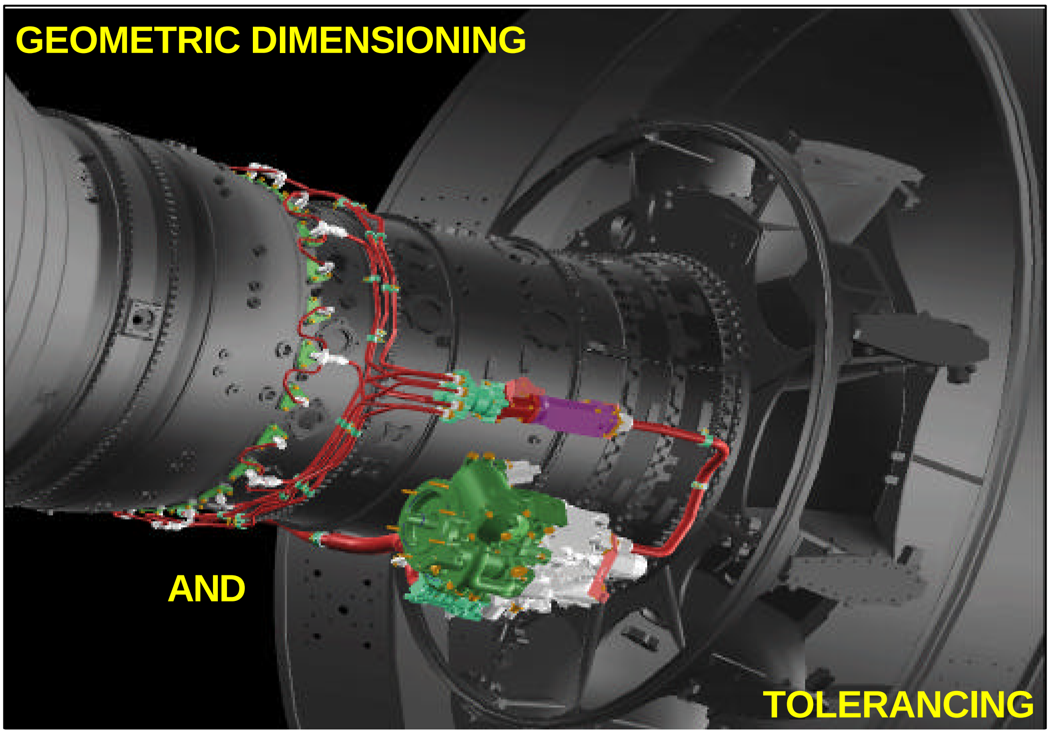


In order to properly plan the processes to make a part, you must first understand the part designer's intention. Geometric dimensioning and tolerancing is a method by which designers specify the geometric form of parts.

The definitions and convention used in this course are based on ISO standard.

It is assumed that you have a basic understanding of dimensional metrology and are familiar with measuring equipment, such as gauge blocks, dial indicators, optical comparators, etc. If you need to revise dimensional metrology then you can refer to:

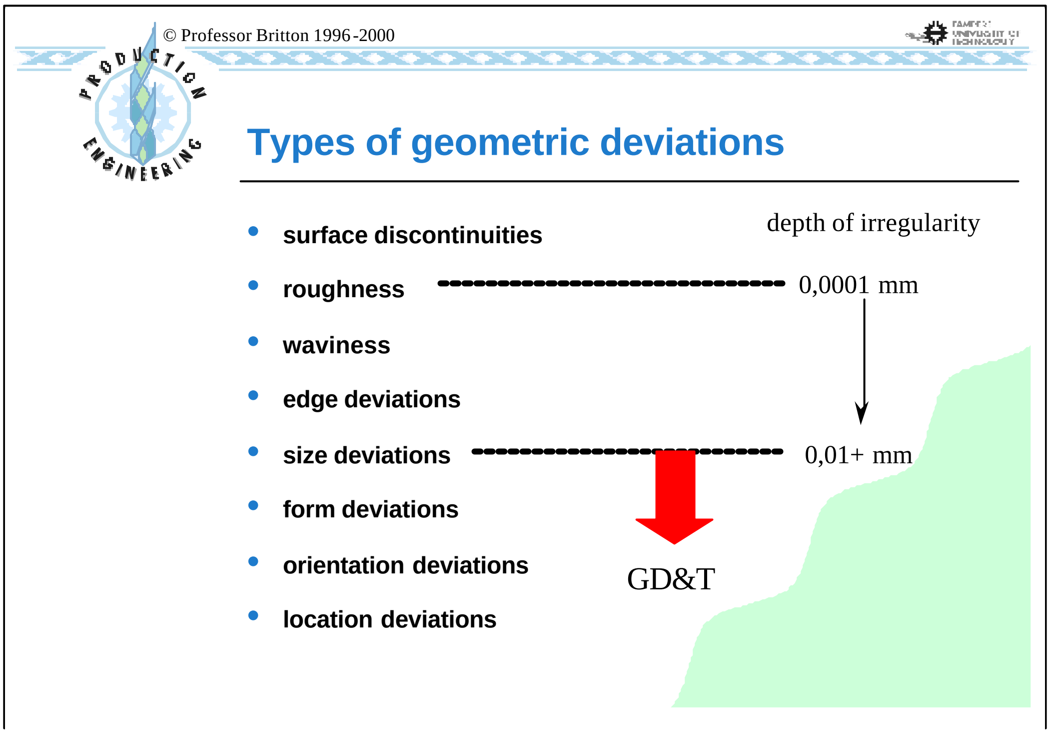
Busch, T. 1989. *Fundamentals of Dimensional Metrology.* Delmar Publishers: New York.

Topics

- **Introduction**
- **Interpreting Geometric Tolerances**
- **Principles of Inspection**

The goal of this part of the course is to describe the different types of geometric tolerances and how actual parts can be inspected to ensure compliance.

At the end of this part of the course you will be able to interpret a drawing or model containing GD&T symbols and know (in principle) how to verify whether a part complies to the design specification.



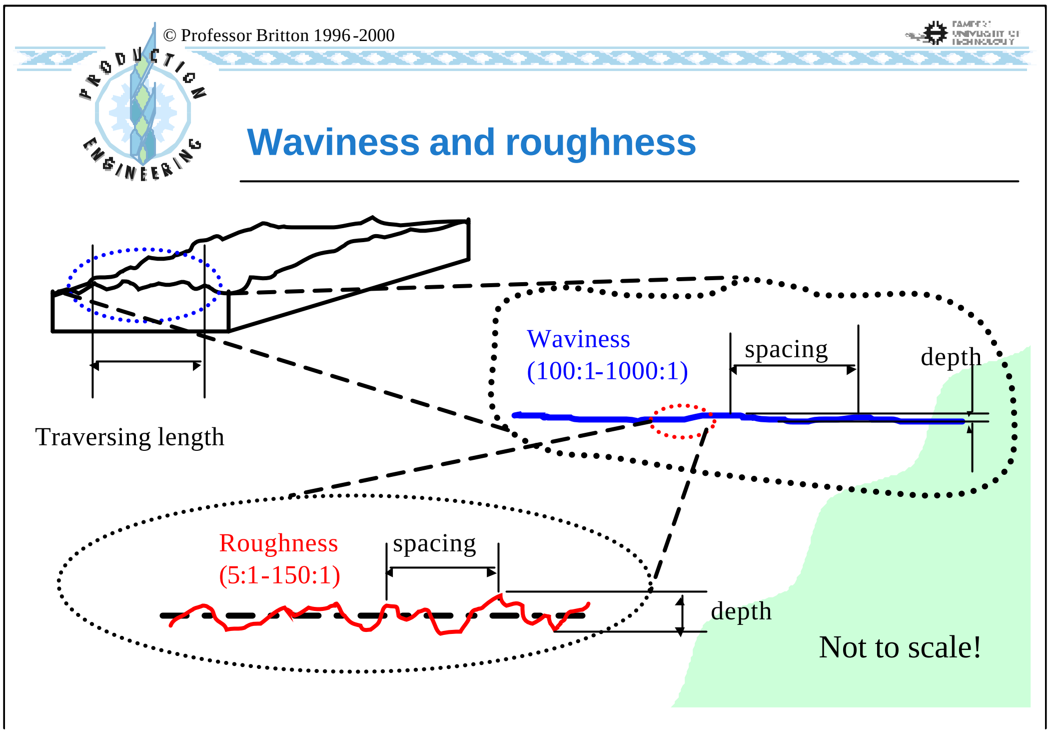
This slide lists the different kinds of geometric deviations. Only the last three are part of Geometric Dimensioning and Tolerancing (GD&T).

Size deviations are controlled by normal dimensioning practices.

Roughness and waviness require separate specifications.

In addition to these requirements the designer also needs to specify the material to be used and the special conditions relating to the material.

Thus complex engineering drawings and models may require you to understand several different conventions, each controlling different aspects of the design.



One technique for distinguishing between roughness, waviness and form control is by the spacing to depth ratio. The slide illustrates this for waviness and roughness.

Roughness is measure at a much smaller scale than waviness and is an indication of very small local imperfections in a surface. Roughness is produced by the direct effect of the cutting process (chip formation), deformation from blasting, crystallization, corrosion and other chemical processes. The spacing to depth ratio between successive peaks is of the order 5:1 to 150:1.

Waviness refers to periodic regularities in the surface of a part, but at a scale smaller than that which is controlled by GD&T. The spacing to depth ratio between successive peaks is of the order 100:1 to 1000:1. It is produced by eccentric fixturing, form deviations in the cutting tool and vibration.

Size deviation

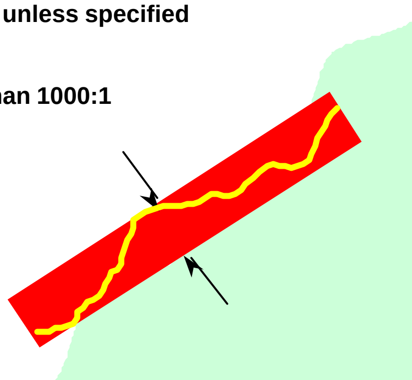
- deviation of actual local size from nominal linear size or from nominal angular size
- actual local linear sizes are assessed by 2-point measurements
- actual local angular sizes are assessed by angular measurements of averaged lines
- assessed over entire geometric element
- produced mainly by imprecise adjustment of machine tool and cutting or process conditions

Size deviation is controlled on engineering drawings and CAD models by stating the nominal size and a tolerance which defines the maximum permissible deviation from the nominal size. The nominal size and tolerance define the permissible design limits (upper and lower) within which all size measurements must lie.

The tolerance values either side of the nominal have the same values (bilateral tolerancing) or different values (unilateral tolerancing). Unilateral tolerancing is used to bias a size towards one of the design limits in order to optimise performance. On the other hand, as you will find out later, manufacturing processes are controlled using bilateral tolerances. Hence it is normal practice to convert unilateral design tolerances to bilateral tolerances during process planning.

Form deviation

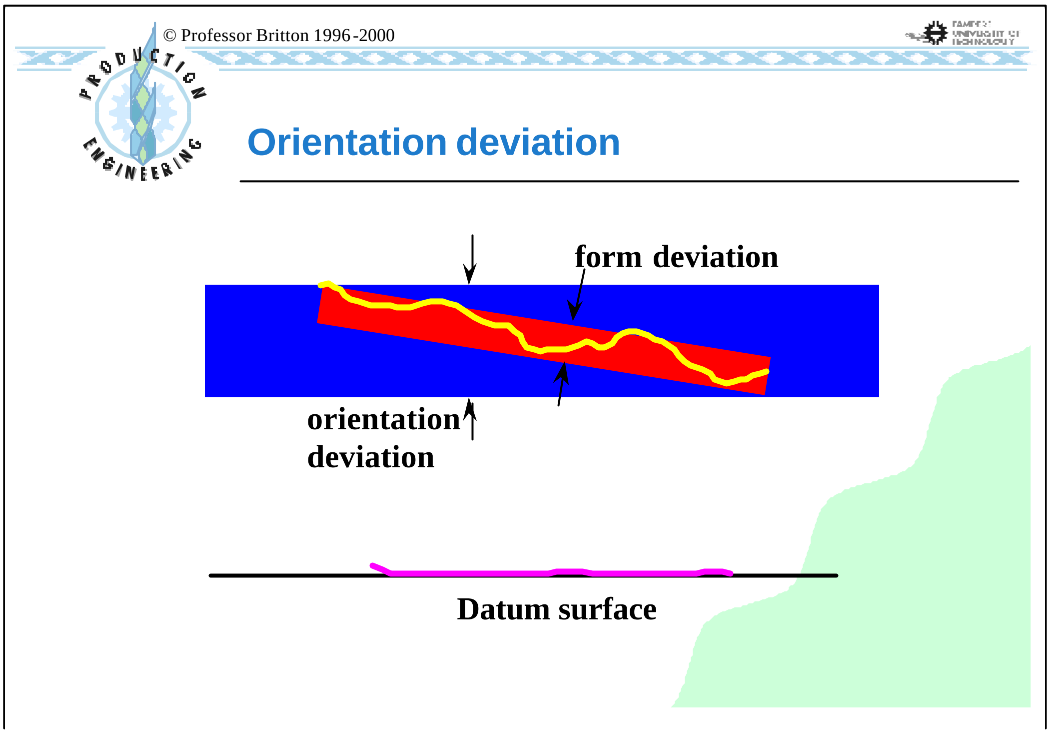
- deviation of a feature (geometrical element, surface or line) from its nominal form
- assessed over entire feature, unless specified otherwise
- spacing/depth ratio greater than 1000:1
- produced by
  - errors in guideways and bearings
  - deflection of cutting tool and fixture
  - fixture errors
  - tool and fixture wear



Form deviation is the deviation of a feature from its nominal (defined) form (shape). The feature being controlled may be a line on a surface, a surface or a geometric element, e.g., an axis. Form deviations are specified without reference to any other features.

All geometric deviations are assumed to apply over the entire feature unless otherwise specified. Thus a straightness control applied to lines on a surface means the entire surface is to be controlled, not just a small part of it.

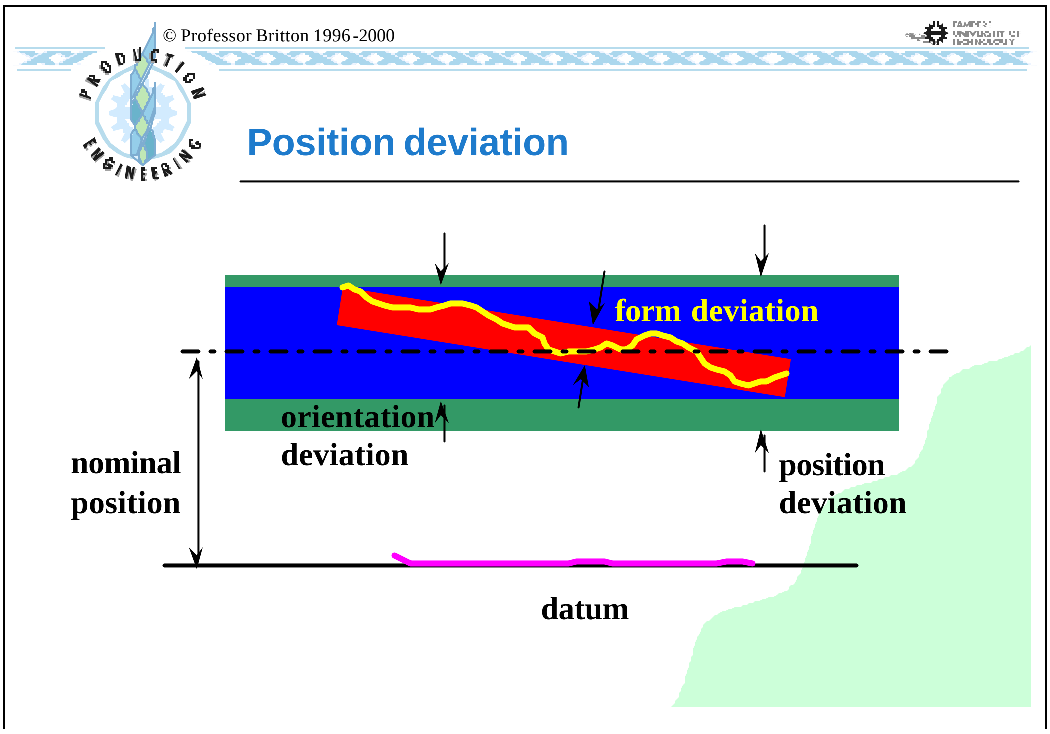
Form deviations are produced by the conditions stated in the slide.



An orientation deviation is a deviation from nominal form AND orientation. An orientation deviation must be related to one or more other features, which are referred to as datums. The datums must be defined first before an orientation tolerance can be specified.

The slide shows a parallel orientation. The feature being controlled must be lie within a tolerance zone that is oriented parallel to the datum surface.

Orientation deviations are produced by the same manufacturing conditions that produce form deviations and in addition can be produced by setup errors when machining datums are changed.

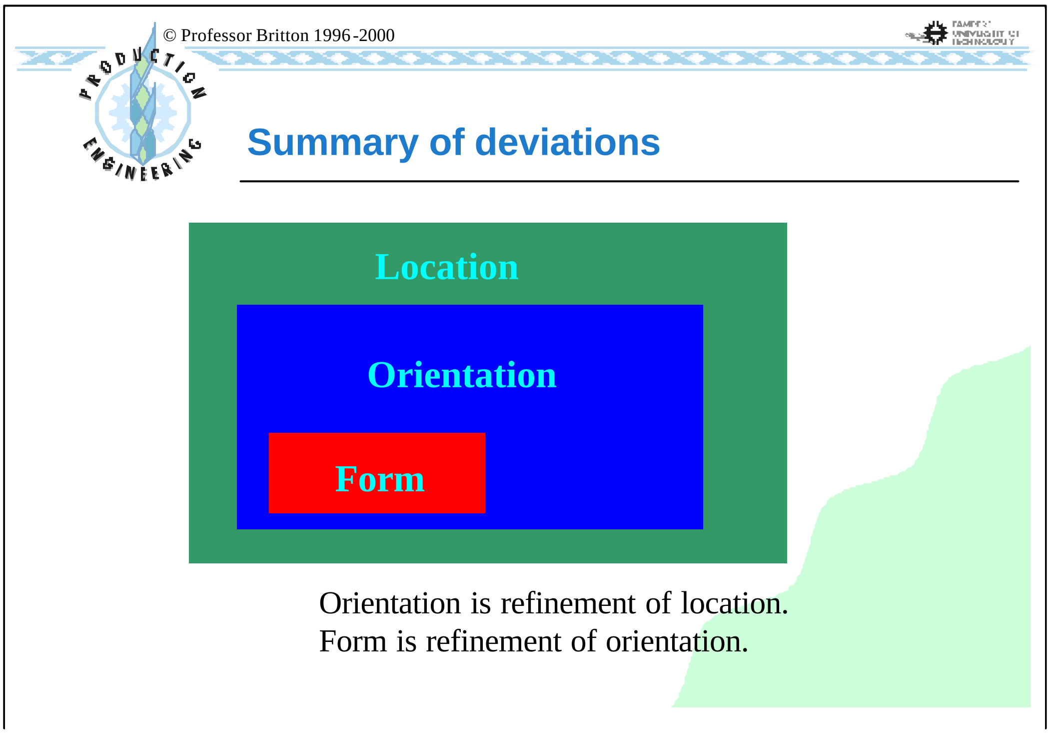


Position deviation is controlled relative to a datum. A nominal position (distance and direction) is given relative to a datum. The postion tolerance defines the maximum permissible deviation from this nominal position. The deviation is shown in green.

Note that position deviation encompasses both orientation deviations and form deviations.

The deviations are asssessed over the feature unless otherwise specified.

Position deviations are produced by the same manufacturing conditions that affect size, form and orientation deviations.



The different kinds of geometric deviations form an hierarchy. Location is the most general form of control. It not only controls location but orientation and form as well. Next is orientation control. This controls both orientation and form.

Orientation is a refinement of location. Orientation control is used when position control is insufficient.

Form is a refinement of orientation. Form control is used when orientation control is insufficient.

Sometimes it is necessary to control several deviations simultaneously. For example it may be necessary to specify both form and orientation tolerances to obtain the necessary shape require to meet the design requirements. This is referred to as composite tolerancing.

Form tolerances must be smaller than orientation tolerances which must be smaller than location tolerances.

© Professor Britton 1996-2000

When to use geometric tolerancing

- **not needed if dimensional tolerances and the manufacturing process provide adequate control**
- **is needed**
  - when part features are critical to function or interchangeability
  - when errors of shape & form must be held within tighter limits than normally expected from the manufacturing process
  - when functional gauging techniques are to be used
  - when datum references are required to ensure consistency between design, manufacture and verification operations
  - when computerization techniques in design and manufacture are used

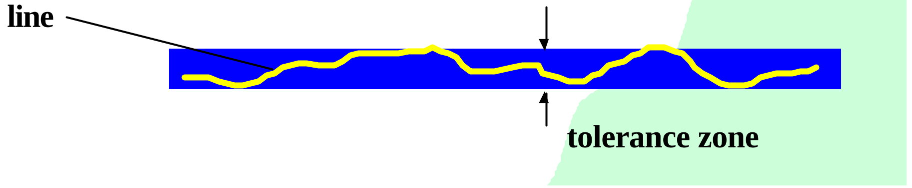
Use of GD&T to control form of a part increases cost because extra control is required for the manufacturing processes and additional inspection is required to ensure compliance. Thus it should only be used when necessary.

In order to apply GD&T properly a designer must know about and understand the manufacturing processes that will be used to make the part. All processes provide geometric control to some degree. The decision to apply GD&T or not depends to a large degree on whether the normal control provided by a process meets the design functional requirements: if it can meet requirements do not use GD&T, otherwise use GD&T.

In addition GD&T could be used to ensure consistency of datum references between design, manufacture and verification operations, and also when computer tools are used during design and manufacture.

Geometric tolerances

- are maximum permissible variation in
  - form
  - orientation
  - location
  - runout
- tolerance value represents the width or diameter of the tolerance zone within which a point, line, surface or feature must lie



There are four different kinds of geometric tolerances:

form: controls the shape of a feature

orientation: controls orientation and form

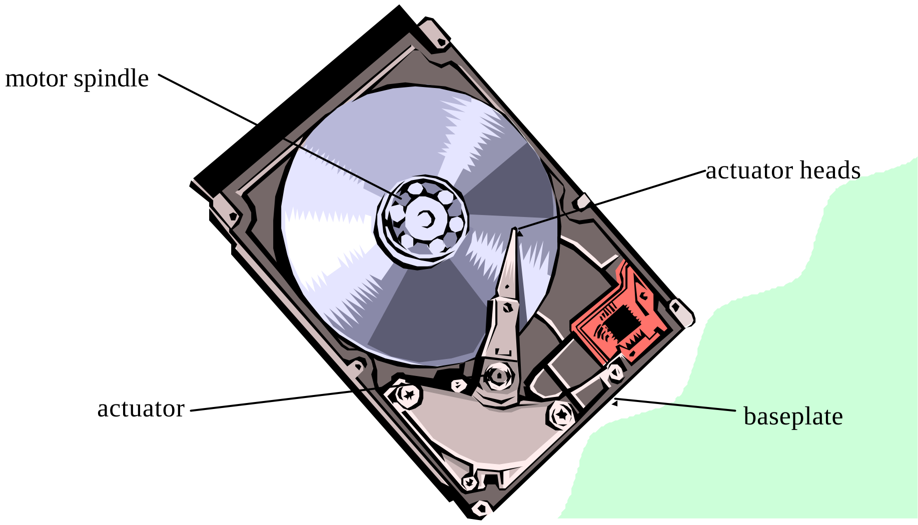
location: controls position, orientation and form

runout: controls position, orientation and form

The tolerance zone is defined by two parallel or concentric surfaces whose distance apart is defined by a width, or the diameter of a circle or sphere used to generate the surfaces. A diameter symbol is used to indicate the latter condition.

© Professor Britton 1996-2000

Geometric deviations in a disk drive assembly



Lets take a disk drive as an example and see what geometric deviations affect its performance.

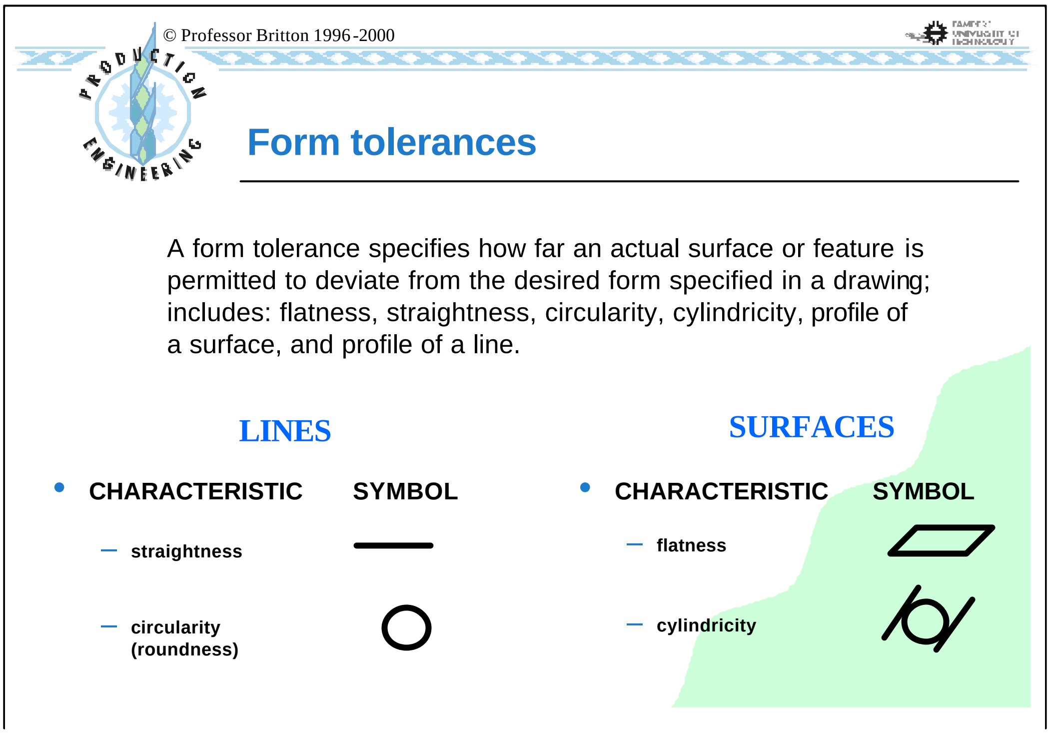
A disk drive consists of disks attached to a motor spindle which spins them around. The motor spindle is mounted in a base plate. An actuator, mounted on a bearing, supports the heads that read data from the disks. The actuator bearing is mounted to the base plate. The following deviations can occur:

- disks relative to disk motor spindle (must be coaxial and perpendicular)
- disk motor spindle relative to base plate support (must be perpendicular)
- base plate motor support relative to base plate actuator support (must be parallel and relative positioning must be accurate)
- actuator bearing relative to base plate support (must be perpendicular)
- actuator relative to actuator bearing (must be coaxial)
- read/write heads relative to actuator (must be perpendicular)
- read/write heads relative to each other (holes must line up)

INTERPRETING GEOMETRIC TOLERANCES

These notes define the symbols and methods for representing geometric tolerances on drawings and engineering models.

The symbols are given in a separate handout.



A form tolerance specifies how far an actual surface or feature is permitted to deviate from the desired form specified in a drawing; includes: flatness, straightness, circularity, cylindricity, profile of a surface, and profile of a line.

Profile Tolerances

LINES

- **CHARACTERISTIC** **SYMBOL**
  - profile of a line

SURFACES

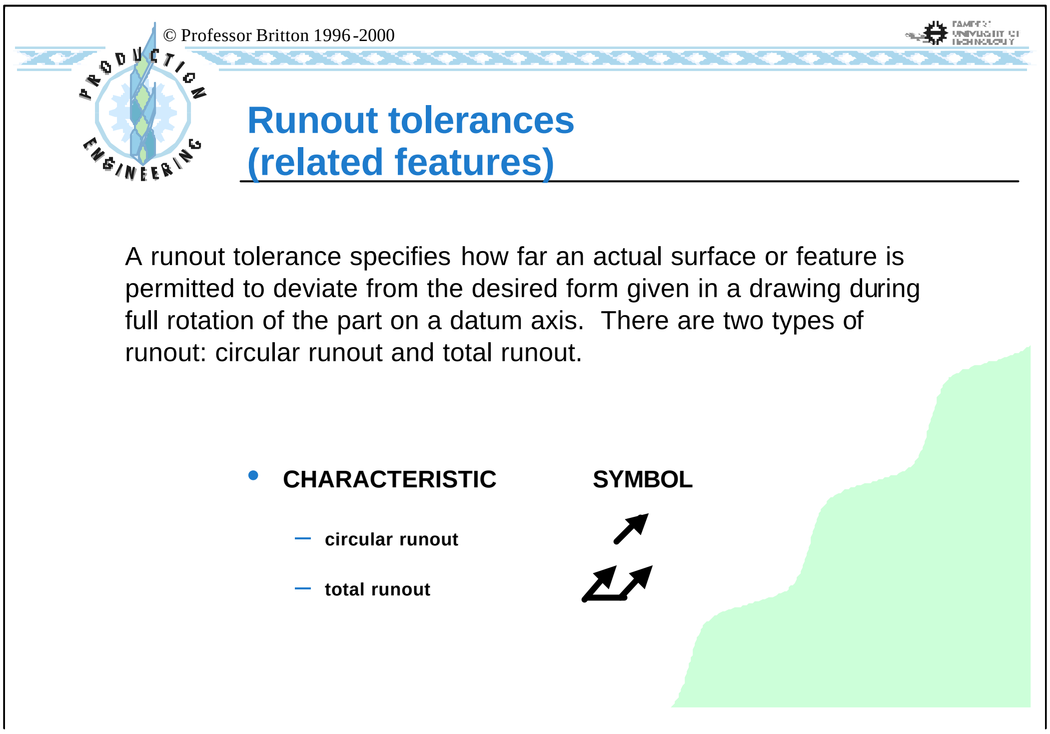
- **CHARACTERISTIC** **SYMBOL**
  - profile of a surface

Orientation tolerances (related features)

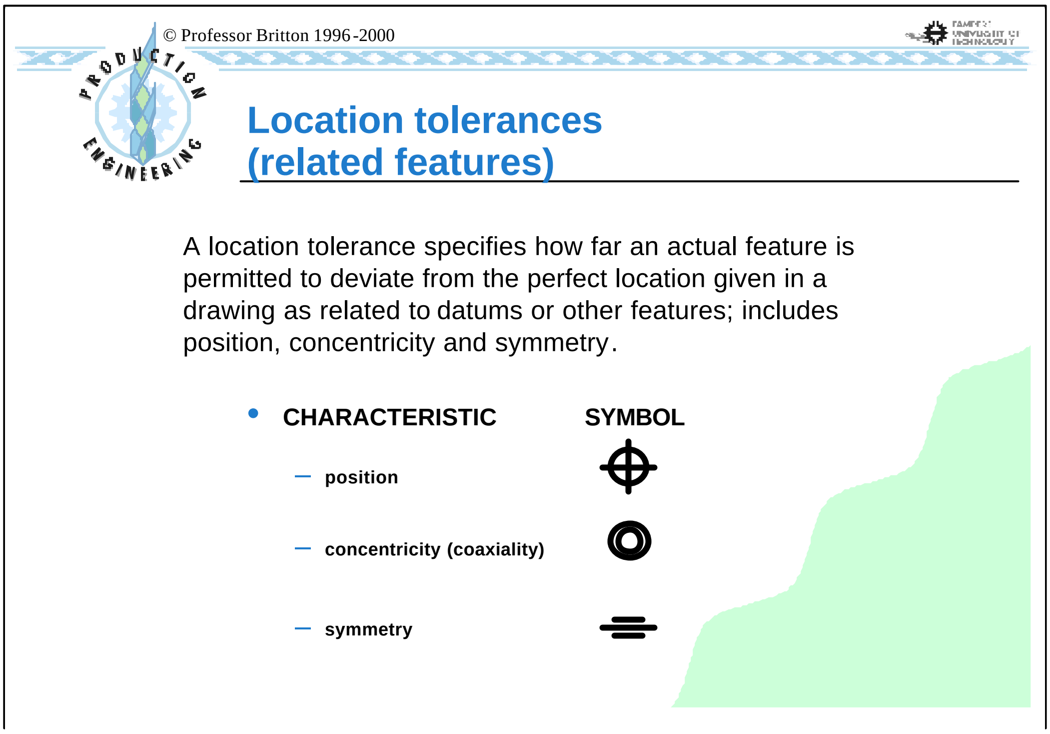
An orientation tolerance specifies how far the actual orientation between two features is permitted to deviate from the perfect orientation given in the drawing; includes perpendicularity, angularity, and parallelism.

CHARACTERISTIC	SYMBOL
angularity	∠
perpendicularity	⊥
parallelism	//

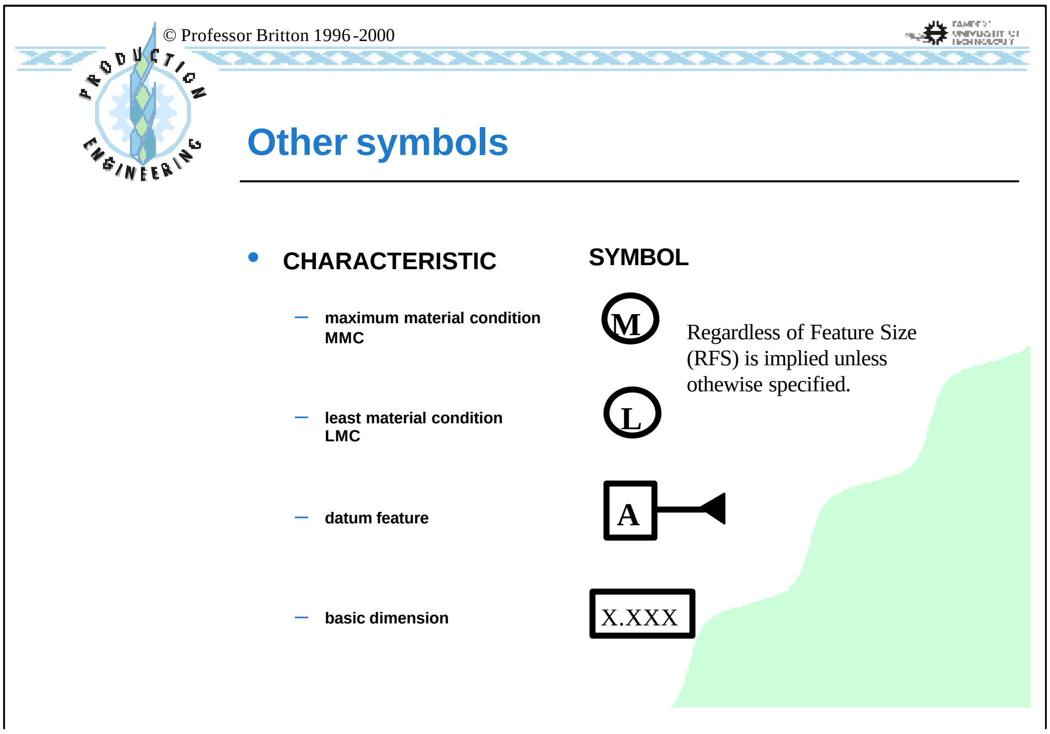
An orientation tolerance specifies how far the actual orientation between two features is permitted to deviate from the perfect orientation given in the drawing; includes perpendicularity, angularity, and parallelism.



A runout tolerance specifies how far an actual surface or feature is permitted to deviate from the desired form given in a drawing during full rotation of the part on a datum axis. There are two types of runout: circular runout and total runout.



A location tolerance specifies how far an actual feature is permitted to deviate from the perfect location given in a drawing as related to datums or other features; includes position, concentricity and symmetry.



The previous slides have shown the main symbols. Additional information can be provided using other symbols. Four important symbols are shown in the slide above.

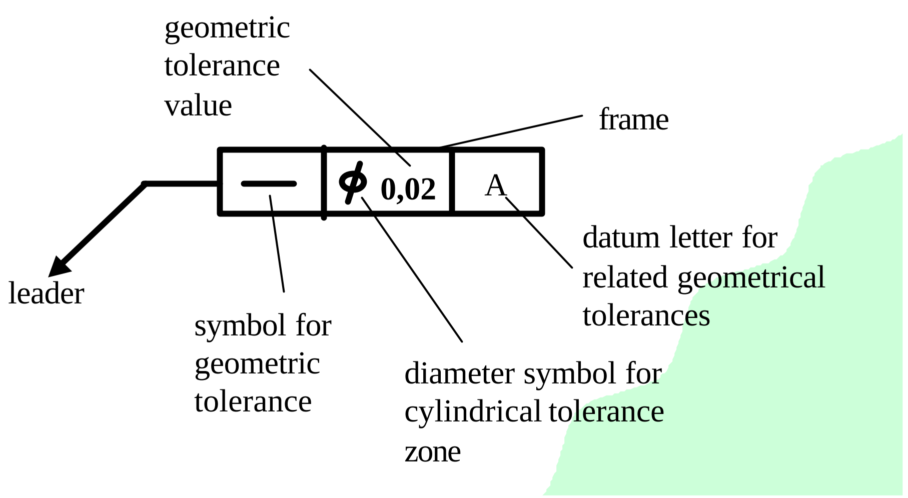
The maximum material condition is used to indicate the size of a feature when the feature is made such that the part contains the maximum amount of material, for example, this would be the largest shaft or the smallest hole. Maximum material condition is very important because manufacturing and inspection costs can be reduced considerably using this specification and yet still ensure design functional requirements are met. This aspect will be discussed in detail later on.

The least material condition indicates the size of a feature when it is made such the part contains the least amount of material, for example, this would be the smallest shaft or the largest hole.

The datum feature symbol shows how to indicate a feature that is to be used as a datum for controlling other features.

Dimensions that are controlled by geometric tolerances are indicated as a basic dimension. The last symbol is the basic dimension symbol.

Feature control frame



The symbols are placed in a feature control frame which points to the feature-being-controlled with an arrow. The parts of the frame are defined above.

Features of size & MMC

- have a size tolerance e.g., diameter of a shaft or hole
- geometric tolerance must be referred to a specific size
- maximum material condition is used to refer to the size limit for the feature when the part contains the maximum amount of material
  - smallest hole
  - largest shaft

Features of size have a size tolerance, e.g., the diameter of a shaft or hole. These different sizes need to be taken into account during GD&T. One way to do this is to use the Maximum Material Condition (MMC). This condition is defined as the size of the feature when the part contains the maximum amount of material – smallest hole or largest shaft.

Parts are combined into assemblies. It is important to ensure that mating parts mate together under all size and GD&T conditions. The MMC ensures this by allowing the position (geometric) tolerance to increase as a feature of size moves away from its MMC towards the Least Material Condition (LMS). That is, the tolerance of size is allowed to be taken up by the position tolerance. This greatly reduces the manufacturing and inspection costs, especially the latter, as functional gauging can be used (see next slide).

MMC is referred to as functional tolerancing as it is based on meeting design functional requirements for mating parts.

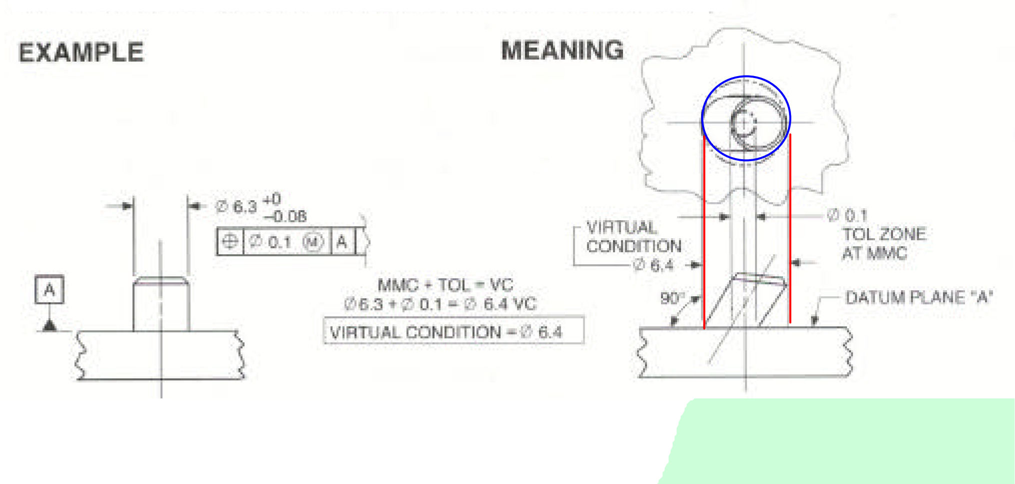
MMC applies to both controlled features and datum features.

Example MMC callout

This slide shows how MMC is indicated in the drawing callout. The interpretation is that hole position is to be measured at MMC. Datum A (another hole) is to be at MMC condition when it is used as a datum for this controlled feature.

The advantage of this arrangement is that two gauge pins can be used. One for datum A and the other for the feature-being-controlled.

Virtual condition for a pin



Reproduced from Foster

<u>Maximum Material Condition</u>: State of a feature when it is at the limit of size such that the part contains the maximum amount of material.

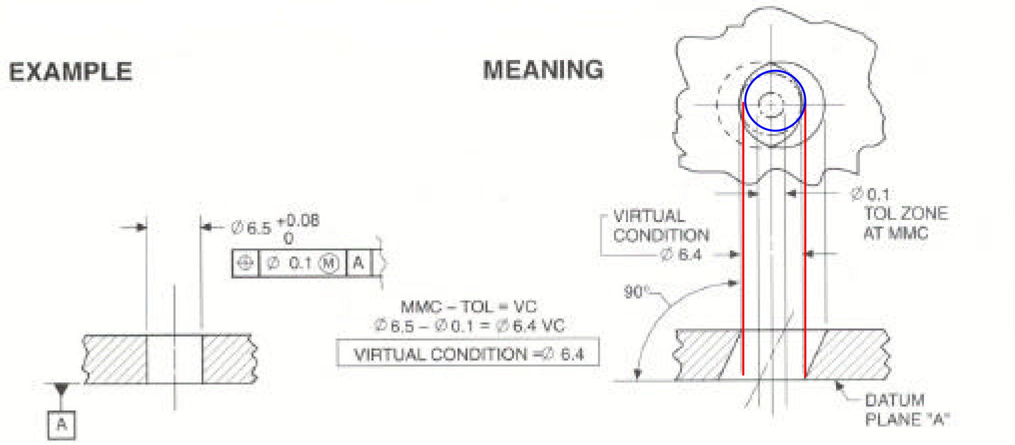
<u>Maximum Material Size</u>: Dimension defining the MMC of a feature; smallest size for a hole and largest size for a shaft.

<u>Virtual Condition (VC)</u>: The size generated by maximum material size and geometric tolerance; the size that guarantees mating. It is the size of the functional gauge pin.

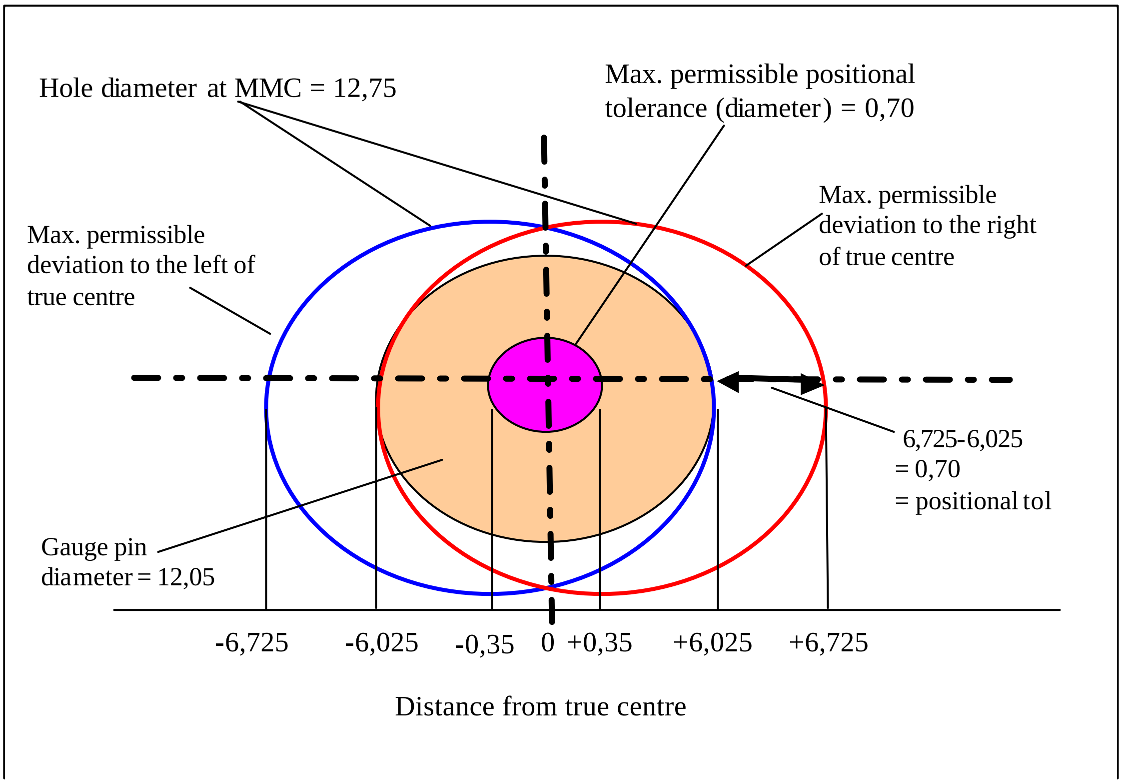
for shaft, VC=MMS+geometric tolerance (this slide)

for hole, VC=MMS-geometric tolerance (next slide)

Virtual condition for a hole



Reproduced from Foster



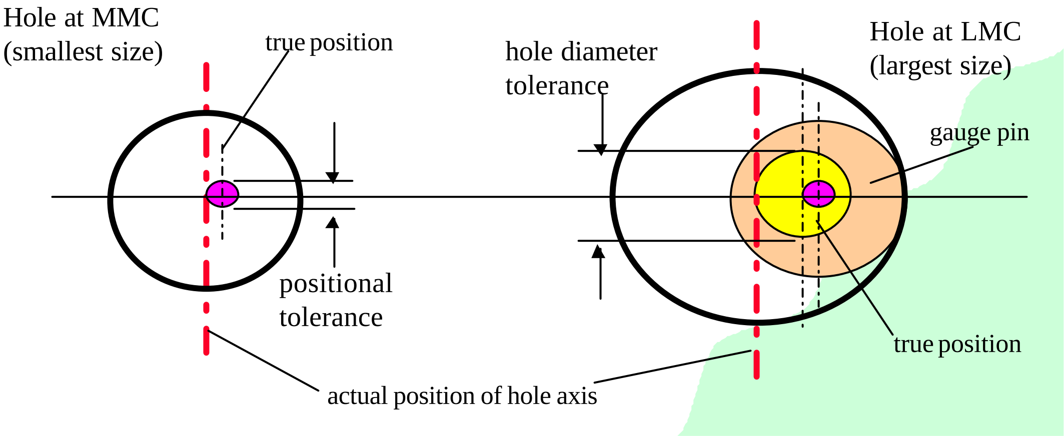
This slide illustrates how MMC works. Assume a hole is at its MMC (smallest diameter) of 12,75 mm and the positional tolerance is 0,70 mm. The mauve circle in the middle is the positional tolerance zone for the axis of the hole. The red circle on the right shows the circumference of the hole when its axis is displaced to the right to the maximum amount permitted by the tolerance zone. The blue circle on the left shows the circumference of the hole when its axis is displaced to the left to the maximum amount permitted by the tolerance zone. The difference in the circumferences is the positional tolerance.

The light orange circle in the middle shows the size of functional gauge pin that can be used to check that the hole meets the mating requirement. The gauge pin is centred at the true position of the hole. You can clearly see that the two circles (red and blue) just touch either side of the gauge pin.

In practice, some tolerance has to be assigned to the manufacture of the gauge and so the design geometric tolerance is reduced accordingly.

MMC for a hole

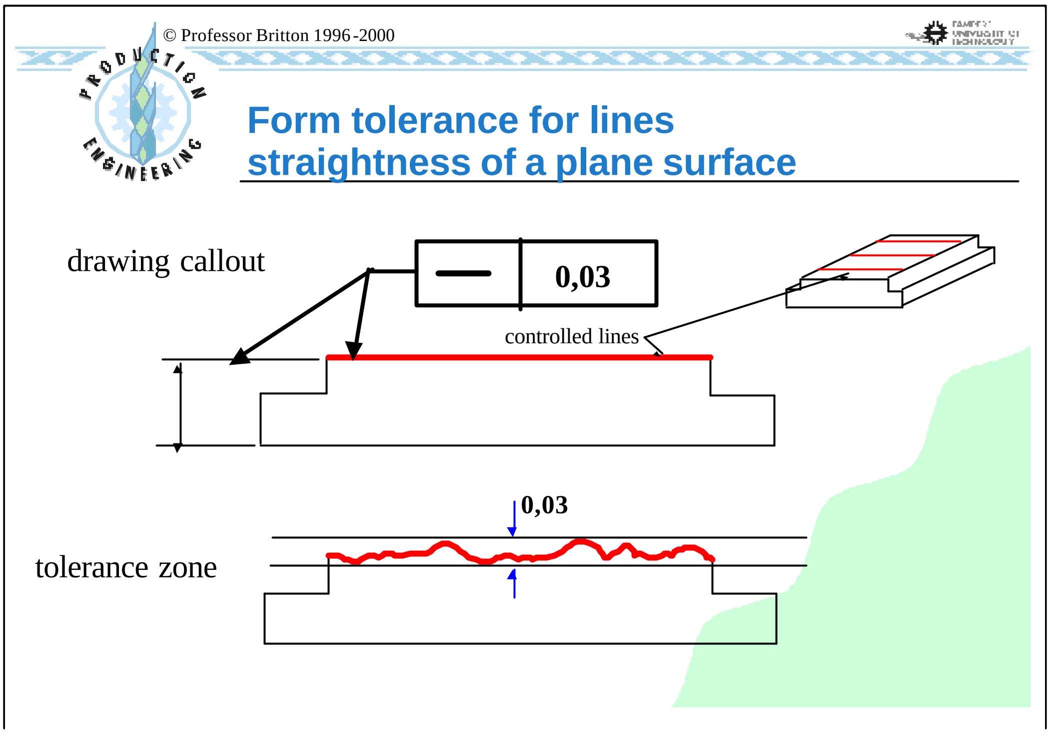
Greatest deviation of hole axis from true position is
(hole diameter tolerance + positional tolerance)/2



This slide shows what happens when the hole increases. The picture on the left shows the hole at MMC. In this condition, the maximum deviation of the hole axis is given by the positional tolerance zone, shown as a mauvecircle. The centre line of the mauve circle indicates the theoretically true position of the hole. The actual hole is displaced to the left as shown.

The picture on the right shows the hole at LMC, with the hole again displaced to the left of true position. The yellow circle represents the amount of permitted tolerance variation for the hole size (diameter). The hole axis can be permitted to vary by an additional amount equal to half the hole diameter tolerance and still meet mating requirements. This can be clearly seen in the slide where the functional gauge pin from the previous slide has been superimposed. The larger hole at LMC just touches the gauge pin and hence the gauge pin will fit in the hole, even though the hole axis has been displaced further than the positional tolerance/2.

With functional gauging the positional tolerance of the hole axis is allowed to vary with change in size. If this is not done and a fixed positional tolerance is used then different gauges have to be made to match each possible hole size. This greatly increases inspection costs. In addition, the costs of manufacture may increase as well as it may be difficult to control both hole size and position simultaneously.

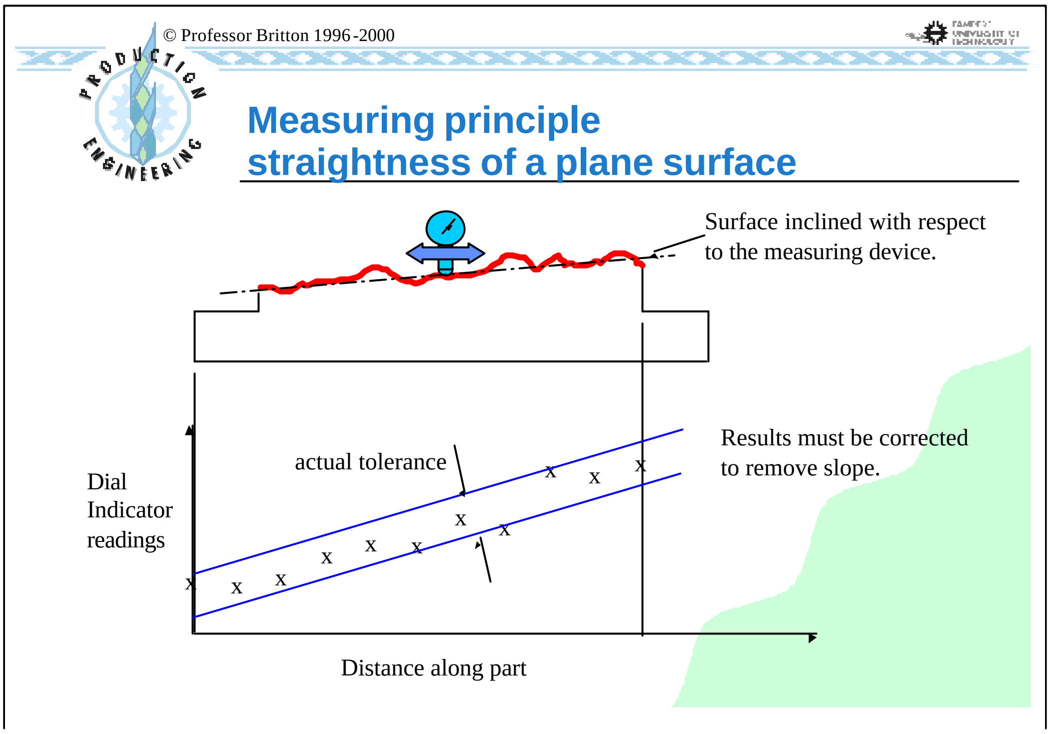


Straightness and flatness are perhaps the two most important types of geometric control. This is because features must be straight or flat before they can be used as datums.

Straightness controls lines drawn on a surface, axes of shafts and holes, and edges of parts. The drawing callout is applied to the view which indicates the profile of the feature-to-be-controlled. The slide above shows a straightness control for lines on a surface. The tolerance zone is defined by two parallel straight lines whose distance apart is given by the tolerance value (0,03). The lines are drawn in a plane parallel to the plane of projection in which the feature is indicated. The actual position of the tolerance zone relative to other features, such as surface A, is not controlled.

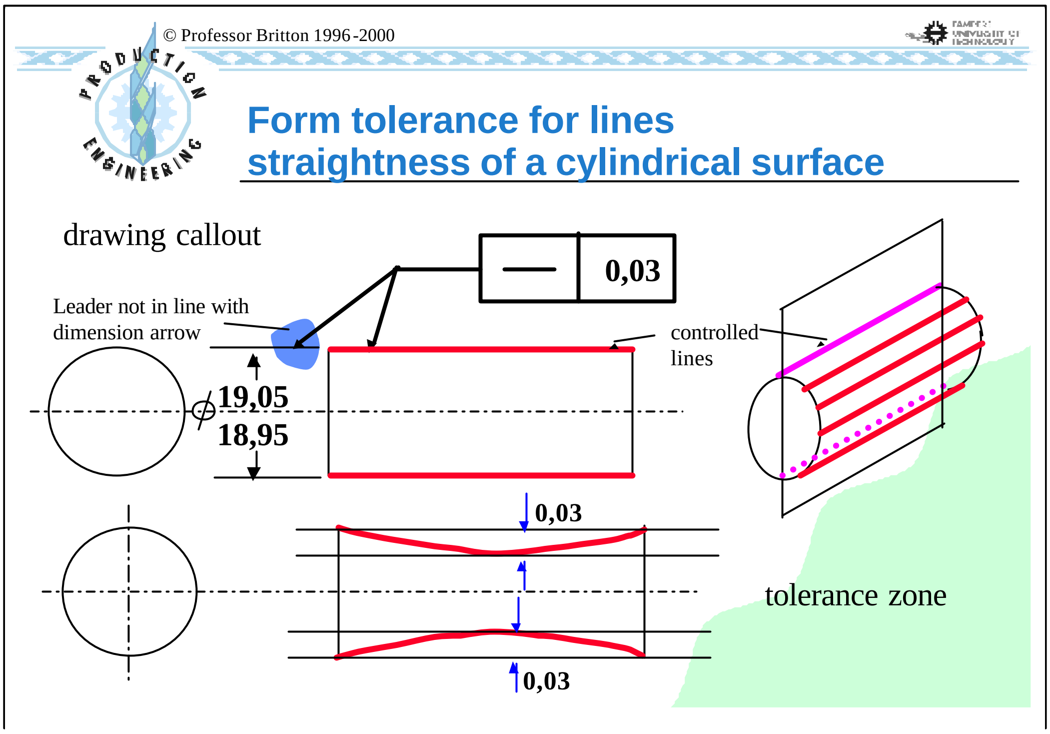
Each line drawn the surface must line within the 0,03 tolerance zone. The lines are drawn in the direction as shown. The callout applies to the entire surface. In practice, it is not cost effective to inspect the entire surface, instead 3 or 4 lines spaced apart would be checked to ensure compliance.

There is one important point to note. The tolerance zone applies to each line on the surface independently. The tolerance zones for different lines may vary in position (up and down) relative to each other. Thus a surface could be wavy, convex or concave and yet still meet the straightness requirement.



This slide shows the measuring principle. The surface-being-measured may be inclined relative to the measuring device. If no correction is made for the slope then the measured tolerance will be larger than the actual value. The slide shows how to correct for slope by plotting results on a graph paper.

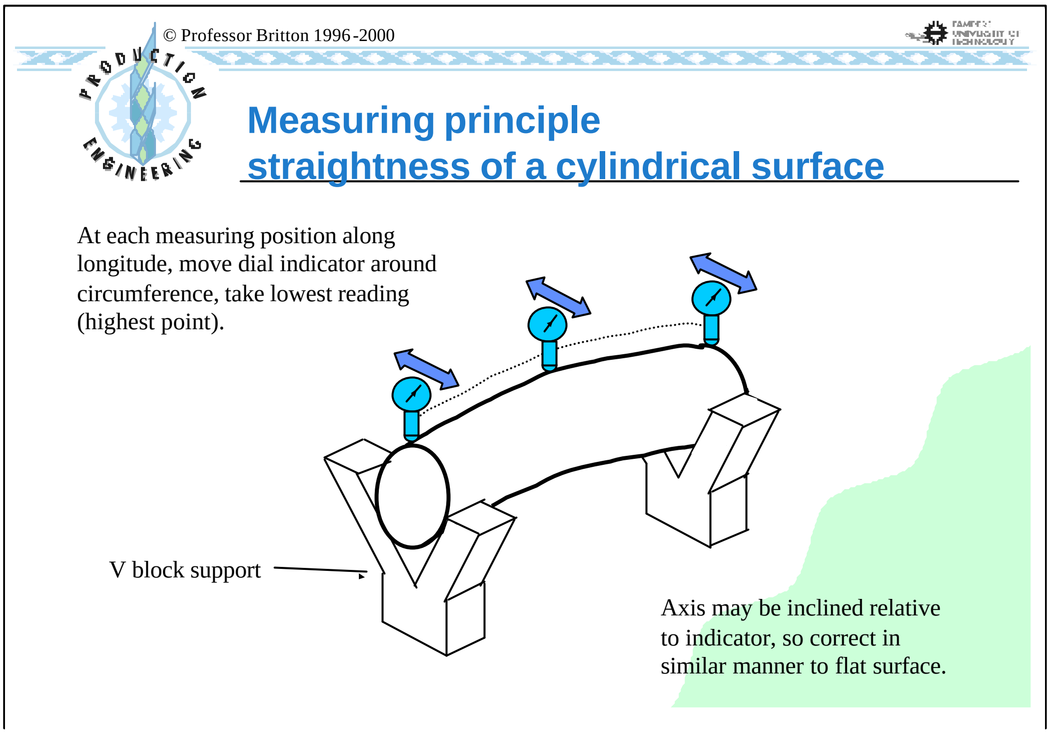
Another technique is to zero the dial indicator at three widely spaced points on the surface-being-measured. To do this the bottom surface of the part must be supported by adjusted supports which can be adjusted to give the zero readings. Once adjustment is completed the dial readings can be used directly to determine the tolerance. The tolerance is given by the difference between the highest and lowest readings.



For cylindrical shapes, such as shafts, pins and holes, there are two different types of straightness control. This slide shows how to control the straightness of the outside surface. For this situation, the leader line indicating the surface-to-be-controlled <u>must not</u> touch the dimension line.

The tolerance zone is given by two lines in a plane through the centre of the part.

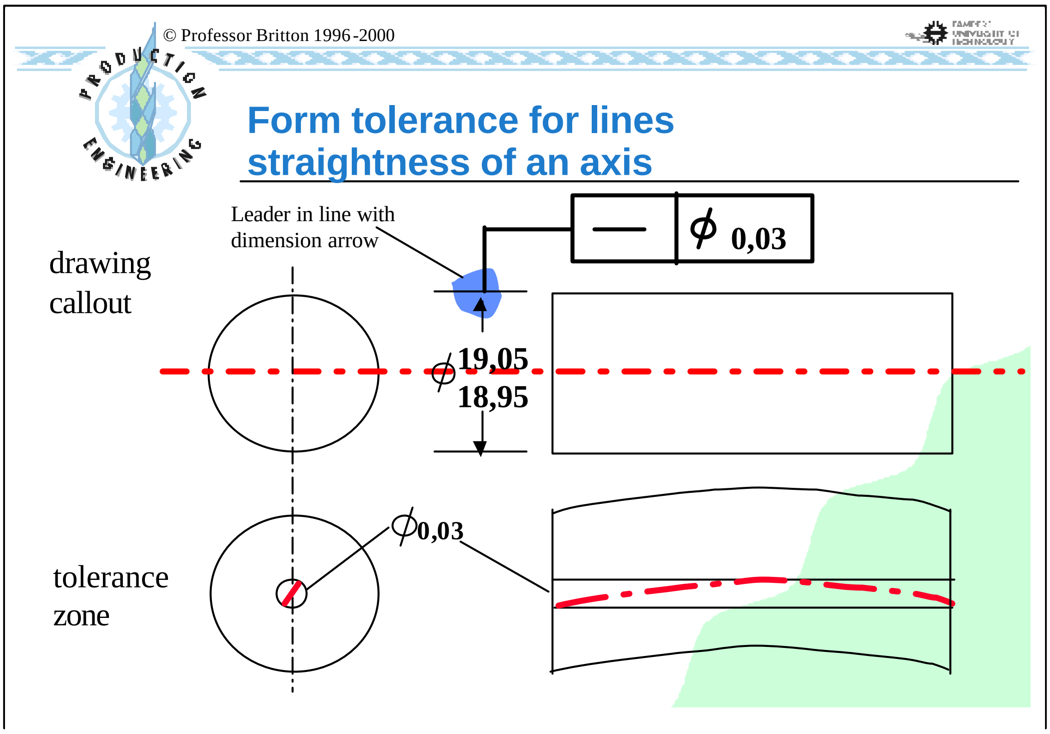
All actual local size (circular elements) of the surface must be within the specified size tolerance (18,95-19,05) and the boundary of perfect form at MMC (19,05). In addition, each longititudinal line of the surface must lie in a tolerance zone defined by two parallel lines with a separation distance equal to the straightness tolerance (0,03).



Measurement of a straightness of a circular surface is more complicated than for a flat surface. A common practice is to use V blocks to support the part, as shown in the slide; this is an approximate method of measurement. V blocks add additional error as the part may not be circular, it could be lobed.

To ensure that the line-being-measured is along the axis the dial indicator is moved across the part at each measuring point. The lowest reading is taken as the measurement (represents the highest point on the part). The part may be inclined relative to the measuring device, so correction needs to be made for this. The technique is the same as for a flat surface.

Complete measurement of the surface is not cost effective. Normal practice is to measure four times around the part, with a spacing of 90 degrees. Each and every measured section (longitudinal line) must lie within the straightness tolerance.



This slide shows how to control the straightness of an axis. The leader for the feature control frame is attached to the dimension line. In addition, a diametral symbol is added to the feature control frame to show that the tolerance zone is a cylinder.

Straightness tolerance of an axis can under RFS or MMC conditions.

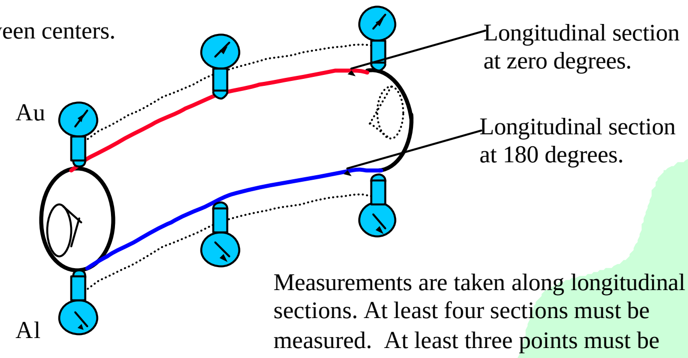
RFS: Each actual local size (circular element) must be within the specified size tolerance (18,95-19,05). The derived median line (estimated actual axis) must lie within a tolerance zone of 0,03 regardless of the size of the feature; i.e, regardless of the diameter of the part. Thus the same tolerance applies to parts at MMC (19,05) and LMC (18,95), and for all sizes in between.

MMC: Each actual local size (circular element) must be within the specified size tolerance (18,95-19,05). The derived median line (estimated actual axis) must lie within a tolerance zone of 0,03 at MMC. If the part is smaller than MMC then the difference in size from MMC can be used to increase the straightness tolerance. Thus the straightness tolerance is different for different sizes of parts. At LMC the straightness tolerance is 0,03 + 0,1 = 0,13.

MMC is used for functional design of assemblies (mating parts) and permits functional gauging, thus reducing manufacturing and inspection costs.

Measuring principle straightness of an axis - RFS

Part is held between centers.



Measurements are taken along longitudinal sections. At least four sections must be measured. At least three points must be measured in each section.

Straightness deviation for each section is Rmax-Rmin, where R=(Au-Al)/2;
Straightness deviation for axis is maximum of section straightness deviations.

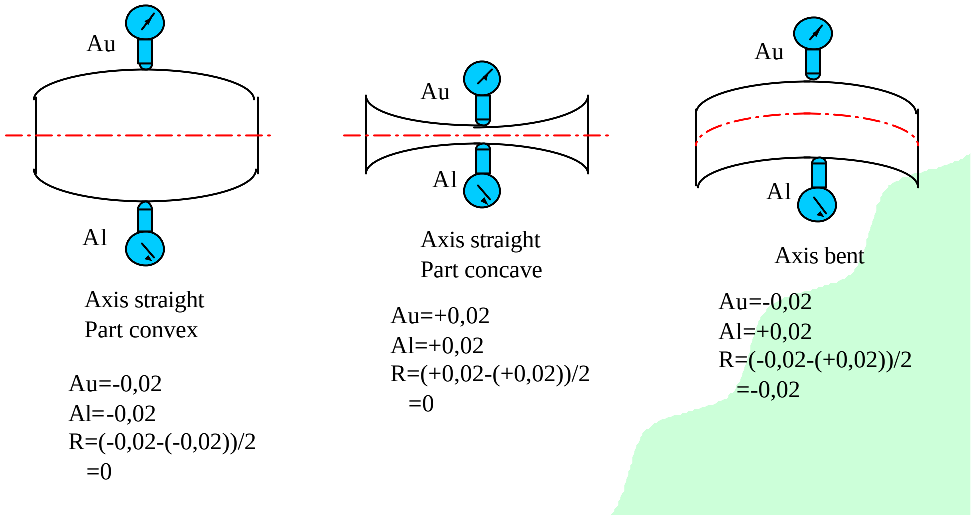
To measure straightness of a shaft axis at RFS two dial indicators are required. The part is held between centers. In each longitudinal section the values R=(Au-Al)/2 are determined. Where Au is the reading of the upper dial indicator and Al is the reading of the lower dial indicator. The difference between Rmax and Rmin within one section represents the straightness deviation of the axis for this section. The straightness deviation of the axis is the maximum of the section deviations; at least 4 sections must be measured. The method shown in this slide is an approximate method for assessing straightness.

Note that it is not possible to measure straightness of an axis directly. The only measurements that can be taken are those on the outside surface of the part. This is true for all measuring methods.

In addition to meeting the straightness requirements circular elements along the part must meet local size feature requirements (must lie between 18,95-19,05 in this case).

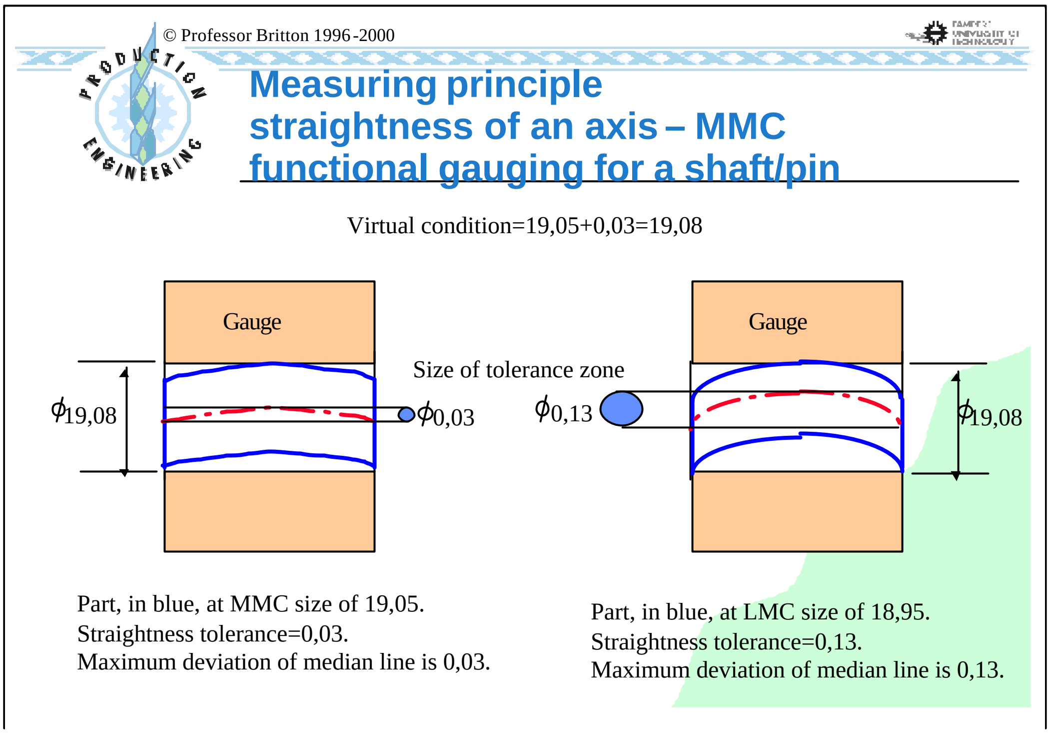
The next slide gives some examples of measurements for different shapes of parts.

Measuring principle straightness of an axis - RFS



This slide illustrates how the straightness is calculated. If the part has a straight axis and is symmetrical about the centreline then the upper and lower dial indicator readings cancel each other.

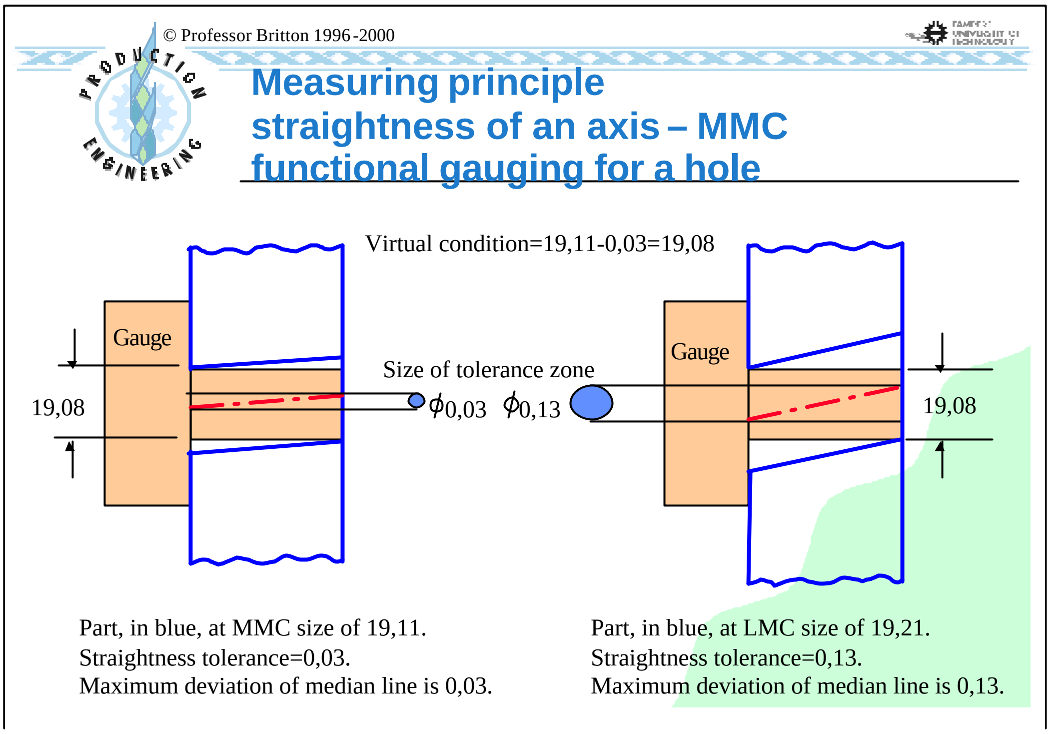
If the axis is bent (not straight) then one indicator will read lower than the other. However each indicator is measuring the error so the sum of the two readings is twice the amount of actual error. The actual error is calculated by dividing the sum by two. This gives an average value estimate for the position of the axis.



When MMC is used to specify the straightness of an axis then functional gauging can be used. The slide shows a shaft (the same example as before). The functional gauge is a ring gauge (circular gauge) with a hole which is sized to the virtual condition of 19,08 mm. The virtual condition is equal to the maximum feature size (19,05 in this case) plus the straightness tolerance (0,03 in this case). The one gauge is used to check all parts irrespective of their size.

If a part is at MMC then the straightness tolerance is 0,03. If a part is at LMC then the straightness tolerance is 0,13. Thus a smaller diameter shaft can be bent more than a larger diameter shaft and yet still mate with its corresponding mating part (a hole in this case). The gauge simulates the mating component. The gauge length must be equal to or greater than the length of the feature-being-measured.

Note that in addition to meeting the gauge requirements, the feature must meet local feature size requirements, i.e., circular elements must lie between 18,95-19,05.

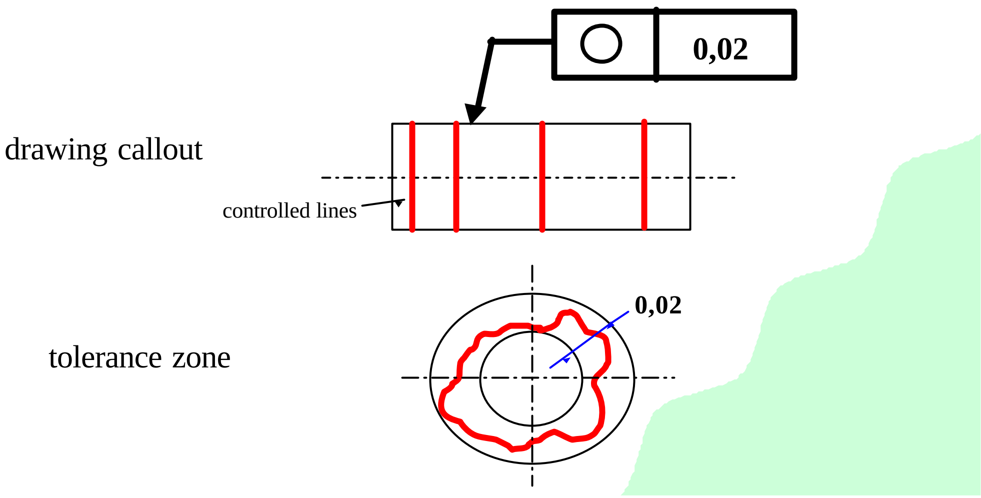


This slide shows how MMC would be applied to a hole to match the shaft of the previous example. The virtual condition is the same as for the shaft. It is the common boundary for mating parts. When a hole is at the MMC (smallest diameter) the straightness tolerance is 0,03. When a hole is at LMC (largest diameter) the tolerance is 0,13.

The gauge must have a length equal to or greater than the feature-being-measured.

Note that in all cases, the gauge will fit the hole. Thus the hole will always mate with the shaft of the previous example.

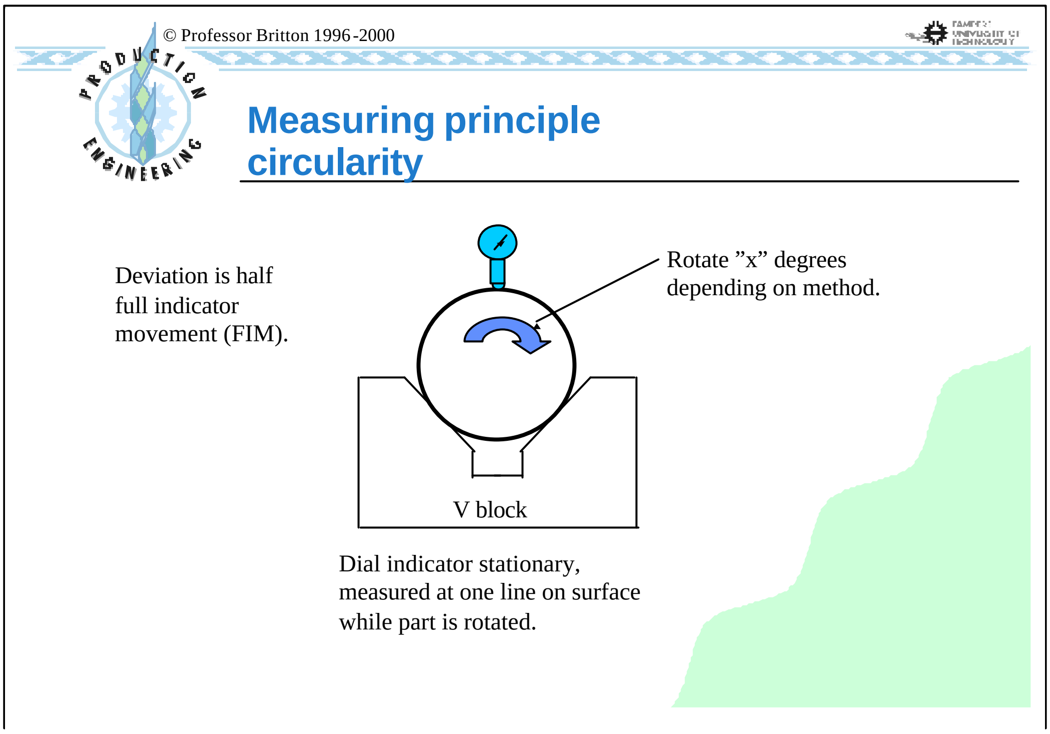
Form tolerance for lines circularity



A circularity tolerance is used to control the roundness of a feature, e.g, the circumference of a shaft or hole. The tolerance zone applies to a line drawn around the circumference of the part at a cross section that is at right angles to the plane of projection. Each line must lie within the tolerance zone, but the tolerance zones are independent. The tolerance zone is defined by two concentric circles set apart by a radial distance equal to the tolerance value (0,02). The position of the tolerance zone relative to the axis of the part is not controlled.

The callout applies to the entire surface, but in practice several lines would be assessed to check compliance.

A part, e.g., shaft, may be bowed or bent and yet still meet this requirement because it does not control the relative positions of the tolerance zones.



The slide shows an approximate method for assessing circularity. The part is rotated with the dial indicator stationary. The full indicator movement (FIM) is determined - it is the difference between the highest and lowest values. The deviation is half this value.

FIM=Amax-Amin.

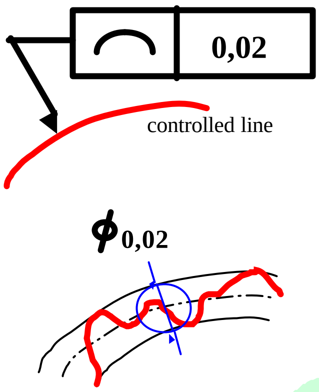
Deviation=FIM/2

With two point measurements, two measurements are taken 180 degrees apart. Two point measurement will not detect lobing of the part. Lobing is known to occur for certain machining operations, e.g., centre less grinding and reaming. Where lobing is suspected three point measurement or other methods should be used. Correction values are used to obtain accurate assessments of circularity when using 3 or more point measurement methods.

Form tolerance for lines profile

drawing callout

tolerance zone

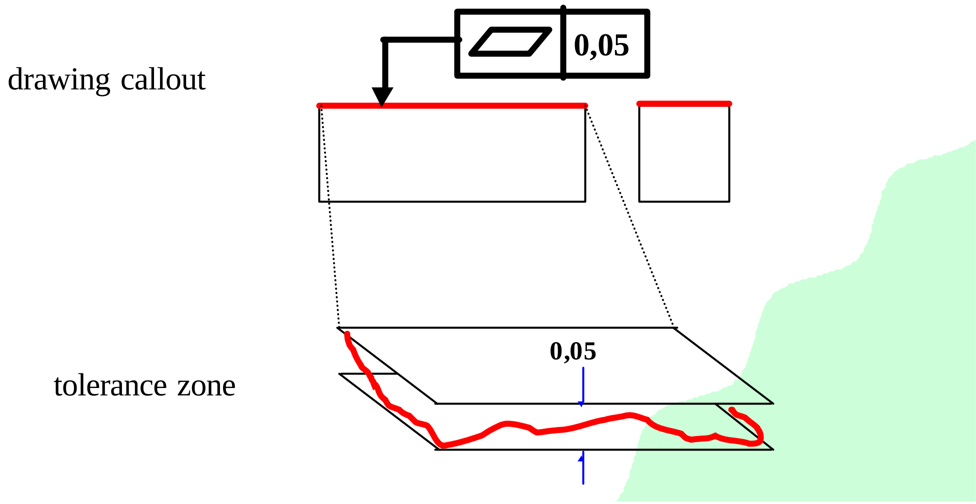


Profile of a line is used to control all forms other than straight and round lines.

In this case the tolerance zone is defined by a two curved lines traced out by a circle whose centre point follows the nominal shape of the profile. Thus this callout requires a diameter symbol.

All points on the line-being-controlled must lie within the tolerance zone. If more than one line is being controlled then the tolerance zones are independent.

Form tolerance for surfaces flatness

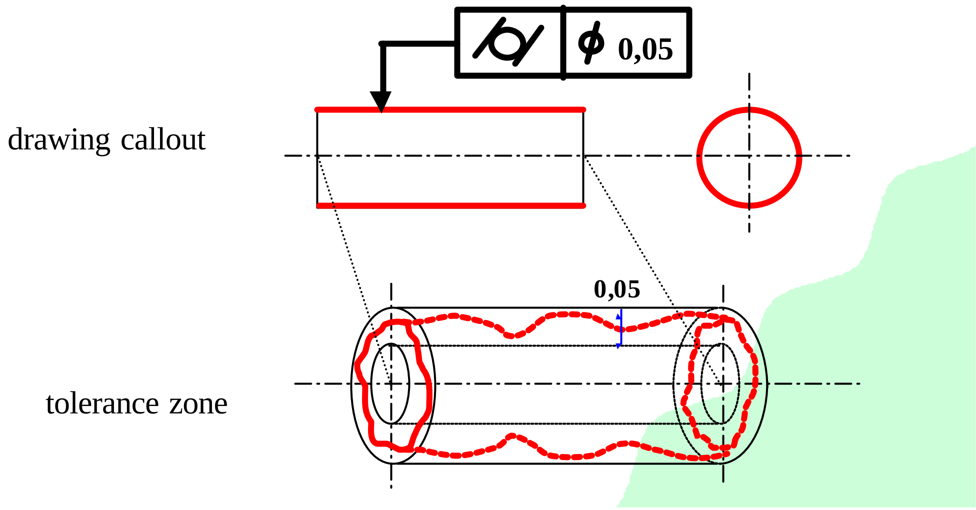


A flatness tolerance applies to surfaces. It is equivalent to two equal straightness tolerances applied at right angles to each other. The tolerance zone is given by two parallel planes whose distance apart is equal to the tolerance value.

ALL points on the surface must lie within the tolerance zone, thus flatness control controls deviations such as waviness, concavity and convexity.

The measuring principle is the same as for straight lines on planes, except that now it is necessary to correct for inclination in two directions.

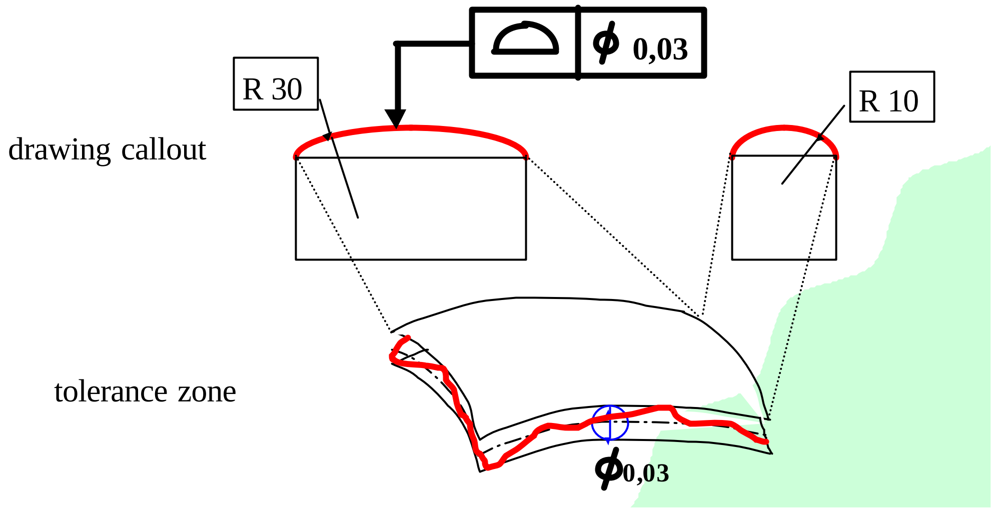
Form tolerance for surfaces cylindricity



Cylindricity controls the roundess of a feature over its entire surface. The tolerance zone is a tube whose thickness is given by the tolerance value. The zone is formed by a 360 degree rotation of a cylinder whose diameter is equal to the tolerance value and whose length is equal to the length of the feature-being-controlled, thus the diameter symbol is used for this tolerance. Cylindricity is verified in a similar manner to circularity.

ALL points on the surface must lie within the tolerance zone, thus cylindricity controls deviations such as concavity and convexity.

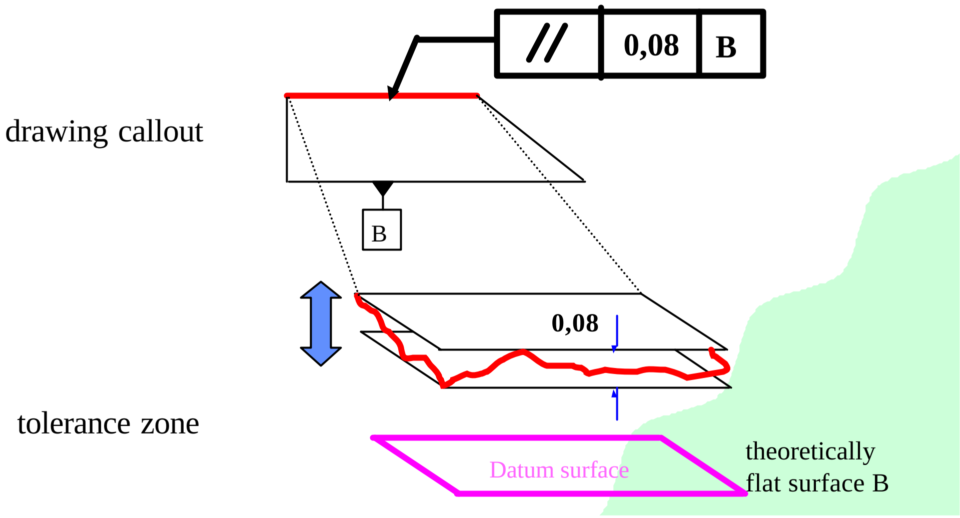
Form tolerance for surfaces profile



The tolerance zone to control the profile of a surface is formed by rolling a sphere such that its center point always lies on the nominal profile of the surface. The extreme points traced by the sphere defines two surfaces whose distance apart is equal to the diameter of the sphere, which is the tolerance value.

Profiles are often verified using shadow graphs. These project an image of the profile onto a large screen which has the tolerance boundaries inscribed on it. The projected profile must lie within these boundaries.

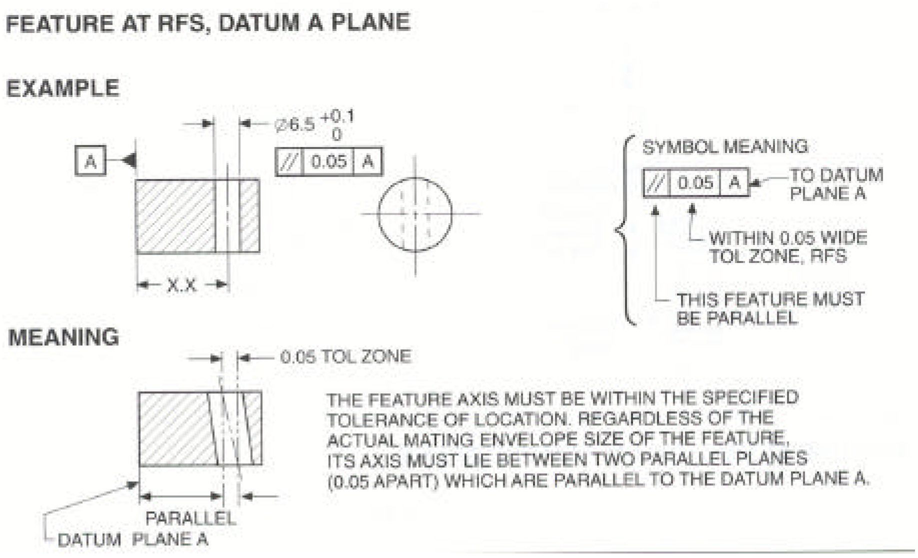
Orientation tolerance parallelism



An orientation tolerance is always specified relative to one or more datums. In the example above, the tolerance zone is defined relative to datum B, which is the bottom surface of the part.

In this case, the tolerance zone is defined two plane surfaces parallel to datum B. All points on the surface-being-controlled must lie within the tolerance zone.

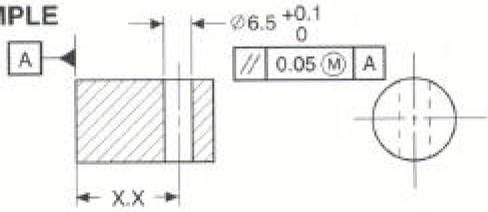
The tolerance zone controls the orientation of the surface relative to datum B but not its position vertically.



Reproduced from Foster

FEATURE AT MMC, DATUM A PLANE

EXAMPLE

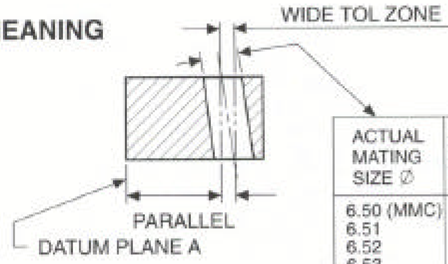


SYMBOL MEANING

// 0.05 Ⓜ A — TO DATUM PLANE A

- WITHIN 0.05 WIDE TOL ZONE AT MMC
- THIS FEATURE MUST BE PARALLEL

MEANING

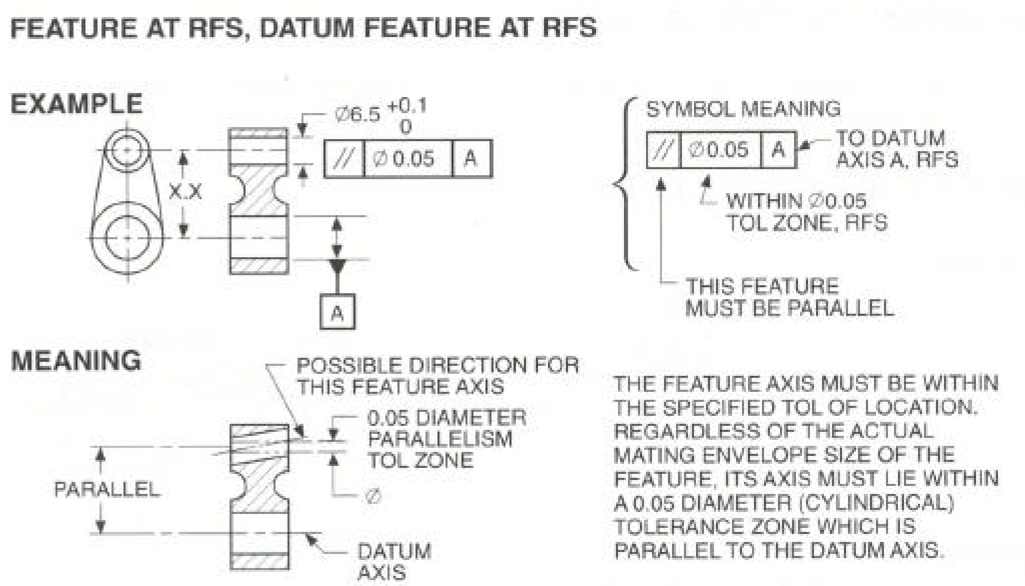


ACTUAL MATING SIZE Ø	PARALLELISM TOL WIDTH ALLOWED
6.50 (MMC)	0.05
6.51	0.06
6.52	0.07
6.53	0.08
6.54	0.09
6.55	0.10
6.60 (LMC)	0.15

FEATURE AXIS MUST BE WITHIN SPECIFIED TOL OF LOCATION. HOLE (AT MMC) AXIS MUST LIE BETWEEN TWO PARALLEL PLANES (0.05 APART) WHICH ARE PARALLEL TO THE DATUM PLANE A.

WHEN FEATURE ACTUAL MATING ENVELOPE SIZE DEPARTS FROM MMC (GETS LARGER), AN INCREASE IN THE PARALLELISM TOL IS ALLOWED EQUAL TO THAT DEPARTURE.

Reproduced from Foster

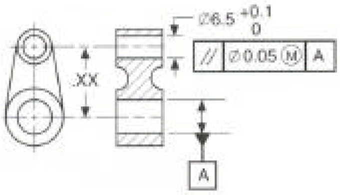


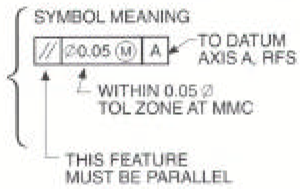
Reproduced from Foster

FEATURE AT MMC, DATUM FEATURE AT RFS

(Note that the ∅ symbol is required to indicate a diameter (cylindrical) tolerance zone.)

EXAMPLE





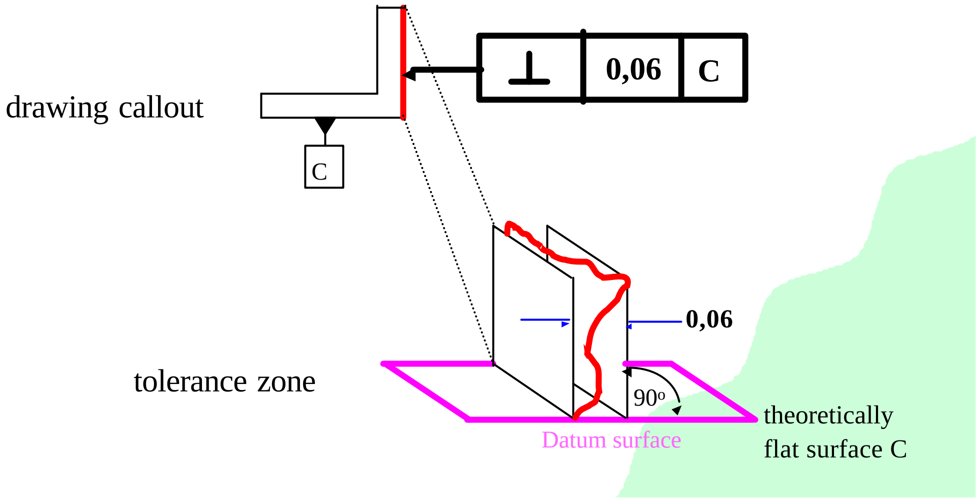
MEANING

ACTUAL MATING SIZE ∅	PARALLELISM TOL DIA ALLOWED
6.5 (MMC)	0.05
6.51	0.06
6.52	0.07
6.53	0.08
6.54	0.09
6.55	0.10
6.6 (LMC)	0.15

THE FEATURE AXIS MUST BE WITHIN THE SPECIFIED TOL OF LOCATION. WHEN THE FEATURE IS AT MAXIMUM MATERIAL CONDITION, 6.5, THE MAXIMUM PARALLELISM TOLERANCE IS 0.05 DIAMETER. WHEN THE FEATURE ACTUAL MATING ENVELOPE SIZE DEPARTS FROM MMC (GETS LARGER), AN INCREASE IN THE PARALLELISM TOL IS ALLOWED EQUAL TO THE AMOUNT OF THAT DEPARTURE.

Reproduced from Foster

Orientation tolerance perpendicularity

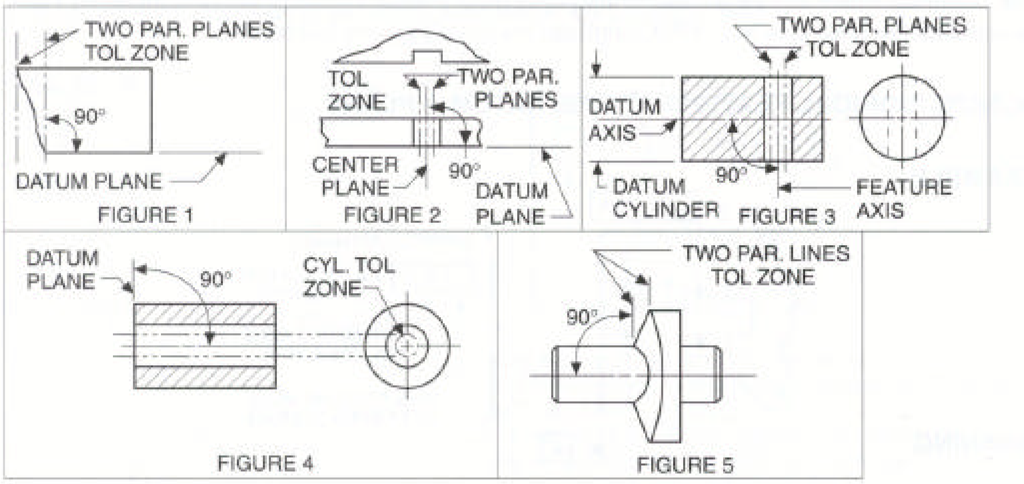


An orientation tolerance is always specified relative to one or more datums. In the example above, the tolerance zone is defined relative to datum C, which is the bottom surface of the part.

In this case, the tolerance zone is defined by two plane surfaces at right angles to datum C. All points on the surface-being-controlled must lie within the tolerance zone.

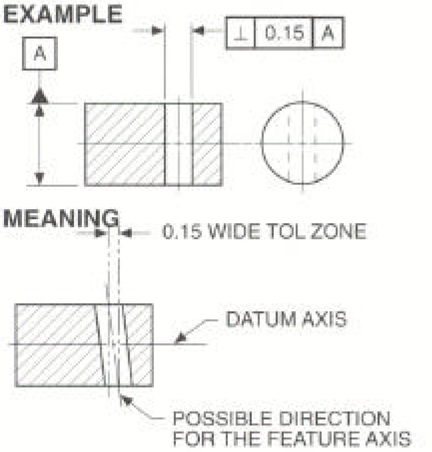
The tolerance zone controls the orientation of the surface relative to datum C but not its position in the horizontal direction.

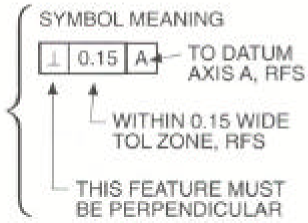
Examples of Perpendicular Tolerance Zones



Reproduced from Foster

CYLINDRICAL FEATURE AT RFS, DATUM A CYLINDER RFS





THE FEATURE AXIS MUST BE WITHIN THE SPECIFIED TOLERANCE OF LOCATION. REGARDLESS OF THE ACTUAL MATING ENVELOPE SIZE, ITS AXIS MUST LIE BETWEEN TWO PLANES (0.15 APART) WHICH ARE PERPENDICULAR TO THE DATUM AXIS.

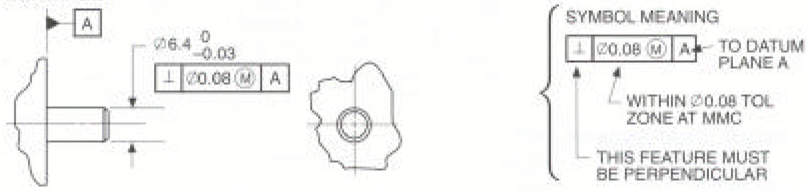
NOTE: This tolerance applies only to the view on which it is specified.

Reproduced from Foster

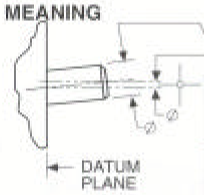
CYLINDRICAL FEATURE AT MMC, DATUM A PLANE

Note that the ∅ symbol is required to indicate a diameter (cylindrical) tolerance zone.

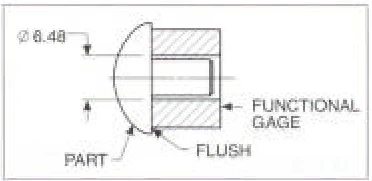
EXAMPLE



MEANING

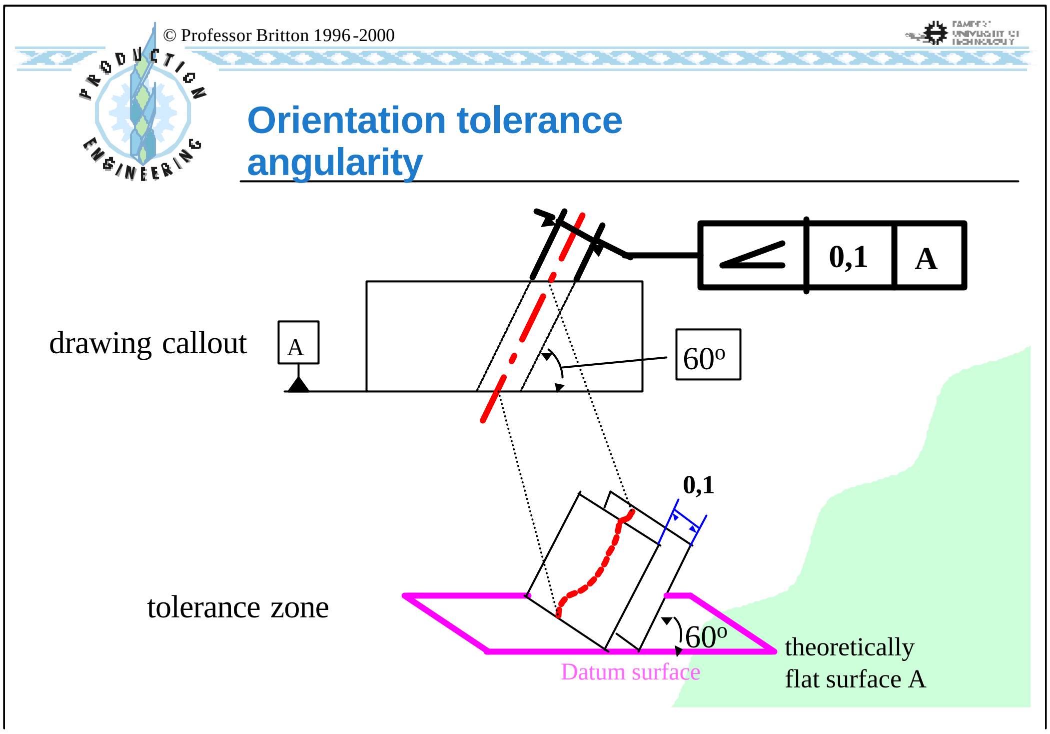


ACTUAL MATING SIZE ∅	PERP TOL ∅ DIA ALLOWED
6.40 MMC	0.08
6.39	0.09
6.38	0.10
6.37 LMC	0.11



THE FEATURE MUST BE WITHIN THE SPECIFIED TOLERANCE OF LOCATION. WHEN THE FEATURE IS AT MMC 6.4, THE PERPENDICULARITY TOLERANCE IS 0.08 DIAMETER. AS THE FEATURE ACTUAL MATING ENVELOPE SIZE DEPARTS FROM MMC (GETS SMALLER), AN INCREASE IN TOLERANCE IS PERMITTED EQUAL TO THAT AMOUNT OF THAT DEPARTURE. VIRTUAL CONDITION IS ∅ 6.48.

Reproduced from Foster



An orientation tolerance is always specified relative to one or more datums. The angle of an orientation tolerance can be any angle except 0º and 90º.

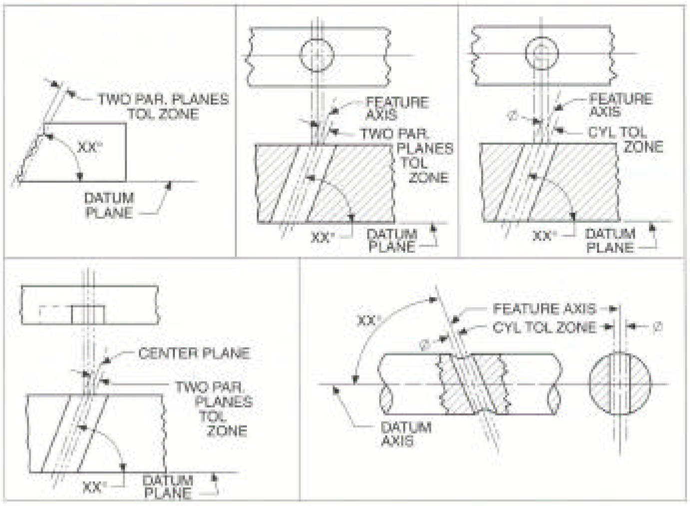
In the example above, the tolerance zone is defined relative to datum A, which is the bottom surface of the part. The tolerance zone is defined by two plane surfaces at set at the theoretically exact angle (60º) to datum A. All points on the hole axis must lie within the tolerance zone. The tolerance zone could also apply to an axis, in which case it may be cylindrical.

The tolerance zone defines the orientation of the axis relative to datum A but not its position in the horizontal direction.

How do we know if a tolerance applies to an axis or a surface for cylindrical shapes? If the tolerance applies to the axis then the leader line will be attached to the dimensioning line.

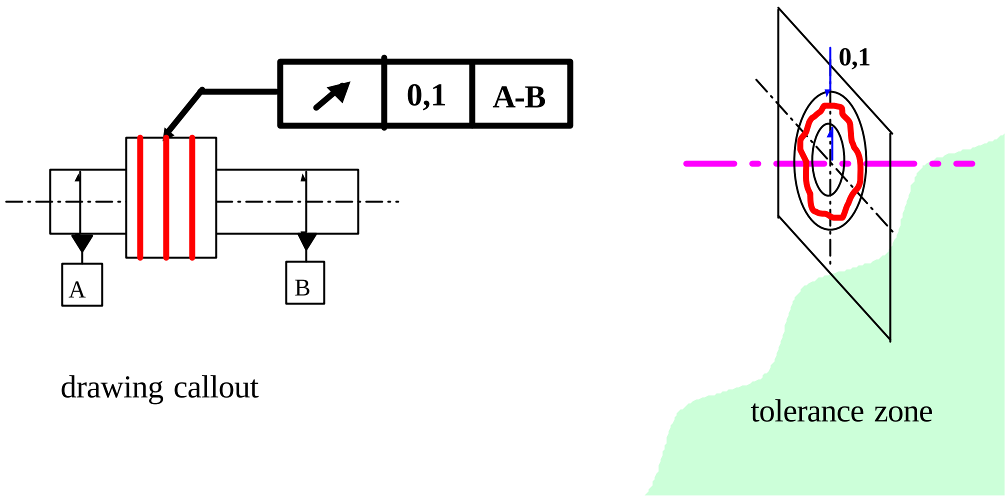
Note that tolerances may apply to features that vary with size. When this happens we need to specify the size of the feature to which the tolerance applies - this will be discussed later under maximum material condition. In the case above, the hole diameter can vary. This variation will affect how the angularity of the axis is measured.

Examples of angular tolerance zones



Reproduced from Foster

Runout circular



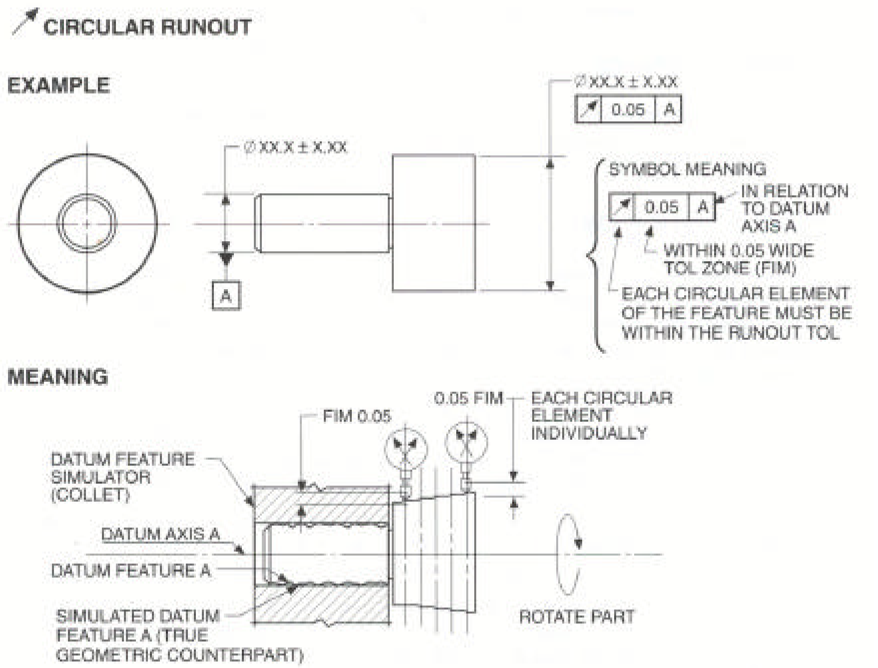
drawing callout

tolerance zone

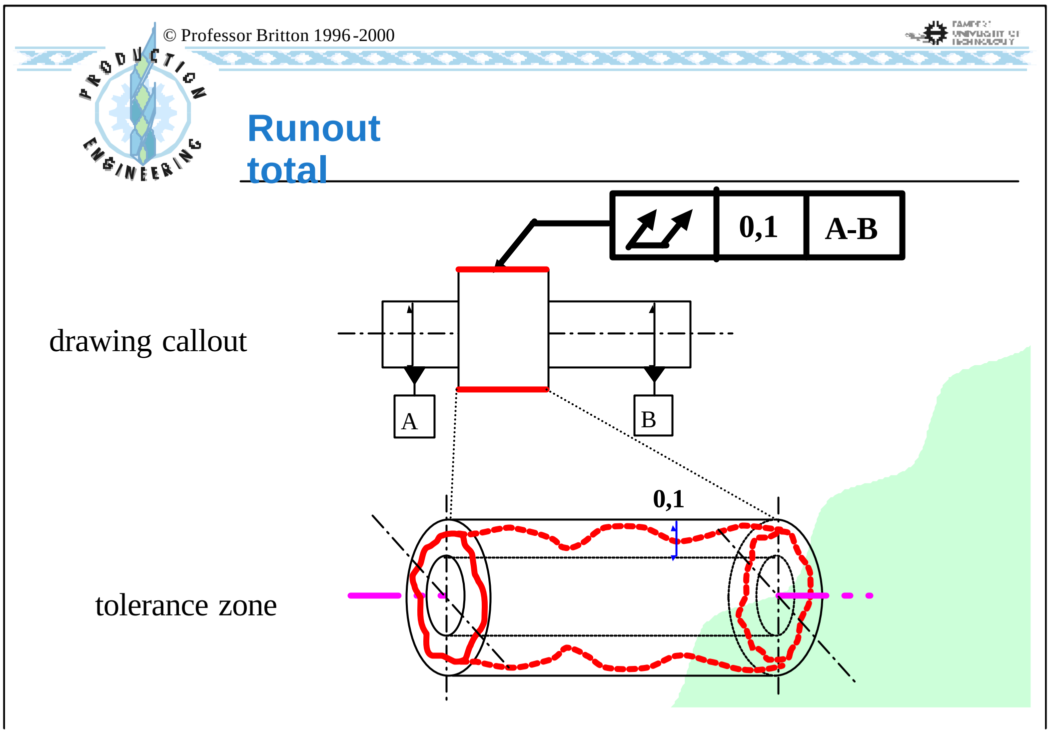
Runout is the composite deviation from the desired form and orientation of a part surface of revolution during full rotation (360°) of the part on a datum axis. A runout tolerance always applies on a RFS basis; size variation has no effect on runout compliance. Surfaces-being-controlled may lie around the datum axis or at 90° to the datum axis. The datum axis may be established by a single diameter, two separate diameters, or a diameter and a face surface at 90° to it.

For circular runout, in each plane perpendicular to the common datum axis A-B the circumference should lie within two circles concentric with axis A-B and set a apart a radial distance of 0,1 mm.

Circular runout controls composite error of circularity, concentricity and circular cross-sectional profile variations. However the tolerance zones between different cross sections can vary in position about the common axis A-B. Thus circular runout does not control convexity, concavity or other deviations along the length of the axis.

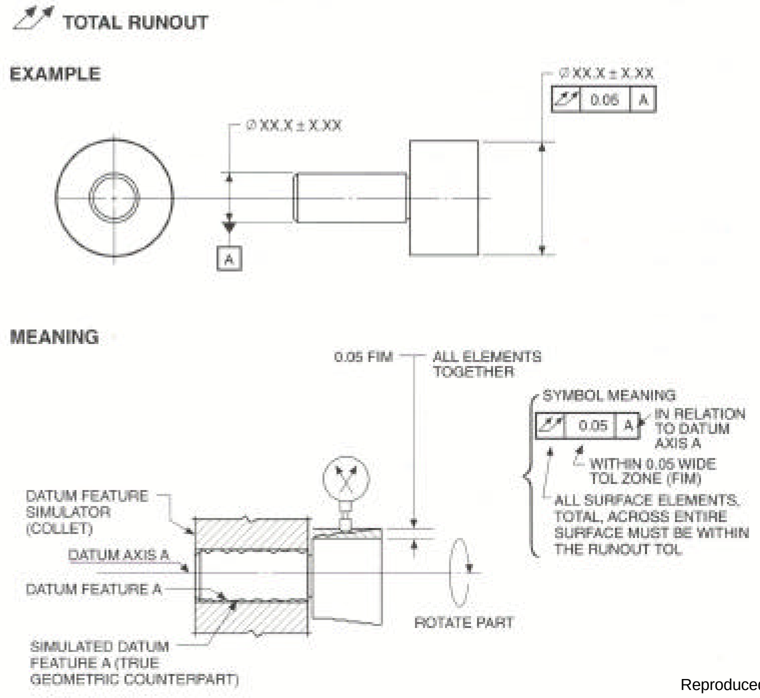


Reproduced from Foster



For total runout the surface should lie within a tube whose thickness is 0,1 mm and whose axis is coaxial with the datum axis A-B.

Total runout controls composite error effect of circularity, cylindricity, straightness, coaxiality, angularity and parallelism (along the axis).



Reproduced from Foster

© Professor Britton 1996-2000

Datums and the Three-Plane Concept

These notes explain how datums are defined and used.

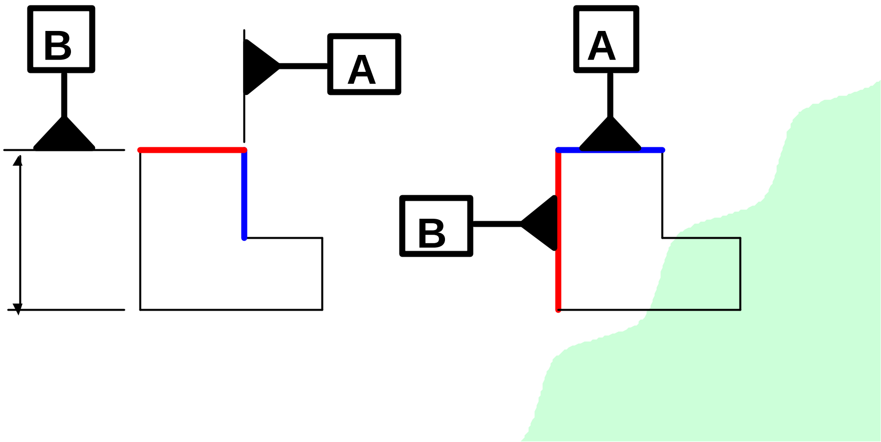
Datum definitions

- **Datum:** **A theoretically point, axis, or plane derived from the true geometric counterpart of a specified datum feature. A datum is the origin from which the location, or geometric characteristics of a part or workpiece are established.**
- **Datum feature*:* Is an actual feature of a part or workpiece used to establish a datum.**

Definitions continued

- **Datum feature simulator:** A surface of adequately precise form (such as a surface plate, a gauge surface or a mandrel) contacting a datum feature and used to establish the simulated datums. A simulated datum feature is the practical application of a datum during manufacture and inspection.
- **Simulated datum:** A point, axis or plane established by processing or inspection equipment such as the following simulators: a surface plate, a gauge surface, or a mandrel.

Application of datum symbol to a surface

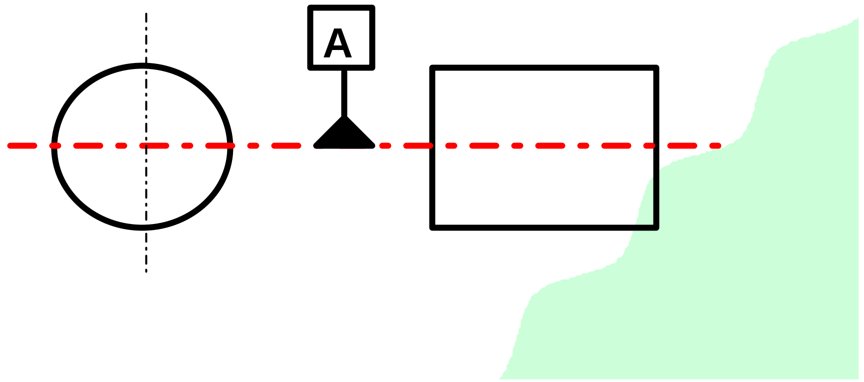


The symbol for a datum is shown in the above slide. The symbol is attached to the datum feature or a leader line from the feature.

Datums are given letter codes starting from the letter A. They follow the order of the alphabet.

© Professor Britton 1996-2000

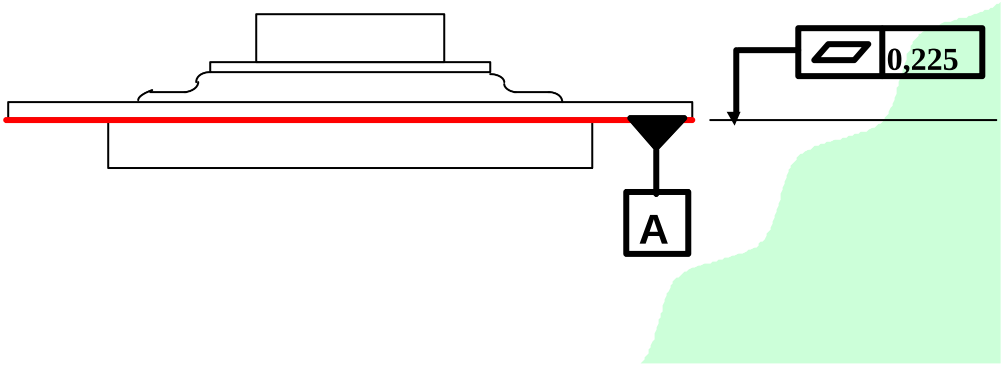
Application of datum symbol to an axis



A datum symbol can also be attached to an axis as shown above.

Note that an axis cannot be determined directly. The position of an axis is approximated by using the circular surface surrounding the axis, i.e., the surface of a hole or shaft.

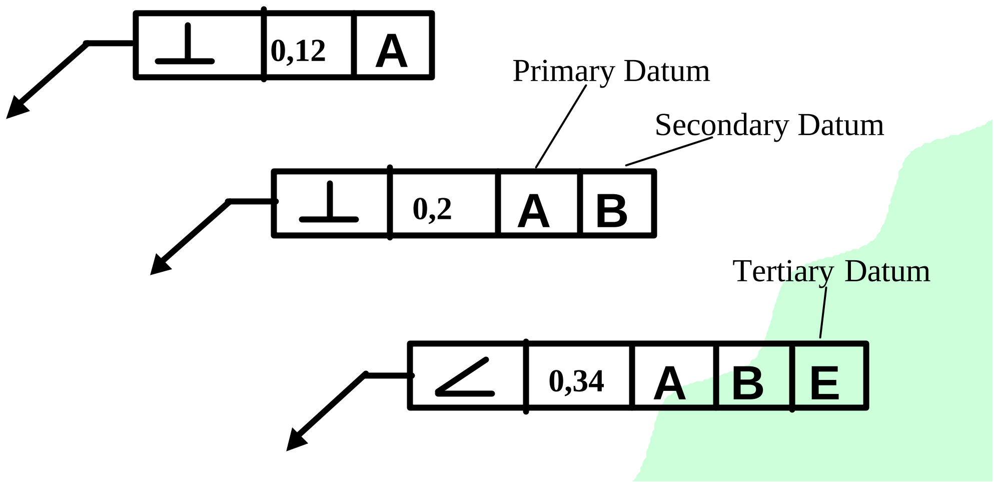
Datum concept example motor spindle



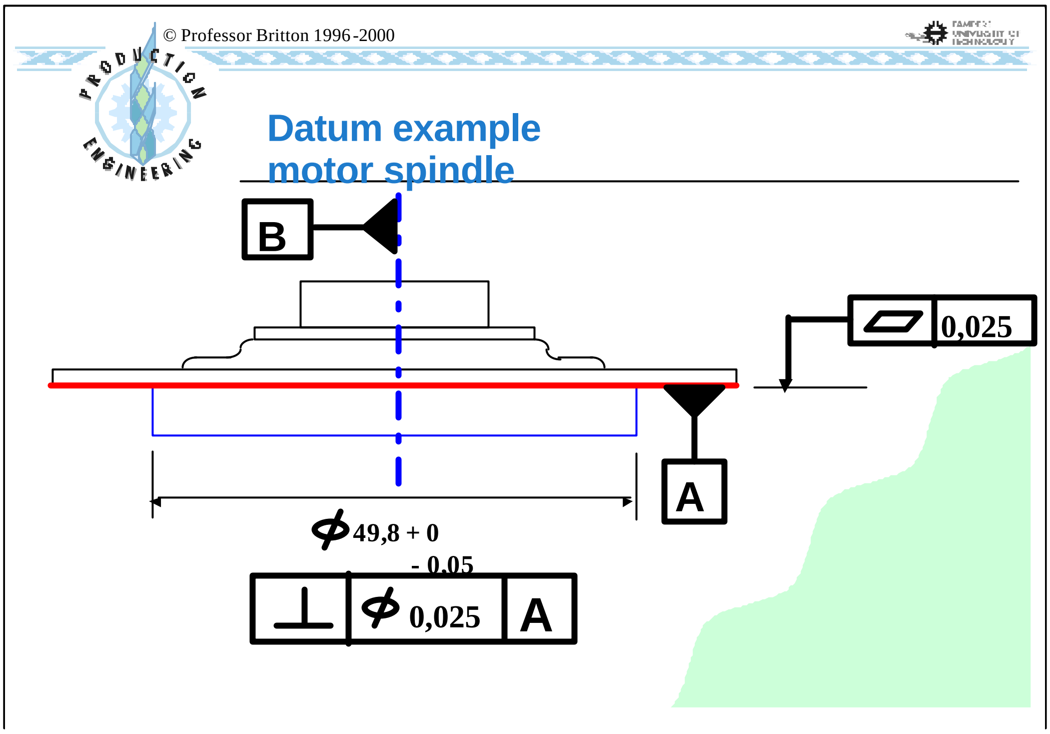
This slide illustrates the use of the symbol. The symbol indicates a surface which is shaped like a disk. This surface is used to support the motor spindle on the baseplate of a disk drive unit.

Note that there is a flatness control on datum A. Without a flatness control on surface A, deviations in surface A would be transferred to features that are to be controlled relative to A. Flatness or straightnesstolerances are normally required for datums.

Referencing datums



One or more datums can be used to control a feature. The different ways of doing this is shown in this slide.

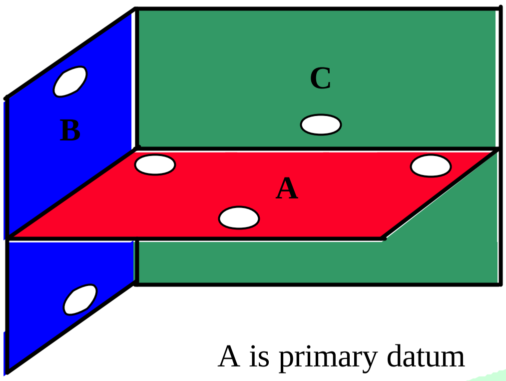


This example illustrates how two datums can be defined relative to each other. Datum A is defined first as is the same as previously defined. The spindle axis is controlled relative to datum A through a perpendicularity tolerance. The spindle axis is defined by the large cylinder at the bottom, shown in blue. This axis is used to control the axis of the top (smaller) cylinder which locates the disks.

The perpendicularity control on datum B provides some control on straightness. However, if this is insufficient for datum B to act properly as a datum then an additional straightness tolerance on the axis would be required.

© Professor Britton 1996-2000

3-2-1 datum system



A is primary datum
B is secondary datum
C is tertiary datum

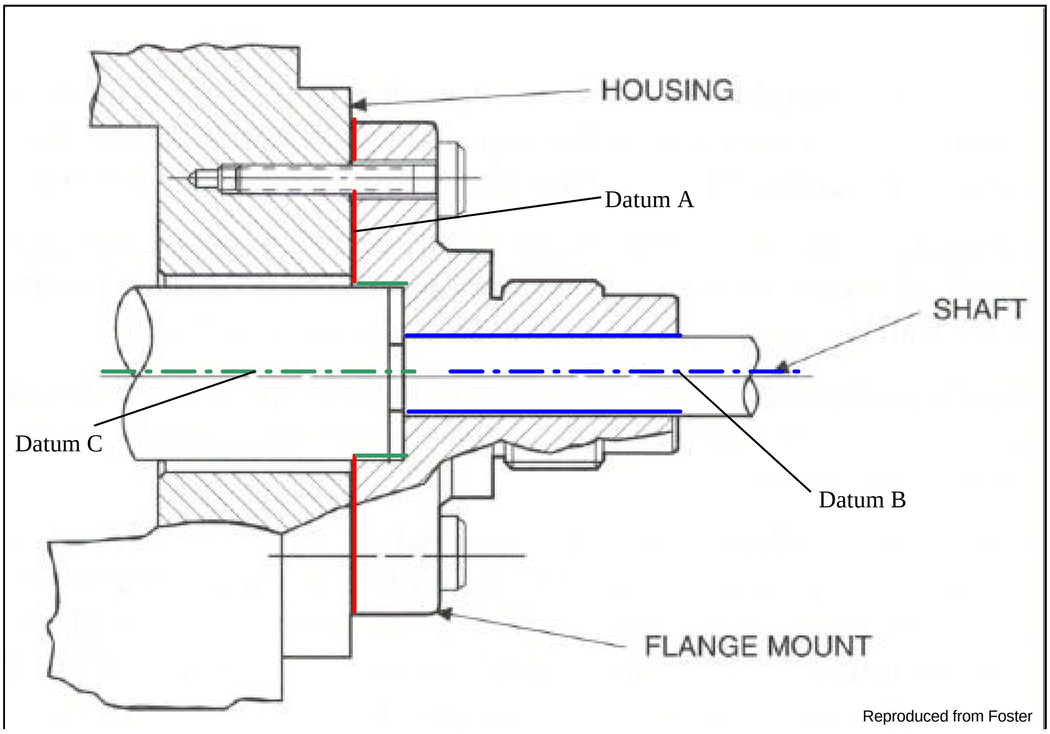
The 3-2-1 datum is illustrated in the slide above. The datum planes are set perpendicular to each other.

The primary datum A provides three point contact to locate a part (it restrains three degrees of freedom – 2 rotation and one translation). The primary datum locates a plane.

The secondary datum B provides 2 point contact (it restrains two degrees of freedom - one rotation and one translation). It locates a line.

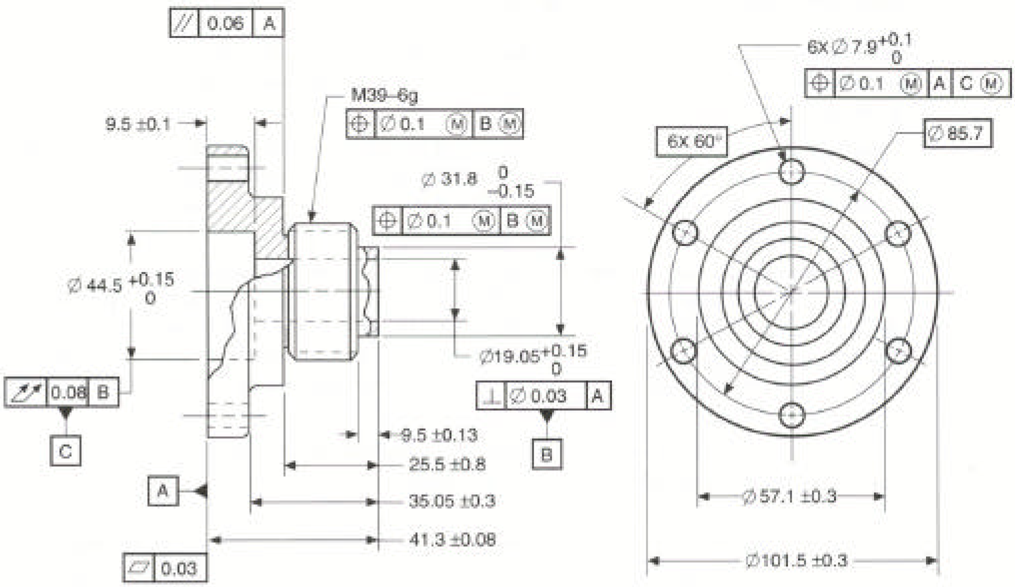
The tertiary datum C provides 1 point contact (it restrains one defgree of freedom – translation). It locates a point.

Thus in total the three datums restrain six degrees of freedom, which is sufficient to locate a part.

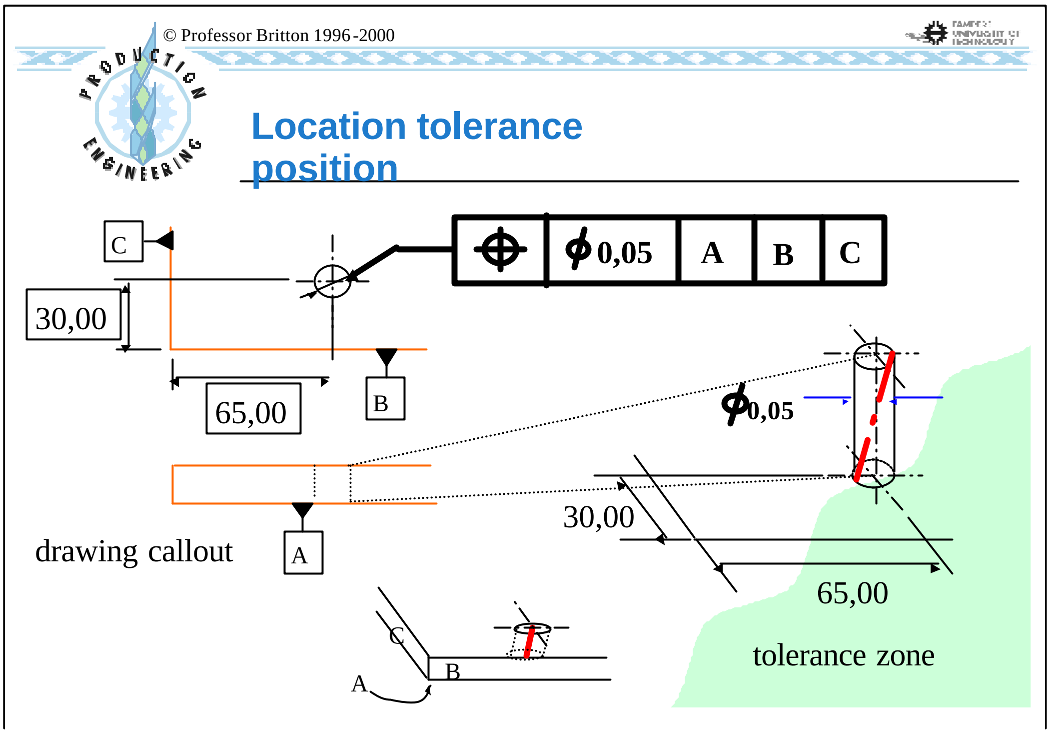


This slide shows an assembly. The next slide shows how datums were selected and how geometric tolerances were specified for the flange, in order to meet design functionality.

EXAMPLE (FLANGE)



Reproduced from Foster

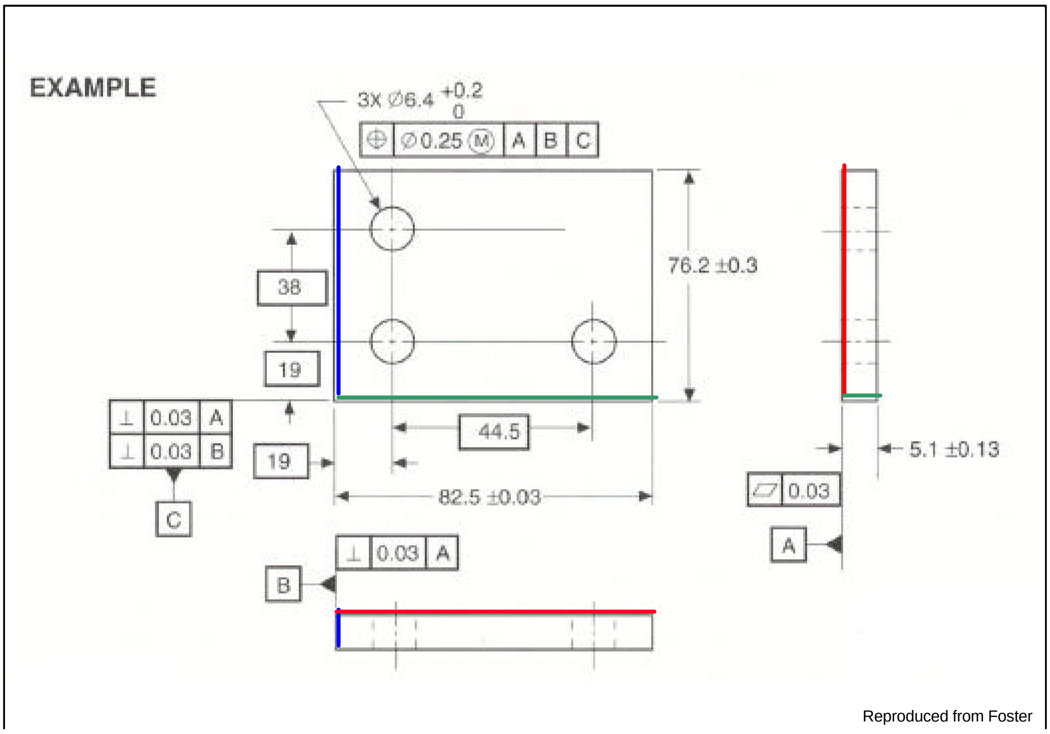


Location tolerances control position as well as orientation and form. Thus for a hole, a location tolerance will control the position of the hole axis relative to specified datums, tilt of the axis of the hole, and form deviations such as convexity or concavity of the axis.

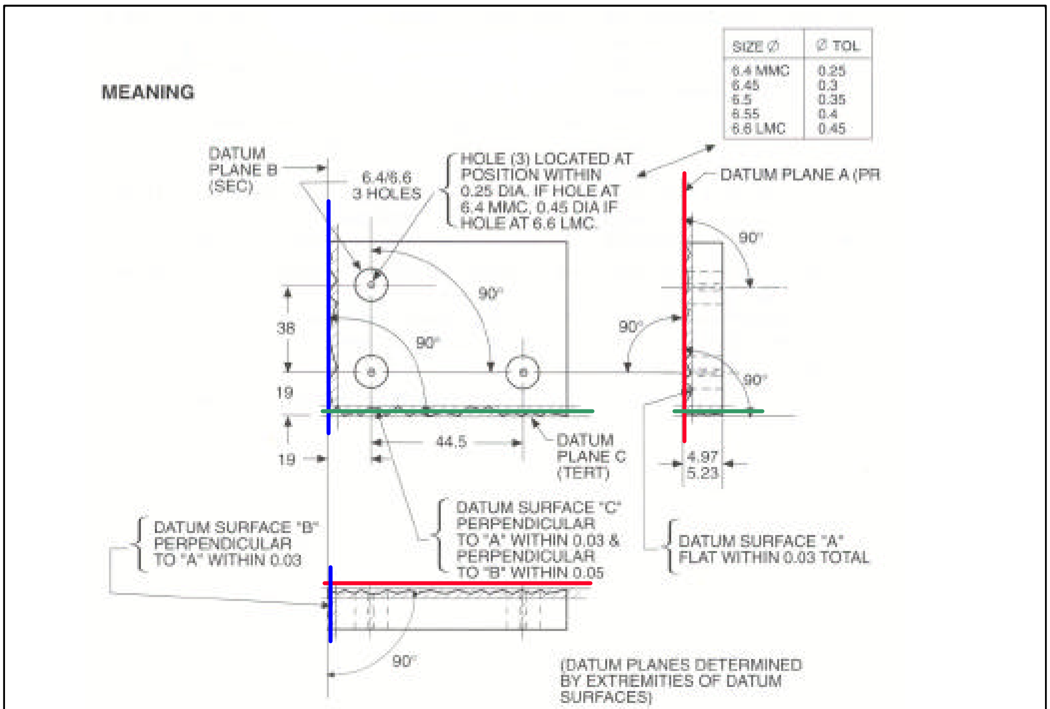
A position tolerance controls the position of a feature relative to one or more datums. The example shows how the axis of a hole is controlled. 3-2-1 datum system has been used with A being the primary datum, B the secondary and C the tertiary (these terms will be defined later on).

The tolerance zone is a cylinder set at right angles to the primary datum with a diameter equal to the tolerance value. The location of the axis of this cylinder is specified by theoretically exact distances (65 mm and 30 mm) from the secondary and tertiary datum surfaces.

All points on the axis of the hole must lie within the tolerance zone. The zone applies over the complete depth of the hole.

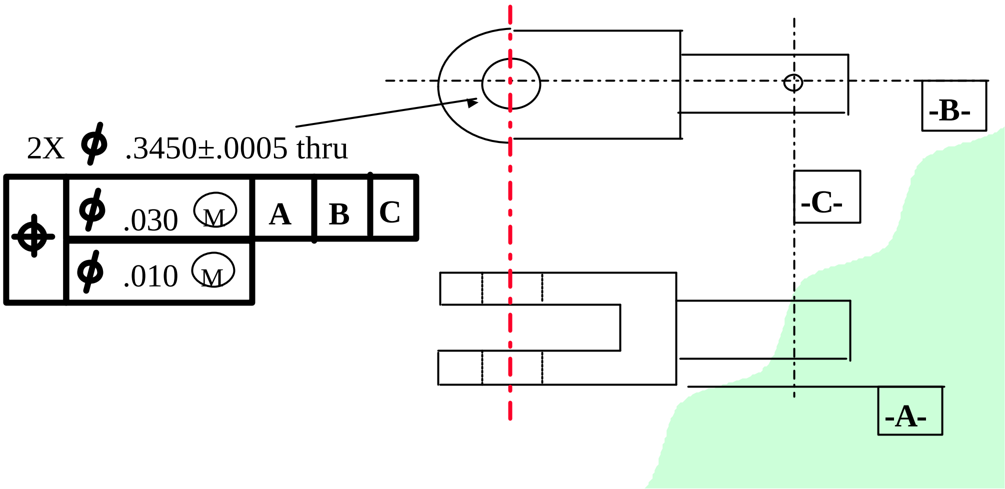


This slide and next give an example of hole positioning, showing how the datums are controlled relative to each other. A is primary datum, B is secondary datum and C is tertiary datum.

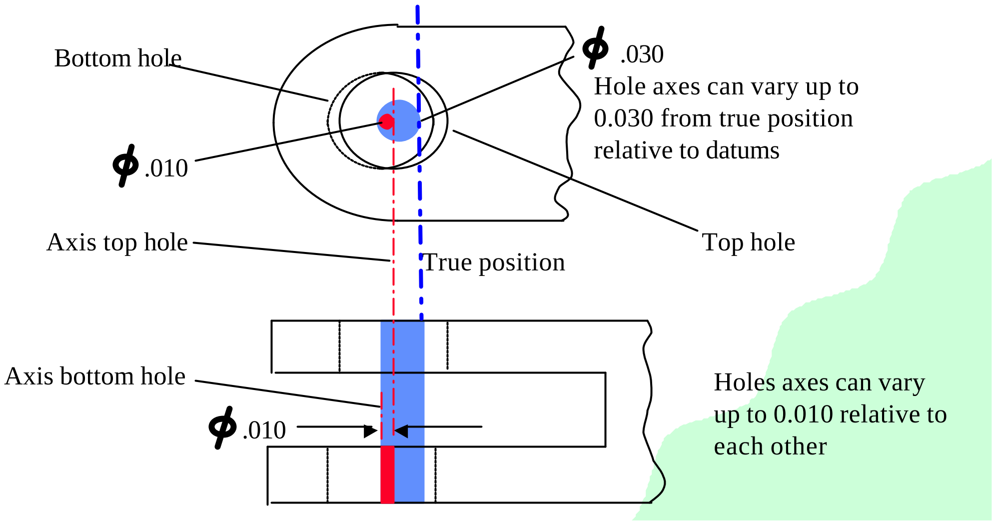


Reproduced from Foster

Composite positioning tolerancing example

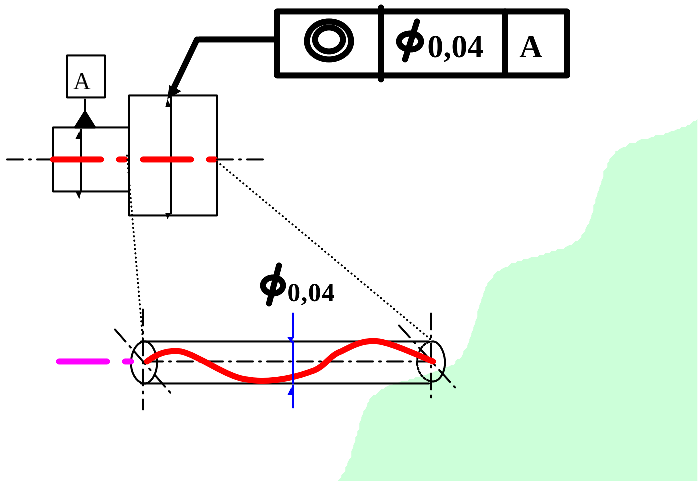


Composite positioning tolerancing interpretation



Location tolerance concentricity

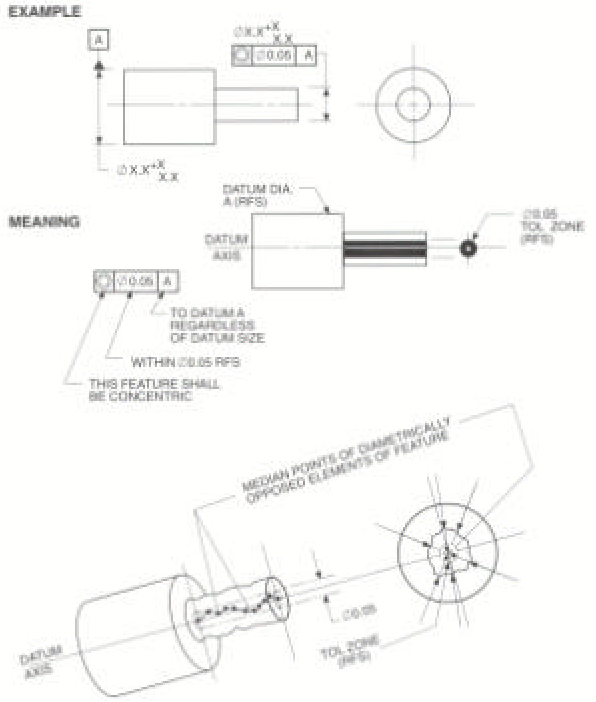
drawing callout



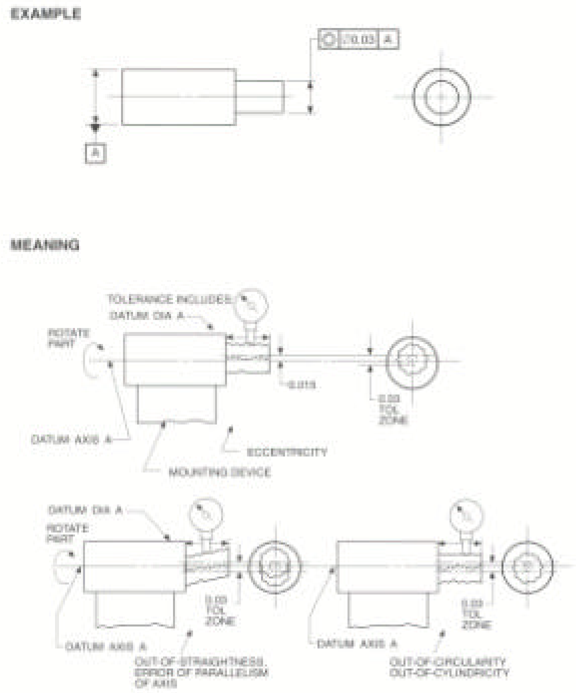
tolerance zone

A concentricity (coaxiality) tolerance controls the axis of one feature relative to the axis of another feature. In this case, the axis of the larger cylinder must lie within a cylindrical tolerance of 0,04 mm of the axis of the smaller cylinder (defined as datum A).

The tolerance zone applies over the full length of the feature-being-controlled.



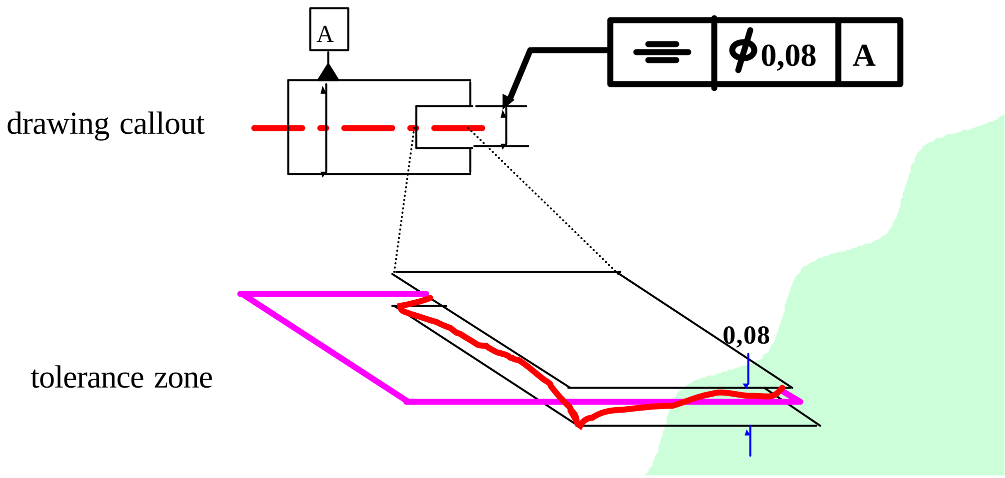
Reproduced from Foster



Reproduced from Foster

This slide shows the errors that are included in a concentricity tolerance. For practical purposes these errors are considered indistinguishable. Hence the total deviation of an axis is a composite of these errors when checking compliance with a concentricity tolerance.

Location tolerance symmetry



The median face should lie within two parallel planes set 0,08 mm apart that are symmetrically positioned about the datum median plane.

Datum A is at RFS.

Measurement Principles

These notes briefly outline the measurement principles used to assess compliance to geometric tolerances.

© Professor Britton 1996-2000

Measurement principles

- workpiece must be correctly aligned in measurement device
- this is defined by minimum alignment requirements
  - for form tolerances
  - for orientation, location and run-out tolerances
  - for roundness and cylindricity

One important point to note is the workpiece must be correctly aligned in the measurement device otherwise large errors in measurement can occur.

The alignment conditions are specified by minimum requirements.

© Professor Britton 1996-2000

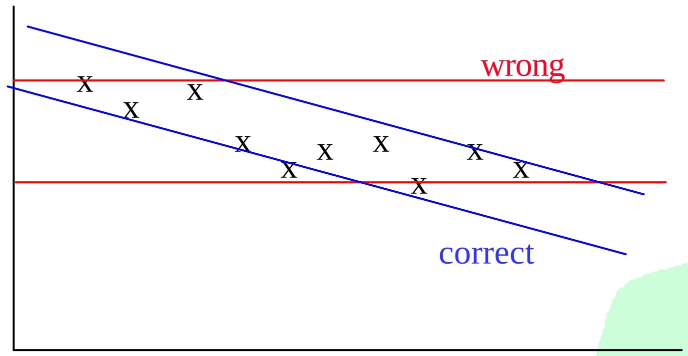
Minimum requirement form tolerance zone

- **for straightness and flatness this requires that distance between parallel lines or planes is a minimum**
- **for roundness and cylindricity this requires that the concentric circles and coaxial cylinders be located so that their radial distance is a minimum**

In addition to the above requirements, roundness and cylindricity require that the axis of the workpiece not be inclined relative to the measurement device. Any inclination will result in an oval shape and increase the estimate of the amount of deviation.

illustration straightness

measurements



position along length of feature

This slide illustrates the wrong way and correct way to measure straightness.

Assume a series of measurements have been taken along a line as indicated. Then the actual amount of deviation of these measurements is given by the distance between the blue lines. The red lines are incorrect.

There are various methods for establishing position and orientation of the blue lines, e.g., least squares regression.

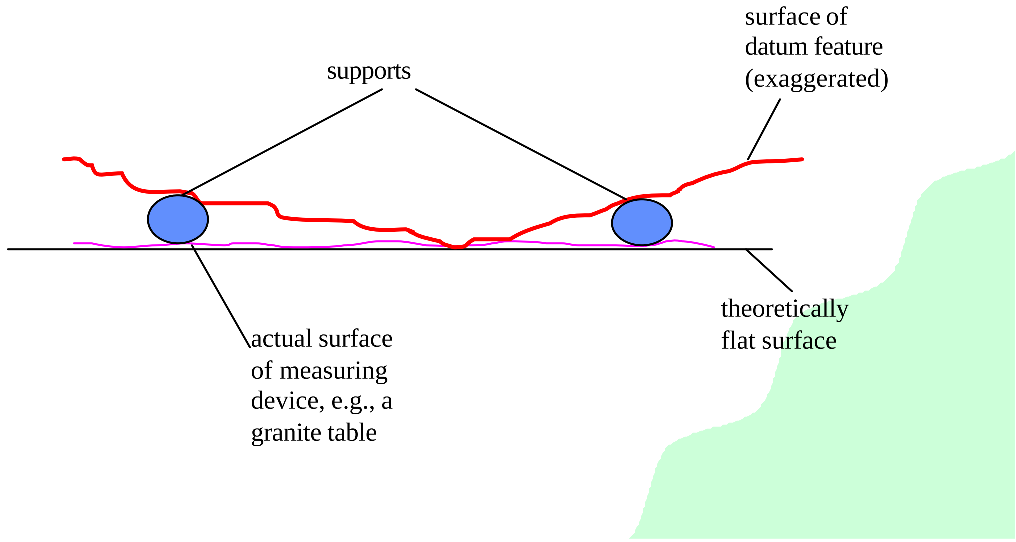
Minimum rock requirement

- deviations for orientation, location, and run-out are measured relative to datums, which must meet the minimum rock requirement
- minimum rock requirement positions the workpiece relative to the datum feature such that the amount of rock (movement) in all directions is equalised

Orientation, location and run-out are measured relative to specified datums. On the drawing callout the datums are theoretically exact surfaces. They have perfect form and position. In practice, the actual surfaces are imperfect. A supposedly flat surface may be convex and able to rock. Similarly a supposedly straight shaft may be barrel shaped and able to rock. The minimum rock requirement is aimed at provide consistent alignment under the different possible conditions that might occur.

When rocking occurs this must be minimised by equalising the amount of movement in all directions. Regression analysis can be used to determine the minimum rock orientation.

illustration
plane datum feature



This slide illustrates the minimum rock requirement for a plane datum feature. The datum feature will be positioned on the measuring device, which should have a surface condition at least an order of magnitude better than the the datum feature.

Supports can be used to minimise the amount of rock of the datum feature as shown above.

Simplified inspection methods

- **substitute a simpler and less precise method for actual method**
- **only use actual method if tolerance zone exceeded**
- **e.g., use run-out as substitute for concentricity**

Some geometric tolerances result in costly inspection methods. In these cases, a simplified inspection method can be used first. If the deviations exceed the tolerance zone then the actual inspection method is used. For example, run-out is easy to measure and so not very expensive. It can be used as a simplified method for concentricity or straightness of axes, which are costly to measure.

Precise verification of the minimum rock requirement is often very costly. Therefore approximate inspection methods are used with the aid of V-blocks, mandrels, centre holes, etc.

These approximate methods will result in a larger estimates of deviations than actually exist. Thus they are conservative; bad parts will be rejected. On the other hand, some marginal good parts will also be rejected. This is considered tolerable given the reduction in inspection costs that can be obtained.