs	Surface weld	Groove weld	Fillet weld	Butt weld	Groove wel d
17	________________is the feature where a feature is divided into identical parts by means of a line or plane.	Straightness	Parallelism	Flatness	Symmetry	Symmetry
18	In the former, one measurement is taken during one revolution while in the later the measuring instrument is moved along the component during several revolutions is called______________ ?	Circular run out	Total run out	Flatness	Angularity	Total run out
19	When a surface is required to be built up with multiple weld passes this is known as	Multiple welding	Groove weld	Seam welding	Surface welding	Surface welding
20	This type of weld is used and understood to fill the depth of a hole unless its depth is indicated	Plug	Slot	Seam	Bend	Plug

UNIT IV

RE-DIMENSIONING AND TOLERANCE CHARTING

Introduction to re-dimensioning to suit manufacturing requirements-manufacturing datum-functional datum. Introduction to tolerance charting

Separate part

Zero True Position Tolerancing

Zero true position tolerancing is a technique adaptable to situation requiring functional interchangeability and maximum tolerance advantage in the feature size, form and position interrelationships. Where mating parts and features are simply to mate up or "GO" and tangent contact of the mating features could occur, zero tolerancing is technically acceptable.

However in some conditions, zero position tolerancing is not appropriate. For example, where specific running clearance, fit or similar special mating feature conditions are required, zero position tolerance will not, in general, be technically applicable. There are other considerations, also, which require evaluation to determine whether or not zero true position tolerancing is applicable. It is an optional method of stating many common true position mating part requirements.

True position tolerances are usually established on the basis of MMC size relationships of mating part features. The feature sizes are the criterion with which the process of developing true position tolerances starts. The designed clearance between the mating components is the basis for the true position tolerances, which are stated on the drawing and applied in the manufacture. When the features specified by the true position tolerances are actually produced, any size departure from the MMC size (ex. enlarging the size of a hole) adds to the permissible true position tolerance.

In zero true position tolerancing the same principles apply, except that the true position tolerancing stated is always a fixed "zero", with all the tolerance placed on the same dimension. This, of course, assumes that the actually produced feature will show some deviation from the MMC, which is then added to the "zero" tolerance to give a working position/form tolerance.

It can be stated that in either conventional or zero methods of true position tolerancing, size, form and position variations are considered simultaneously as a composite value. This is really the fundamental principle (along with the MMC principle) on which functional true position tolerancing is based. The reason for this is the fact that related mating part features perform their function in the space limitations provided, regardless of whether that space is derived from size, form, or position variation.

In the use of zero true position tolerancing a situation arises when a produced part with a true position hole pattern might be acceptable to a functional gauge, yet be reject able on the basis of a low limit "GO" size violation, with the result that functionally good parts might be scraped. As stated true position tolerance may be use only for form and position variations.

Conventional true position tolerancing the stated size tolerance can be used for size, form and true position variables as the feature size departs from MMC, whereas a stated true position tolerance may be used only for form and position variables. Size tolerance variation of the features from MMC size can thus add to the true position tolerance; but according to standard practices unused true position variations cannot be added to size tolerance.

The above principle is best described by referring to the CONVENTIONAL TP APPLICATION example. The notation at the bottom of the illustration states that if the hole is produced in perfect location, its size will be permitted to exceed the low limit 6.375(MMC) size down to the virtual size of 6.250. The virtual size is developed from the MMC size of the hole, 6.375, minus the stated true position tolerance, 0.125. This is, of course also the functional gage pin size, and represents the mating part feature at its "worst" condition of assembly.

Further analysis of zero tolerancing, however, reveals drawbacks that tend to temper some of its advantages.

1. For the less experienced and uninitiated user, zero tolerancing represents a psychological barrier; the zeros may give a false impression of the "perfection" expected.
2. The designer may feel that he is relinquishing excessively broad discretion to the production departments, thus abdicating design responsibilities in favor of production such as large size tolerances

In Fig.1, one of the holes illustrated in the .000 method, example is shown with reference to the gage pin (or simulated mating part component). It is seen that the zero true position specification requires a perfect part (perfect form and perfect position) when at MMC, or virtual size.

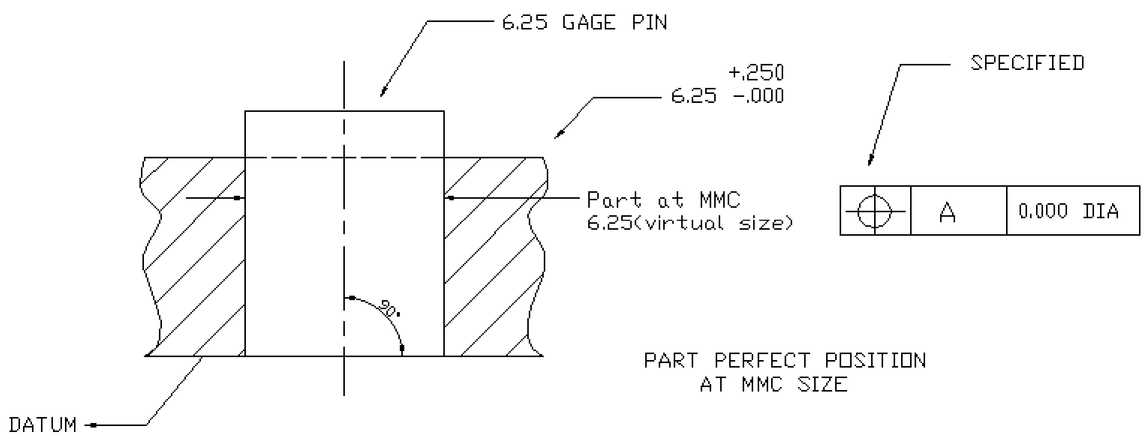


Fig.1 Zero True Position Method

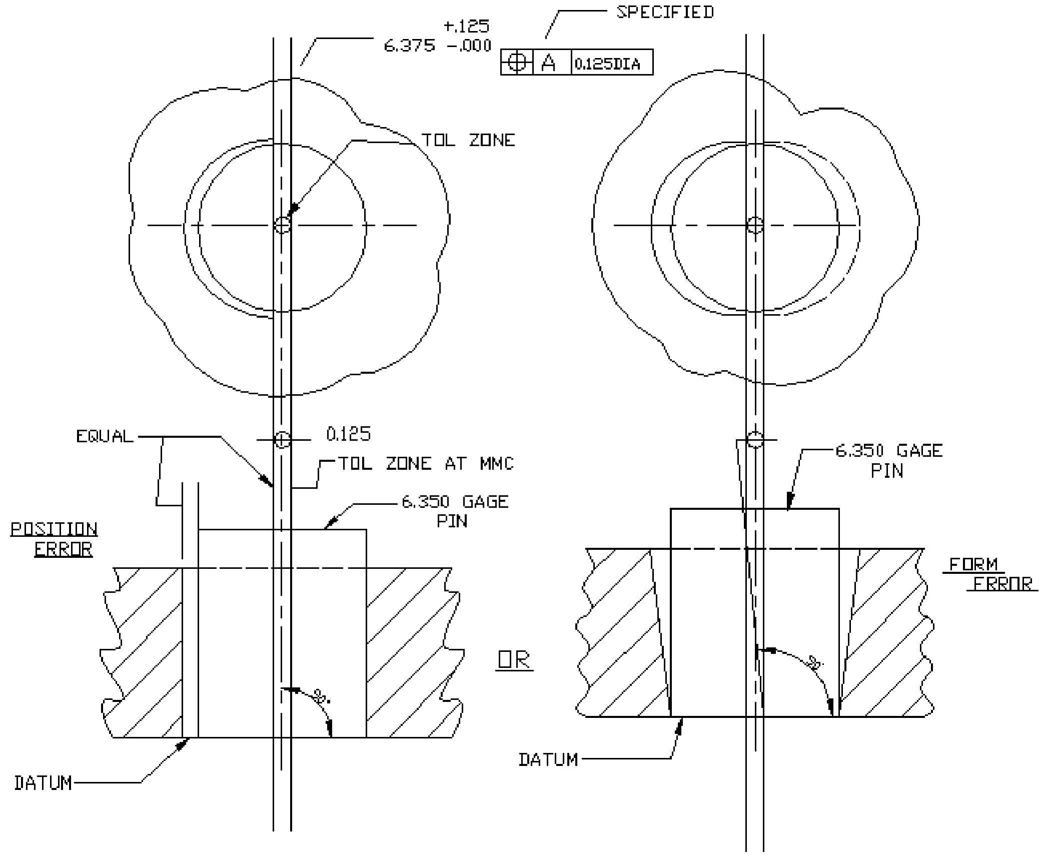


Fig. 2 Conventional Tp Method

Since there must be some clearance between the hole and the inserted mating component or they will not assemble, there is an immediate deviation from the perfect "zero - clearance - zero -interference" situation and some tolerance is acquired.

Fig 2 illustrates the "conventional" method and the established true position tolerance. The tolerance of 0.125 will permit either position or form error (or a combination of both) to this extent when the feature is at MMC. With the same size gage pin as in Fig 1, we see that true position tolerance of 0.125 plus the size tolerance of 0.005 is equivalent to the 0.250 size tolerance obtained by zero method in Fig 1.

As an example imaging $6.250^{+0.000}_{-0.038}$ on the locating dowels, and $6.300^{+0.038}_{-0.000}$ on the locating holes. Using the convention true position "fixed fasteners" method, the calculations are,

MMC size hole - 6.300
MMC size dowel - 6.250

0.050

0.025 -TP tol. On both hole and dowel

The actual true position tolerance in production on both parts would be somewhere between 0.025 and 0.050 (increase due to MMC departure). A functional gage pin size to check the holes between 6.275(hole MMC 6.300, minus TP 0.025 which gives 6.275).

Since the gage pin represents the worst condition (virtual size) of the mating dowel at 6.275, the hole size could be acceptable functionally at 6.275; yet this exceeds the stated hole size low limit.

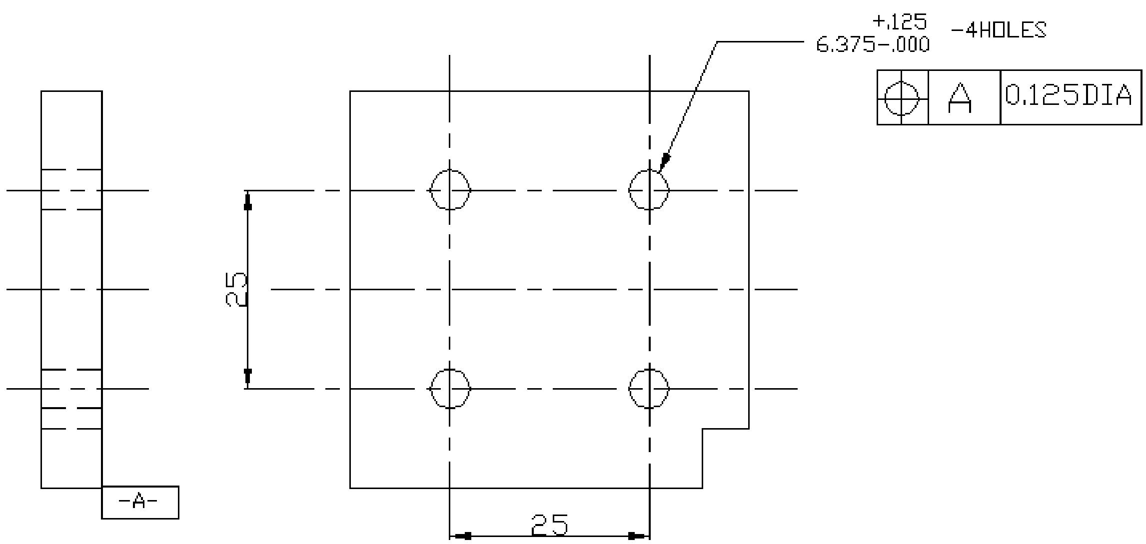
The dowel size, too, could be functional at 6.275 which represents the mating part hole at the worst condition (virtual size).this exceeds the stated dowel size high limit.

However, the 0.000 TP method can provide more total tolerance and yet guarantee proper control if stated as,

$6.275^{-0.000}_{-0.063}$ (Dowel) and $6.275^{+0.063}_{-0.000}$ (Hole)

Comparison of two methods in terms of the full tolerance range difference between the hole and dowel which determines usable size, form, and position tolerance as shown below

Conventional True Position Application Compared With .000 True Position Tolerances



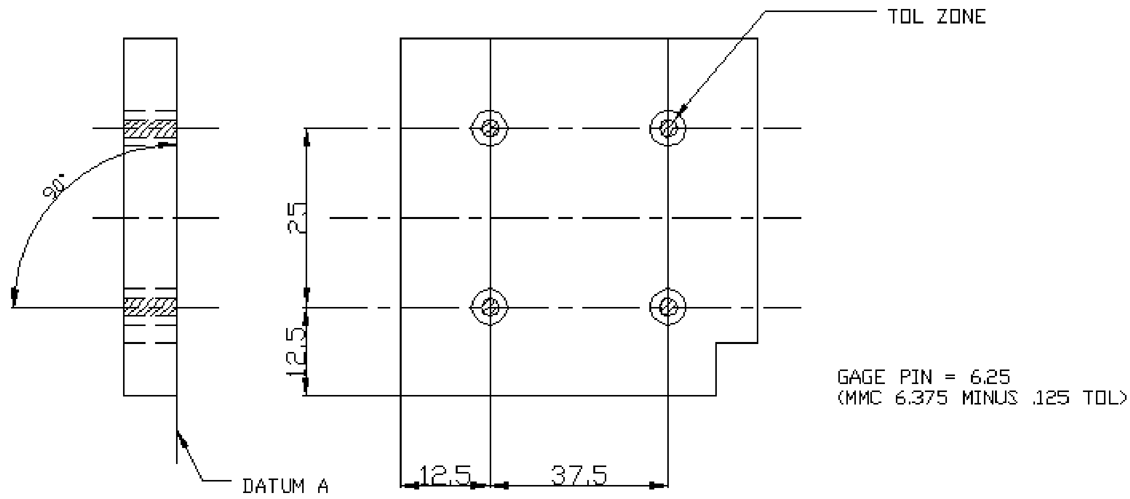


Fig. 3 True Position (Conventional) As Drawn

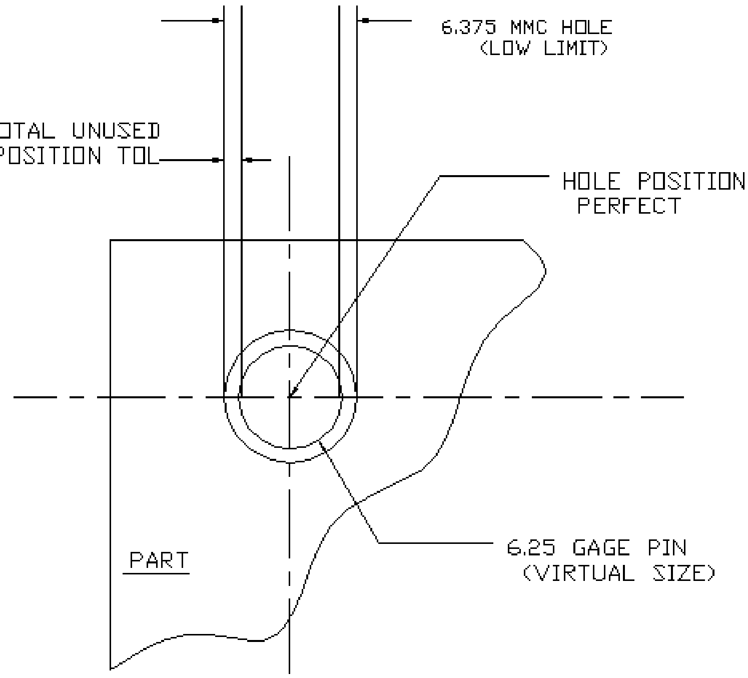
Table 1:

ACCEPTABLE TRUE POSITION TOLERANCES

ACTUAL FEATURE SIZE	TP TOL
6.375	0.125
6.400	0.150
6.425	0.175
4.450	0.200
6.475	0.225
6.500	0.250

Interpretation

Assuming The Gage Pin Represents The Worst Mating Condition, As Position Location Approaches Perfect, It Is Evident That The Hole Size Could Go Down To 6.250 (0.125 Below ,6.375 Low Limit Of Hole) And Still Pass The Gage Pins. However, Parts Below The Low Limit Hole Size Of 6.375 Would Be Rejected On Size, But They Are Good Parts.



True Position

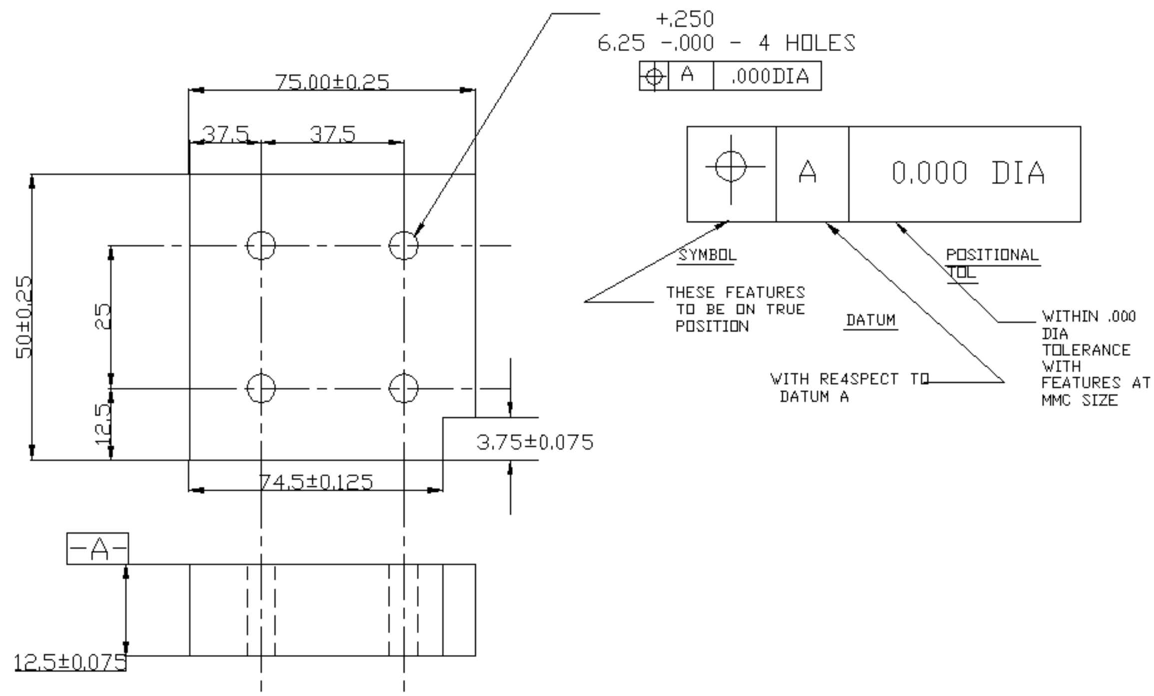


Fig.4 Zero True Position as Drawn

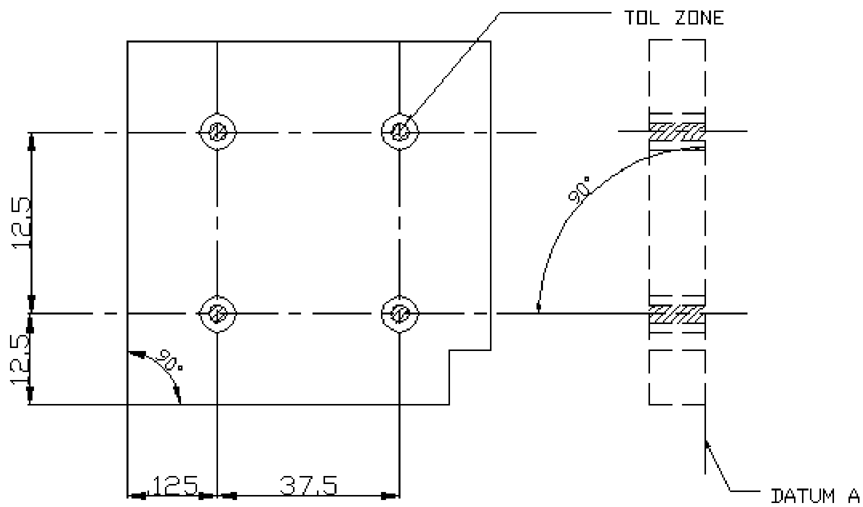


Fig.5 Interpretation

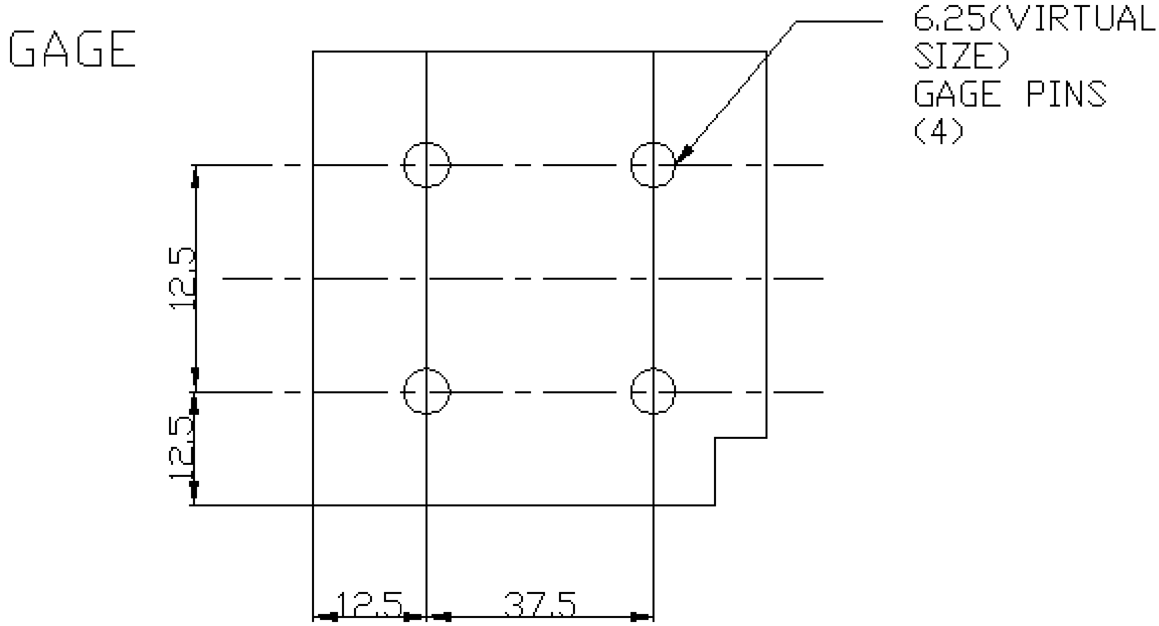


Fig.6 Virtual Size Gage

Table 2:
Acceptable Zero True Position Tolerances

ACTUAL FEATURE SIZE	TP TOL
6.250	0.000
6.275	0.025
6.300	0.050
6.325	0.075
6.350	0.100
6.375	0.125 **MMC**
6.400	0.150
6.425	0.175
6.450	0.200
6.475	0.225
6.500	0.250 **LMC**

FUNCTIONALLY SATISFIED GAUGE PIN SIZE ACCEPTED BY ZERO TRUE POSITION TOL.

Functional Gauge

Functional gauge is the one of the application of true position tolerancing, used for checking the functional worthiness of the component. If the sizes and the relative positions of the features are to be inspected, the most widely used method is to use the functional gauge.

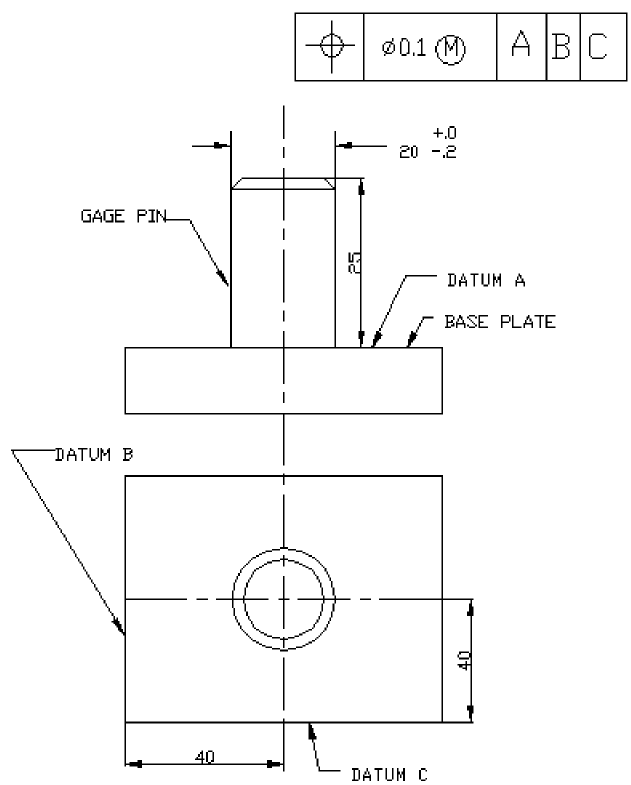


Fig.6 Functional Gauge

Fig 6 shows the functional gauge to be used for this purpose and the component to be inspected. The size of the holes and their relative positions are to be checked for acceptance.

The component is inserted over the gauge and it is passed through the gauge, the component is accepted otherwise the component is rejected.

Datum 'A' is called primary datum on which the peg should be located. Datum 'B' is called the secondary datum which represents the position of peg from certain datum generally from itself.

Now the pin size is $500^{+0.0}_{-5.0}$

For IT grade 6, the tolerance value for dia 502.1 is given by 9μm.A value of 0.125 can be obtained in jig boring machine.

The true position value is given as

$$5.025^{+0.250}_{-0.000}$$

This method is used when the inspecting quantity is large. As it is expensive to produce a functional gauge it cannot be used for job shop type production.

Paper Layout Gauging

The paper layout gauging technique has been used for inspecting the components of job shop type, one of the applications of true position tolerancing, by overlapping the component diagram with that of blue print diagram. When the inspecting quantity is less then it is best to use paper layout gauging. The term, 'paper' is used to indicate that the technique has been used to inspect the components with paper.

The technique for measurement of paper layout gauging is as follows. The tolerances are drawn in one of the transparent sheet. The tolerance zones are obtained from the blue print diagram. The component is measured and the axis of the features is marked on other layer. Now both these layers are overlapped. Both the tolerance zone and axis is visible. On overlapping if the axis of the features lies within tolerance zones, then the component is accepted, if not the paper containing component axis is moved until the axis come within their respective tolerance zones. If the component hole centre does not lie within their true position tolerance zones, it is said that the component is rejected

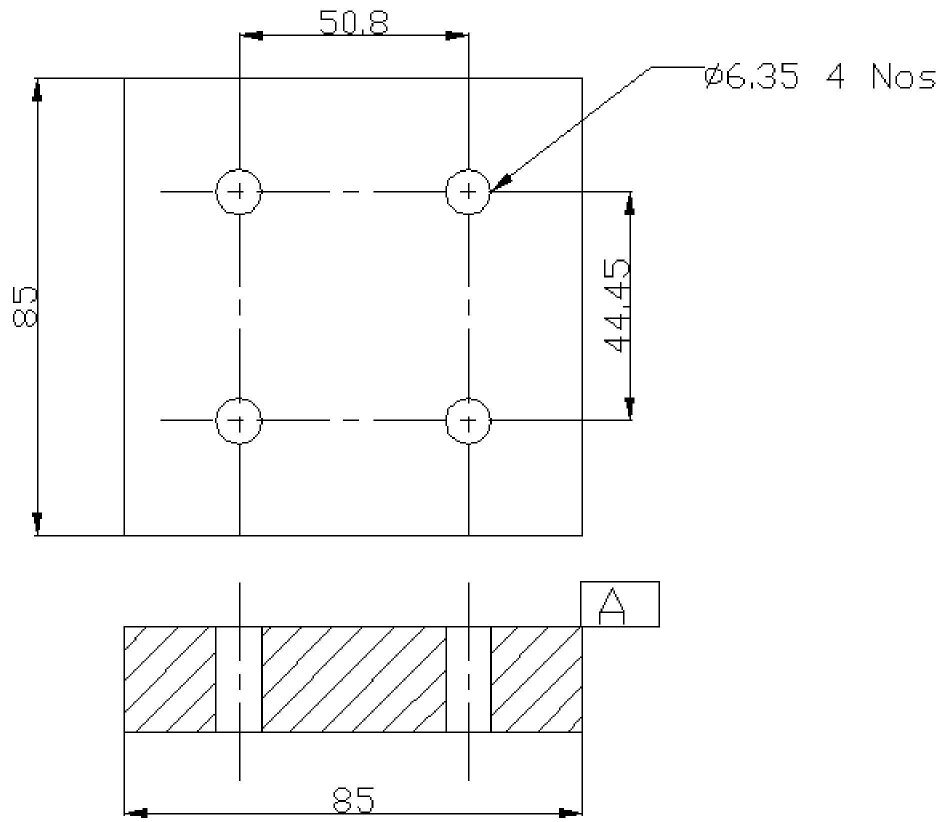


Fig.7 Jig Plate

The figure 7 shows the blue print drawing of jig plate. In this jig plate four holes are to be drilled by means of a machining technique. The axis positions as well as the diameter of the four holes of the component are measured.

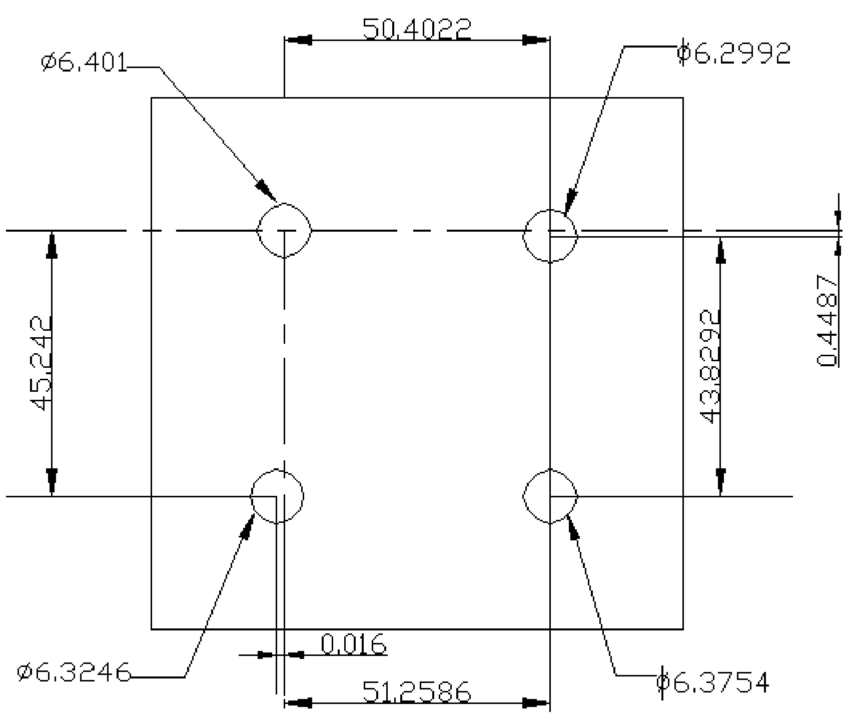


Fig.8 Component Dimensions Of Jig Plate

The figure 8 shows the component dimensions of the first sample of the jig plate. All the hole size lie within the limit (upper and lower limit).

Upper limit = 6.4262 mm

Lower limit = 6.2738 mm

To verify whether the hole position are within the true position tolerance zone, PAPER LAYOUT GAUGING technique is used. The maximum material condition of the (MMC) of the jig plate is arrived as 0.1778 mm. That is when the jig plate is having the maximum material, the sizes of the holes are minimum. It is the maximum material condition. In this state there is no bonus tolerance for the true position tolerance zone.

The true position tolerance zones of the holes are drawn on the layer. The positions of these tolerance zones are the blue print drawing dimensions. The layer is named as 'MASTER'. The centers of the holes on the component have been drawn on a different layer by name 'COMPONENT'. The Master layer remains stationary and the Component layer is moved or rotated to bring the centers of the holes within their respective true position tolerance zones. The component layer is moved in x and y directions, so that all the four component hole centers has been brought within their respective true position tolerance zones.

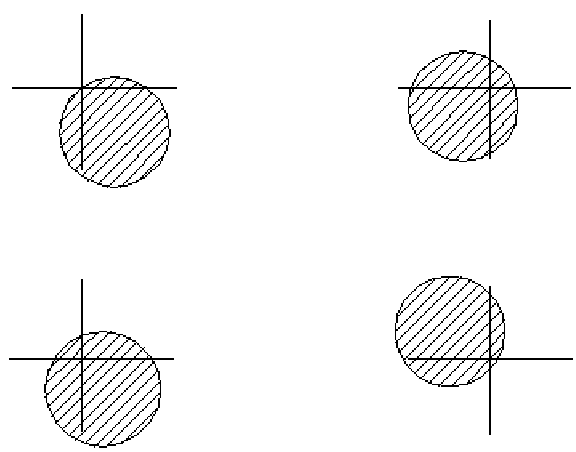


Fig.9 Hole Centres Within Tolerance Zone

Here, in this case the component dimensions are conformed with the blue print drawing dimensions and the component is ACCEPTED.

Compound Assembly

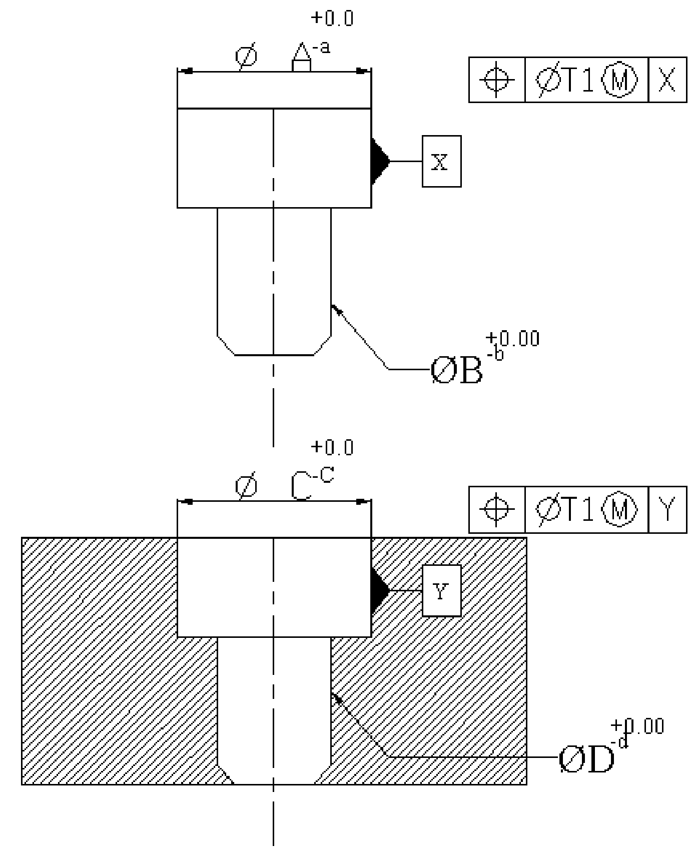


Fig.10 Gauge Pin And Hole Assembly

The compound assembly in the figure shows the pin and the hole. The gage pin is of dia A and its peg of dia B should be placed in hole of dia C and dia D. The datum face X on A coincides with datum face Y on C. The axis passes through the centre line of pin and hole. The datum X and Y both have the true position tolerance with tolerance T1 and T2 respectively at MMC.

Let us consider the worst-case condition of assembly. The position of the peg is offseted or tilted in the opposite manner in pin and hole. The worst-case analysis is done to calculate the maximum misalignment in the assembly.

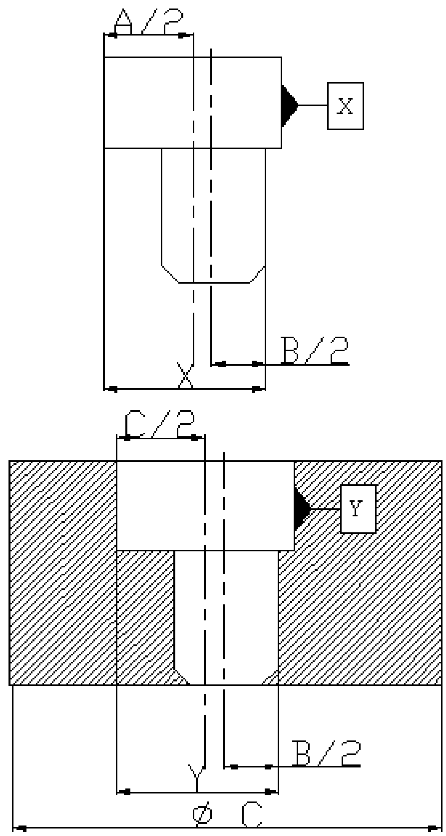


Fig.11 Tolerances in Hole and Pin

Peg B is offset by a distance T1/2 in positive direction and hole D is offset by a distance T2/2 in negative direction from the central axis. By calculating the tolerance values the maximum misalignment can be,

$$A1/2 + T1/2 + B/2 \leq C/2 - T2/2 + D/2$$

$$= \quad A + T1 + B \leq C - T2 - D$$

Rearranging,

$\mathbf{T1+T2+W_C = (C-A) + (D-B)}$ **,**

Where W_c=Working clearance

Which means, the maximum misalignment is equal to the sum of tolerances.

TRUE POSITION THEORY

Tolerances Of Position

Tolerances Of Position State The Permissible Variation In The Specified Position Of Feature In Relation To Some Other Feature Or Datum.

Tolerances of position refer to true position, concentricity and symmetry.

The course of the discussion on positional tolerancing, more detail on maximum material condition, datums, basic dimensions, and the interrelationship of positional and form tolerancing will be introduced.

Position tolerances involve features of size and relationships of centerlines, centre planes, axes, etc. At least two features are required, one of which is a size feature, and before position tolerancing is valid. Where function or interchangeability mating part features is involved, the MMC principle may be introduced to great advantage. Perhaps the most widely used and best example of the application of this principle is true position.

Of true position and the maximum material condition concept provide some of major advantages of the geometric tolerancing system.

Definition:

True position is a term used to describe the perfect (exact) location of a point, line, or plane (normally the center) of a feature in relationship with a datum reference or other feature.

True Position Tolerance

A true position tolerance is the total permissible variation in the location of feature about its true position. For cylindrical features (holes and bosses) the true position tolerance is the diameter (cylindrical) of the tolerance zone within which the axis of the feature must lie, the centre of the tolerance zone being at the True position. For other features (slots, tabs, etc.) The true position tolerance is the total width of the tolerance zone within which the center plane of the feature must lie, the center plane of the zone being at the true position.

True Position Theory

We shall now examine the true position theory as typically applied to a part for purposes of function or interchangeability. As a means of describing this theory we shall first compare the true position system with the bilateral or coordinate system.

Imagine a part with four holes in a pattern, which must line up with a mating part, to accept screws, pins, rivets, etc. to accomplish assembly, or four holes in a pattern to accept the pins, dowels, or studs of a mating part to accomplish assembly.

The top figure at the right shows tile part with a hole pattern dimensioned and toleranced using a coordinate system. The bottom figure shows the same part dimensioned using the true position system. Comparing the two approaches, we find the following differences:

1. The derived tolerance zones for the hole centers are square in the coordinate system and round in the true position system.
2. The hole center location tolerance in the top figure is a part of the coordinates (the 50.000mm and the 45.000mm dimensions). In the bottom figure, however, the location tolerance is associated with the hole size dimension and is shown in the feature control symbol at the right. The 50.000mm and 45.000mm coordinates are retained in the true position application, but are stated as BASIC or exact values.

For this comparison, the 0.126mm square coordinate tolerance zone has been converted to an equivalent 0.172mm true position tolerance zone. The two tolerance zones are super- imposed on each other in the enlarged detail.

The black dots represent possible inspected centers of this hole on eight separate piece parts. We see that if the coordinate zone is applied, only three of the eight parts are acceptable. However, with the true position zone applied, six of the eight parts appear immediately acceptable.

The true position diameter shaped zone can be justified by recognizing that the .007 diagonal is unlimited in orientation. Also, a cylindrical hole should normally have a cylindrical tolerance zone.

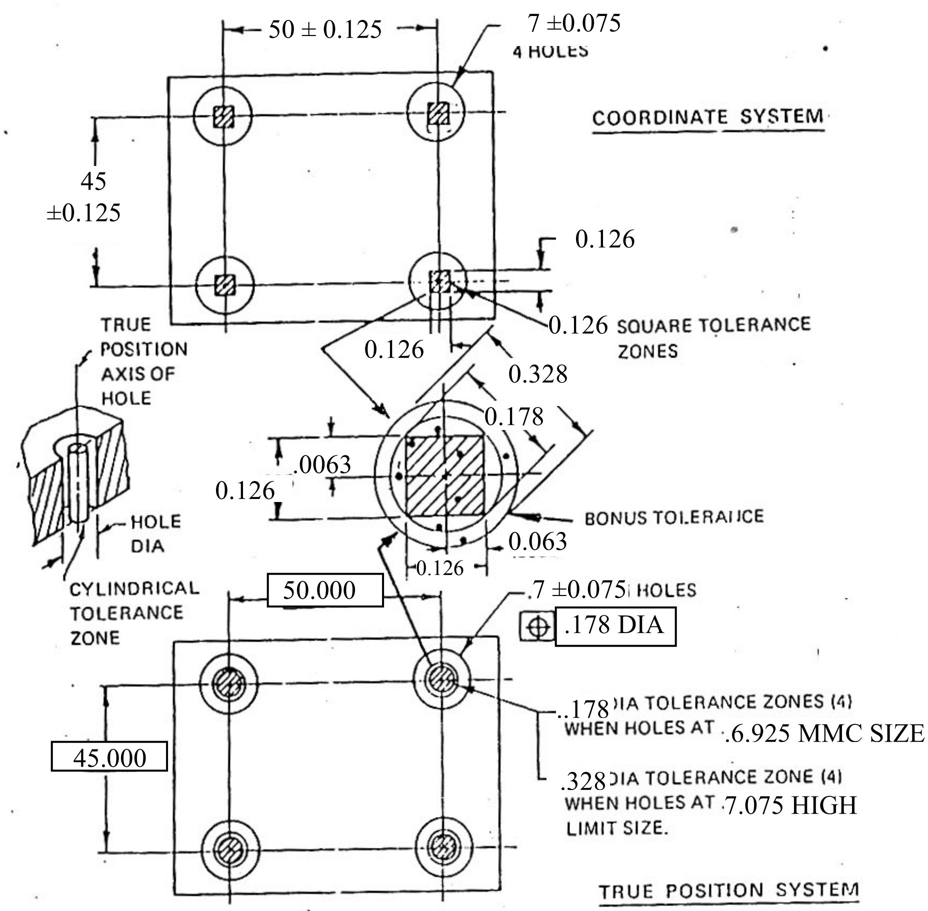


Fig-1 Comparision of Co-ordinate and True Position Systems

Closer analysis of the representative black dots and their position with respect to desired exact location clearly illustrates the fallacies of the coordinate system when applied to a part such as that illustrated.

Dot in the upper left diagonal corner of the square zone, and the dot on the left side of the square zone are in reality at nearly the same distance from the desired hole center. However, in terms of the square coordinate zone, the hole on the left unacceptable by a wide margin, whereas the upper left hole is acceptable.

Then, that a hole produced off center under the coordinate system has greater tolerance if the shift is on the diagonal, and not in the horizontal or vertical section.

Realizing that the normal function of a hole relates to its mating feature in any direction (i.e., a hole vs. a round pin). We see that the square zone restriction seems unreasonable and incorrect. Thus the true position tolerance zone, which recognizes and accounts for unlimited orientation of round or cylindrical features as they relate to one another, is more realistic and practical.

In normal applications of true position principles, the tolerance is derived of course, from the design requirement, not from converted coordinates. (The maximum material sizes of the features (hole and mating component) are used to determine this tolerance).

Thus the 0.178mm true position tolerance of Fig. 1 would normally be based on the MMC size of the hole (6.025mm). As the hole size deviates from the MMC size, the position of the hole is permitted to shift off its "true position" beyond the original tolerance zone to the extent of that departure. The "bonus tolerance" of 0.328mm illustrates the possible true position tolerance should the hole be produced, for example, to its high limit size of 7.075mm

Although we have considered only one hole to this point in the explanation the same reasoning applies to all the holes in the pattern. Note that true position tolerancing is also a non-cumulative type of control in which each hole relates to its own true position and no error is accumulated from the other holes in the pattern.

True position tolerancing is usually applied on mating parts in cases where function and interchangeability are the considerations. It provides greater production tolerances, ensures design requirements, and provides the advantages of functional inspection practices as desired.

Functional gaging techniques. Familiar to a large segment of industry through many years. of application, are fundamentally based on the true position concept. It should be clearly understood, however, that functional gages are not mandatory in fulfilling true position requirements.

Functional gages are used and discussed in this text for the dual purpose of explaining the principles involved in true position tolerancing and also to introduce the functional gage technique as a valuable tool. A functional gage can be considered as a simulated master mating part at its worst condition.

True position, although a positional tolerance, also includes form tolerance elements in composite. For example, as shown in the illustration, ‘perpendicularity is invoked as part of the control to the extent of the diameter zone, actually as a “cylindrical” zone, for the depth of the hole. Further, the holes in the pattern are parallel to one another within the true position tolerance. Various other elements of form are included as a part of the composite functional control provided by true position tolerancing.

True Position System

The example at the right further clarifies the true position theory; two of the holes on the part shown in the previous examples are enlarged to illustrate the actual effect of feature size variation on the positional location of the features.

Fig.2 shows the two 7 ±0.075 mm holes at a MMC size (or the low limit of their size tolerance) of 6.925 mm and with their centers perfectly located in the 0.178mm diameter true position tolerance zone. The drawing illustrates the mating part situation represented by a functional or' fixed pin. The gage pins are shown undersize an amount equal to the positional tolerance of 0.178; i.e. at 6.725mm diameter. This represents the maximum permissible offset of the holes within their stated positional tolerance then the hole is at MMC size of 6.925mm.

Fig.3 shows the two 6.925mm MMC holes offset in opposite directions to the maximum permissible limits of the 0.178mm true position tolerance zone. Note that we illustrate the worst condition: the edges of the holes are tangent to the diameters of the simulated mating part or gage pins. The holes are within tolerance and, as can be seen, would satisfactorily pass the simulated mating part condition as represented by the gage pins.

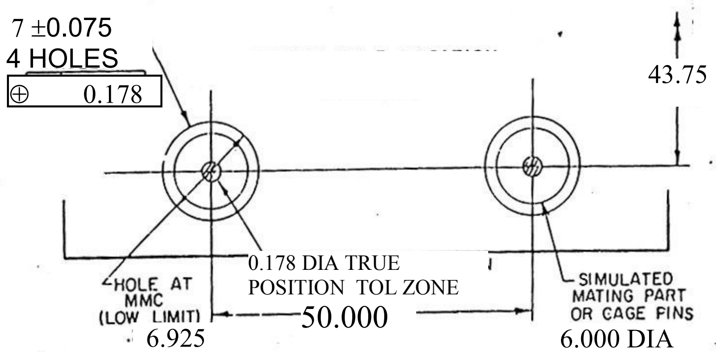


Fig – 2 Perfect hole location at MMC

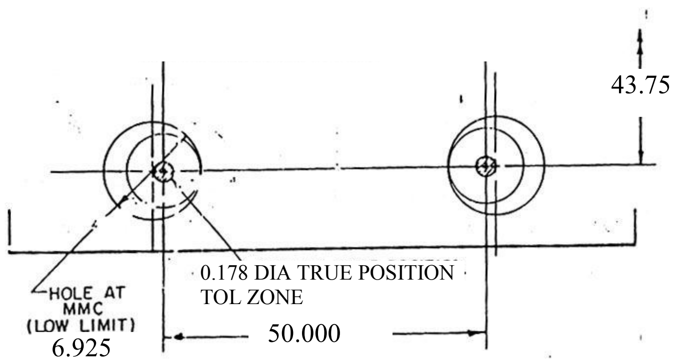


Fig – 3 Holes offset at MMC

In Fig. 4 the 7 ±0.075mm holes have been produced to the opposite, or high limit (minimum or least material condition) size of 7.075mm. It can now be seen that when we retain the same offset and tangency of the holes and mating part of the gage pins as shown in Fig - 3, the produced centers of the holes are allowed to shift beyond the original 0.178mm tolerance zone to a resulting .328 diameter tolerance zone still providing an acceptable situation.

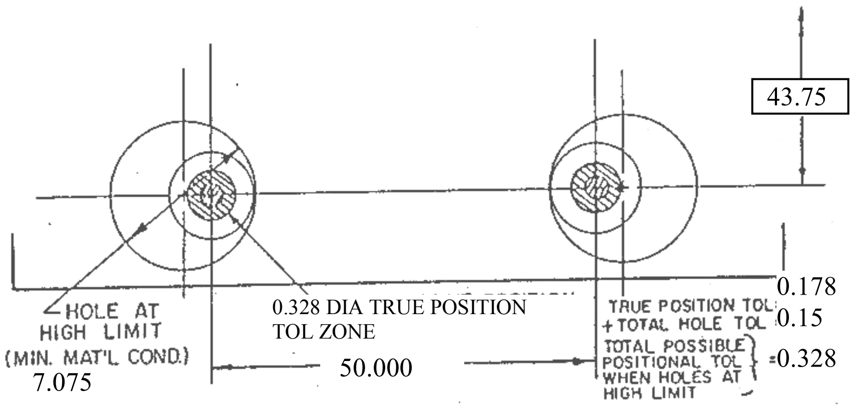


Fig -4 Holes offset at Hole high limit size (Min material condition)

The foregoing illustrates the interrelationship of size and position tolerances, which is utilized in true position dimensioning and tolerancing.

Although in this example we have used only two of the holes, the same reasoning applies to all the holes in the pattern; similarly, each individual hole could be offset within its tolerance zone in any direction around 360° and provide an acceptable situation.

It should be noted that a functional or fixed pin gage such as the one used here to explain the true position theory could be used only to check the positional location of the holes. Positional tolerance can be added as the holes increase in size or depart from MMC size within their size tolerance range. Hole size tolerance, however must be held within the tolerances specified on the drawing and must be checked individually and separately from the positional check.

The "diameter" or cylindrical tolerance zone callout has been used in this illustration and in all others in this text. Note, however, that the "radius" callout may also be used. See GENERAL RULES section, "Shape of Tolerance Zone for Positional or Form Tolerance."

Merits of True Position:

1. It represents logical tolerance zone.
2. By true position tolerancing, cumulative errors of dimensions eliminated.
3. By true position measurements not only the dimensional offsets, the tilt or angular error, which can be allowed, are measured.

$$\tan \theta = \frac{\text{Tolerance diameter}}{\text{Width of the plate.}}$$

4. The bonus tolerance can also predicted in true position tolerancing. Bonus tolerance is the extra tolerance, which is available due to variation the material condition of the hole.

Mating Parts – Floating Fastener

True position tolerancing techniques are most effective and appropriate in mating Part situations. The illustrations on page 8, in addition to demonstrating the calculations required, also emphasize the importance of decisions at the design stage to recognize and initiate the true position principles.

The mating parts shown in the illustration on page 8 are to be interchangeable. Thus the calculation of their true position tolerances should be based on the two parts and their interface with the fastener in terms of MMC sizes.
The two parts are to be assembled with four screws. The holes in the two parts are to line up sufficiently to pass the four screws at assembly. Since the four screws ("fasteners") are separate components, they are considered to have some "float" with respect to one another. The colloquial term, "floating fastener" application, has been popularly used to describe this situation.

The calculations are shown in the upper right corner of the illustration on page 8. Also, note that, in this case, the same basic dimensions and true position tolerances are used on both parts. They are, of course, separate parts and are on separate drawings.

The true position tolerance calculations are based on the MMC sizes of the holes and the screws. The maximum material basis then sets the stage for maximum producibility, interchangeability, functional gaging (if desired), etc., at production. As seen from the illustration, part acceptance tolerances will increase as the hole sizes in the parts are actually produced and vary in size as a departure form MMC. From the 0.4mm diameter tolerance calculated, the tolerance may increase to as much as .0.55mm dependent upon the actually produced hole size. It should be noted that clearance between the mating features (in this case hole and screw) is the criterion for establishing the true position tolerances.

Simultaneously with these production advantages, the design is protected since it has been based upon the realities of the hole and screw sizes as they interrelate at assembly and in their function. Thus, as parts are produced, assembly is ensured, and the design function is carried out specifically as planned.

A possible function gage is also shown in the illustration on page 8. The 4.750mm gage pin diameters are determined by the MMC size of the hole, 5.15.mm, minus the stated true position tolerance of 0.400mm. In our example, the same functional gage can be used both parts. Functional gages are, of course, not required with true position application, but they do, however, provide an effective method of evaluation where desired.

Referring to the true position tolerance calculations, if more than two parts are assembled in a floating fastener application, we must determine the true position tolerance to ensure that any two parts and the fastener will mate properly. Calculate each part to mate with the fastener using the illustrated formula and MMC sizes.

The calculation on the illustrated parts on page 8 shows a balanced tolerance application in which the total permissible true position tolerance of the holes on the two parts is the same, i.e., 0.400mm. The total true position tolerances can, however, the distributed as desired: for example, if one part specifies only 0.250mm of the 0.4mm tolerance available

for each part, 0.15mm may be added to the specified true position tolerance of the mating part.
Distribution of the tolerance may, if desired, be adjusted as shown as shown below:

(If specified on one part) → | ⌖ | 0.250DIA | ⌖ | 0.250 DIA | ← (May be specified on mating part)

Originally calculated tol	0.400	0.400
If tol assigned as above	(-) 0.250	(+) 0.150
Tol unused	0.150	0.550

The clearance holes on these parts are all specified as same size. Where they are specified <u>different</u> sizes, the total true position tolerance is equal to the <u>average</u> diametral clearance between mating holes and fasteners. As example is shown below:

MMC hole	5.000
MMC fastener	4.750
	0.250

$$\frac{0.250 + 0.4}{2} = 0.325$$

Other part

MMC hole	5.150
MMC fastener	4.750
	0.400

⌖	0.325 DIA

(specified on both parts)

Or, each part can be calculated separately for the allowable positional displacement based on the difference between the MMC of the hole and fastener. If one part in our example had 5.075 ±0.075mm – 4 holes specified, the method below would be used:

MMC hole	5.000
MMC fastener	4.750
	0.250

⌖	0.250 DIA

Other part

MMC hole	5.150
MMC fastener	4.750
	0.400

⌖	0.400 DIA

The true position tolerance calculation method illustrated assumes the possibility of a zero interference-zero clearance condition of the mating part features at extreme tolerance limits. Additional compensation of the calculated tolerance values should be considered as necessary relative to the particular application.

Formulas used as a basis for the true position floating fastener calculation are:

To calculate true position tolerance with fastener and hole size known:

$$T = H - F$$

Where T = tolerance, H = MMC hole, and F = MMC fastener

Where the hole size or fastener size is to be derived from an established true position tolerance, the formula is altered to:

$$H = T + F$$

$$F = H - T$$

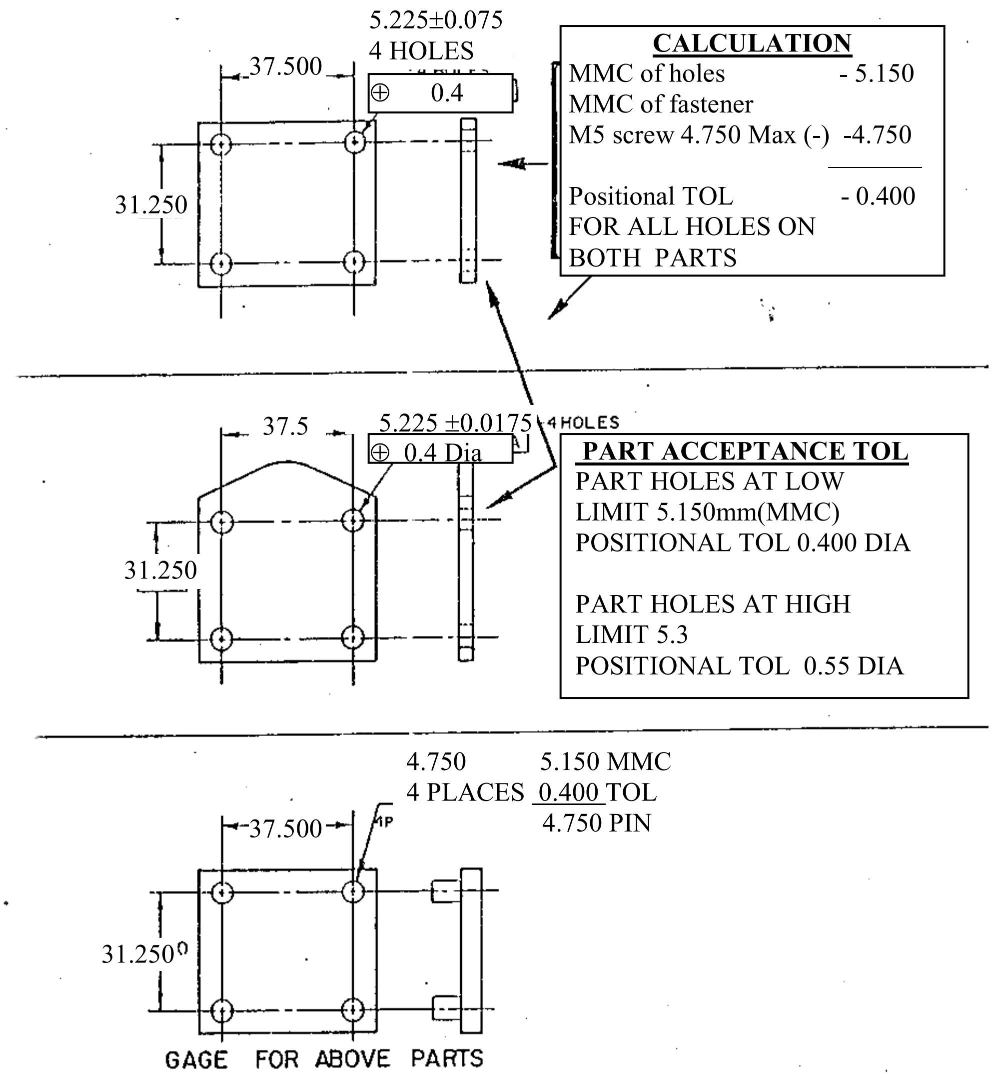


Fig- 5 Mating Parts - Floating Fastener

Mating Parts- Fixed Fastener

When one of two mating parts has “fixed fastener”, such as the threaded studs in this example, the “fixed fastener” method is used in calculating true position tolerances.

The term “fixed fastener” is a colloquialism popularly used to describe this application. Both the term and the technique are applied to numerous other manufacturing situations such as locating dowels and holes, tapped holes, etc.

The advantages of the MMC principle as described in the foregoing “floating fastener” application also apply here. However, with a “fixed fastener” application, the difference between the MMC sizes of mating features must be divided between the two features, since the two mating features must share the total true position tolerance. In this example, the two mating features (actually four of each in each pattern) are the studs and the clearance holes. The studs must fit through the holes at assembly.

Again, we see that the clearance of the mating features as they relate to each other at assembly determines the true position tolerances. When one feature is to be assembled within another on the basis of the MMC sizes and “worst” condition of assembly, the clearance, or total tolerance, must be divided for assignment to each of the mating part features. In this case, the derived 0.4 mm was divided equally, with 0.2 mm diameter true position tolerance assigned to each mating part feature (stud and hole). The total tolerance of 0.4 mm can be distributed to the two parts as desired, so long as the total is 0.4 mm (e.g., 0.250+0.150, 0.275 +0.125, etc). This decision is made at the design stage, however, and must be fixed on the drawing before release to production.

Application of the MMC principle to situation of this type guarantees functional interchangeability, design integrity, maximum production tolerance, functional gaging (if desired), and uniform understanding of the requirements.

As the part features of both parts are produced, any departure in size from MMC will increase the calculated true position by an amount equal to that departure. Thus, for example, the true position tolerance of the upper part could possibly increase up to 0.35 mm, and that of the lower part up to 0.325 mm dependent upon the amount of departure from their MMC sizes. However, parts must actually be produced and sizes established before the amount of increase in tolerance can be determined.

Functional gages (shown below each part in the illustration) can be used for checking and, although their use is not a must, they provide a very effective method of evaluation if desired. Note that the functional gages resemble, the mating parts; as a matter of fact, functional gages simulate mating parts at their worst condition.

The functional gage pins of the upper part are determined by the MMC hole size minus the stated true position tolerance. Gage tolerances are not shown, although they may be imagined to be on the order of (4.953 + 0.005 - .000) mm for pin size, and ±0.005 mm on between pin locations. Local gage practices would prevail.

The functional gage on the lower part of the illustration contains holes instead of pins. The gage hole sizes are determined by the MMC (O.D.) size of the 10-32 pins plus the stated true position tolerance. The tolerances are similar to those of the above pin gage.

Tolerances on the order of (4.948+. 0002 - .005) mm for hole size, and ±0.005 mm between holes could be applied, depending on local gage practices.

It should be noted that the term MAJOR DIA is used beneath the true position callow on the lower part. In the absence of a this special notation of exception, ANSI Y14.5 Rule 4 would have invoked the tolerance on the basis of the pitch diameter of the threads. The major diameter (or O.D.) of the thread was the desired criterion in this example.

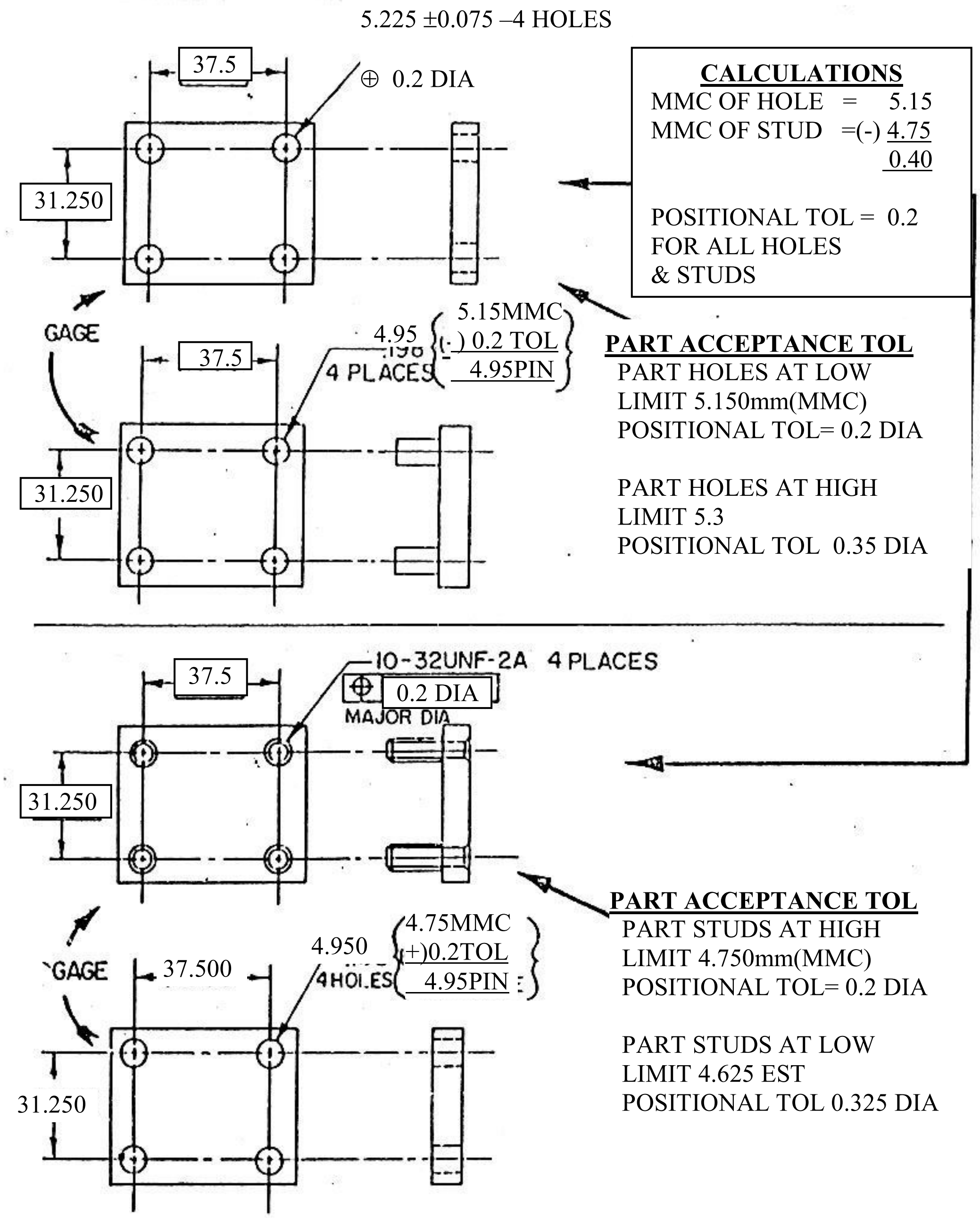


Fig- 6 Mating Parts- Fixed Fastener

The calculations on these parts illustrate a balanced tolerance application in which the total permissible true position tolerance of the two parts is equally divided, for example, 0.200mm on each part. The total true position tolerance can, however, be distributed as desired, as discussed earlier.

If more than two parts are assembled in a fixed fastener application, each part containing clearance holes must be calculated to mate with the part with the fixed features.

The true position tolerance calculation method illustrated assumes the possibility of a zero interference – zero clearance condition of the mating part features at extreme tolerance limits. Additional compensation of the calculated tolerance values should be considered as necessary relative to the particular application.
Formulas used as a basis for the true position fixed fastener (or locator) calculations are:

$$T = \frac{H - F}{2}$$

MMC hole = H
MMC fastener = F
(Or pin, dowel, etc)
Tolerance = T

Where the hole size or fastener (or pin, dowel, etc) size is to be derived from an established tolerance, the formula is altered to:

$$H = F + 2T$$
$$F = H - 2T$$

Mating Parts – Fixed Fastener

This illustration shows true position tolerancing applied to two mating parts with a round hole pattern. The same reasoning applies here as in the preceding examples except that the basic dimensions are angular (45° angles, 8 places) and a diameter (the 37.500 mm diameter).

These two parts again are of the fixed fastening type, the studs of the lower part being the fixed elements. To determine the positional tolerances for each part, the MMC of the hole and the MMC of the stud are used to determine the total positional tolerance. This is divided by two to give the positional tolerance value for each part. The total value may be divided as desired, as previous described.

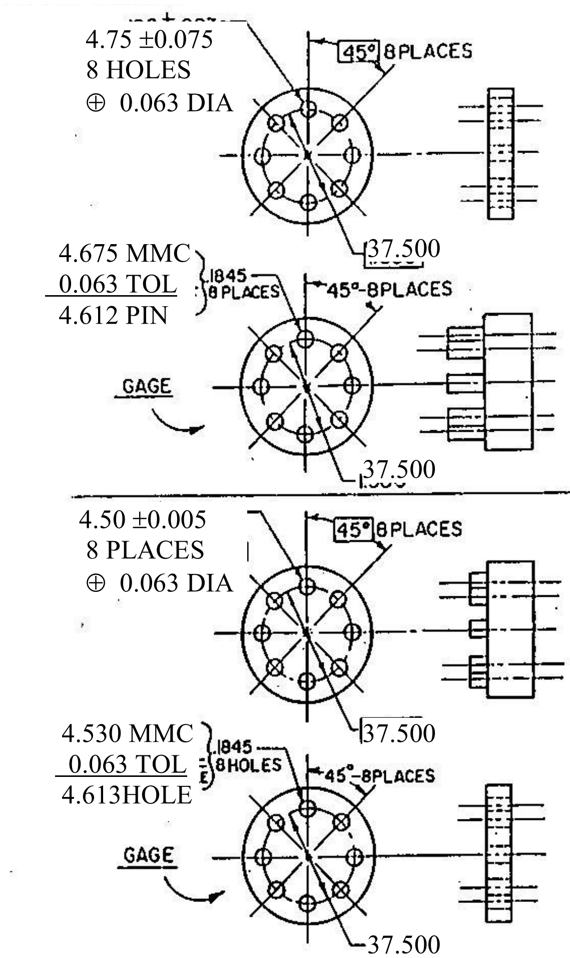
Note again how the positional tolerance increases as the holes in the upper part and the studs in the lower part depart from their MMC sizes, that is, when the holes get larger and the pins get smaller during the production process.

Functional gages are shown in the illustration for parts. Note that the pins in the upper gage are calculated to the MMC or low limit of the holes in the part (which is 4.675 mm in this case) minus the positional tolerance (.063 mm), resulting in the 4.612 mm gage pin size.

The lower gage is calculated in reverse, using the MMC or high limit of the studs, 4.550 mm plus the positional tolerance, resulting in the 4.612 mm gage hole size.

These calculations illustrate a balanced tolerance application in which the total permissible true position tolerance of the two parts is equal divided, for example, 0.063 mm on each part. The total true position tolerance can, however, be distributed as desired, for example, 0.050 mm on one part, 0.075 mm on the other, etc., so long as it totals the tolerance calculated (in this case 0.125 mm).

The true position tolerance calculation method illustrated here and in preceding examples assumes the possibility of a zero interference – zero clearance condition of the mating part features at extreme tolerance limits. Additional compensation of the calculated tolerance value should be considered as necessary relative to the particular application.



CALCULATIONS

MMC OF HOLE = 4.675
MMC OF STUD =(-) 4.550
0.125

POSITIONAL TOL =0.063
FOR ALL HOLES
& STUDS

PART ACCEPTANCE TOL

PART HOLES AT LOW
LIMIT 4.675mm(MMC)
POSITIONALTOL=.0625DIA
PART HOLES AT HIGH
LIMIT 4.825
POSITIONAL TOL 2.125DIA

PART ACCEPTANCE TOL

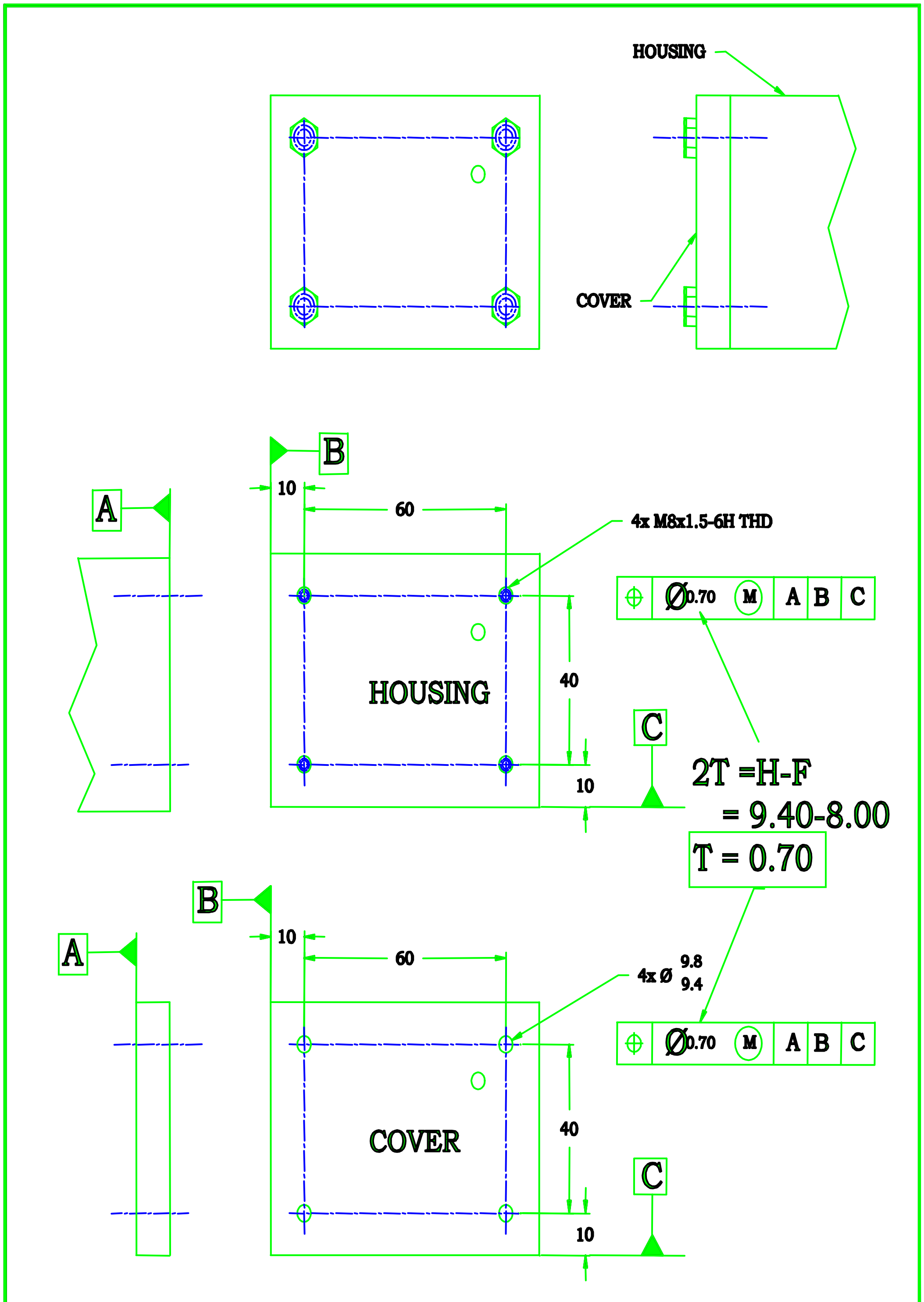
PART STUDS AT HIGH
LIMIT 4.550mm(MMC)
POSITIONAL TOL= .063DIA

PART STUDS AT LOW
LIMIT 4.450
POSITIONAL TOL 0.163 DIA

Fig- 7 Mating Parts- Fixed Fastener

Example : 1

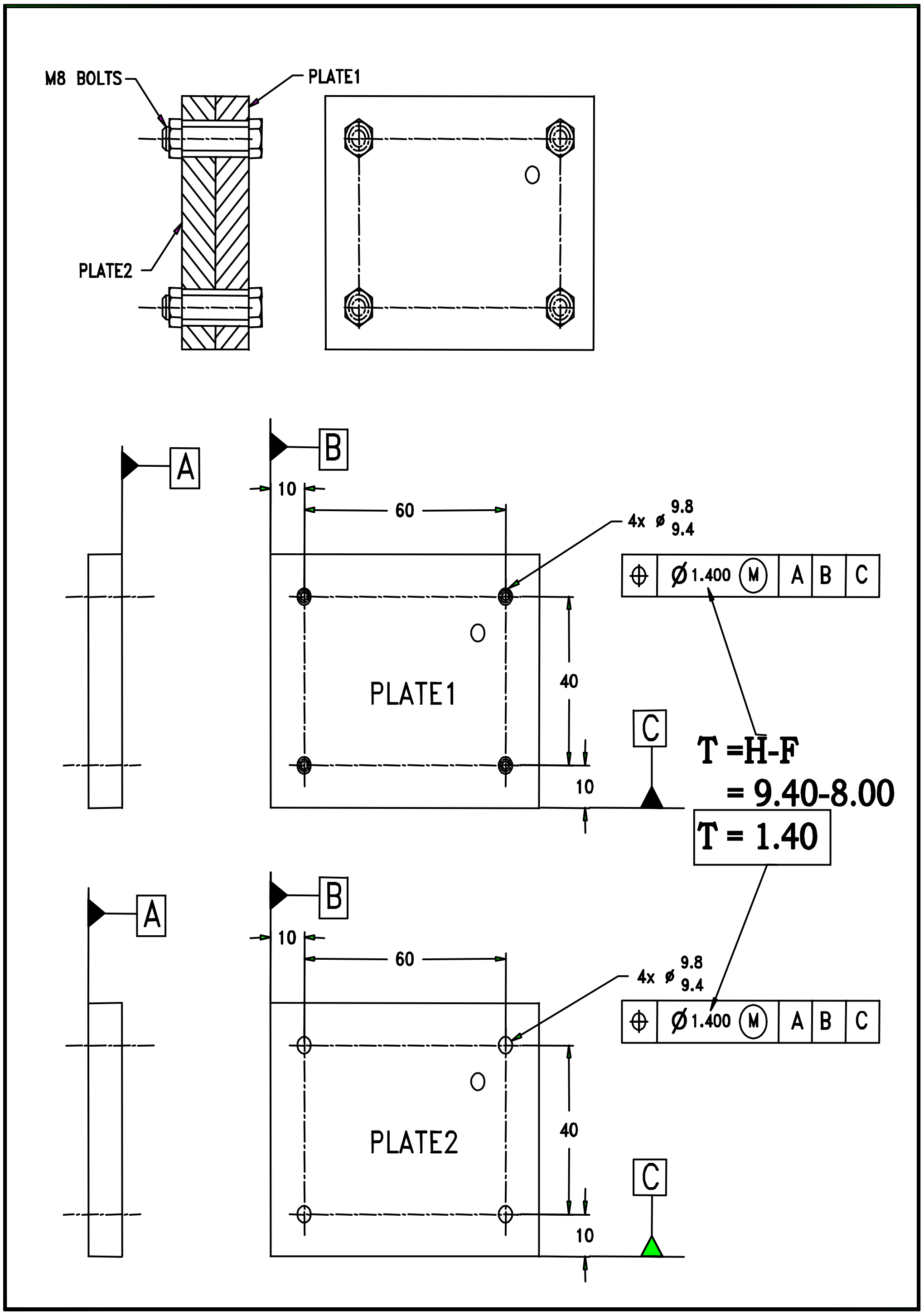
Using the fixed fastener formula calculate the position tolerance Values for the clearance holes & the tapped holes.



Example: 2

`

Using the floating fastener formula, calculate the position tolerance values for the clearance holes in both the parts.
(Assume the MS fasteners to be perfect)



MMC With Respect To A Center Plane And Related To A Datum Feature

True position relationships are normally associate with round holes or features and establish a cylindrical tolerance zone around theoretically exact axes. The cylindrical tolerance zone is not applicable to slots, dial markings, tabs, etc., for which noncumulative tolerance and MMC aspects of true position may also be desired.

Such features may be allowed to vary with respect to a true position center plane rather than an axis. The true position tolerance zone is a total wide zone with one half the total tolerance assigned to each side of the true position center plane.

In this example, we present two mating parts in order to illustrate the calculations and relationships. The top part could be either a thin metal part or a type of drive shaft with three tab projections. The mating part below might be a sleeve or collar, which must fit the upper part. The side views are not shown.
Both parts have corresponding datum reference diameters, which are related, in turn, to the true position features of each part. The datums are identified by the letter A in the datum identification symbol. The true position feature control symbol for the top part (Example 3) reads, "these features (3 tabs) must be at true position with respect to datum A (at MMC) with in 0.150 mm total wide zone with the feature at MMC size." Although the symbol used is the same as that for cylindrical zones, there is no confusion, since the drawing always clearly shows the feature being dimensioned.

The feature control symbol for the bottom part (Example 4) reads, "these features (3 slots) must be at true position with respect to datum A (at MMC) within 0.150 mm total wide zone with the feature at MMC size."

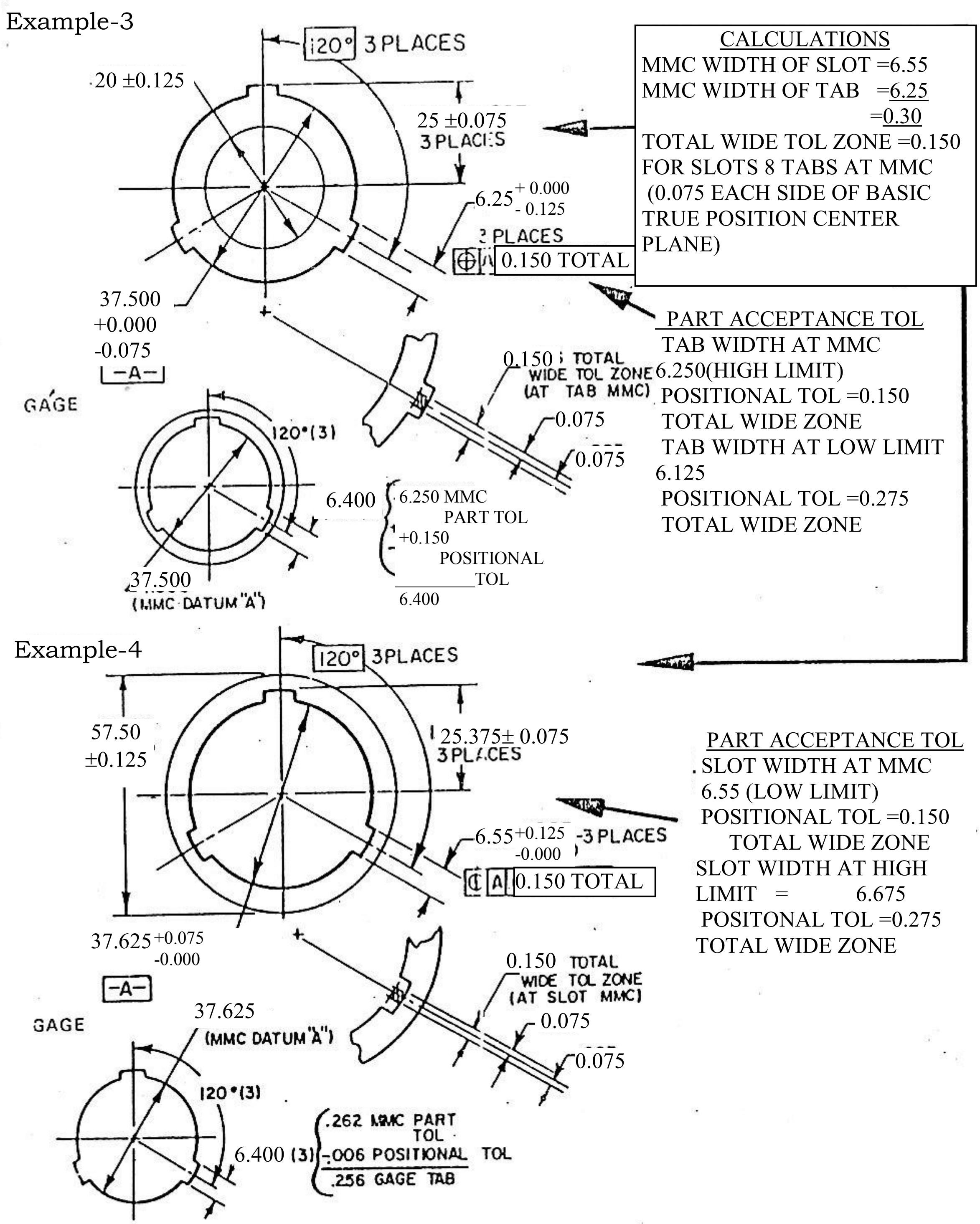
Note that the tolerance zones are <u>not</u> cylindrical but are total widths (parallelepiped) equally disposed about, and parallel to, the center plane as established by the 120 ° basic angles and extending the full depth and length of the produced feature.
The width of the tolerance zone is always total and is equally disposed on either side of the basic true position center plane. In this case, the total wide zone is 0.150 mm, with 0.075 mm each side of the basic center plane.

The calculations of the true position tolerance zone for mating parts of this type are shown at the upper right. They are based on the same reasoning as previously discussed for "fixed fasteners" using cylindrical features. The tolerance zone in this case are, however, not cylindrical.

As in any true position calculation, the MMC <u>sizes</u> of the two mating features are used to determine their individual positional tolerances. The MMC width of the tab, 6.250 mm, is subtracted from the MMC width of the slot, 6.550 mm, giving a combined clearance of 0.3 mm. This is divided by the fixed factor 2 to give the total tolerance zone for each mating part feature at MMC. As previously discussed on round feature true position calculations, the total combined tolerance (in this case 0.3 mm) may be divided as desired in other combinations, (e.g., 0.2 and 0.1, 0.175 and 0.125 mm etc).

In example 3, the notation “Part Acceptance Tolerance” indicates that the total positional tolerance zone increases from 0.150 to 0.275 mm the actually produced tab width reduces from MMC of 6.250 to 6.125 mm.

The same is true for Example 4. The slot width positional tolerance increases to 0.275 mm as the slot is produced to the high limit size of 6.675 mm simulated gages are also shown.



MMC – Non Cylindrical Part Features

This illustration shows a pair of mating parts involving non-cylindrical features. Part 1 is to fit with in the opening of part 2.

Part 1 has a width of (25 + .000), - 0.150 mm which is to fit within the (25.125 + 0.075), - .000 mm opening width on part 2. Simultaneously, the (12.500 + 0.100), -.000 mm slot on part 1 is to fit onto the (12.375 + .000 -0.075) mm projection on part 2.

The (12.500 + 0.100), -.000 mm slot on part 1 has a true position feature control symbol which states, "this feature is to be at true position with respect to datum A when datum A is at MMC size, within 0.125 mm MMC size of the feature". The width of the part is established as datum A.

Part 2 has an identical true position feature control symbol on the (12.375 + .000), - 0.075 mm dimension, and the (25.125 + 0.075), - .000 mm opening is established as datum A.

Figure 8 (a) shows the relationship of these two parts as they would appear if both parts were produced perfectly at the feature MMC sizes. Note the common center or median planes established on both parts. The parts are assembled in Fig. 8 (b).

Figure 9 (a) illustrates the slot feature on part 1 offset the maximum permissible amount of 0.063 mm at the extreme of the 0.126 mm total tolerance zone when the part is at MMC size. Also, the mating projection of part 2 (b) is shown offset in the opposite direction the maximum permissible amount of 0.063 mm at the extreme of the 0.126 mm total tolerance zone when the part is at MMC size.

Figure 9 (b) shows the assembly of the two parts. They still assemble satisfactorily. Figure 9 also emphasizes that the 0.126 mm total tolerance zone, as stated in the symbol boxes on parts 1 and 2, applies at the MMC size of the features and is the maximum tolerance permissible under this condition.

Figure 10 illustrates the increase in the permissible total true position tolerance zone as the feature sizes depart from MMC to the opposite extreme of MINIMUM (or LEAST) MATERIAL CONDITION. For part 1 (Fig .10a), with the slot at its high limit size of 12.600 mm and the datum width at its low limit of 24.850 mm, the permissible true position tolerance zone becomes 0.376 mm total or a 0.188 mm offset off the median plane of the slot with respect to the datum median plane.

For part 2 in Fig. 10 (b), with the projection at its low limit of 12.300 mm and the datum opening width at its high limit of 25.200 mm, the true position tolerance zone becomes 0.276 mm total or a 0.138 mm offset off the median plane of the projection with respect to the datum median plane.

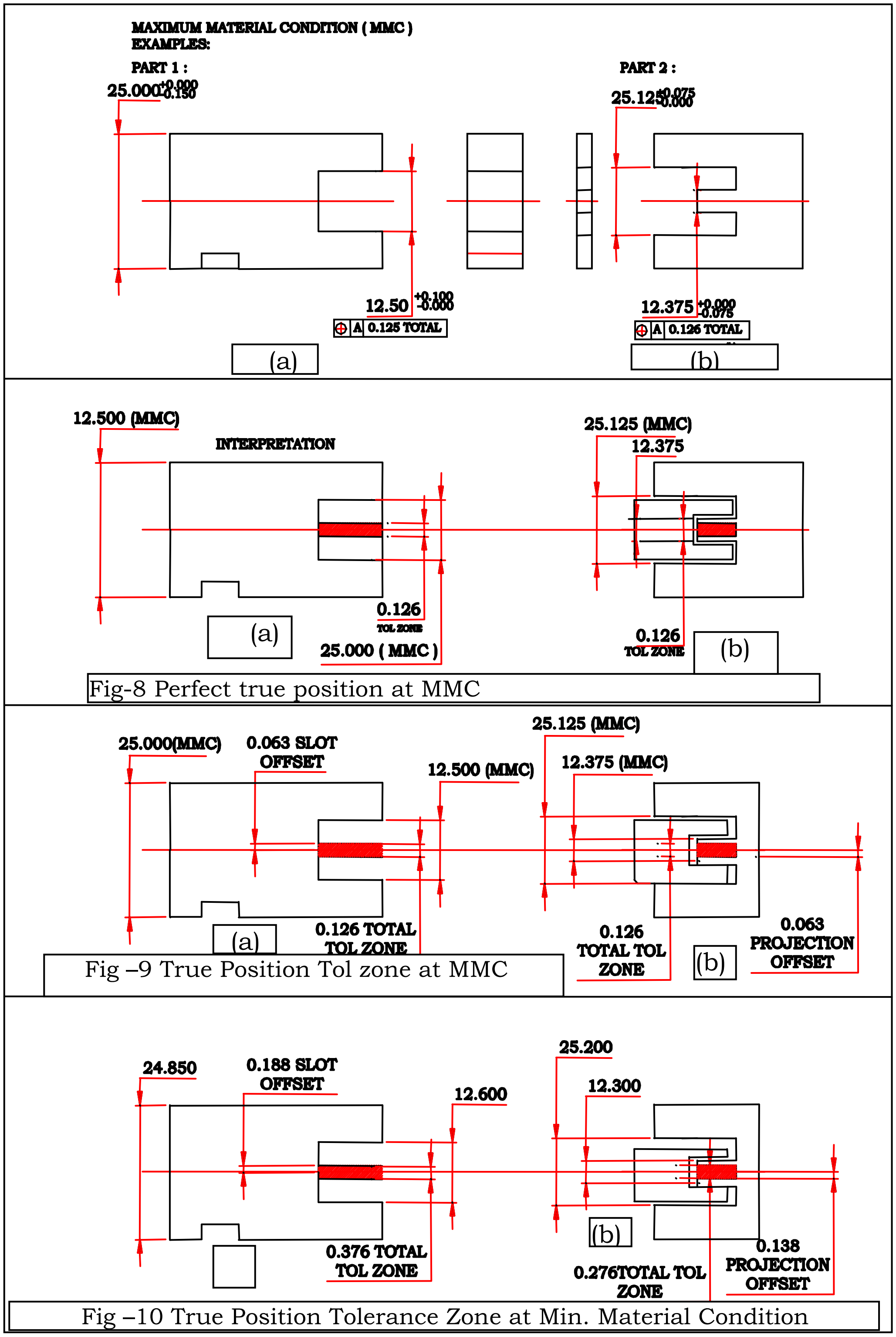


Fig-8 Perfect true position at MMC

Fig –9 True Position Tol zone at MMC

Fig –10 True Position Tolerance Zone at Min. Material Condition

Figure 10 (b) shows the assembly of the two parts under these conditions. They still assemble satisfactorily with considerably more clearance as a result of the feature size variation to size limits opposite MMC, or their MINIMUM MATERIAL CONDITION.

From this illustration it is evident that true position MMC applications permit greater tolerance and ensure a satisfactory fit of mating parts. For example, the possible tolerance on part 1 has been increased from 0.125 to 0.375 mm, and on part 2 from 0.125 to 0.275 mm. The actual tolerance to be realized is, of course, dependent upon the sizes to which the concerned features are actually produced.

MMC Calculations to Determine Tolerance

In this example we present the calculated required to determine the true position tolerance for the mating parts shown in the previous example.

Since one part is to fit within the other, the first step is to determine the clearance of the features and which feature is to receive the true position tolerance. In this case, it seemed more functional to control the true position of the slot in part 1 and the true position of the projection in part 2. The clearance of the two mating part features is to be 0.125 mm minimum. The projection on part 2 is 12.375 mm and the slot on part 1 is larger at 12.500 mm. these are MMC sizes, or the largest projection possible on part 2 and the smallest slot possible on part 1.

The width features on both parts are also given 0.125 mm clearance at MMC size of the features and are selected as the datum features for each part.

Under the subheading, TRUE POSITION TOLERANCE CALCULATIONS, the 12.375 mm MMC size of the projection o part 2 is subtracted from the 12.500 mm MMC size of the slot on part 1. This results in a difference of 12.500 mm. Next the 25 mm MMC datum projection feature of part 1 is subtracted from the 25.125 mm MMC datum slot of part 2, resulting in a difference of .125 mm.

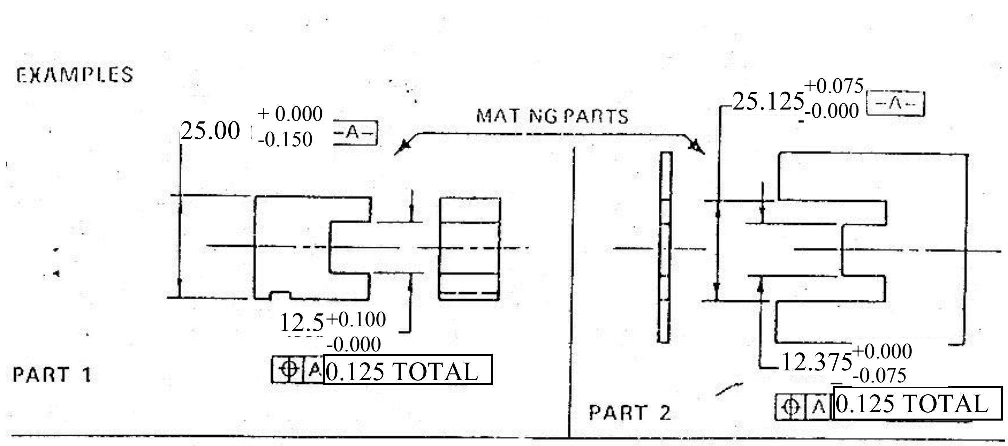
The .125 mm result of the first calculation and the 0.125 mm result of the second calculation are added to give the 0.250 mm total combined true position tolerance for both parts and their interrelated features. This total tolerance is then divided to establish the required true position tolerance on <u>each</u> individual part. How we allocate the total tolerance is optional, so long as it totals the calculated combined tolerance, in this case 0.250 mm

For the purposes of this example, the 0.250 mm total tolerance was divided evenly, with 0.125 mm selected as the true position tolerance for the 12.500 mm slot on part 1 and the 12.375 mm projections on part 2. These two figures, 0.125 plus 0.125, total 0.250 mm and comply with the 0.250 mm allowable total combined true position tolerance calculated.

Once the true position tolerance is established for both mating part features based on their relationship to each other and to common datum axes, possible extra true position tolerance for each part may be determined as shown in the lower half of the figure.

To do this calculation, we must first determine the relationship of one mating part feature to another and, then, we must consider each of these part features individually with respect to the size variations, which could occur within their size tolerances. As has been shown, the size of features affects their location tolerances, and it is this fact that makes true position tolerancing advantages, since it permits economical product with greater tolerances and ensures assembly of the mating parts.
On part 1, the permissible tolerance may be increased from 0.125 mm up to 0.375 mm and on part 2 the permissible tolerance may be increased from 0.125 up to 0.275 mm. The actual tolerance permissible in each case is, of course, dependent on the actual sizes of the features as produced.

This method of calculating true position tolerance assumes the possibility of zero clearance – zero interface fits of mating part features if all at extreme tolerance limits. It also assumes parallel orientation or permissible float of one part to the other at assembly. Additional compensation of the calculated tolerance values should be considered as necessary for any particular application or where additional datum orientation may restrict this float.



True Position Tolerance Calculations

MMC size slot (part 1) = 12.500
MMC size projection (part 2) = (-)12.375
0.125 -- -- -- 0.125
MMC size datum slot (part 2) = 25.125 (+)0.125
MMC size datum projection = (-) 25.000 0.250

(part 1) 0.125

TOTAL tolerances
To be divided as desired
To establish required true
Position tolerance on each
Individual part."can be any
Combination which total 0.250"

E.g., selected 0.125 mm for part 1 and 0.125 mm for part 2.

Extra Tolerance For Each Part

Part 1

Permissible slot true position tolerance as feature size depart from MMC:

Stated true position tolerance with slot at 12.500 MMC=	0.125
Plus total 12.500mm slot size tolerance	+ 0.100
True position tolerance with datum width at 25 mm MMC =	0.225
Plus total 25 mm datum width size tolerance	+ 0.150
Total true position tolerance with slot and datum width at Minimum material condition (largest slot, smallest datum width) =	0.375

Part 2

Permissible projection true position tolerance as feature sizes depart from MMC:

Stated true position tolerance with projection at 12.375mm MMC=	0.125
Plus total 12.375 mm projection size tolerance	+ 0.075
True position tolerance with datum opening at 25.125 mm MMC =	0.200
Plus total 25.125mm datum slot size tolerance	+ 0.075
Total true position tolerance with projection and datum opening At Minimum material condition (smallest projection, largest datum opening) =	0.275

Definitions:

Virtual Condition:

A constant boundary produced by the combined effects of the MMC Size and geometric tolerance. It represents the worst-case condition of assembly at MMC.

Virtual Hole Size:

This is the maximum size of the gage pin, which enters the hole with true position errors at maximum material condition.

Virtual Hole Size = Dia of the hole at MMC - True position error

Virtual Shaft Size:

This is the maximum size of the gage pin, which accepts the shaft with true position errors at maximum material condition.

Virtual Shaft Size = Dia of the shaft at MMC - True position error

Projected Tolerance Zone:

A projected tolerance zone applies to a hole in which a pin, stud, screw etc., is to be inserted. It controls the perpendicularity of the hole to the extent of the projection from the hole and as it relates to the mating part clearance. The projected tolerance zone extends above the surface of the part to the functional length of the pin, stud and screw relative to its assembly with the mating part.

The projected tolerance zone method prevents the condition shown in the figure-11 where interference could possibly exist with conventional true position tolerancing. The variation from perpendicularity of the bolt passing through the mating part is of concern. Therefore the location and perpendicularity of the tapped hole is of the importance in so far as it affects this extended portion of the bolt. The projected tolerance zone method figure -12 eliminates this interference.

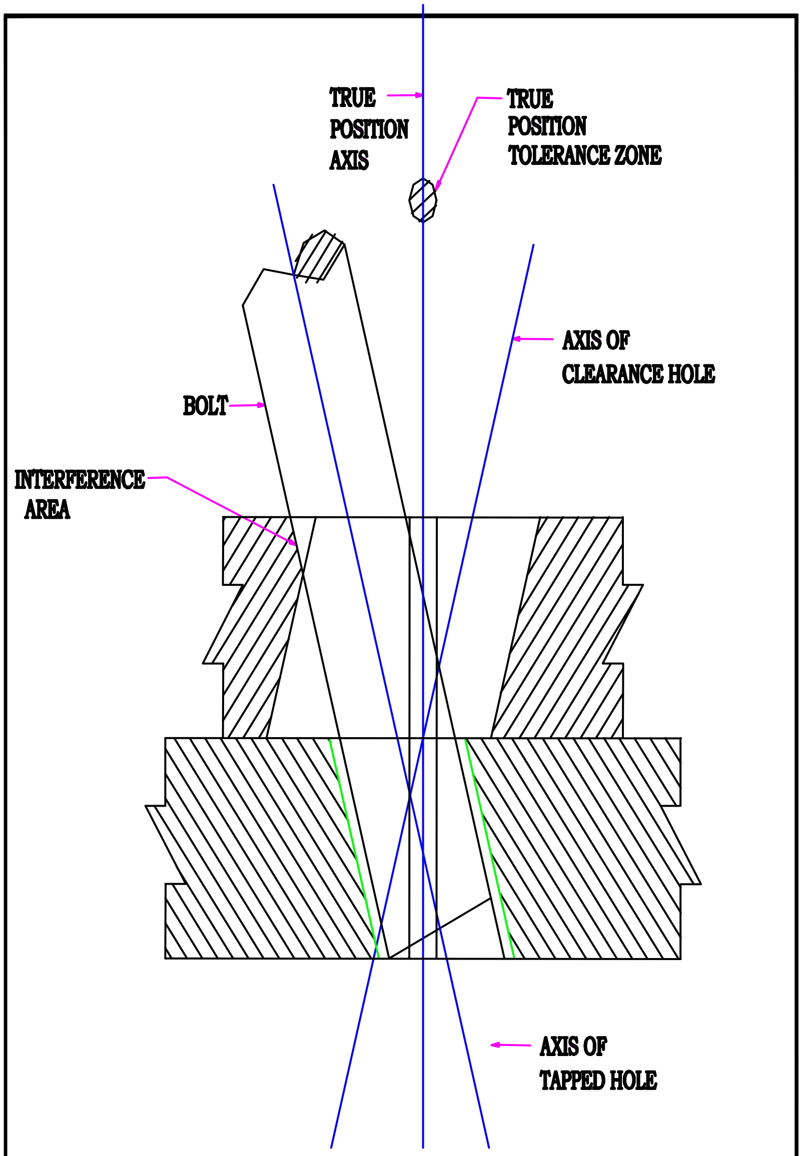


Fig – 11 Interference with conventional
True Position Tolerance

With this method, we can use conventional "fixed fastener" calculations to determine the true position tolerance. Furthermore, specifying by this method means that gaging techniques will simulate the mating part relationship, and the projected perpendicularity error will, therefore, be accounted for in the tolerance and in the gaging.

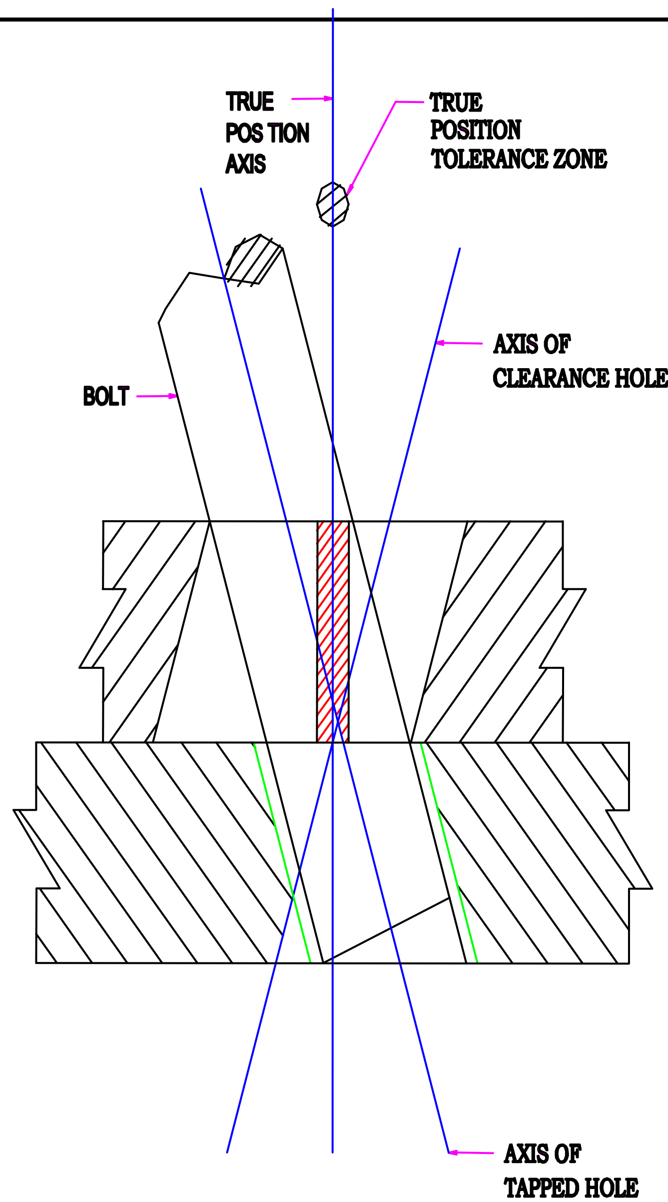


Fig –12 Elimination of interference with the Projected Tolerance Zone method

Zero True Position Tolerancing

Zero true position tolerancing is a technique adaptable to situation requiring functional interchangeability and maximum tolerance advantage in the feature size, form and position interrelationships. Where mating parts and features are simply to mate up or "GO" and tangent contact of the mating features could occur, zero tolerancing is technically acceptable.

However in some conditions, zero position tolerancing is not appropriate. For example, where specific running clearance, fit or similar special mating feature conditions are required, zero position tolerance will not, in general, be technically applicable. There are other considerations, also, which require evaluation to determine whether or not zero true position tolerancing is applicable. It is an optional method of stating many common true position mating part requirements.

True position tolerances are usually established on the basis of MMC size relationships of mating part features. The feature sizes are the criterion with which the process of developing true position tolerances starts. The designed clearance between the mating components is the basis for the true position tolerances, which are stated on the drawing and applied in the manufacture. When the features specified by the true position tolerances are actually produced, any size departure from the MMC size (ex. enlarging the size of a hole) adds to the permissible true position tolerance.

In zero true position tolerancing the same principles apply, except that the true position tolerancing stated is always a fixed "zero", with all the tolerance placed on the same dimension. This, of course, assumes that the actually produced feature will show some deviation from the MMC, which is then added to the "zero" tolerance to give a working position/form tolerance.

It can be stated that in either conventional or zero methods of true position tolerancing, size, form and position variations are considered simultaneously as a composite value. This is really the fundamental principle (along with the MMC principle) on which functional true position tolerancing is based. The reason for this is the fact that related mating part features perform their function in the space limitations provided, regardless of whether that space is derived from size, form, or position variation.

In the use of zero true position tolerancing a situation arises when a produced part with a true position hole pattern might be acceptable to a functional gauge, yet be reject able on the basis of a low limit "GO" size violation, with the result that functionally good parts might be scraped. As stated true position tolerance may be use only for form and position variations.

Conventional true position tolerancing the stated size tolerance can be used for size, form and true position variables as the feature size departs from MMC, whereas a stated true position tolerance may be used only for form and position variables. Size tolerance variation of the features from MMC size can thus add to the true position tolerance; but according to standard practices unused true position variations cannot be added to size tolerance.

The above principle is best described by referring to the CONVENTIONAL TP APPLICATION example. The notation at the bottom of the illustration states that if the hole is produced in perfect location, its size will be permitted to exceed the low limit

6.375(MMC) size down to the virtual size of 6.250. The virtual size is developed from the MMC size of the hole, 6.375, minus the stated true position tolerance, 0.125. This is, of course also the functional gage pin size, and represents the mating part feature at its “worst” condition of assembly.

Further analysis of zero tolerancing, however, reveals drawbacks that tend to temper some of its advantages.

1. For the less experienced and uninitiated user, zero tolerancing represents a psychological barrier; the zeros may give a false impression of the “perfection” expected.
2. The designer may feel that he is relinquishing excessively broad discretion to the production departments, thus abdicating design responsibilities in favor of production such as large size tolerances

In Fig.1, one of the holes illustrated in the .000 methods, example is shown with reference to the gage pin (or simulated mating part component). It is seen that the zero true position specification requires a perfect part (perfect form and perfect position) when at MMC, or virtual size.

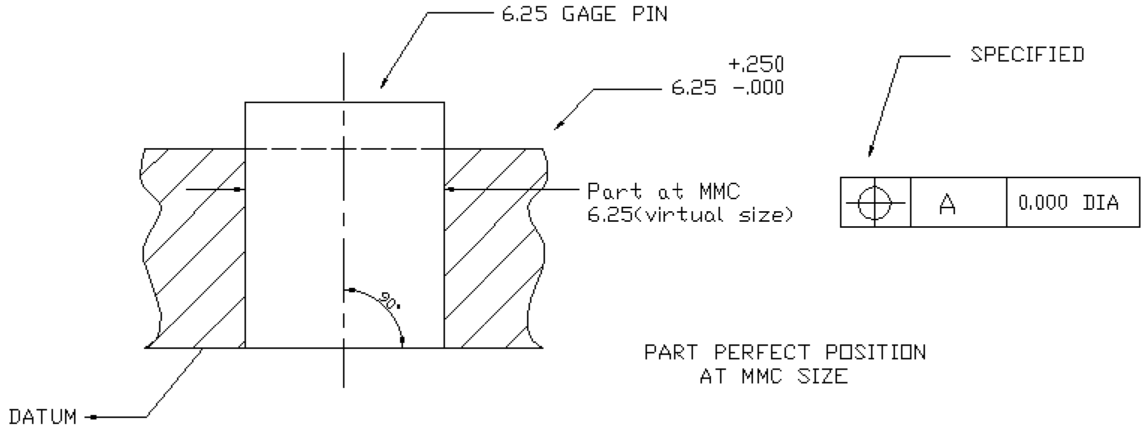


Fig.1 Zero True Position Method

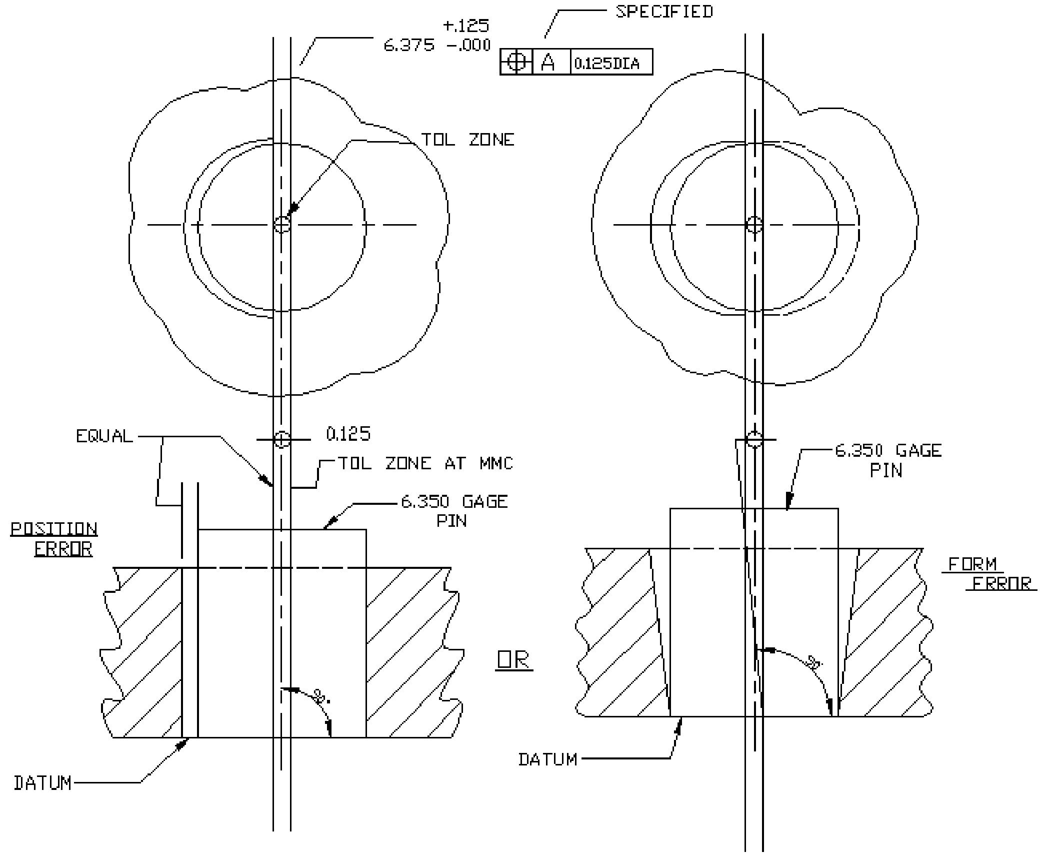


Fig. 2 Conventional Tp Method

Since there must be some clearance between the hole and the inserted mating component or they will not assemble, there is an immediate deviation from the perfect "zero - clearance - zero -interference" situation and some tolerance is acquired.

Fig 2 illustrates the "conventional" method and the established true position tolerance. The tolerance of 0.125 will permit either position or form error (or a combination of both) to this extent when the feature is at MMC. With the same size gage pin as in Fig 1, we see that true position tolerance of 0.125 plus the size tolerance of 0.005 is equivalent to the 0.250 size tolerance obtained by zero method in Fig 1.

As an example imaging $6.250^{+0.000}_{-0.038}$ on the locating dowels, and $6.300^{+0.038}_{-0.000}$ on the locating holes. Using the convention true position "fixed fasteners" method, the calculations are,

MMC size hole - 6.300
MMC size dowel - 6.250

0.050

0.025 -TP tol. On both hole and dowel

The actual true position tolerance in production on both parts would be somewhere between 0.025 and 0.050 (increase due to MMC departure). A functional gage pin size to check the holes between 6.275(hole MMC 6.300, minus TP 0.025 which gives 6.275).

Since the gage pin represents the worst condition (virtual size) of the mating dowel at 6.275, the hole size could be acceptable functionally at 6.275; yet this exceeds the stated hole size low limit.

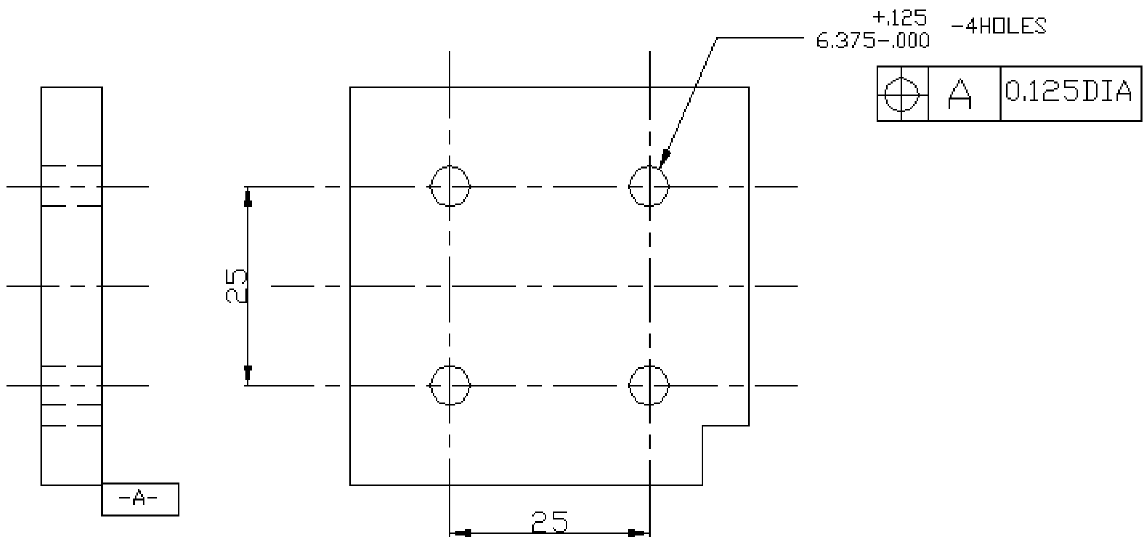
The dowel size, too, could be functional at 6.275 which represents the mating part hole at the worst condition (virtual size).this exceeds the stated dowel size high limit.

However, the 0.000 TP method can provide more total tolerance and yet guarantee proper control if stated as,

$6.275^{-0.000}_{-0.063}$ (Dowel) and $6.275^{+0.063}_{-0.000}$ (Hole)

Comparison of two methods in terms of the full tolerance range difference between the hole and dowel which determines usable size, form, and position tolerance as shown below

Conventional True Position Application Compared With .000 True Position Tolerances



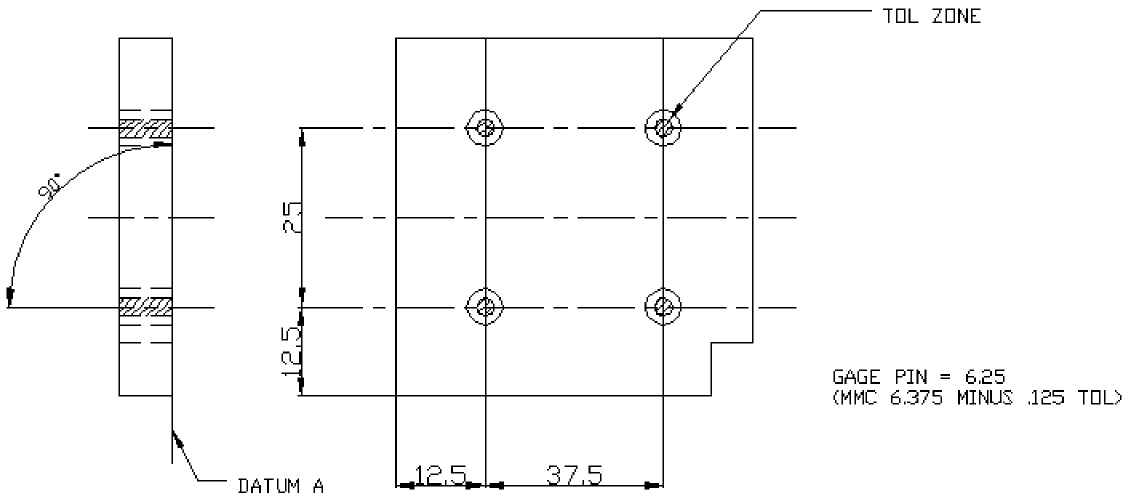


Fig. 3 True Position (Conventional) As Drawn

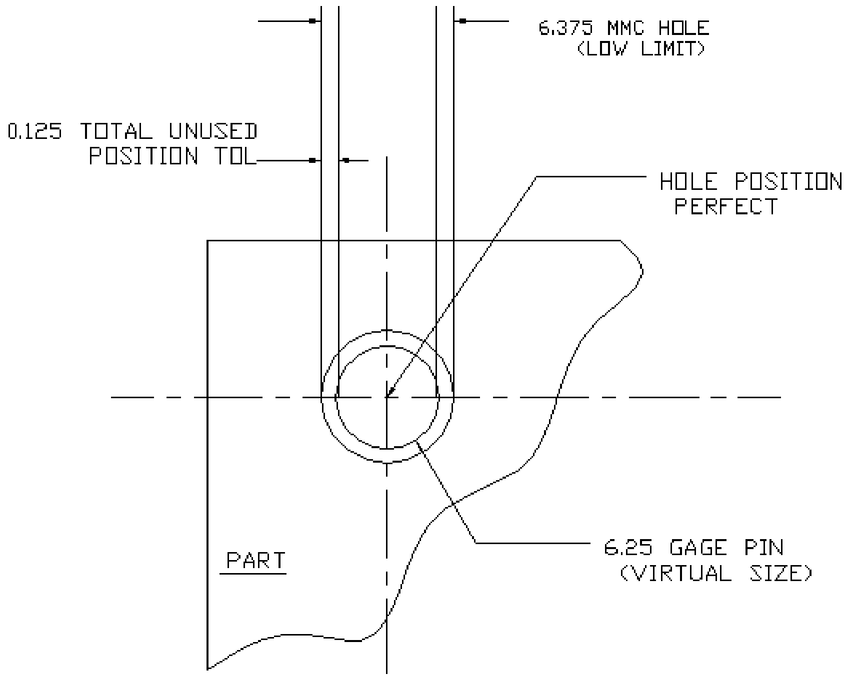
Table 1:

ACCEPTABLE TRUE POSITION TOLERANCES

ACTUAL FEATURE SIZE	TP TOL
6.375	0.125
6.400	0.150
6.425	0.175
4.450	0.200
6.475	0.225
6.500	0.250

Interpretation

Assuming The Gage Pin Represents The Worst Mating Condition, As Position Location Approaches Perfect, It Is Evident That The Hole Size Could Go Down To 6.250 (0.125 Below ,6.375 Low Limit Of Hole) And Still Pass The Gage Pins. However, Parts Below The Low Limit Hole Size Of 6.375 Would Be Rejected On Size, But They Are Good Parts.



True Position

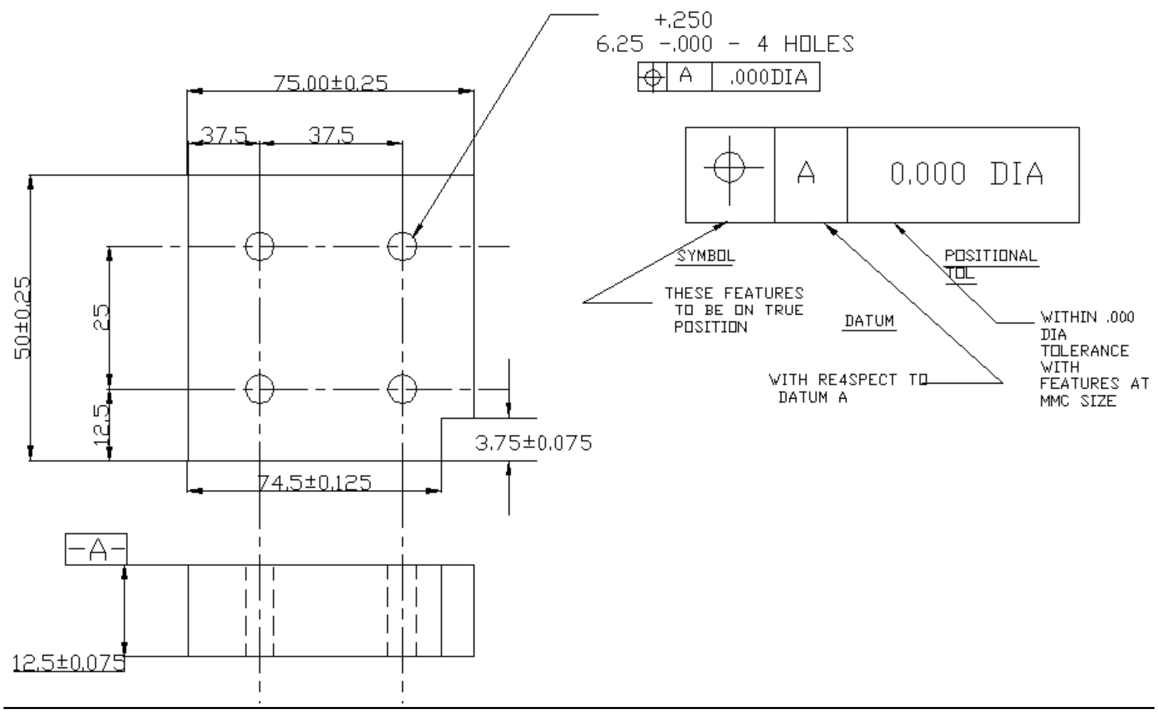


Fig.4 Zero True Position as Drawn

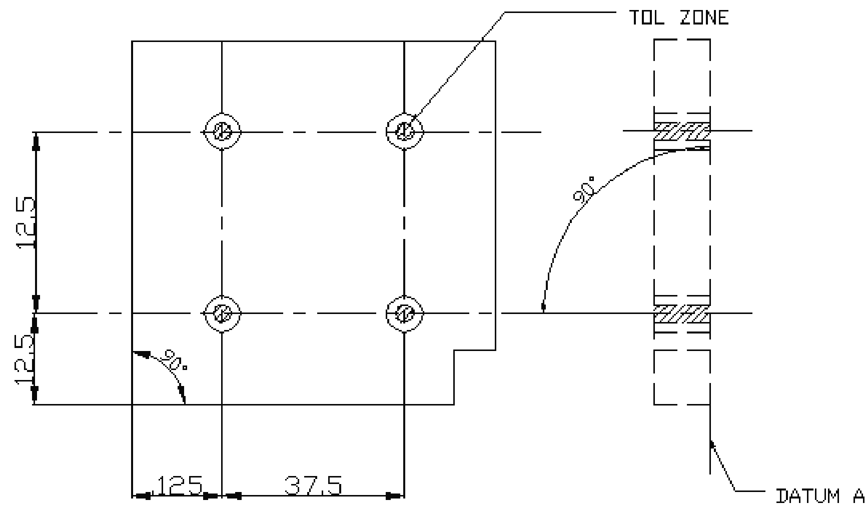


Fig.5 Interpretation

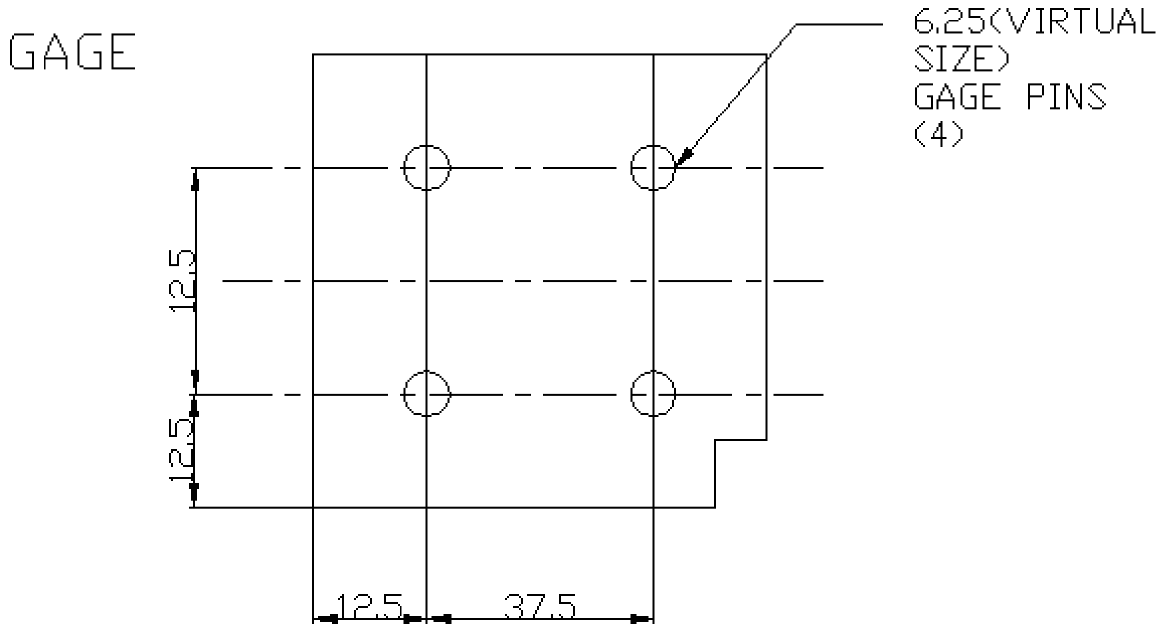


Fig.6 Virtual Size Gage

Table 2:

Acceptable Zero True Position Tolerances

ACTUAL FEATURE SIZE	TP TOL
6.250	0.000
6.275	0.025
6.300	0.050
6.325	0.075
6.350	0.100
6.375	0.125 **MMC**
6.400	0.150
6.425	0.175
6.450	0.200
6.475	0.225
6.500	0.250 **LMC**

Functional Gauge

Functional gauge is the one of the application of true position tolerancing, used for checking the functional worthiness of the component. If the sizes and the relative positions of the features are to be inspected, the most widely used method is to use the functional gauge.

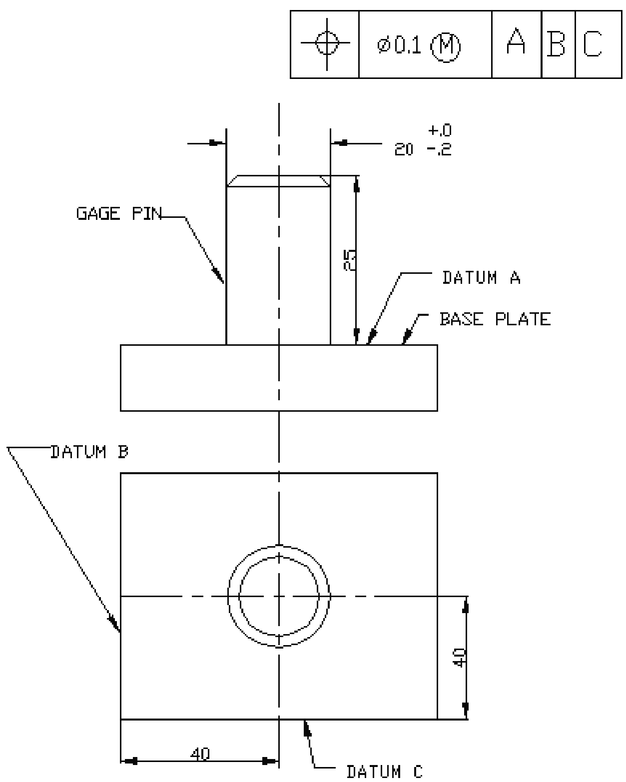


Fig.6 Functional Gauge

Fig 6 shows the functional gauge to be used for this purpose and the component to be inspected. The size of the holes and their relative positions are to be checked for acceptance.

The component is inserted over the gauge and it is passed through the gauge, the component is accepted otherwise the component is rejected.

Datum 'A' is called primary datum on which the peg should be located. Datum 'B' is called the secondary datum which represents the position of peg from certain datum generally from itself.

Now the pin size is $500^{+0.0}_{-5.0}$

For IT grade 6, the tolerance value for dia 502.1 is given by 9µm.A value of 0.125 can be obtained in jig boring machine.

The true position value is given as

$$5.025^{+0.250}_{-0.000}$$

This method is used when the inspecting quantity is large. As it is expensive to produce a functional gauge it cannot be used for job shop type production.

Paper Layout Gauging

The paper layout gauging technique has been used for inspecting the components of job shop type, one of the applications of true position tolerancing, by overlapping the component diagram with that of blue print diagram. When the inspecting quantity is less then it is best to use paper layout gauging. The term, 'paper' is used to indicate that the technique has been used to inspect the components with paper.

The technique for measurement of paper layout gauging is as follows. The tolerances are drawn in one of the transparent sheet. The tolerance zones are obtained from the blue print diagram. The component is measured and the axis of the features is marked on other layer. Now both these layers are overlapped. Both the tolerance zone and axis is visible. On overlapping if the axis of the features lies within tolerance zones, then the component is accepted, if not the paper containing component axis is moved until the axis come within their respective tolerance zones. If the component hole centre does not lie within their true position tolerance zones, it is said that the component is rejected

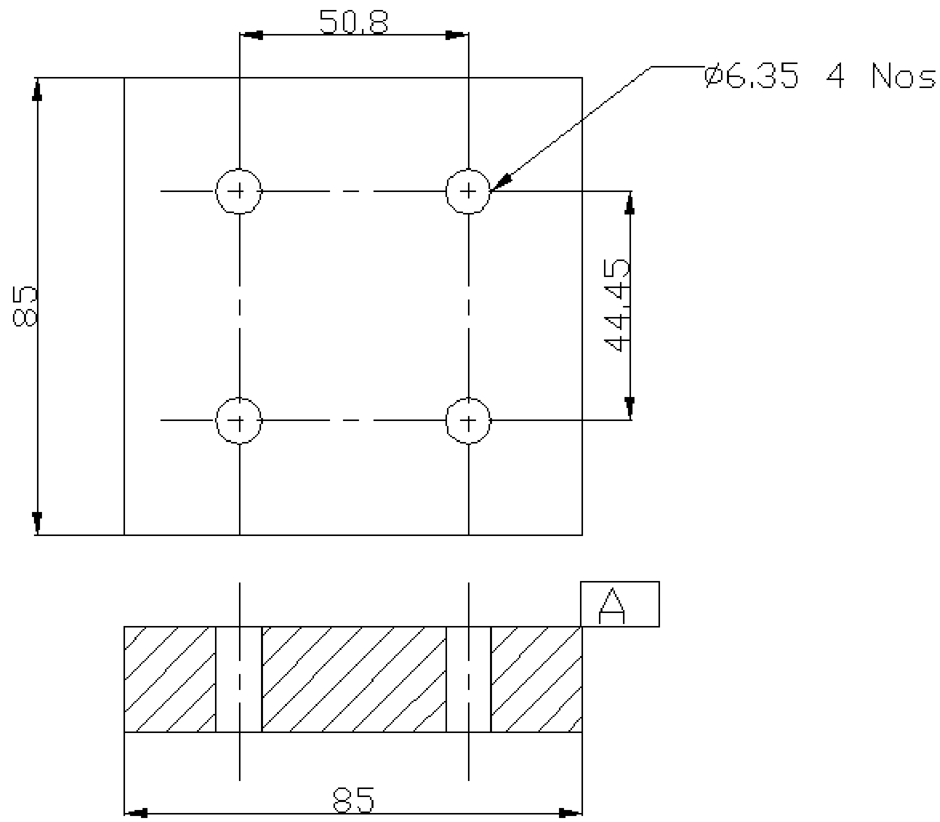


Fig.7 Jig Plate

The figure 7 shows the blue print drawing of jig plate. In this jig plate four holes are to be drilled by means of a machining technique. The axis positions as well as the diameter of the four holes of the component are measured.

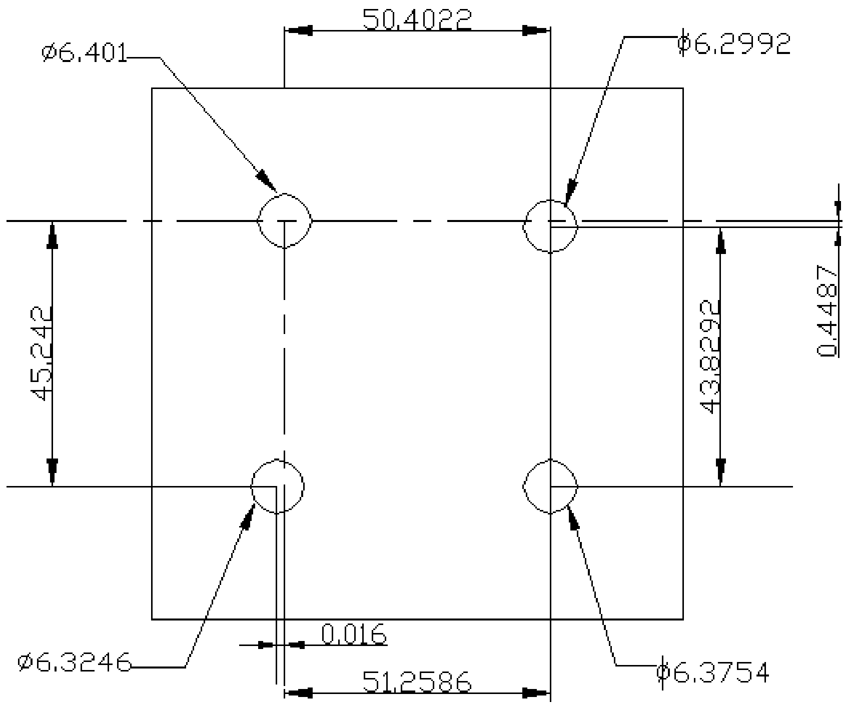


Fig.8 Component Dimensions Of Jig Plate

The figure 8 shows the component dimensions of the first sample of the jig plate. All the hole size lie within the limit (upper and lower limit).

Upper limit = 6.4262 mm

Lower limit = 6.2738 mm

To verify whether the hole position are within the true position tolerance zone, PAPER LAYOUT GAUGING technique is used. The maximum material condition of the (MMC) of the jig plate is arrived as 0.1778 mm. That is when the jig plate is having the maximum material, the sizes of the holes are minimum. It is the maximum material condition. In this state there is no bonus tolerance for the true position tolerance zone.
The true position tolerance zones of the holes are drawn on the layer. The positions of these tolerance zones are the blue print drawing dimensions. The layer is named as 'MASTER'. The centers of the holes on the component have been drawn on a different layer by name 'COMPONENT'. The Master layer remains stationary and the Component layer is moved or rotated to bring the centers of the holes within their respective true position tolerance zones. The component layer is moved in x and y directions, so that all the four component hole centers has been brought within their respective true position tolerance zones.

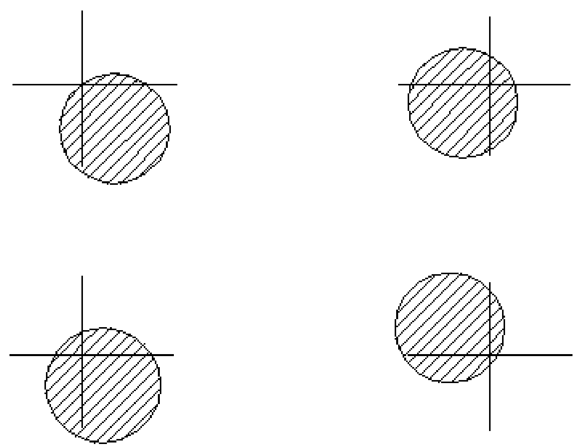


Fig.9 Hole Centres Within Tolerance Zone

Here, in this case the component dimensions are conformed with the blue print drawing dimensions and the component is ACCEPTED.

Compound Assembly

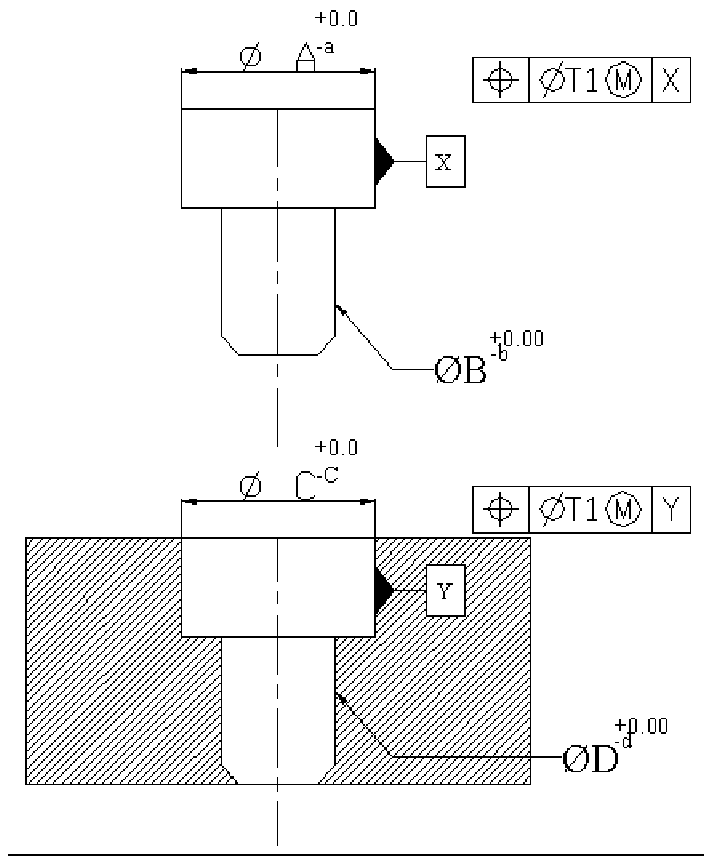


Fig.10 Gauge Pin And Hole Assembly

The compound assembly in the figure shows the pin and the hole. The gage pin is of dia A and its peg of dia B should be placed in hole of dia C and dia D. The datum face X on A coincides with datum face Y on C. The axis passes through the centre line of pin and hole. The datum X and Y both have the true position tolerance with tolerance T1 and T2 respectively at MMC.

Let us consider the worst-case condition of assembly. The position of the peg is offseted or tilted in the opposite manner in pin and hole. The worst-case analysis is done to calculate the maximum misalignment in the assembly.

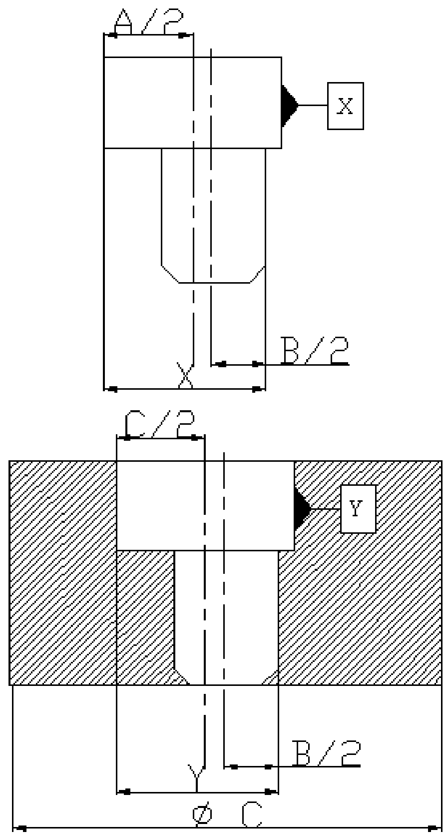


Fig.11 Tolerances in Hole and Pin

Peg B is offset by a distance T1/2 in positive direction and hole D is offset by a distance T2/2 in negative direction from the central axis. By calculating the tolerance values the maximum misalignment can be,

$$A1/2 + T1/2 + B/2 \leq C/2 - T2/2 + D/2$$
$$= \quad A + T1 + B \leq C - T2 - D$$

Rearranging,

$T1+T2+W_C = (C-A) + (D-B)$,

Where W_c=Working clearance

Which means, the maximum misalignment is equal to the sum of tolerances.

UNIT – IV Multiple Choice Questions and Answers

S. No	Question	Option a	Option b	Option c	Option d	Answer
1	What is meant by roughness?	Minute succession of hills of different height	Minute succession of valleys and hills of different height and varied spacing	Minute succession of valleys and hills of same height and same gap	Minute succession of valleys of different depth	Minute succession of valleys and hills of different height and varied spacing
2	Surfaces produced by straight and cylindrical grinding tools tend to create which type of roughness?	Regularly spaced but directional roughness	Regularly spaced but non directional roughness	Irregularly spaced but directional roughness	Irregularly spaced but non directional roughness	Irregularly spaced but directional roughness
3	In tolerance charting technique, the arrow denotes________	surface to be machined	surface to be measured	un-machined surface	none of the above	surface to be machined
4	Data's needed for tolerance charting	Process capability of processes	Stock requirements of each machining process	Stage drawing	All of these	All of these
5	Which of the following is necessary for the complete study of	Measurement of all the components of elements	Analysis of all the component element	Assessment of the effects of combined texture	Measurement and analysis of all the components	Measurement and analysis of all the components

	surface roughness?				and assessment of combined texture	and assessment of combined texture
6	Which of the following is true for measurement of surface roughness?	3 dimensional geometry can be easily measured	Direction of measurement is perpendicular to the lay	Direction of measurement is parallel to the lay	Direction of measurement is parallel to the direction of the predominant surface marking	Direction of measurement is perpendicular to the lay
7	In tolerance charting technique, the dot shows________	surface to be machined	surface to be measured	un-machined surface	none of the above	surface to be measured
8	In tolerance charting machine to dimension indicates	Indirectly achieved dimensions	Directly achieved dimensions	Raw material dimension	Finished part dimension	Directly achieved dimensions
9	How much a stylus instrument can be magnified to plot or find minute irregularities?	50 times	500 times	5000 times	50,000 times	50,000 times
10	Which of the following is true about	It is a mechanical instrument	It is an electrical instrument	It is a mechanical cum optical instrument	It is an optical instrument	It is a mechanical cum optical instrument

	Tomlinson surface meter?					
11	In tolerance charting technique,the column line no. indicates	To refer previous line	To refer next line	To refer dimensions involved	None of the above	To refer previous line
12	In tolerance charting balance dimension indicates	Indirectly achieved dimensions	Directly achieved dimensions	Raw material dimension	Finished part dimension	Indirectly achieved dimensions
13	What do you mean by dominant spacing?	Distance between successive peaks when irregularities are comparatively uniform in shape and size	Distance between middle point of successive hills when irregularities are comparatively uniform in shape and size	Distance between middle point of successive valleys when irregularities are comparatively uniform in shape and size	Distance between successive peaks when irregularities are regardless of shape and size	Distance between successive peaks when irregularities are comparatively uniform in shape and size
14	Which of the following is used for the direct measurement of surface quality and commonly used in U.S.A.?	Profilometer	Tomlinson surface meter	Talysurf	Replica method	Profilometer
15	In tolerance charting technique,the	To refer previous line	To refer next line	To refer dimensions involved	None of the above	To refer dimensions involved

	column lines involved indicates					
16	Tolerance charting the term ORR no. indicates	Operation sequence	Operation number	Operation research	None of the above	Operation number
17	Which of the following parameter is important for specifying surface roughness?	Size of irregularity	Spacing of irregularity	Height of irregularities	Height, spacing and form of irregularitie s	Height, spacing and form of irregularities
18	Which of the following is necessary for the complete study of surface roughness?	Measuremen t of all the components of elements	Analysis of all the component element	Assessment of the effects of combined texture	Measureme nt and analysis of all the components and assessment of combined texture	Measuremen t and analysis of all the components and assessment of combined texture
19	In tolerance charting technique adapted for arriving	The stage dimensions	The machining sequence	The maching allowance	Both 1 and 2	The stage dimensions
20	Tolerance charting is used to	Avoid insufficient and more material	Avoid rejection	Optimize the stock removal	All of these	All of these

UNIT V

COST ESTIMATION

Preparation of Process chart for a given component-estimation of setting time and machining time-estimation of material cost, labour cost and overhead cost based on supplied data.

Process Flow chart and process mapping

A **process flow chart** is a process analysis tool which maps out a process and its steps through a set of standardizedflow chart symbols. The process flow chart is an initial step in process re-engineering and continuous improvement/kaizen initiatives that help understand the different process steps, the sub-steps within these and the nature of these steps. It is a similar approach to value stream mapping where the value stream of a product or service is mapped from raw materials to customer distribution. Some process flow charts can be very complex and used in engineering design and plant designs, these are usually known as schematic diagrams and use a different set of symbols and provide more detail of the process.

Process Flowchart symbols

The symbols are used to represent a value adding task (a rectangle), an arrow represents a material / WIP movement, a square with a rounded side is the symbol for a delay, and an inverted triangle represents an inventory holding area. The process flowchart can contain a variety of information, depending on its intended use and complexity. It is up to the user to collect the required information to put on the flow chart.

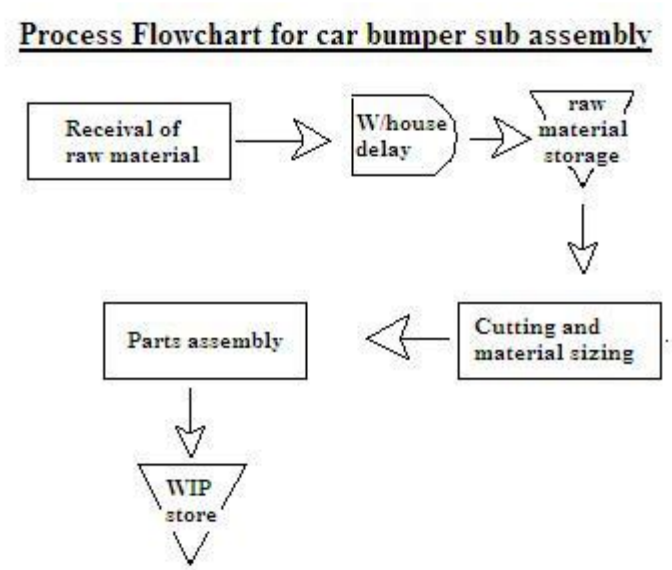
Uses of process flowcharts

The use of process flow charts are usually used to identify problems within a production process and inefficiencies. A list is presented below:

- -Understanding a productive process
- -Identifying process bottlenecks and excessive waiting periods within the process
- -Identify redundant or unnecessary process tasks
- -Identify potential improvements in labour productivity
- -Identify inefficient inventory management and unnecessary inventory costs

The items above are the main reasons why process engineers or industrial engineers involved in improving process performance will map a process or value stream with a process flow chart. Lean teams also use them for similar reasons and as a basis for their kaizen initiatives to identify sources of waste to eliminate to make any process more efficient.

Example of a process flow chart



Process Chart Symbols:

The recording of the facts about the job in a process chart is done by using standard symbols. Using of symbols in recording the activities is much easier than writing down the facts about the job. Symbols are very convenient and widely understood type of short hand. They save a lot of writing and indicate clearly what is happening.

1. Operation

A large circle indicates operation. An operation takes place when there is a change in physical or chemical characteristics of an object. An assembly or disassembly is also an operation. When information is given or received or when planning or calculating takes place it is also called operation.

Example 1.1

Reducing the diameter of an object in a lathe. Hardening the surface of an object by heat treatment.

2. Inspection

A square indicates inspection. Inspection is checking an object for its quality, quantity or identifications.

Example 1.2

Checking the diameter of a rod. Counting the number of products produced.

3. Transport

An arrow indicates transport. This refers to the movement of an object or operator or equipment from one place to another. When the movement takes place during an operation, it is not called transport.

Example 1.3

Moving the material by a trolley

Operator going to the stores to get some tool.

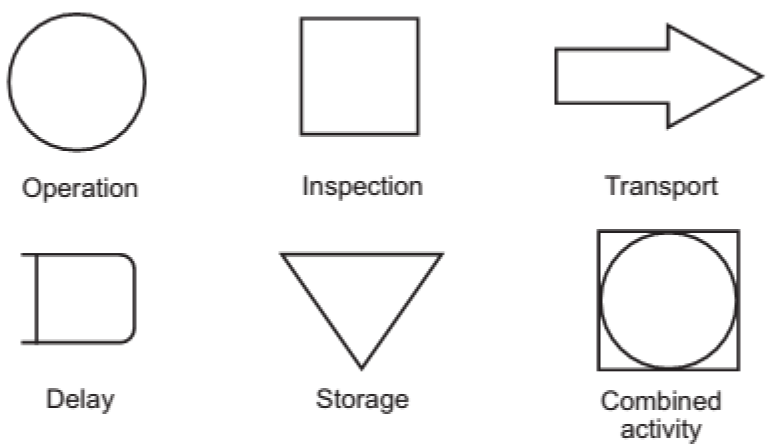


Fig. 1.1: Process chart symbols

4. Delay or temporary storage

A large capital letter D indicates delay. This is also called as temporary storage. Delay occurs when an object or operator is waiting for the next activity.

Example 1.4

An operator waiting to get a tool in the stores.

Work pieces stocked near the machine before the next operation.

5. Permanent storage

An equilateral triangle standing on its vertex represents storage. Storage takes place when an object is stored and protected against unauthorized removal.

Example 1.5

Raw material in the store room.

6. Combined activity

When two activities take place at the same time or done by the same operator or at the same place, the two symbols of activities are combined.

Example 1.6

Reading and recording a pressure gauge. Here a circle inside a square represents the combined activity of operation and inspection.

Operation Process Chart

An operation process chart is a graphic representation of the sequence of all operations and inspections taking place in a process. It is also known as outline process chart. It gives a bird's eye view of the overall activities. Entry points of all material are noted in the chart. An example of operation process chart is shown in the figure 1.2. Here the process of manufacture of electric motor is shown.

The conventions followed in preparing the chart are

1. Write title at the top of the chart.

2. Begin the chart from the right hand side top corner.

3. Represent the main component at the right extreme.

4. Represent the sequence of operations and inspections by their symbols. Connect them by vertical flow lines.

5. Record the brief description of the activity to the right side of the symbols.

6. Note down the time for each activity to the left of the symbol.

7. Number all operations in one serial order. Start from the right hand top (from number 1).

8. Similarly number all inspections in another serial order (starting from 1).

9. Continue numbering, till the entry of the second component

10. Show the entry of purchased parts by horizontal lines.

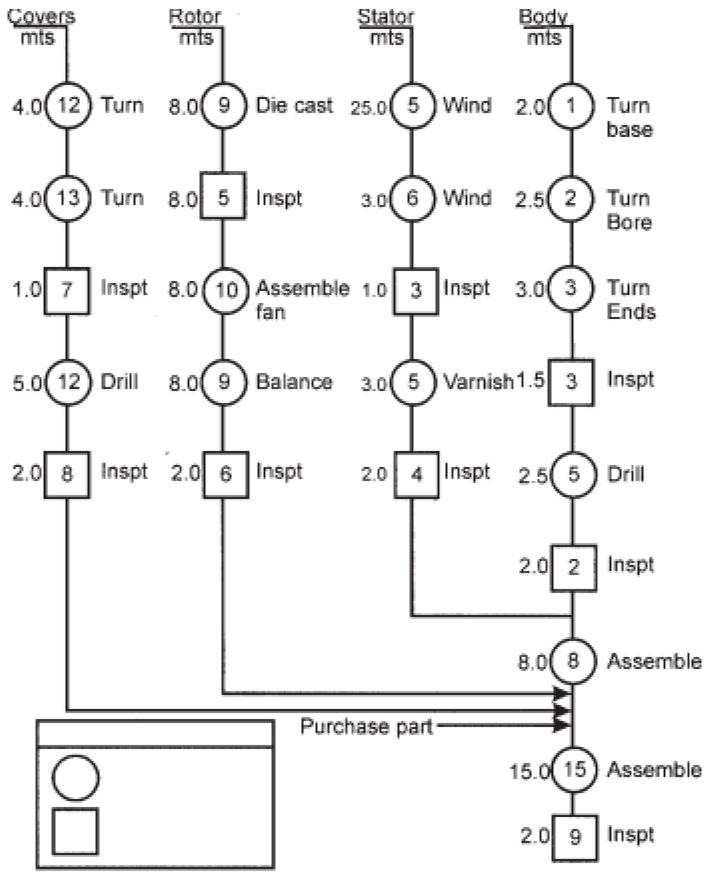


Fig. 1.2: Operation process chart

1.5.3 Flow Process Chart

A flow process chart is a graphical representation of the sequence of all the activities (operation, inspection, transport, delay and storage) taking place in a process. Process chart symbols are used here to represent the activities. There are three types of flow process charts.

They are

1. Man type flow process chart

This flow process chart records what the worker does.

2. Material type flow process chart

This flow process chart records how the material is handled or treated.

3. Equipment type flow process chart

This flow process chart records how the equipment or machine is used.

Example 1.7

The activities of a stenographer in preparation of a letter are recorded in the operator type flow process chart shown in figure 1.3.

Chart No. : 001 Date :
Job : Typing A letter Charted by: ....
Chart begins : Steno in her seat Chart ends-putting the typed letter in the way
Method : Present/Proposed

Sl. No.	*Description of the activities*	*Distance*	*Time in Sec.*	*Symbols*	*Remarks*
				O ☐ ⇒ D ∇	
1.	Steno in her seat	-	-		
2.	Hears the bell	-	3		
3.	Goes to manager's room	6m	10		
4.	Takes down dictation	-	120		
5.	Returns to her seat	6m	10		
6.	Prepares typewriter	-	15		
7.	Types the letter	-	150		
8.	Checks the matter	-	40		
9.	Goes to manager's room	6m	10		
10.	Waits till the manager signs	-	20		
11.	Returns to her seat	6m	10		
12.	Types envelope	-	20		
13.	Puts the letter inside envelope	-	5		
14.	Puts the envelope in dispatch tray	-	5		

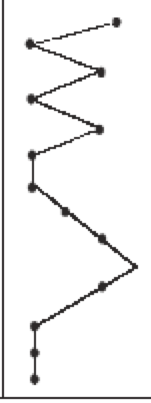


Fig. 1.3: *Flow process chart—operator type*

The chart records the activities of the steno. Here, the manager calls the steno and dictates a letter. The steno takes notes of the letter, types it, gets the signature of the manager and sends it for dispatching. These activities are shown in the chart. This is operator type flow process chart. Considering the message in the letter as material, we can prepare the material type flow process chart.

Similarly, considering the type writer as the equipment, we can prepare the equipment type flow process chart.

General guidelines for making a flow process chart

1. The details must be obtained by direct observation—charts must not be based on memory.
2. All the facts must be correctly recorded.
3. No assumptions should be made.
4. Make it easy for future reference.
5. All charts must have the following details:
   a) Name of the product, material or equipment that is observed.
   b) Starting point and ending point.
   c) The location where the activities take place.
   d) The chart reference number, sheet number and number of total sheets.
   e) Key to the symbols used must be stated.

Cost estimation may be defined as the process of forecasting the expenses that must be incurred to manufacture a product. These expenses take into consideration all expenditures involved in design and manufacturing with all the related service facilities such as pattern making, tool making as well as portion of the general administrative and selling costs. Cost estimates are the joint product of the engineer and the cost accountant.

Estimating is the calculation of the costs which are expected to be incurred in manufacturing a component in advance before the component is actually manufactured. Costing may be defined as a system of accounts which systematically and accurately records every expenditure in order to determine the cost of a product after knowing the different expenses incurred in various department.

REASONS FOR DOING ESTIMATES

Cost estimates are developed for a variety of different reasons. The most important reasons are shown below. Should the product be produced? When a company designs a new product, a detailed estimate of cost is developed to assist management in making an intelligent decision about producing the product. This detailed estimate of cost includes an estimate of material cost, labour cost, purchased components and assembly cost.

In addition to product cost, many other elements must be estimated. These include all tooling costs. A cost estimate must be developed for jigs, fixtures, tools, dies and gauges. Also, the cost of any capital equipment must be entered into the estimate. These figures are usually supplied through quotation by vendors. An estimate of this nature will include a vast amount of details, because if management approves the project, the estimate now becomes the budget.

Estimates as temporary work standards. Many companies that produce product in high volume, such as automotive companies, will use estimates on the shop floor as temporary work standards. Temporary work standards are replaced with time studied work standards as rapidly as possible.

Cost control

A job shop (contract shop) will use a cost estimate for cost control purposes because lot of sizes are small and job shops seldom estimate work standards for what they produce. This use of an estimate for this purpose is different from temporary standards in that it uses the “meet or beat” philosophy.

Make-or-buy decision

When a company sets out to produce a new product, many components in the bill of materials re subject to a make-or-buy decision. A cost estimate is developed for comparison purposes. There are usually considerations aside from piece part cost. These may include tooling cost, vendor quality, and vendor delivery.

Determine selling price

An estimate is used to determine selling price. The estimate is always a reflection of actual cost. In most organizations the marketing department has the responsibility of establishing a selling price, which can be substantially different from the cost estimate. There are many reasons for this. For example, a contract shop might be willing to sell the first order at something less than the estimate to develop a new customer.

Check vendor quotes (purchase analysis)

An estimating function is often established for the sole purpose of checking vendor quotations on outsourced work. One automobile company has an entire department of cost estimators devoted to this task.

DEFINITION

Cost estimating may be defined as the process of forecasting the expenses that must be incurred to manufacture a product. These expenses take into consideration all expenditures involved in design and manufacturing with all the related service facilities such as pattern making tool making, as well as a portion of the general administrative and selling costs.

Cost estimating also includes predetermination of the quantity and quality of material, labour required etc. Estimating requires highly technical knowledge about manufacturing methods and operation times etc.

Cost estimates are the joint product of the engineer and the cost accountant, and involve two factors.

1. Physical data.

2. Costing data.

1. Physical data

The engineer as part of his job of planning and manufacturing determines the physical data.

2. Costing data

The cost accountant compiles and applies the costing data.

ELEMENTS OF COST INTRODUCTION

For the successful functioning of an industrial enterprise, one of the most important onsideration is to reduce the cost of manufacture of the product or article, as much as possible without affecting the quality. This will help in earning higher profits. To achieve the idea of reducing cost one must be familiar with elements which make up the total cost of a product. The total cost is made up of three main elements (figure 3.1).

3.12 MATERIAL COST

Material cost consists of the cost of materials which are used in the manufacture of product. It is divided into the following:

Direct Material Cost

It is the cost of those materials which are directly used for the manufacture of the product and become a part of the finished product. This expenditure can be directly allocated and charged to the manufacture of a specific product or job and includes the scrap and waste that has been cut away from original bar or casting.

The procedure for calculating the direct material cost is as follows:

(i) From the product drawing, make a list of all the components required to make the final product.

(ii) Calculate the volume of each component from the drawing dimensions after adding machining allowances, wherever necessary.

(iii) The volume of component multiplied by the density of material used gives the weight of the material per component.

(iv) Add process rejection and other allowances like cutting allowance to get the gross weight per component.

(v) Multiply the gross weight by the rate of material per unit weight to get the cost of raw material per component.

(vi) The cost of raw material for all the components is, similarly, calculated and added up which gives the cost of direct material for the product.

Indirect Material Cost

In addition to direct materials a number of other materials are necessary to help in the conversion of direct materials into final shape. Though these materials are consumed in the production, they don't become part of the finished product and their cost cannot be directly booked to the manufacture of a specific product. Such materials are called indirect materials. The indirect materials include oils, general tools, greases, sand papers, coolants, cotton waste etc. The cost associated with indirect materials is called indirect material cost.

In some cases certain direct materials like nails, screws, glue, putty etc. are used in such small quantity that it is not considered worthwhile to identify and charge them as direct materials. In such cases these materials are also charged as indirect materials.

Depending upon the product manufactured, the same may be direct materials for one concern

and indirect materials for others.

LABOUR COST

It is the expenditure made on the salaries, wages, overtime, bonuses, etc. of the employees of the enterprise. It can be classified as

Direct Labour Cost

Direct labourer is one who actually works and processes the materials to convert it into the final shape. The cost associated with direct labour is called direct labour cost. The direct labour cost can be identified and allocated to the manufacture of a specific product. Examples of the direct labour are the workers operating lathes, milling machines or welders, or assemblers in assembly shop. The direct labour cost may be allocated to a product or job on the basis of time spent by a worker on a job.

EXPENSES

Apart from material and labour cost in each factory there are several other expenditures such as cost of special layouts, designs, etc. hire of special tools and equipments; depreciation charges of plants and factory building; building rent; cost of transportation, salaries and commissions to salesman etc. All these expenditures are known as overheads or expenses. So, from above it is clear that except for direct material and direct labour cost, all other expenditures are known as expenses. The expenses include indirect material cost and indirect cost and such other expenses.

Direct Expenses

Direct expenses also known as chargeable expenses include any expenditure other than direct material or direct labour incurred on a specific cost unit. These are the expenses which can be charged directly to a particular job and are done for that specific job only. For example, hire of special tools and equipment, cost of special jigs and fixtures or some special patterns and its maintenance cost, costs of layouts, designs and drawings or experimental work on a particular job etc.

Indirect Expenses (Overheads)

These are known as overhead charges, burden or on cost. All the expenses over and above prime cost are indirect expenses. Overhead is the sum of indirect labour cost, indirect material cost and other expenses including service which cannot be conveniently charged to specific cost unit. These can be further classified as

1. Production expenses/Factory expenses. 2. Administrative expenses. 3. Selling expenses. 4. Distribution expenses

TYPES OF ESTIMATE

Estimates can be developed in a variety of different ways depending upon the use of the estimates and the amount of detail provided to the estimator. Importance of understanding estimating methods. Every estimator should understand every estimating method and when to apply each, because no one estimating method will solve all estimating problems.

4.1.1 Guesstimates

Guesstimates is a slang term used to describe as estimate than lacks detail. This type of estimate relies on the estimators experience and judgment. There are many reasons why some estimates are developed using his method. One example can be found in the tool and die industry. Usually, the tool and die estimator is estimating tool cost without any tool or die drawings. The estimator typically works from a piece part drawing and must visualize what the tool or die looks like. Some estimators develop some level of detail in their estimate. Material cost, for example, is usually priced out in some detail, and this brings greater accuracy to the estimator by reducing error. If the material part of the estimate has an estimating error of plus or minus 5 per cent and the reminder of the estimate has an estimating error of plus or minus 10 per cent, the overall error is reduced.

4.1.2 Budgetary

The budgetary estimate can also be a guesstimate but is used for a different purpose. The budgetary estimate is used for planning the cost of a piece part, assembly, or project. This type of estimate is typically on the high side because the estimator understands that a low estimate could create real problems.

COST ESTIMATION

Using Past History

Using past history is a very popular way of developing estimates for new work. Some companies go to great lengths to ensure that estimates are developed in the same way actual cost is conducted. This provides a way past history in developing new estimates. New advancements in group technology now provide a way for the microcomputer to assist in this effort.

Estimating in Some Detail

Some estimators vary the amount of detail in an estimate depending on the risk and dollar amount of the estimate. This is true in most contract shops. This level of detail might be at the operation level where operation 10 might be a turning operation and the estimator would estimate the setup time at 0.5 hours and the run time at 5.00 minutes. The material part of the estimate is usually calculated out in detail to reduce estimating error.

Estimating in Complete Detail

When the risk of being wrong is high or the dollar amount of the estimate is high, the estimator will develop the estimate in as much detail as possible. Detailed estimates for machinery operations, for example, would include calculations for speeds, feeds, cutting times, load and unload times and even machine manipulations factors. These time values are calculated as standard time and adjusted with an efficiency factor to predict actual performance.

Parametric Estimating

Parametric estimating is an estimating method developed and used by trade associations. New housing constructions can be estimated on the basis of cost per square. There would be different figures for wood construction as compared with brick and for single strong construction as compared with multilevel construction.

Some heat-beating companies price work on a cost per pound basis and have different cost curves for different heat-treating methods.

Project Estimating

Project estimating is by far the most complex of all estimating tasks. This is especially true if the project is a lengthy one. A good example of project estimating is the time and cost of developing a new missile. The project might take 5 years and cost millions of dollars. The actual manufacturing cost of the missile might be a fraction of the total cost. Major projects of this nature will have a PERT network to keep track of the many complexities of the project. A team of people with a project leader is usually required to develop a project estimate.

5.0 INTRODUCTION—PRODUCTION COST ESTIMATION

In this rapid developing and competitive age it is necessary for a factory that the advance information about the cost of a job or a manufacturing order to be put through should be available before taking up actual production. Estimating which is predetermination of cost is mainly concerted with factory owner. It helps him to decide about the manufacturing the selling prices.

Estimation is calculation of cost when are expected to be incurred in manufacturing a component in advance before the component is actually manufactured.

5.1 ESTIMATION OF MATERIAL COST

5.1.1 Determination of Material Cost

To calculate the material cost of the product the first step is to study drawing of the product and split it into simple standard geometrical shapes and to find the volume of the material in the product and then to find the weight. The volume is multiplied by density of the metal used in the product.

The exact procedure to find the material cost is like this:

1. Study the drawing carefully and break up the component into simple geometrical shapes. (Cubes, prisms, cylinders, etc.)

2. Add the necessary machining allowances on all sides which are to be machined.

3. Determine the volume of each part by applying the formulae of mensuration.

4. Add the volumes of all the simple components to get total volume of the product.

5. Multiply the total volume of the product by the density of the material to get the weight of the material.

6. Find out the cost of the material by multiplying the cost per unit weight to the total weight of the material.

5.1.2 Mensuration in Estimating

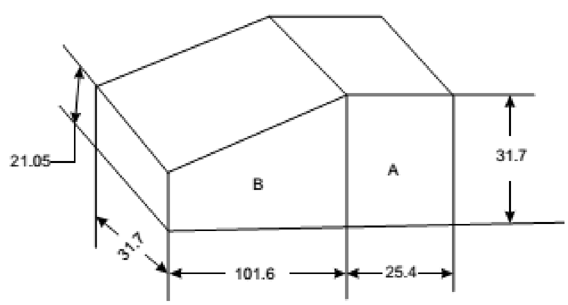
Introduction Mensuration is the science which deals with the calculation of length of lines, areas of surfaces and volumes of solids by means of mathematical rules and formulae. An estimator is often required to calculate the length, area of volume of a job he is going to perform. Hence, he must be thoroughly acquainted with the rules and formulae of mensuration.

The general formulae for calculating the volume of a simple solid having a uniform crosssectional area throughout in the direction normal to the section considered, is to find the product of the cross-sectional area and the length of the solid in the direction normal to the section considered. To calculate the volume of a complex solid, it should be divided into a number of sample geometric solids. The volume of all these parts are calculated separately and then added together to get the total volume.

The volume of a solid of revolution, as generated by the rotation of a plane area about a given axis in its plane, is equal to the product of the area of the revolving section and the length of the path covered by its centroid in describing a circle about the axis. This theorem was given by Guldinus. Volume of a circular ring, a half-round rib surrounding the boss of a fly wheel, and Vgroove of a V-belt pulley may be calculated by Guildinus theorem.

Problem:

The wedge shown in the figure 5.1 below is to be made of 38.1 mm diameter stock by forging. Calculate the length of the bar if the volume remains unchanged.



Solution

Divide the wedge into components A, B. Consider component A. It is a simple RECTANGLE.

Hence, Surface area $A_A = l \times b$

and Volume $V = l \times b \times h = A \times h$

Now, $V_A = l \times b \times h = A \times h$

$= 31.7 \times 25.4 \times 31.7$

$V_A = 25{,}524.206 \text{ mm}^3$

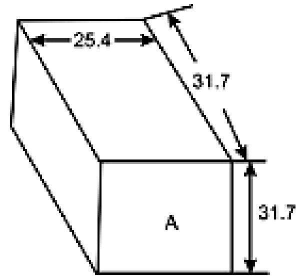


Fig. 5.2

Consider component B

Surface area $A_B = \frac{1}{2}(a + b)\,h$

Here $a = 21.05$

$b = 31.7$

$h = 101.6$

Hence, $A_B = 2679.7$

Volume. $V_B = A_B \times \text{Thickness}$

Volume of part B, $V_B = 2679.7 \times 31.7$

$= 84{,}946.49$

$V_B = 84{,}946.49 \text{ mm}^3$

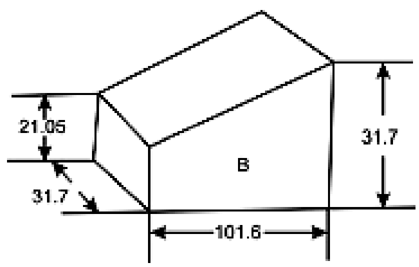


Fig. 5.3

Total volume of wedge

$V = V_A + V_B$

$= 110{,}470.696 \text{ mm}^3$

$V = 110470.696 \text{ mm}^3$

Now, volume of the cylindrical bar stock (from which the wedge is to be forged) is given as

$V_1 = \frac{\pi}{4} d^2 l$ d = Dia of stock

$= \frac{\pi}{4}(38.1)^2 l$ l = Length of stock

$V_1 = 1140.091\, l$

As volume does not change $V_1 = V$

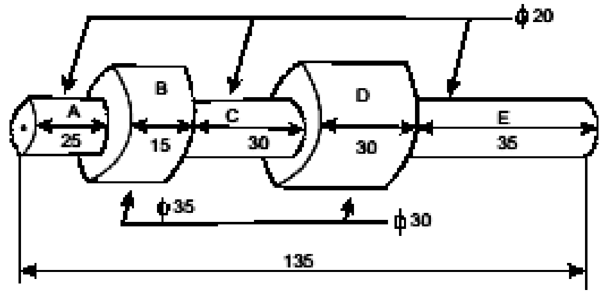
$1140.091\, l = 114070.696$

$l = 96.89 \simeq 96.9 \equiv 97 \text{ mm}$

Thus, the length of the bar stock is 97 mm or 9.7 cms.

Problem 2

Estimate the weight of the aluminium shaft. Aluminium weighs 9 gms/cm3. Also evaluate the material cost if 1 kg Aluminium costs Rs. 8 kg.



Solution

Let us disintegrate the given shaft into five different sections as shown below consider section A.

$$V_A = \frac{\pi}{4} d^2 l = \frac{\pi}{4} (20)^2 \times 25$$

$$V_A = 10,000 \times \frac{\pi}{4} = 7853 \text{ mm}^3$$

Volume of section B $\quad V_B = \frac{\pi}{4} d^2 l$

$$= \frac{\pi}{4} (35)^2 \times 15$$

$$V_B = 14431.69 \text{ mm}^3$$

Volume of section C $\quad V_c = \frac{\pi}{4} d^2 l$

$$= \frac{\pi}{4} (20)^2 \times 30$$

$$V_D = 21205.75 \text{ mm}^3$$

Volume of section E $\quad V_E = \frac{\pi}{4} (20)^2 \times 35$

$$= 10,995.57 \text{ mm}^3$$

$$V_E = 10,995.57 \text{ mm}^3$$

Total volume of the shaft $\quad V = V_A + V_B + V_C + V_d + V_E$

$$= 63,911.776 \text{ mm}^3$$

$$V = 63.911 \text{ cm}^3$$

Weight of the material of the shaft $\quad W = 63.911 \times 9 = 575.205 \text{ g}$

$$= 0.575 \text{ kg}$$

The weight of the material is 0.585 kgs.

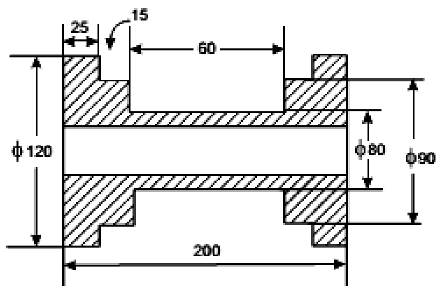
Material cost (cm)

$$cm = 0.575 \times 8 = \text{ Rs. } 4.6$$

Hence, the cost of material required to forge a shaft is Rs. 4.6/kg.

Problem 3:

The following figures show brasses for a particular bearing. Determine material cost and weight if brass weighs 8.4 g/cm3 and 1 kg brass costs Rs. 27.50.



Solution

The brasses diagram is split into individual volumes and the total volume is found as

$$V = \left(2 \times \frac{\pi}{4} \times 120^2 \times 25\right) + \left(2 \times \frac{\pi}{4} \times 90^2 \times 15\right)$$

$$+ \left(\frac{\pi}{4} \times 80^2 \times 60\right) - \left(\frac{\pi}{4} \times 40^2 \times 200\right)$$

$$= (806.60 \text{ mm}^3)$$

$$V = 806.60 \text{ cm}^3$$

-

Weight of brass required (W),

$$W = 806.66 \times 8.4 = 6775.4 \text{ g}$$

$$= 6.775 \text{ kg}$$

Thus, 6.775 kgs of brass the required to manufacture the brasses

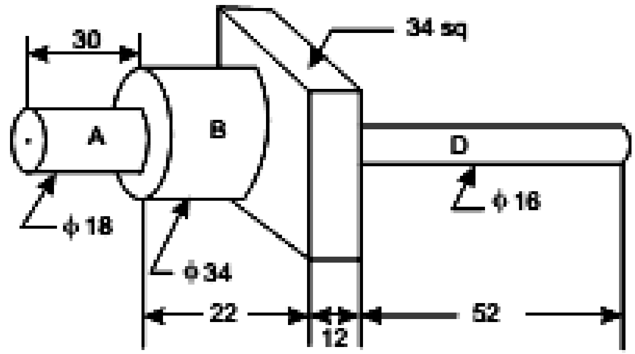
Cost of required brasses (C)

$$C = 6.775 \times 27.50 = 186.32 \approx 187 \text{ Rs./kg}$$

The cost of the required brasses are Rs. 198.

Problem 4:

Estimate the weight of steel is gms/cc if about 200g of steel are required to forge a shaft as shown.



Solution

Divide the shaft into four sections.

Total volume of solid

$$V = V_A + V_B + V_C + V_D$$

$$V_A = \frac{\pi}{4}(18)^2 \times 30 = \frac{\pi}{4} \times 324 \times 40$$

$$= 7634.070 \text{ mm}^3 = 7.63 \text{ cm}^3$$

$$V_A = 7.63 \text{ cm}^3$$

$$V_B = \frac{\pi}{4}(34)^2 \times 22 = 19{,}973.8 \text{ mm}^3$$

$$V_B = 19.973 \text{ cm}^3$$

$$V_C = 34 + 34 + 12 = 13{,}872 \text{ mm}^3$$

$$V_C = 13.87 \text{ cm}^3$$

$$V_D = \frac{\pi}{4}(16)^2 \times 52 = 10{,}455.2 \text{ mm}^3$$

$$V_D = 10.45 \text{ cm}^3$$

Total volume $V = V_A + V_B + V_C + V_D$

$$= 51.923$$

Weight of required steel material is 200 gms

$$W = 200 \text{ kgs} = 200 \text{ gms}$$

Also $W = V + (\text{Weight of steel mg/cc})$

$\therefore$ Weight of steel in g/cc $= \dfrac{200}{51.923} = 3.85 \text{ g/cc}$

Hence, steel weight 3.85 gms/cc approximately.

UNIT – V Multiple Choice Questions and Answers

S. No	Question	Option a	Option b	Option c	Option d	Answer
1	Cost estimation include(s) the following expenditure(s)	Pattern making	Tool making	Selling expenses	All of these	All of these
2	To calculate the probable cost of the product, knowledge of following factors involves	Production time required	Use of previous estimates of comparable parts	Effect of change in facilities on costing rates	All of these	All of these
3	The following is cost of indirect materials	Lubricating oil	Octroi	Import duties	Insurance	Lubricating oil
4	The payment made to the following is cost of direct labour.	Machinist	Supervisor	Inspector	Sweeper	Machinist
5	Cost accounting is a specialized branch of accounting which deals with	classification, recording, allocation and control of costs	classification, processing, allocation and directing	classification, recording, planning and control of costs	classification, recording, allocation and directing	classification, recording, allocation and control of costs
6	Expenditure incurred on material, labor, machinery, production and inspection are summed up to find the	Total cost of product	Selling price of product	Factory cost of product	None of the above	Factory cost of product
7	The payment made to the following is cost of indirect labor.	Time keeper	Welder	Molder	Turner	Time keeper

8	Cost of preparing drawings for the manufacture of a particular product is	Cost of direct labor	Cost of indirect labor	Direct expenses	Indirect expenses	Direct expenses
9	Which of the following calculate the actual cost of product	Cost estimation	Costing	Both (and (	None of the above	Costing
10	The cost data provide invaluable information for taking the following managerial decision(s)	To make or buy	To own or hire fixed asset	Determining the expansion or contraction policy	All of the above	All of the above
11	The following is also known as overhead costs or on costs.	Cost of direct labor	Cost of indirect labor	Direct expenses	Indirect expenses	Indirect expenses
12	The following is(are) the overhead cost(s)	Factory expenses	Selling expenses	Distribution expenses	All of the above	All of the above
13	Match the following Type of costing Type of industry a. Job costing 1. Utility services b. Process costing 2. Automobile industry c. Departmental costing 3. ship building d. Operating costing 4. paper making	a-2, b-3, c-1, d-4	a-3, b-4, c-2, d-1	a-4, b-2, c-1, d-3	a-3, b-2, c-1, d-4	a-3, b-4, c-2, d-1
14	The method of unit costing is adopted by	Transport services	Steel industry	Mines	Bicycle industry	Mines
15	All such expenses which are incurred for creating and enhancing the demands for the products are	Selling expenses	Administrative expenses	Distribution expenses	All of the above	Selling expenses

16	Prime cost=	cost of direct labour + cost of direct material + direct expenses	cost of indirect labour + cost of indirect material + direct expenses	cost of direct labour + cost of direct material + indirect expenses	cost of indirect labour + cost of direct material + indirect expenses	cost of direct labour + cost of direct material + direct expenses
17	____ costing is a type of job costing.	Multiple	Operating	Unit	Batch	Batch
18	The following is cost of direct materials	Freight charges	grease	coolant	cotton waste	Freight charges
19	Factory Cost =	Prime cost + factory expenses	Prime cost + administrative expenses	Prime cost + selling expenses	Prime cost + distribution expenses	Prime cost + factory expenses
20	The following is(are) the method(s) to increase profit	Increase the sales price	Increase the market	Reduce total cost	All of the above	All of the above