**ROHINI** COLLEGE OF ENGINEERING AND TECHNOLOGY

**ACCREDITED WITH NAAC A+ GRADE**

**DEPARTMENT OF MECHANICAL ENGINEERING**

# ME 3592 – METROLOGY AND MEASUREMENTS

**SEMESTER: 05**

**III YEAR**

**REGULATION 2021**

**PREPARED BY**

**MR.P. NAVIN JASS, M.E, (Ph.D).,**

**AP/MECH,**

**RCET.**

# UNIT – IV METROLOGY OF SURFACES

Fundamentals of GD & T- Conventional vs Geometric tolerance, Datums, Inspection of geometric deviations like straightness, flatness, roundness deviations; Simple problems – Measurement of Surface finish – Functionality of surfaces, Parameters, Comparative, Stylus based and Optical Measurement techniques, Filters, Introduction to 3D surface metrology- Parameters.



**Prepared By**

P. Navin Jass,

Assistant Professor,

Department of Mechanical Engineering,

Rohini College of Engineering & Technology.

## 4.1 Fundamentals of Geometric Dimensioning and Tolerancing

The designer could discuss with the manufacturing personnel (die designer, foundry foreman, machinist, and inspectors) what features were to be contacted to establish the so called "centerlines" that were used on the drawing to locate features such as holes and keyways.

GD&T was the solution to this major problem. GD&T provides a designer the tools to have clear, concise, and consistent instructions as to what is required. It eliminates ambiguities hence everyone that is involved with the part will not have to interpret the dimensioning.

Tolerance can be defined as the magnitude of permissible variation of a dimension or other measured value or control criterion from the specified value. It can also be defined as the total variation permitted in the size of a dimension, and is the algebraic difference between the upper and lower acceptable dimensions. It is an absolute value.

It is compilation of symbols and rules that efficiently describe and control dimensioning & tolerancing for all drawings (castings, machined components, etc.). It is documented in ASME Y14.5M which has the symbols, rules, and simple examples. Also, ASME Y14.8 has guidance for casting and forging drawings.

### 4.1.1 NEED OF GD&T

With functional assemblies, multi-part products, or parts with complex functionality, it is crucial that all components work well together. All relevant fits and features need to be specified in a way that impacts the manufacturing process and its related investments the least, while still guaranteeing functionality. Tightening tolerances by a factor two can raise the costs twofold or even more, due to higher reject rates and tooling changes. GD&T is the system that allows developers and inspectors to optimize functionality without increasing cost.

The most important benefit of GD&T is that the system describes the design intent rather than the resulting geometry itself. Like a vector or formula, it is not the actual object but a representation of it.

For example, a feature standing at 90 degrees to a base surface can be toleranced on its perpendicularity to that surface. This will define two planes spaced apart, that the center plane of the feature must fall within. Or, when drilling a hole, it makes the most sense to tolerance it in terms of alignment to other features.

Describing product geometry related to its intended functionality and manufacturing approach is ultimately simpler than having to describe everything in linear dimensions. It also provides a communication tool with manufacturing vendors, customers, as well as quality inspectors.

When performed well, GD&T even allows statistical process control (SPC), reducing product reject rates, assembly failures, and the effort needed for quality control, saving organizations substantial resources. As a result, multiple departments are able to work more in parallel because they have a shared vision and language for what they want to achieve.

### 4.1.2 WORKING GD&T

Engineering drawings need to show the dimensions for all features of a part. Next to the dimensions, a tolerance value needs to be specified with the minimum and maximum acceptable limit. The tolerance is the difference between the minimum and maximum limit. For example, if we have a table that we would accept with a height between 750 mm and 780 mm, the tolerance would be 30 mm.

However, the tolerance for the table implies that we would accept a table that is 750 mm high on one side and 780 mm on the other, or has a waved surface with 30 mm variation. So to appropriately tolerance the product, we need a symbol communicating the design intent of a flat top surface. Therefore we have to include an additional flatness tolerance in addition to the overall height tolerance.

a cylinder with a toleranced diameter will not necessarily fit into its hole if the cylinder gets slightly bent during the manufacturing process. Therefore it also needs a straightness control, which would be difficult to communicate with traditional plus-minus tolerancing. Or a tube that has to seamlessly match a complex surface that it's welded to requires a surface profile control.

Besides individual tolerances, engineers must take into account system-level effects. when a part comes out with all dimensions at their maximum allowed value, does it still meet overall requirements such as product weight and wall thicknesses? This is called the **Maximum Material Condition** (MMC), while its counterpart is the **Least Material Condition** (LMC).

The standards do not only pertain to designers and engineers but also to quality inspectors by informing them how to measure the dimensions and tolerances. Using specific tools such as digital micrometers and calipers, height gauges, surface plates, dial indicators, and a coordinate measuring machine (CMM) are important to tolerancing practice.

When measuring and defining a part, the geometry exists in a conceptual space called the Datum Reference Frame (DRF). This is comparable to the coordinate system at the origin of a space in 3D modeling programs. A datum is a point, line or plane that exists in the DRF and is used as a starting place for measuring.

### 4.1.3 GD&T GUIDE LINES

An engineering drawing has to accurately convey the product without adding unnecessary complexity or restrictions. The following guidelines are helpful to consider:

- Clarity of a drawing is the most important, even more so than its accuracy and completeness. To improve clarity, draw dimensions and tolerances outside of the part's boundaries and applied to visible lines in true profiles, employ a unidirectional reading direction, convey the function of the part, group and/or stagger dimensions, and make use of white space.
- Always design for the loosest feasible tolerance to keep costs down.
- Use a general tolerance defined at the bottom of the drawing for all dimensions of the part. Specific tighter or looser tolerances indicated in the drawing will then supersede the general tolerance.

- Tolerance functional features and their interrelations first, then move on to the rest of the part.
- Whenever possible, leave GD&T work to the manufacturing experts and do not describe manufacturing processes in the engineering drawing.
- Do not specify a 90-degree angle since it is assumed.
- Dimensions and tolerances are valid at 20 °C / 101.3 kPa unless stated otherwise.

## 4.1.4 DATUM

In Geometric Dimensioning and Tolerancing (GD&T), a "datum" refers to a theoretical perfect point, line, plane, or axis used as a reference to establish the coordinate system and control the allowable variations of other features on a part or component. Datums are essential for defining the part's geometric relationship and orientation within the assembly. They serve as the foundation for dimensioning and tolerancing other features on the part.

### 4.1.4.1 Datums in GD&T:

**Datum Features:** A datum feature is a physical feature on the part that serves as the reference for establishing the datum. Common datum features include flat surfaces, holes, cylinders, and bosses.

**Primary, Secondary, and Tertiary Datums:** Datums are categorized into three levels: primary, secondary, and tertiary. Primary datums are the first datums established and typically relate to the fundamental features needed to control the part's orientation. Secondary datums refine the orientation further, and tertiary datums are used for additional control as necessary.

**Datum Reference Frame:** The collection of datums used to define the part's coordinate system is called the datum reference frame (DRF). It consists of the datum features identified by letters (e.g., A, B, C) and the coordinate axes they create.

**Datum Precedence:** Datum precedence rules dictate the order in which datums are established in the DRF. Primary datums take precedence over secondary, and secondary take precedence over tertiary. The order in which datums are specified affects how subsequent tolerances are interpreted.

**Datum Targets**: Datum targets are specific areas or points on the part where the part's features are to be constrained relative to the datum reference frame. Datum targets are used when features' tolerance zones are controlled by their relationship to the datum features.

**Material Condition Modifier:** Datums can be associated with material condition modifiers, such as Maximum Material Condition (MMC) or Least Material Condition **(LMC), to control the tolerance zone based on the part's actual material size.**

**Datum Shift:** Datum shift is the cumulative effect of deviations of the established datums from their theoretically perfect locations. It can affect how other features' tolerances are applied, especially when secondary and tertiary datums are used.

GD&T provides a systematic way to define and control part features by referencing them to the datum reference frame. This ensures that parts are manufactured and assembled with precise orientation and location, leading to improved interchangeability and functional performance. Using datums effectively in GD&T helps avoid ambiguity in engineering drawings and fosters clear communication among designers, manufacturers, and quality control personnel.

### 4.1.5 GD&T SYMBOLS

GD&T is feature-based, with each feature specified by different controls. GD&T symbols fall into five groups:

- **Form controls** specify the shape of features, including:
    - Straightness is divided into line element straightness and axis straightness.
    - Flatness means straightness in multiple dimensions, measured between the highest and lowest points on a surface.

- Circularity or roundness can be described as straightness bent into a circle.
- Cylindricity is basically flatness bent into a barrel. It includes straightness, roundness, and taper, which makes it expensive to inspect.

- **Profile controls** describe the three-dimensional tolerance zone around a surface:
  - Line Profile compares a two-dimensional cross-section to an ideal shape. The tolerance zone is defined by two offset curves unless otherwise specified.
  - Surface Profile creates through two offset surfaces between which the feature surface must fall. This is a complex control typically measured with a CMM.
- **Orientation controls** concern dimensions that vary at angles, including:
  - Angularity is flatness at an angle to a datum and is also determined through two reference planes spaced the tolerance value apart.
  - Perpendicularity means flatness at 90 degrees to a datum. It specifies two perfect planes the feature plane must lie in between.
  - Parallelism means straightness at a distance. Parallelism for axes can be defined by defining a cylindrical tolerance zone by placing a diameter symbol in front of the tolerance value.
- **Location controls** define feature locations using linear dimensions:
  - Position is the location of features relative to one another or to datums and is the most used control.
  - Concentricity compares the location of a feature axis to the datum axis.
  - Symmetry ensures that non-cylindrical parts are similar across a datum plane. This is a complex control typically measured with a CMM.
- **Runout controls** define the amount by which a particular feature can vary with respect to the datums:
  - Circular Runout is used when there is a need to account for many different errors, such as ball-bearing mounted parts. During inspection, the part is rotated on a spindle to measure the variation or 'wobble' around the rotational axis.

- Total Runout is measured on multiple points of a surface, not just describing the runout of a circular feature but of an entire surface. This controls straightness, profile, angularity, and other variations.

**Table 4.1 GD&T SYMBOLS**

| Sr. No. | Symbol | Geometric Characteristics | Tolerance Type | Datum Referencing |
|---|---|---|---|---|
| 1 | ⏥ | Flatness | Form (No Relation between Features) | Not Required |
| 2 | — | Straightness | | |
| 3 | ⌭ | Cylindricity | | |
| 4 | ○ | Circularity (Roundness) | | |
| 5 | ⊥ | Perpendicularity | Orientation (No Relation between Features) | Required |
| 6 | // | Parallelism | | |
| 7 | ∠ | Angularity | | |
| 8 | ⌖ | Position | Location | Required |
| 9 | ◎ | Concentricity | | |
| 10 | ⌯ | Symmetry | | |
| 11 | ↗ | Circular Runout | Runout | Required |
| 12 | ⌰ | Total Runout | | |
| 13 | ⌓ | Profile of a Surface | Profile | Optional |
| 14 | ⌒ | Profile of a Line | | |

## 4.1.6. Feature Control Frame (FCF)



The Feature Control Frame is the notation to add controls to the drawing. The leftmost compartment contains the geometric characteristic. In the example above, it is a location control but it can contain any of the control symbols. The first symbol in the second compartment indicates the shape of the tolerance zone. In this example, it is a diameter as opposed to a linear dimension. The number indicates the allowed tolerance.

Next to the tolerance box, there are separate boxes for each datum feature that the control refers to. Here, the location will be measured related to datum B and C. Next to the tolerance or a datum feature is an optional encircled letter, the feature modifier.


The following possibilities can occur:

- **M** means that the tolerance applies in the Maximum Material Condition (MMC)
- **L** means that the tolerance applies in the Least Material Condition (LMC)
- **U** indicates an unequal bilateral tolerance, i.e. for a 1 mm tolerance it may specify it as minus 0.20 and plus 0.80.
- **P** means that the tolerance is measured in a Projected Tolerance Zone at a specified distance from the datum.
- No symbol installs the tolerance regardless of feature size (RFS)

For this example, if the part is not in MMC, a bonus tolerance can be added proportionally to the deviation from MMC. So if a part is at 90% MMC, the tolerance will also loosen by 10%.

## 4.2 CONVENTIONAL TOLERANCING VS GEOMETRIC TOLERANCING

### Traditional Tolerance or Conventional Tolerancing or Coordinate Tolerance

Conventional tolerance pedals the size of each dimension with only number value and text messages. For example, when the designer wants to place an order of sheet parts, the size tolerance-based instructions are given as shown in Figure 4.8. This part is non-conforming defective product. For this part, parallelism is not mentioned in the drawing. The fault lies with the tolerance instruction by the designer and not with the manufacturer.

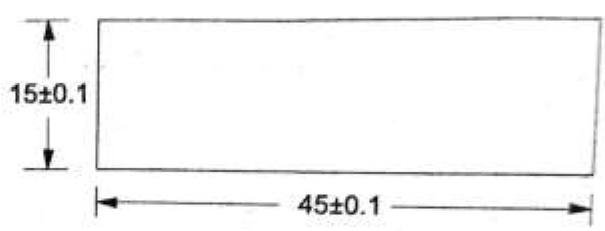



**FIG 4.1 Example 1 for conventional tolerance**

With coordinate tolerancing, the dimensions of the part are given but the drawing does not specify how the Pan is to be set up for measurement. It may lead to wrong measurement values for the same part. For the part drawing in Figure as an example, the coordinate dimensions give the location of two holes. Based on the drawing, the location of these holes could be measured in multiple ways because it is not specified on the drawing from where the measurement should be started.

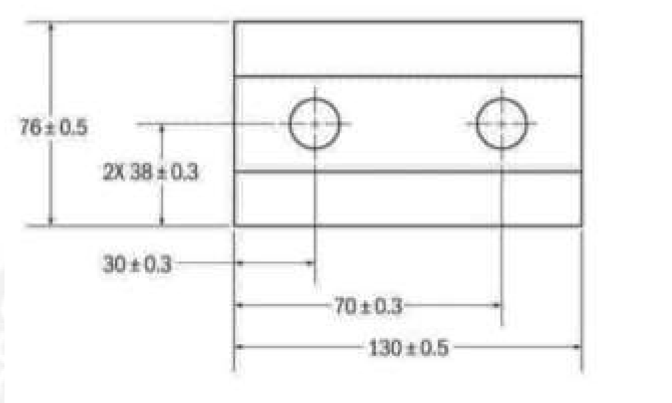



**FIG 4.2: Example 2 for conventional tolerance**

Figure 4.10 and Figure 4.11 shows in two ways that this part could be set up for measurement. It can be seen that the ends of the manufactured part are not perpendicular to the long edge of the part. It is due to the measurement location of the holes in setup I would result in different values than what will be measured in setup 2. So, this problem can be resolved by GD&T using datum.

In another example Figure 4.12, a drawing of a part with four holes are shown using coordinate dimensions. Here also, the measurement setup is not specified on the drawing which Similar to above case. So, it results the different setups with different measurement values for the location of the holes.

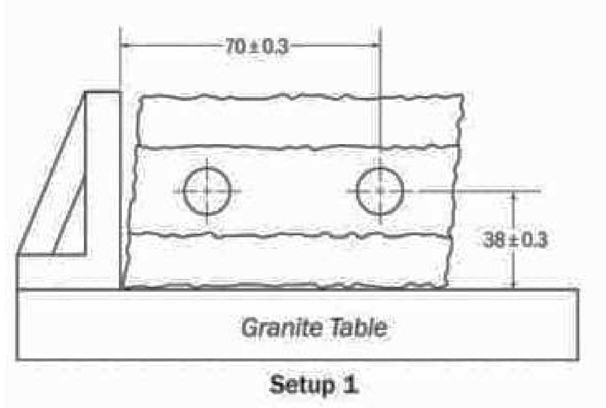



**FIG 4.3** Measurement setup 1

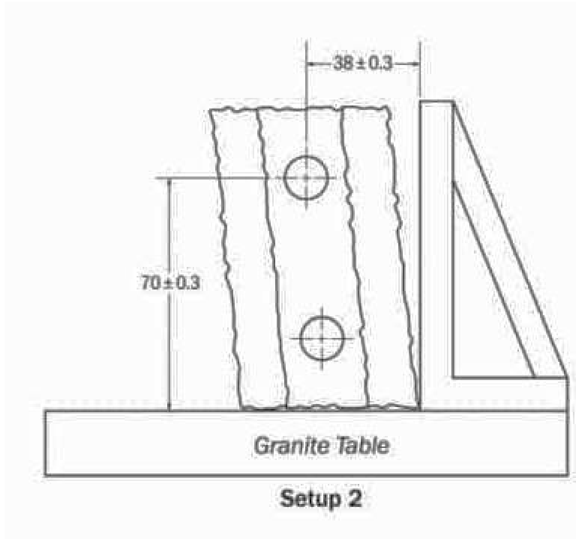



**FIG 4.4** Measurement setup 2

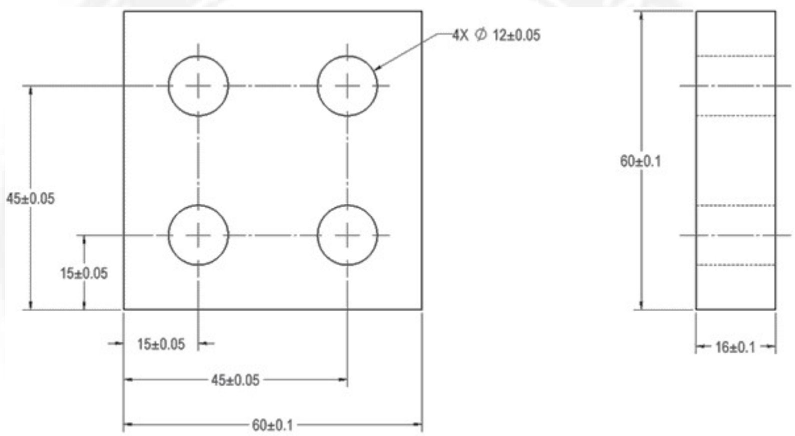



**FIG 4.5 Part drawing using conventional tolerance**

## 4.2.1 Geometric tolerance

Geometric tolerance controls the shape and positional relationship. GD&T is based on principle on independency.

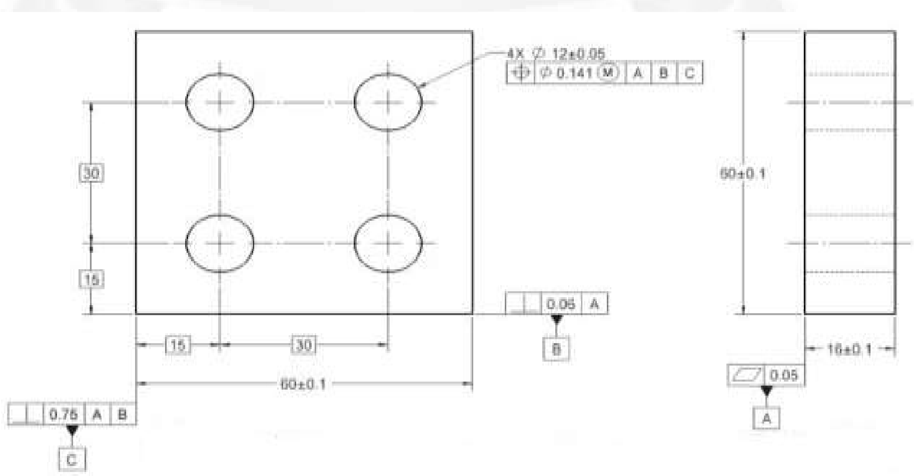
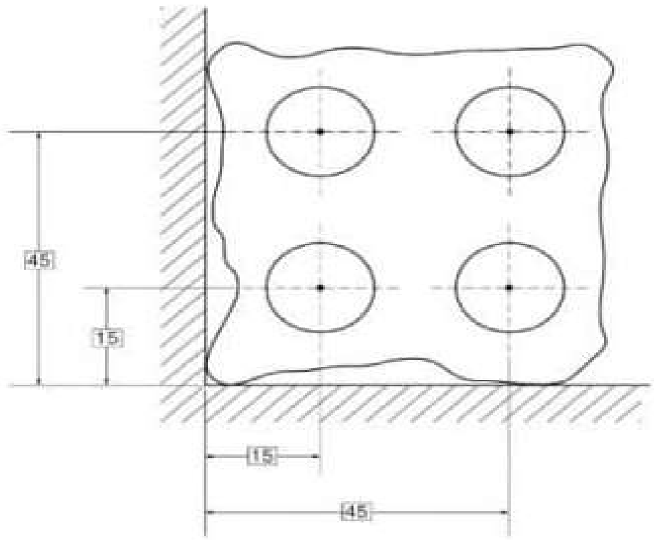



**FIG 4.6 Part drawing using geometric tolerance**




Measurement Setup

### 4.7 Datum reference frame and measurement setup

To inspect this part, you would use a datum feature simulator. A datum feature simulator is a piece of inspection equipment with a surface that contacts the datum feature (part surface) to simulate a perfect surface/datum. The datum reference frame and the measurement setup for this part as specified by the drawing are shown in Figure.

### 4.2.2 Bonus Tolerance Ensured by GD&T

In GD&T, **bonus tolerance** is a modification of a GD&T tolerance that under certain conditions increases the tolerance, hence the term “bonus”. when the maximum material condition (MMC) symbol is used to modify a GD&T tolerance, bonus tolerance becomes available. The MMC modifier implies that the GD&T tolerance can be increased if the manufacturing process can control the related feature-of-size. The bottom drawing shows how controlling the feature-of-size can allow for more straightness tolerance.

The logic here is if the feature-of-size is smaller than the MMC, than additional variation in the straightness should not prevent the successful assembly of this pin to its mating hole. For this reason, the manufacturing team may choose to intentionally make the part smaller in order to avoid the need to tightly control the straightness. Sometimes this leads to lower manufacturing costs since the part does not require special tools and equipment to control its geometry.

#### 4.2.2.1 Advantages

- Open up tolerance zone
- Offers flexibility to manufacturers to get more tolerances.
- Bonus Tolerance increases the part acceptance ratio.
- Assures part interchangeability
- Reduce part cost

#### 4.2.2.2 Disadvantages

- Part dimensional quality is a concern
- A tight fit is not possible
- Precision assembly is a big problem.

### 4.2.3 COMPARISON Of Conventional Tolerance and Geometric Tolerance

**FIG 4.2 COMPARISON Of Conventional Tolerance and Geometric Tolerance**

| Sl. No | Conventional Tolerance | Geometric Tolerance |
|---|---|---|
| 1. | It represents in a text form | It is a combination of symbols and characters to represent dimensions. |
| 2. | It is more clear to everyone because text is specified in drawing. | It ensures a clear and concise standard to explain tolerances. |
| 3. | Measurement might go wrong | Precise measurement can be ensured. |
| 4. | A dimensional tolerance is the total amount a specific dimension permitted to vary between maximum and minimum limits of size | It is the maximum or minimum variation from true geometric form or position which may be permitted in manufacture. |

| | | |
|---|---|---|
| 5. | Bonus tolerance cannot be obtained because there is no flexibility. | Bonus tolerance can be ensured due to more flexibility in measurements. |

## 4.3 INSPECTION OF GEOMETRIC DEVIATIONS

### 4.3.1 Straightness Measurement

**"A line is said to be straight over a given length, if variation in distance of all points lying on line from two planes perpendicular to each other and parallel to general direction of line, remains within the specified tolerance limits".**

The reference planes are so chosen that, their intersection is parallel to the straight line joining two points, which are located on the line to be tested and close to ends of the length under measurement. Tolerance on straightness of a line is defined as, "the maximum deviation in relation to the reference straight line going to the two extremities or ends of line under examination".

#### 4.3.1.1 Test for Straightness by Using Spirit Level and Auto-collimator

- Tests for straightness can be carried out by using spirit level or autocollimator.
- The above instruments determine the straightness of any surface by measuring the relative angular position of various adjacent sections of surface to be tested.
- For this purpose, initially a straight line is drawn on the surface under test. Then this drawn line is divided into number of equidistant sections.
- If spirit level is used, then length of each section should be equal to length of base of spirit level.
- If auto-collimator is used, then length of each section should be equal to length of base of plane reflector.

• Generally, the bases of spirit level block or reflector are fitted with two legs (or feets), such that,

(i) Feets or legs have line contact with the surface under test, and

(ii) Entire surface of base does not touch the surface under test.

This ensures that, angular deviation obtained is between the two specified points.

**(i) Spirit Level:**

• The block of spirit level is moved linearly on the surface to be tested, in number of steps. Every step chosen is equal to the pitch distance between centre lines of two feets.

• When block of spirit level is kept on a perfectly flat surface, we observe that, vapour bubble is resting at the middle and topmost position of glass tube indicating zero reading on the scale engraved on glass tube.

• But, when the block of spirit level is moved over surface test, then this vapour bubble moves away from the middle position due to irregularities in straightness of surface. This variation in the spirit level is measured, which gives the angular variation in the direction of block.

• Angular variation can be correlated in terms of the difference of height between two points by knowing the least count of level and length of the base.

• Limitation of spirit level: It can be used to find out variation in straightness of horizontal surface only.

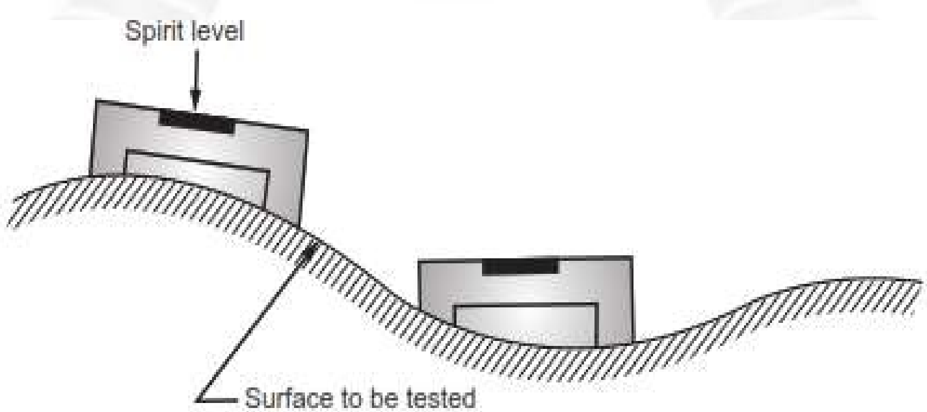



**Fig. 4.8** Test for Straightness by Using Spirit Level

[source: Metrology and Quality Control, Vinod Thombre Patil, Pg. No 212]

**(ii) Auto-collimator:**

• Auto-collimator can be used to measure the straightness of surface in any plane.

• Auto-collimator is placed at a distance of 0.5 to 0.75 metre from the surface to be tested. It is held in desired position on any rigid support, which is independent of the surface to be tested.

• A parallel beam from auto-collimator is projected along the length of surface to be tested.

• A block resting on two legs or feets is placed on the surface under test.

• Block carries a plane vertical reflector mounted on its extreme left side in such a way that, face of reflector is facing auto-collimator.

• Plane reflector and auto-collimator are arranged in such a way that,

(i) image of cross wires of the collimator appears very near to the centre of eyepiece, and

(ii) linear movement of reflector over the entire length of surface under test is completed.

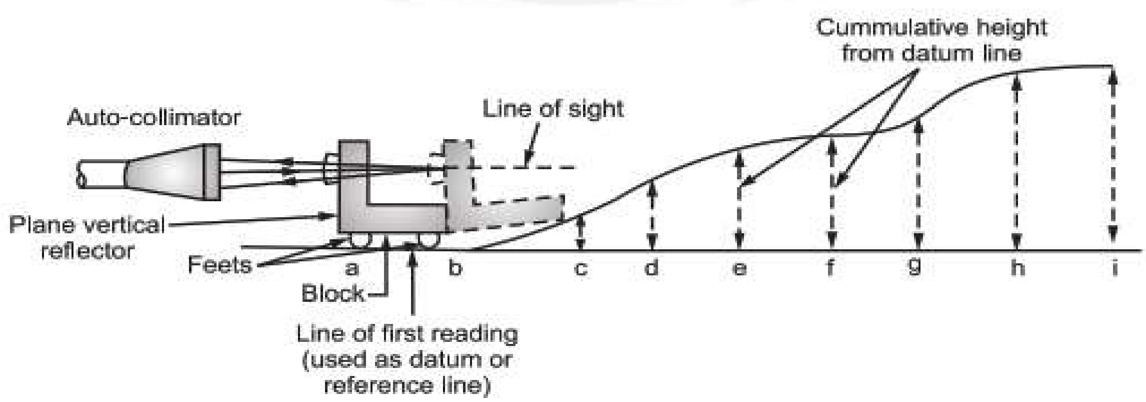



**Fig. 4.9** Auto-collimator checking straightness

• Now, the reflector is moved towards the other end of surface in steps equal to the centre distance between the two legs or feets. During this movement, the tilting of reflector is noted down in seconds from the eyepiece.

1 second of arc = 0.000006 mm/mm

• Now, the reflector is set at first position a-b (perfectly flat and straight) and first micrometer reading is noted down. This line is labelled as 'a-b' is treated as datum line or reference line. Successive readings at positions b-c, c-d, d-e …… and so on, are taken, till the plane reflector completes its linear movement over the entire length of surface under test.

**4.3.2 Flatness measurement**

Machine tool tables, which hold workpieces during machining, should have a high degree of flatness. Many metrological devices like the sine bar invariably need a perfectly flat surface plate.

Flatness error may be defined as the minimum separation of a pair of parallel planes that will just contain all the points on the surface. Figure 10.7 illustrates the measure of flatness error a. It is possible, by using simple geometrical approaches, to fit a best-fit plane for the macro surface topography. Flatness is the deviation of the surface from the best-fit plane.

According to IS: 2063-1962, a surface is deemed to be flat within a range of measurement when the variation of the perpendicular distance of its points from a geometrical plane (this plane should be exterior to the surface to be tested) parallel to the general trajectory of the plane to be tested remains below a given value. The geometrical plane may be represented either by means of a surface plane or by a family of straight lines obtained by the displacement of a straight edge, a spirit level, or a light beam. While there are quite a few methods for measuring flatness,

such as the beam comparator method, interferometry technique, and laser beam measurement, the following paragraphs explain the simplest and most popular method of measuring flatness using a spirit level or a clinometer.

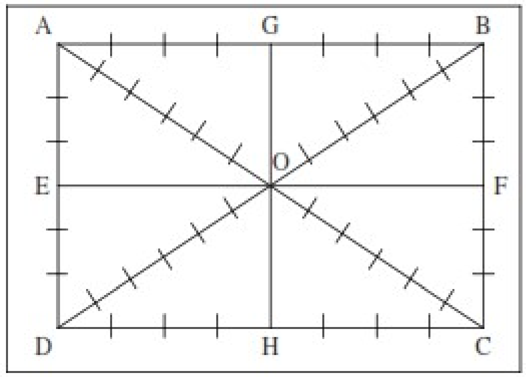



**Fig. 4.10** Grid lines for flatness Test

#### 4.3.2.1 Measurement of flatness error

Assuming that a clinometer is used for measuring angular deviations, a grid of straight lines, as shown in Fig. 4.3., is formulated. Care is taken to ensure that the maximum area of the flat table or surface plate being tested is covered by the grid. Lines AB, DC, AD, and BC are drawn parallel to the edges of the flat surface; the two diagonal lines DB and AC intersect at the centre point O. Markings are made on each line at distances corresponding to the base length of the clinometer.

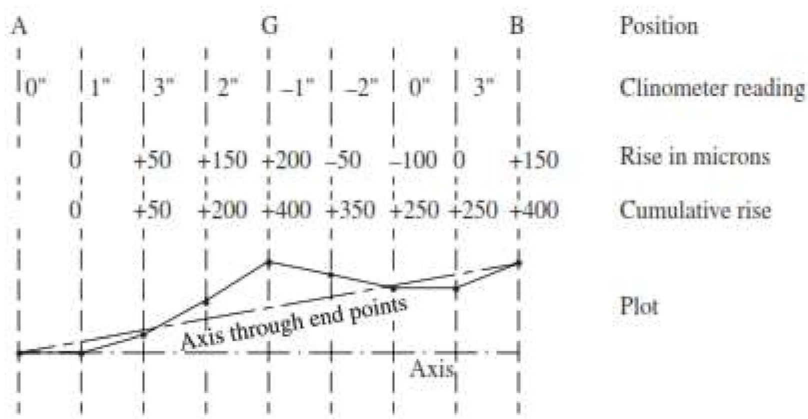



**Fig. 4.11** Straightness plot for line AB

The following is a step-by-step procedure to measure flatness error:

1. Carry out the straightness test, as per the procedure described in Chapter 5, on all the lines and tabulate the readings up to the cumulative error column. Figure 10.9 gives an example of line AB.

2. We know that a plane is defined as a 2D entity passing through a minimum of three points not lying on the same straight line. Accordingly, a plane passing through the points A, B, and D is assumed to be an arbitrary plane, relative to which the heights of all other points are determined. Therefore, the ends of lines AB, AD, and BD are corrected to zero and the heights of points A, B, and D are forced to zero.

3. The height of the centre 'O' is determined relative to the arbitrary plane ABD. Since O is also the mid-point of line AC, all the points on AC can be fixed relative to the arbitrary plane ABD. Assume A = 0 and reassign the value of O on AC to the value of O on BD. This will readjust all the values on AC in relation to the arbitrary plane ABD.

4. Next, point C is fixed relative to the plane ABD; points B and D are set to zero. All intermediate points on BC and DC are also adjusted accordingly.

5. The same procedure applies to lines EF and GH. The midpoints of these lines should also coincide with the known midpoint value of O.

6. Now, the heights of all the points, above and below the reference plane ABD, are plotted as shown in Fig. 4.5. Two lines are drawn parallel to and on either side of the datum plane, such that they enclose the outermost points. The distance between these two outer lines is the flatness error.

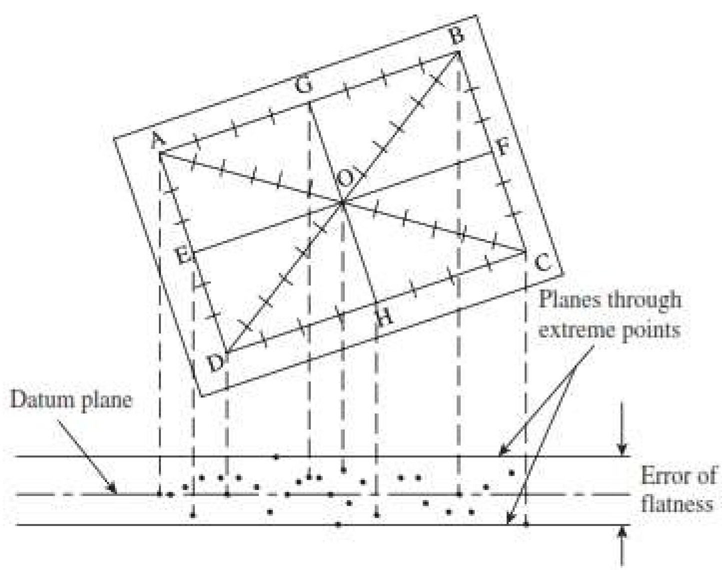



**Fig. 4.12** Plot of heights of all points with reference to the datum plane ABD

Some authors argue that the reference plane ABD that is chosen in this case may not be the best datum plane. They recommend further correction to determine the minimum separation between a pair of parallels that just contains all the points on the surface. However, for all practical purposes, this method provides a reliable value of flatness error, up to an accuracy of 10 µm.

## 4.4 Roundness measurement

Roundness is defined as a condition of a surface of revolution. Where all points of the surface intersected by any plane perpendicular to a common axis in case of cylinder and cone.

Roundness is a geometric aspect of surface metrology and is of great importance because the number of rotational bearings in use is much more than that of linear bearings. Many machine parts, such as a machine spindle or the hub of a gear, have circular cross sections; these parts should have roundness with a very high degree of accuracy.

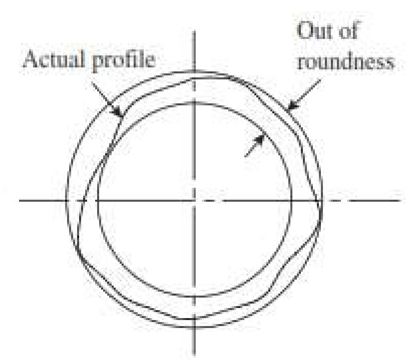



**Fig. 4.13 Out of roundness**

Roundness is defined as a condition of surface of revolution where all points of the surface intersected by any plane perpendicular to a common axis are equidistant from the axis. It is obvious that any manufactured part cannot have perfect roundness because of limitations in the manufacturing process or tools; we need to determine how much deviation from perfect roundness can be tolerated so that the functional aspects of the machine part are not impaired. This leads to the definition of out of roundness as a measure of roundness error of a part. It is the radial distance between the minimum circumscribing circle and the maximum inscribing circle, which contain the profile of the surface at a section perpendicular to the axis of rotation.

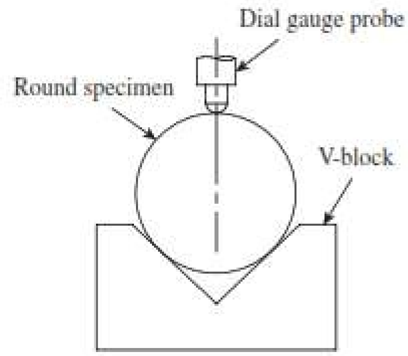



**Fig. 4.14 Use of a V-block for measuring out of roundness**

Roundness error can be measured in various ways. Accessories required for the measurement comprise a surface plate, a V-block, and a dial gauge with a stand. The V-block is kept on the surface plate, and the cylindrical work part is positioned on the V-block. Care should be taken to ensure that the axis of the work part is parallel to the edges of the 'V' of the V-block. The dial gauge is mounted on its stand and the plunger is made to contact the surface of the work part. A light contact pressure is applied on the plunger so that it can register deviations on both the plus and minus sides. The dial gauge reading is set to zero. Now the work part is slowly rotated and the deviations of the dial indicator needle on both the plus and minus sides are noted down. The difference in reading for one complete rotation of the work part gives the value of out of roundness.

### 4.4.1 Devices used for measurement of roundness

1) Diametral gauge.

2) Circumferential conferring gauge => a shaft is confined in a ring gauge and rotated against a set indicator probe.

3) Rotating on center

4) V-Block

5) Three-point probe.

6) Accurate spindle.

#### 4.4.1.1. Diametral method:

- The measuring plungers are located 180° a part and the diameter is measured at several places.
- This method is suitable only when the specimen is elliptical or has an even number of lobes.
- Diametral check does not necessarily disclose effective size or roundness.
- This method is unreliable in determining roundness.

**Circumferential confining gauge:**

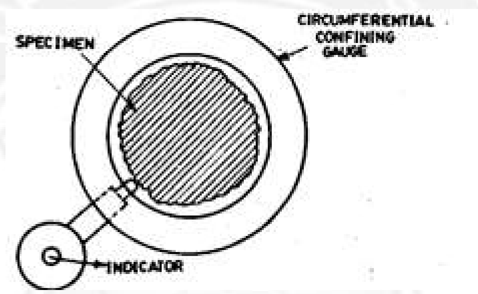



**Fig. 4.15 Circumferential confining gauge**

- It is useful for inspection of roundness in production.
- This method requires highly accurate master for each size part to be measured. The clearance between part and gauge is critical to reliability.
- This technique does not allow for the measurement of other related geometric characteristics, such as concentricity, flatness of shoulders etc.

**Rotating on centers:**

- The shaft is inspected for roundness while mounted on center.
- In this case, reliability is dependent on many factors like angle of centers, alignment of centres, roundness and surface condition of the centres and centre holes and run out of piece.
- Out of straightness of the part will cause a doubling run out effect and appear to be roundness error,

**V-Block:**

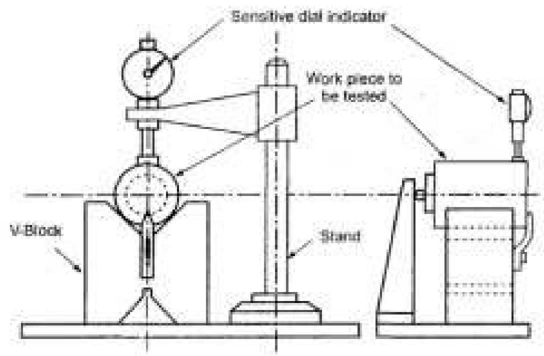



**Fig. 4.16 V-Block**

- The V block is placed on surface plate and the work to be checked is placed upon it.
- A diameter indicator is fixed in a stand and its feeler made to rest against the surface of the work. The work is rotated to measure the rise on fall of the work piece.
- For determining the number of lobes on the work piece, the work piece is first tested in a 60° V-Block and then in a 90° V-Block.
- The number of lobes is then equal to the number of times the indicator pointer deflects through 360° rotation of the work piece.

**Limitations:**

a) The circularity error is greatly by affected by the following factors.

(i) If the circularity error is i\e, then it is possible that the indicator shows no variation.

(ii) Position of the instrument i.e. whether measured from top or bottom.

(iii) Number of lobes on the rotating part.

b) The instrument position should be in the same vertical plane as the point of contact of the part with the V-block.

c) A leaf spring should always be kept below the indicator plunger and the surface of the part.

### 4.4.1.2. Three-point probe

- The fig. shows three probes with 120° spacing is very, useful for determining effective size they perform like a 60° V-block.
- 60° V-block will show no error for 5 a 7 lobes magnify the error for 3-lobed parts show partial error for randomly spaced lobes.

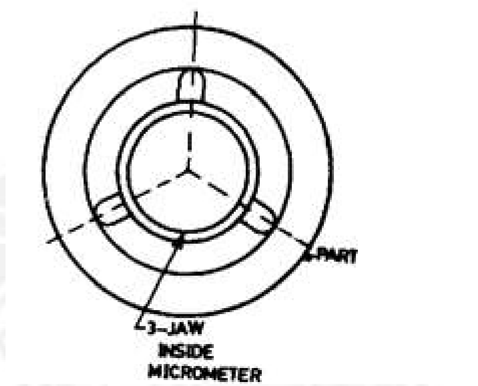



**Fig. 4.17 Three-point probe**

### 4.4.1.3 Roundness measuring spindle

There are following two types of spindles used.

### 4.4.1.4 Overhead spindle:

- Part is fixed in a staging plat form and the overhead spindle carrying the comparator rotates separately from the part.
- It can determine roundness as well as camming (Circular flatness). Height of the work piece is limited by the location of overhead spindle.
- The concentricity can be checked by extending the indicator from the spindle and thus, the range of this check is limited.

### 4.4.1.5 Rotating table:

Spindle is integral with the table and rotates along with it. The part is placed over the spindle and rotates past a fixed comparator

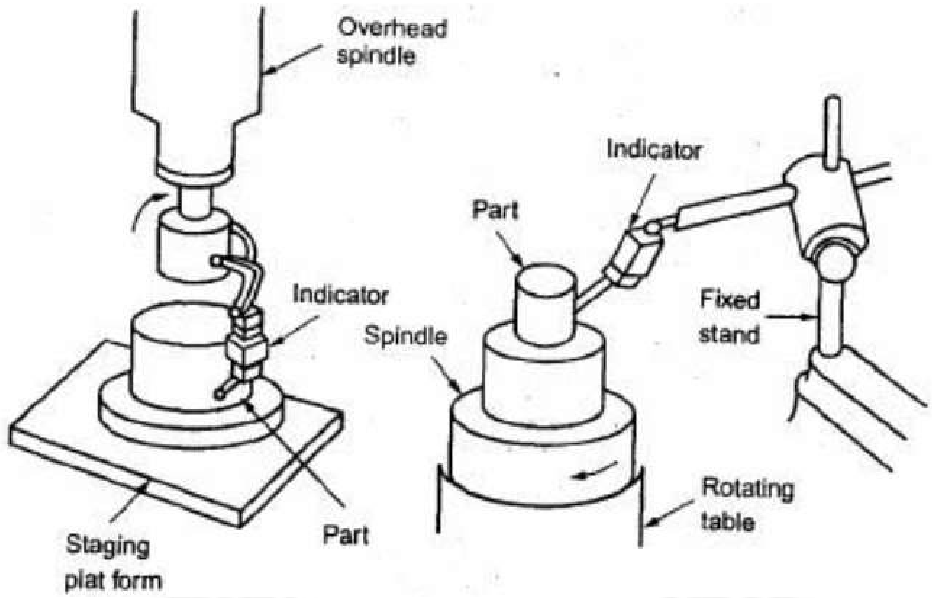



**Fig. 4.18 Roundness measuring spindle Rotating table**

### 4.4.1.4.3 Modern Roundness Measuring Instruments

- This is based on use of microprocessor to provide measurements of roundness quickly and in a simple way; there is no need of assessing out of roundness. Machine can do centering automatically and calculate roundness and concentricity, straightness and provide visual and digital displays.
- A computer is used to speed up calculations and provide the stand reference circle.

**(i) Least square circle:**

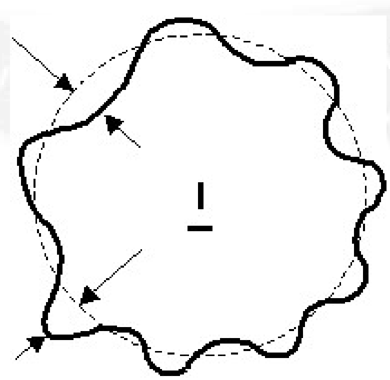


**Fig. 4.19 Least square circle**

- The sum of the squares of a sufficient no. of equally spaced radial ordinates measured from the circle to the profile has minimum value.

- The center of such circle is referred to as the least square center. Out of roundness is defined as the radial distance of the maximum peak from the circle (P) plus the distance of the maximum valley from this circle

**(ii) Minimum zone or Minimum radial separation circle:**

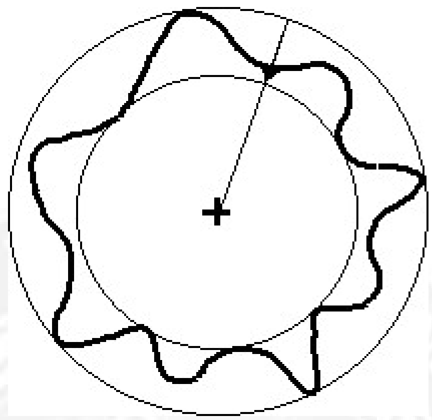


**Fig. 4.20 Minimum zone circle**

- These are two concentric circles. The value of the out of roundness is the radial distance between the two circles.
- The center of such a circle is termed as the minimum zone center. These circles can be found by using a template.

**(iii) Maximum inscribed circle:**

- This is the largest circle. Its center and radius can be found by trial and error by compare or by template or computer. Since V = 0 there is no valleys inside the circle.

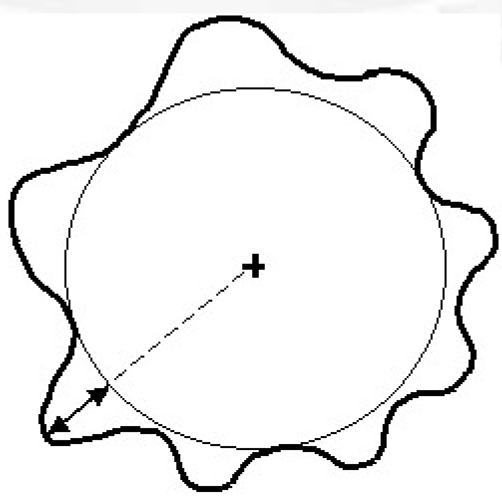


**Fig. 4.21 Maximum inscribed circle**

### (iv) Minimum circumscribed circles:

- This is the smallest circle. Its center and radius can be found by the previous method since P = 0 there is no peak outside the circle.
- The radial distance between the minimum circumscribing circle and the maximum inscribing circle is the measure of the error circularity. The fig shows the trace produced by a recording instrument.

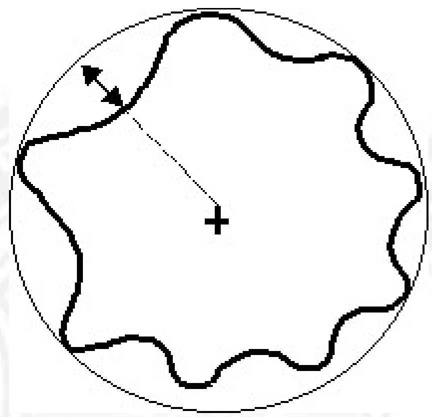


**Fig. 4.22 Minimum circumscribed circles**

- This trace to draw concentric circles on the polar graph which pass through the maximum and minimum points in such way that the radial distance be minimum circumscribing circle containing the trace or the n inscribing circle which can fitted into the trace is minimum.
- The radial distance between the outer and inner circle is minimum is considered for determining the circularity error.
- Assessment of roundness can be done by templates.
- The out off roundness is defined as the radial distance of the maximum peak (P) from the least square circle plus the distance of the maximum valley (V) from the least square circle.
- All roundness analysis can be performed by harmonic and slope analysis.

## 4.5 SURFACE FINISH MEASUREMENT

### 4.5.1 Introduction:

- When we are producing components by various methods of manufacturing

process it is not possible to produce perfectly smooth surface and some