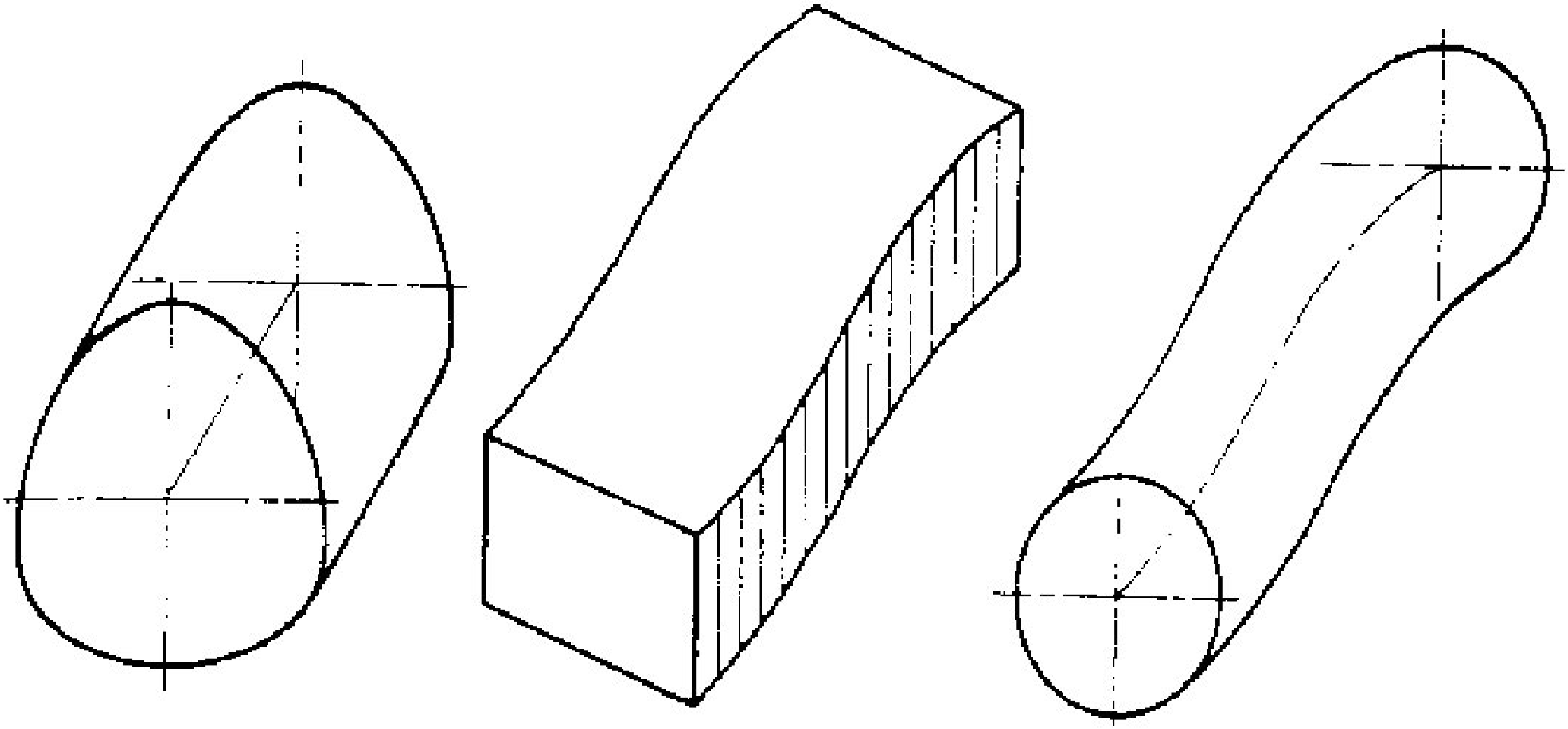
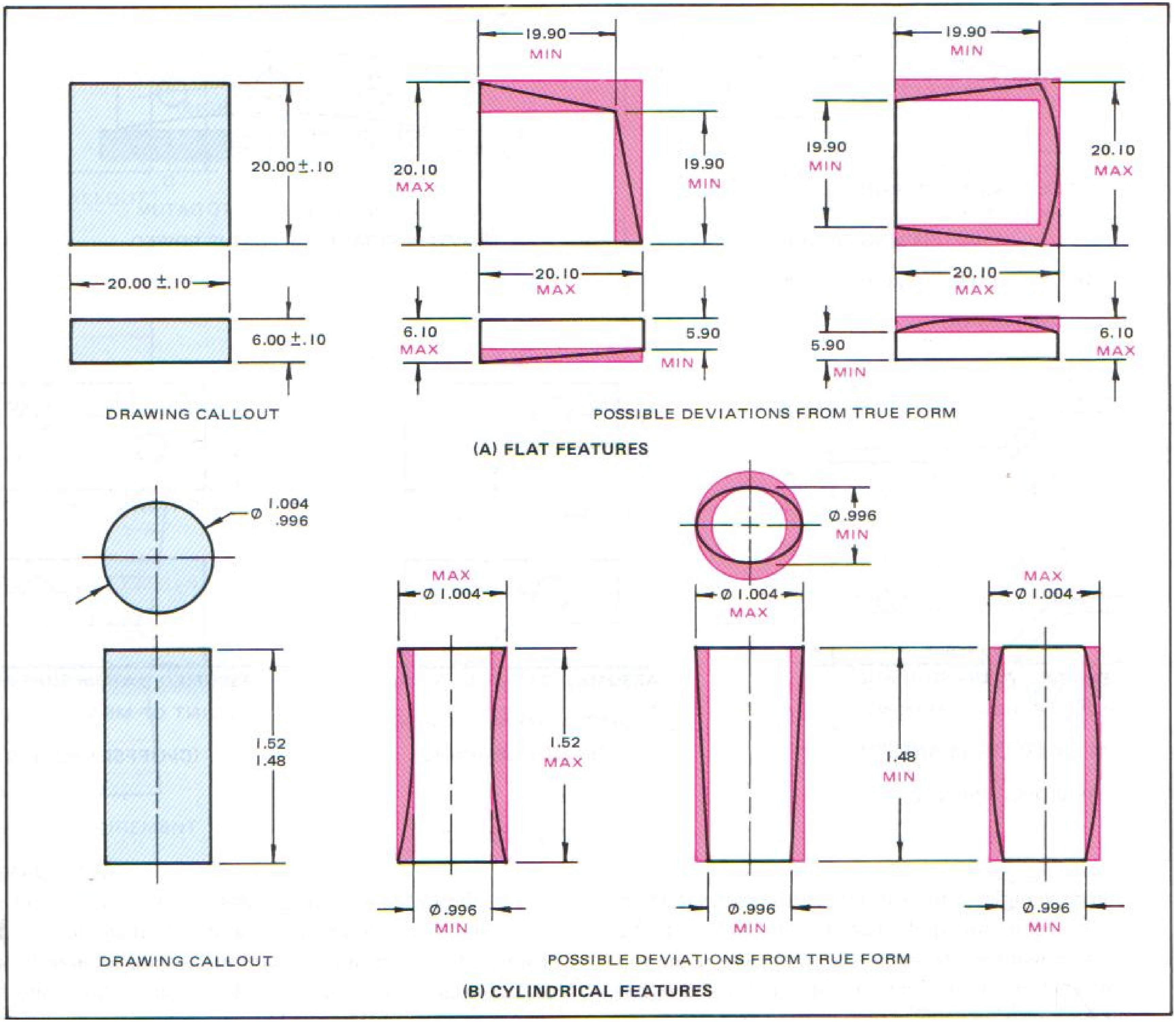
# Geometrical Tolerances




20.00 ±.10
20.00 ±.10
6.00 ±.10
DRAWING CALLOUT
19.90 MIN
20.10 MAX
19.90 MIN
20.10 MAX
6.10 MAX
5.90 MIN
19.90 MIN
19.90 MIN
20.10 MAX
20.10 MAX
5.90 MIN
6.10 MAX
POSSIBLE DEVIATIONS FROM TRUE FORM
(A) FLAT FEATURES
Ø 1.004 .996
1.52 1.48
MAX Ø 1.004
1.52 MAX
Ø .996 MIN
Ø .996 MIN
Ø 1.004 MAX
1.48 MIN
Ø .996 MIN
MAX Ø 1.004
Ø .996 MIN
DRAWING CALLOUT
POSSIBLE DEVIATIONS FROM TRUE FORM
(B) CYLINDRICAL FEATURES

# What is a Geometrical Tolerance?

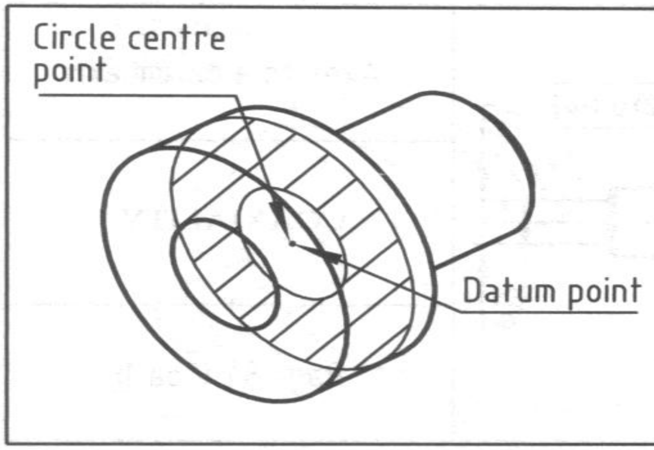
A geometrical tolerance is the maximum allowable variation of form or position of a feature. This is controlled by defining the size and shape of a tolerance zone. The specified part of the feature must be within this tolerance zone.

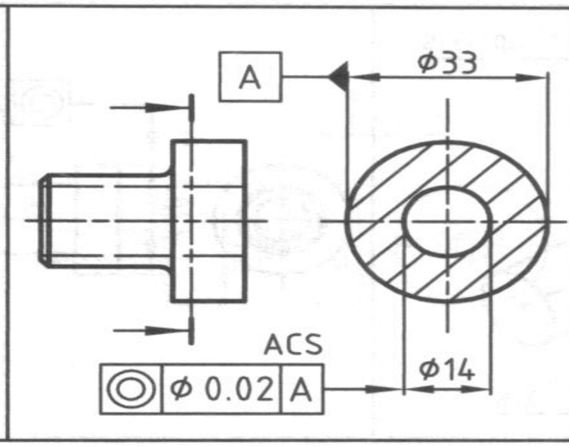
Geometrical Tolerancing can best be described as a language of symbols placed on technical drawings to adequately define the allowable variation of part geometry.

The current internationally accepted standard in Geometrical Tolerancing is ISO1101: 2004

# When to use a Geometrical Tolerance

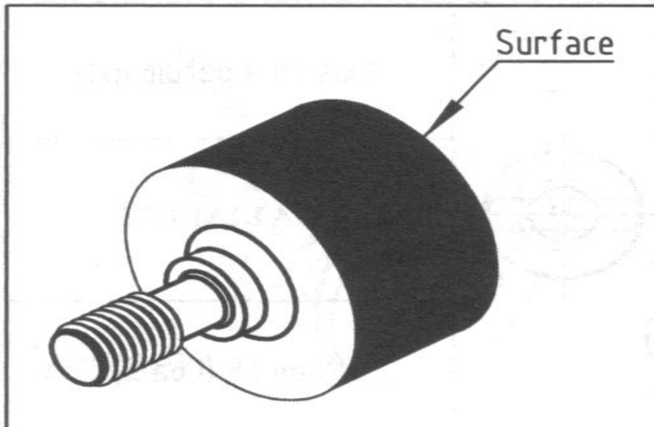
- The size tolerance of a dimension has a certain amount of control over form and attitude but if a better degree of control is required then geometrical tolerances should be used
- Position of a feature is also controlled by geometrical tolerances.
- The use of geometrical tolerances can increase manufacturing costs, so they should only be used when necessary.

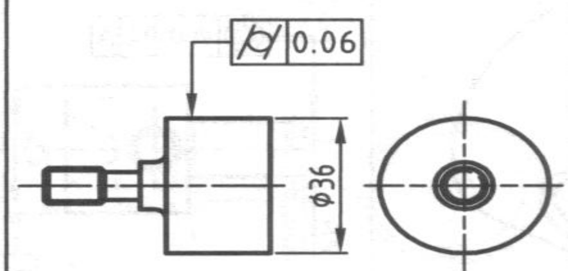



Circle centre point to a datum point

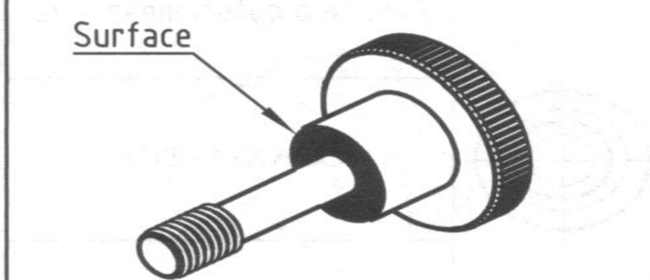
CONCENTRICITY

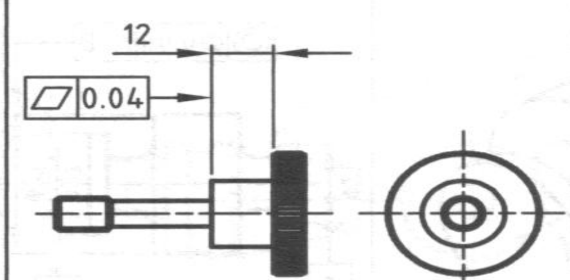



Surface

CYLINDRICITY



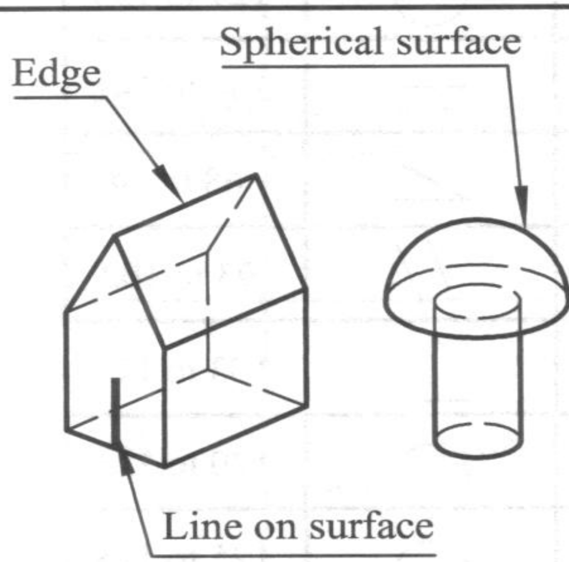

Surface

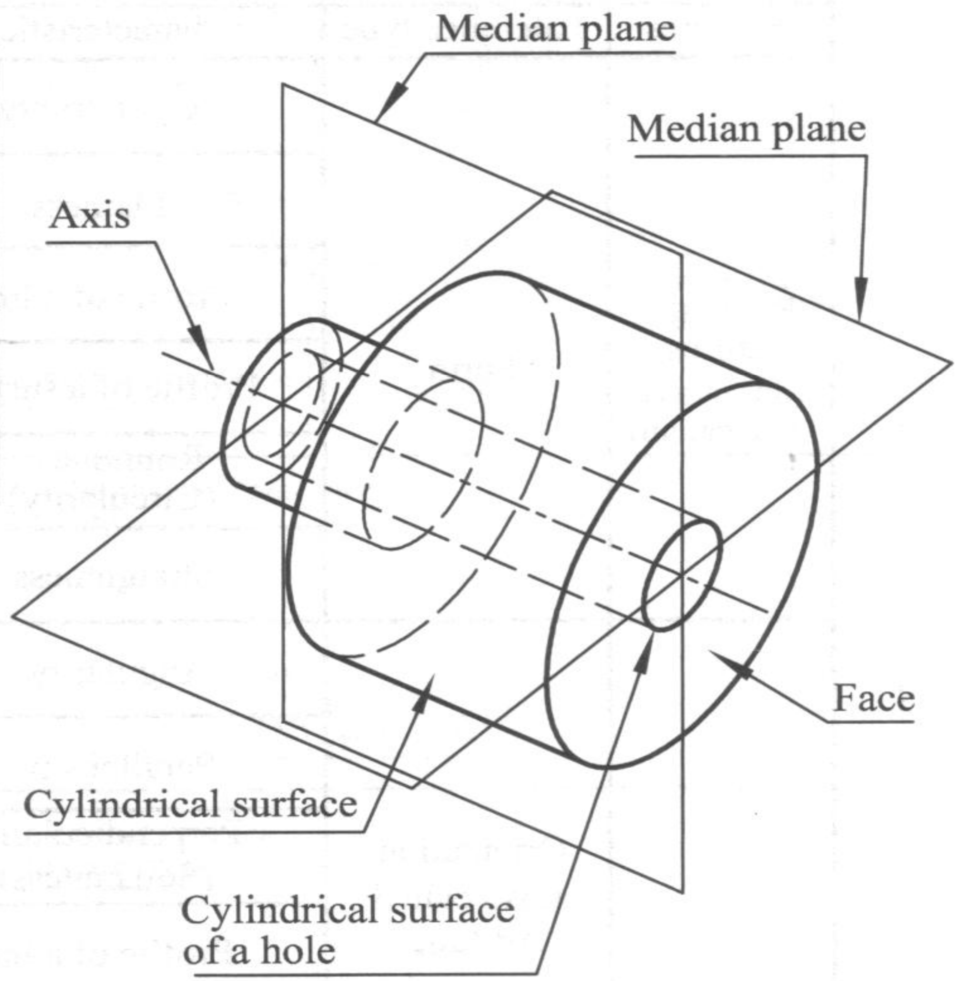
FLATNESS

# SINGLE FEATURES

The following sketches illustrate some of the single features that could be on a component.

| Some examples of single features |
|---|
| An axis |
| A cylindrical surface |
| A cylindrical surface of a hole |
| An edge |
| A face |
| A line on a surface |
| A median plane |
| A spherical surface |

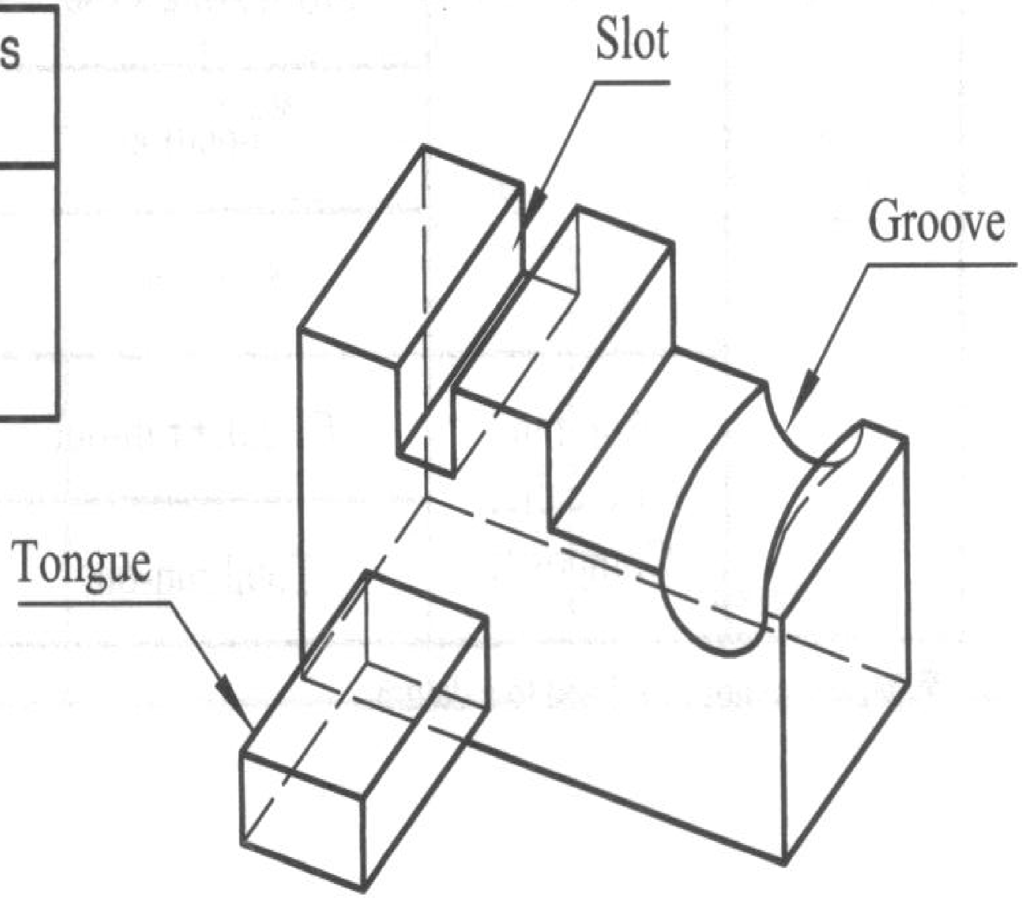


## Combinations of Single Features

The following sketch illustrates some combinations of single features that could be on a component.

| Some examples of combinations of single features |
|---|
| A groove<br>A slot<br>A tongue |

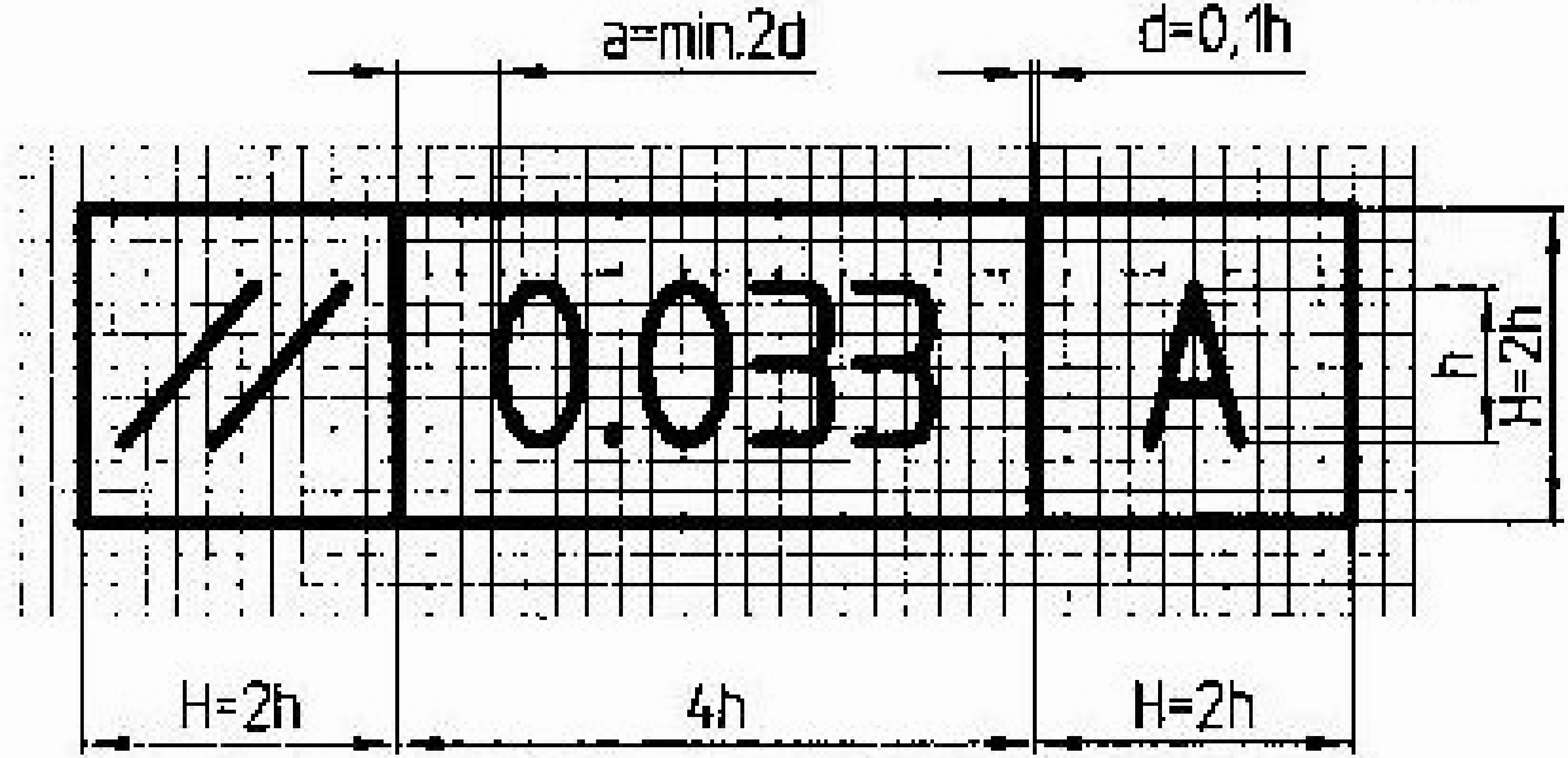


Geometric characteristic symbols

| Features | Tolerance type | Characteristic | Symbol |
|---|---|---|---|
| For single features (not related to a datum) | Form | Cylindricity | ⌭ |
| | | Flatness | ⏥ |
| | | Profile of a line | ⌒ |
| | | Profile of a surface | ⌓ |
| | | Roundness (Circularity) | ○ |
| | | Straightness | — |
| For related features (related to a datum) | Orientation (also called Attitude) | Angularity | ∠ |
| | | Parallelism | ∥ |
| | | Perpendicularity (Squareness) | ⊥ |
| | Location | Concentricity | ◎ |
| | | Coaxiality | ◎ |
| | | Position | ⌖ |
| | | Symmetry | ⌯ |
| | Run-out (also called Composite) | Circular run-out | ↗ |
| | | Total run-out | ⌰ |

# SIZE OF FEATURE CONTROL FRAME

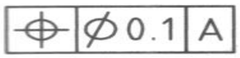
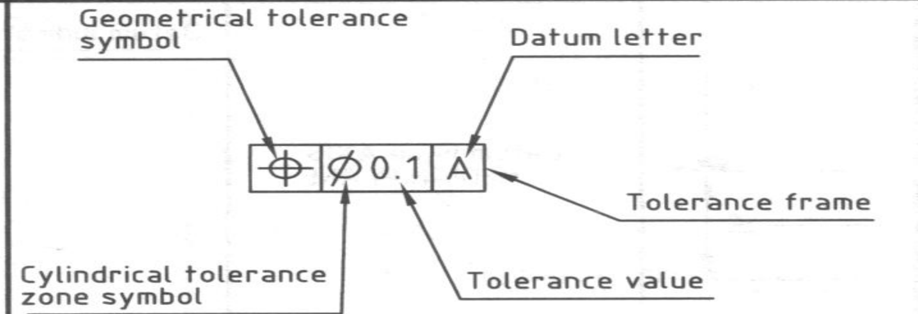
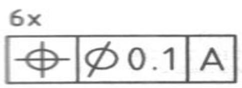
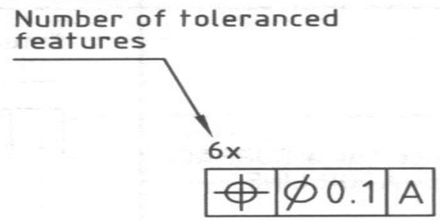
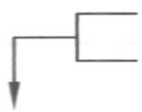
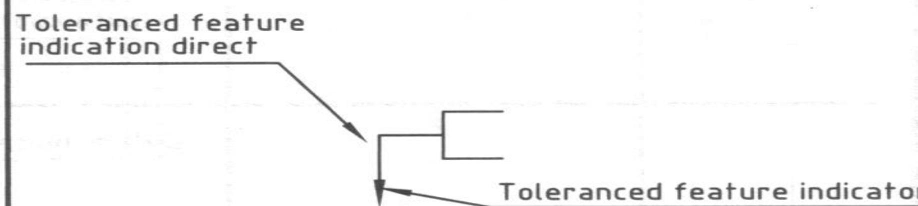
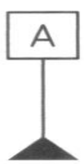
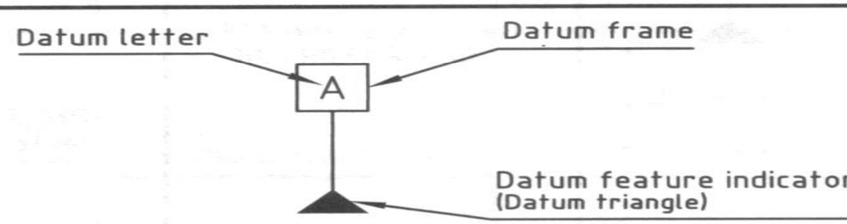
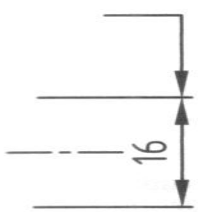
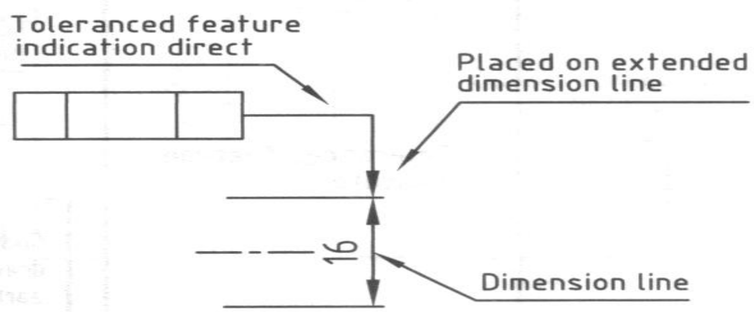
TS 10844



## ŞEKİL VE KONUM ÇEŞİTLERİYLE SEMBOLLERİ (TS 1304 ISO 1101)

| Eleman Cinsi | Özelliği | Adı | Sembol |
|---|---|---|---|
| TEK ELEMANLAR | ŞEKİL | Doğrusallık | ⏤ |
| | | Düzlemsellik | ⏥ |
| | | Dairesellik | ○ |
| | | Silindiriklik | ⌭ |
| | | Bir çizginin şekli | ⌒ |
| | | Bir yüzeyin şekli | ⌓ |
| BİRBİRLERİYLE İLGİLİ ELEMANLAR | YÖN | Paralellik | ∥ |
| | | Diklik | ⊥ |
| | | Eğiklik (Açısallık) | ∠ |
| | KONUM | Bir elemanın konumu | ⌖ |
| | | Ortak merkezlilik,eksenlilik | ◎ |
| | | Simetriklik | ⌯ |
| | YALPALAMA | Yalpalama | ↗ |
| | | Toplam yalpalama | ⌰ |

Symbols for geometrical tolerancing

| Symbol on the drawing | Description | Interpretation |
| --- | --- | --- |
|  | Tolerance frame (also known as a feature control frame) |  |
|  | Multiple tolerance frames |  |
|  | Toleranced feature indicator |  |
|  | Datum indicator |  |
|  | Tolerance for an AXIS or a MEDIAN PLANE |  |

## The feature controlled

The feature controlled by the tolerance is indicated by a leader line connecting it to the tolerance frame. At the toleranced feature the leader line terminates in an arrow head which is positioned as follows:

(1) On the outline of the feature or on an extension of the outline *but not at a dimension line* when the tolerance refers to the line itself or to the surface represented by the line (Fig. 4).

(2) On the outline or on a projection line of the feature *at a dimension* line when the tolerance refers only to the axis or median plane of the feature so dimensioned (Figs. 5 and 6).

(3) On the axis or median plane when the tolerance refers to the common axis or median plane of all features lying on that axis or median plane (Figs. 7, 8 and 9).

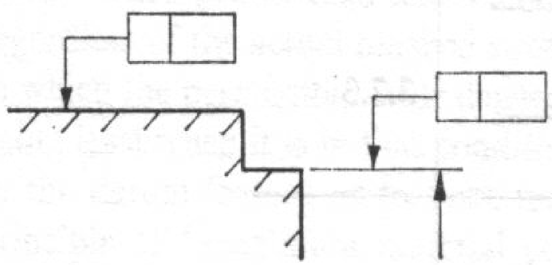


Fig. 4

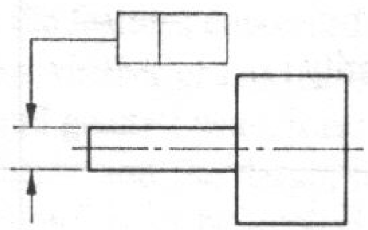


Fig. 5

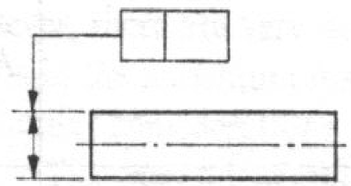


Fig. 6

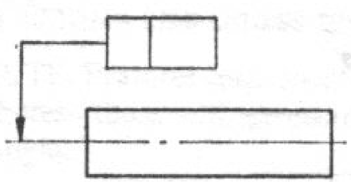


Fig. 7

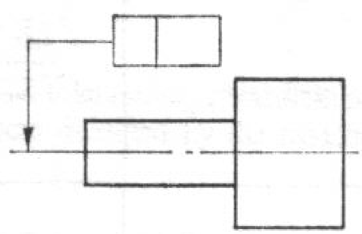


Fig. 8

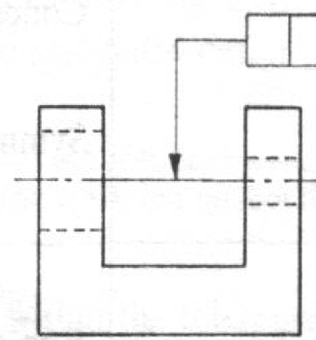


Fig. 9

## Tolerances applicable to restricted lengths of features

If a tolerance is applied only to a particular part of a feature, that part is indicated as in Fig. 25.

If the tolerance value over the whole of the considered length or surface is to be qualified by a smaller tolerance on any specified shorter length lying anywhere within it, this is expressed as in Fig. 26 where the tolerance value 0.05 applies to any length 100. This convention used for a feature which is a surface means that the tolerance value is applicable to all lines of the specified length lying in any position and in any direction on the surface.

If the tolerance is only specified in the form of a tolerance value applicable to a specific length as in Fig. 27, an accumulation of tolerance over the whole length or surface is permitted.

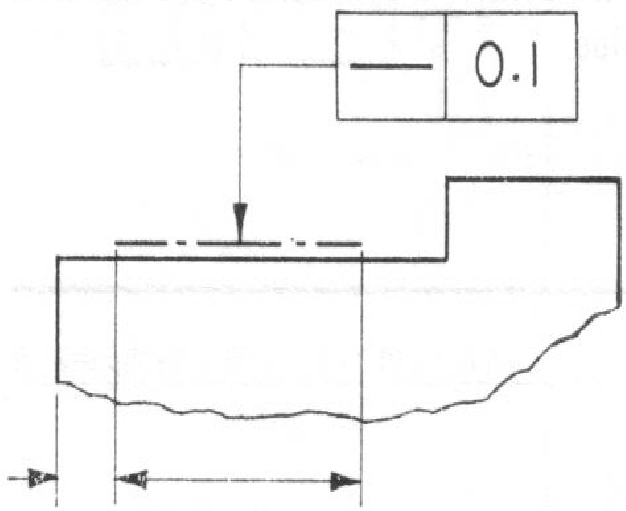



Fig. 25

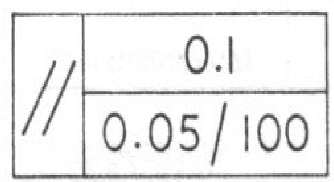



Fig. 26

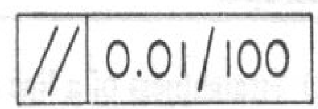
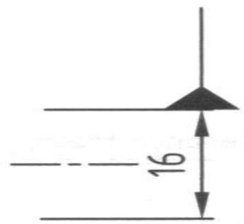
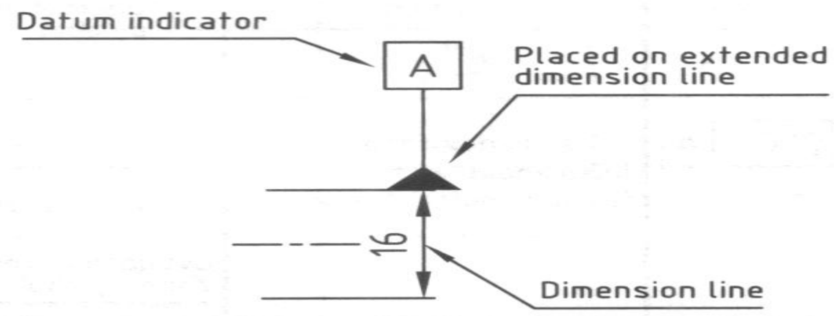
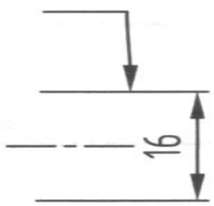
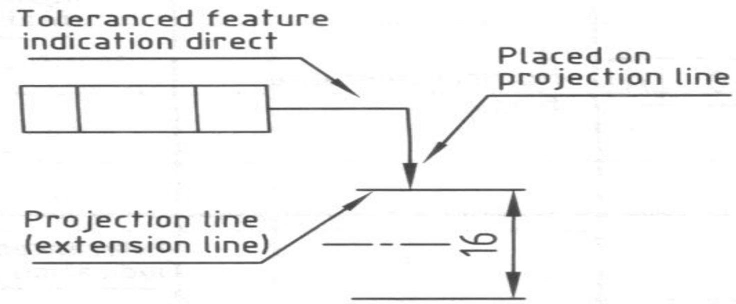
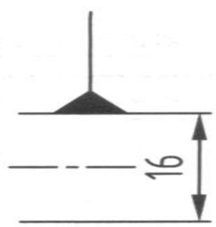
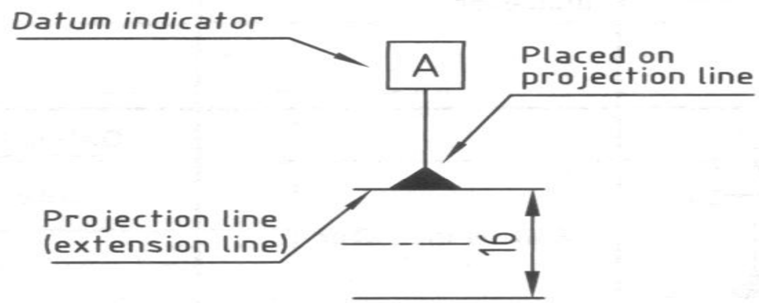
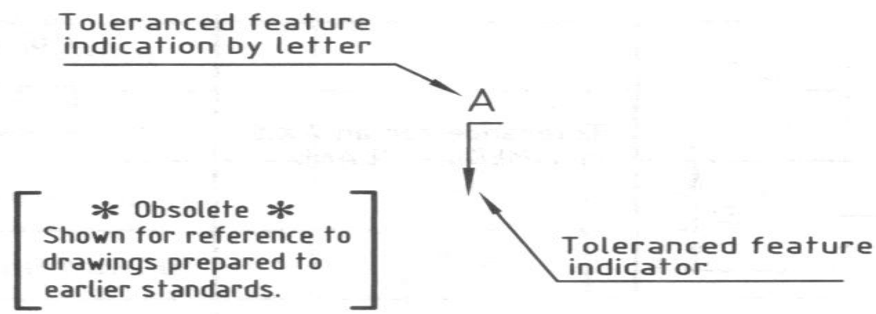



Fig. 27

Symbols for geometrical tolerancing

| Symbol on the drawing | Description | Interpretation |
|---|---|---|
|  | Datum is an AXIS or MEDIAN PLANE |  |
|  | Tolerance for a SURFACE or GENERATOR LINE |  |
|  | Datum is a SURFACE or GENERATOR LINE |  |
|  | Toleranced feature indicator |  |

## Datum features

The datum feature(s) is (are) indicated by a leader line from the tolerance frame terminating in a solid triangle the base of which lies as follows:

(1) On the outline of the feature or an extension of the outline (but not at a dimension line), when the datum feature is the line or surface itself (Fig. 10).

(2) On the projection line at the dimension line or on the axis when the datum feature is the axis or median plane of the whole component (Figs. 12 and 13) or of the part so dimensioned (Fig. 11).

(3) On the common axis or median plane of two or more features (Figs. 14 and 15).

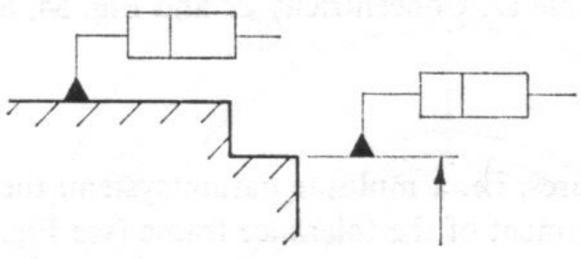


Fig. 10

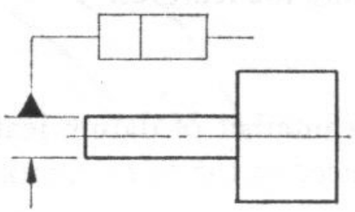


Fig. 11

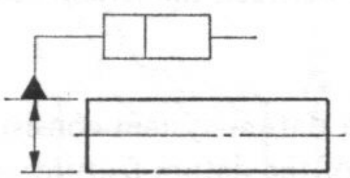


Fig. 12

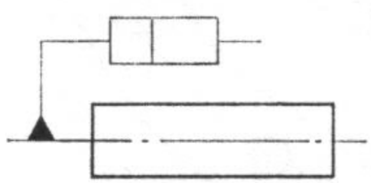


Fig. 13

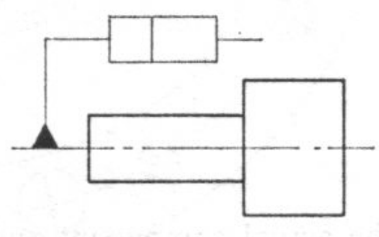


Fig. 14

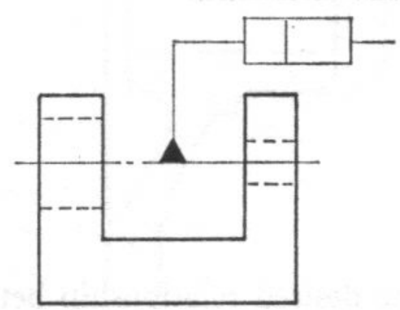
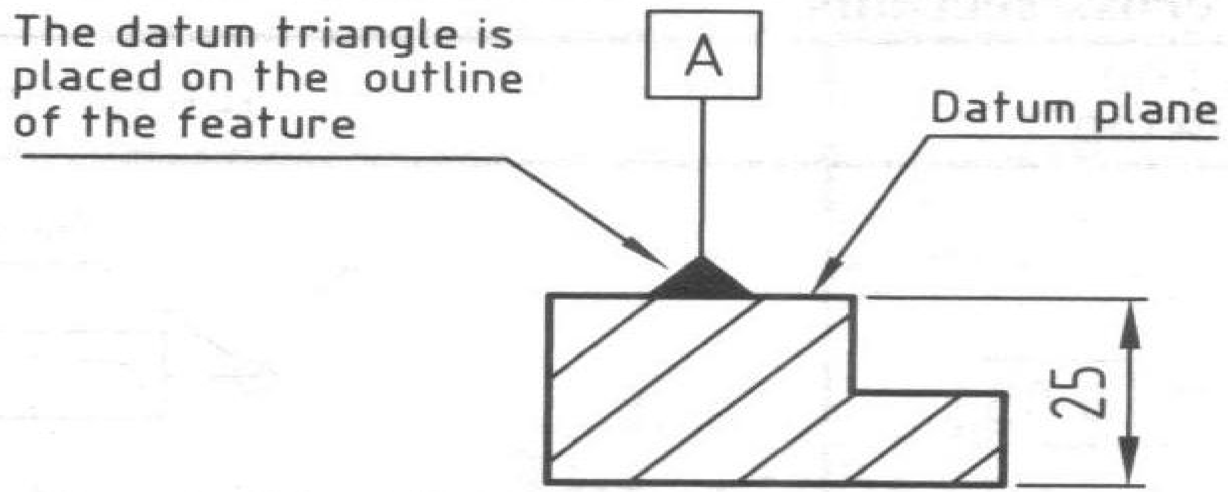


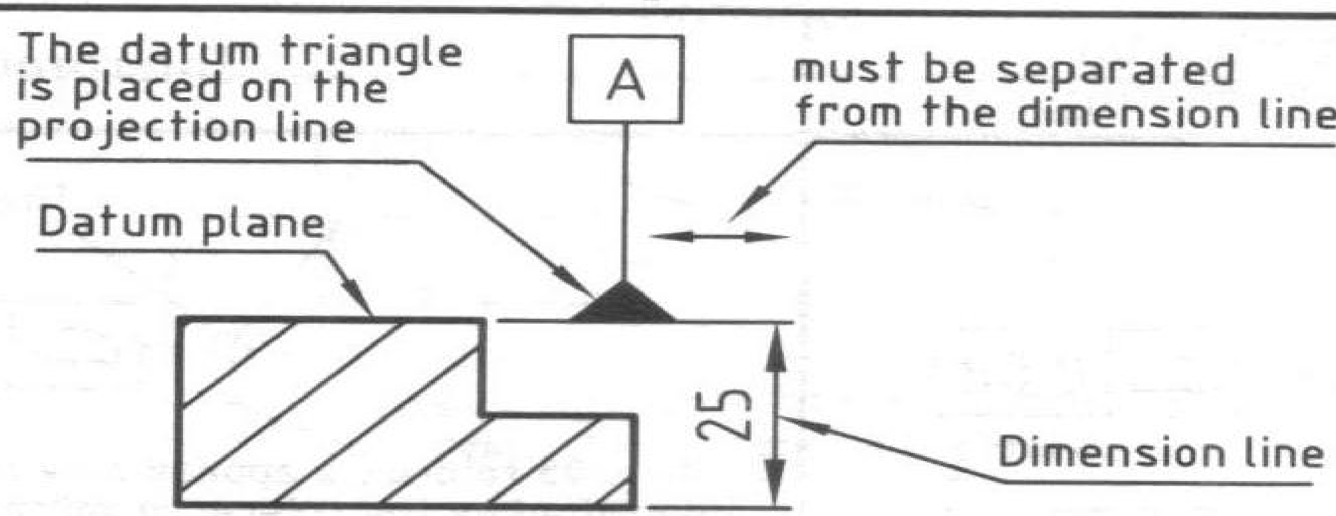
Fig. 15

Interpretation




**datum is a plane or line**

Interpretation



If the tolerance frame cannot be connected in a clear and simple manner with the datum feature, a capital letter* in a frame is connected to the datum feature

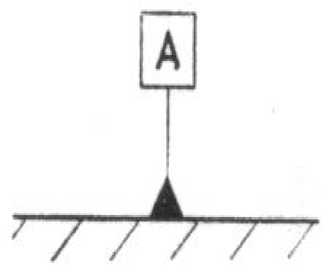



Fig. 16

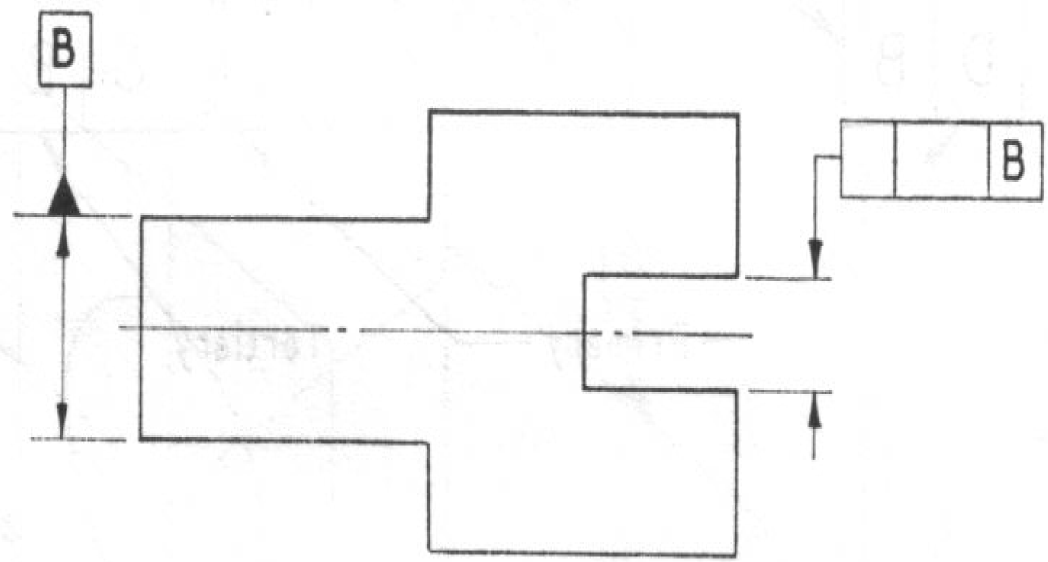
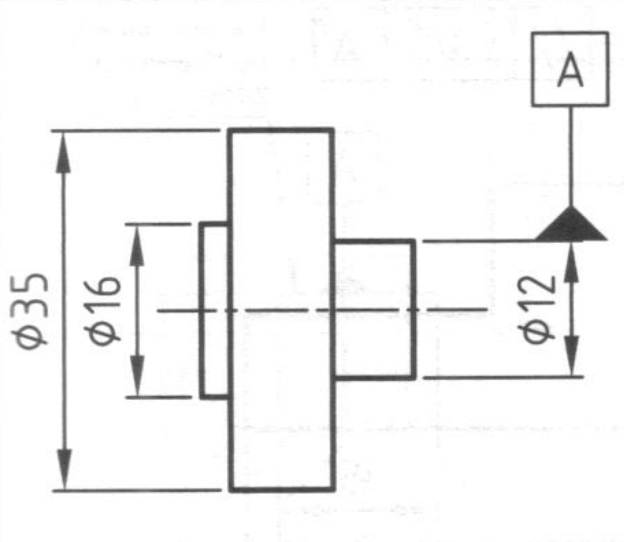
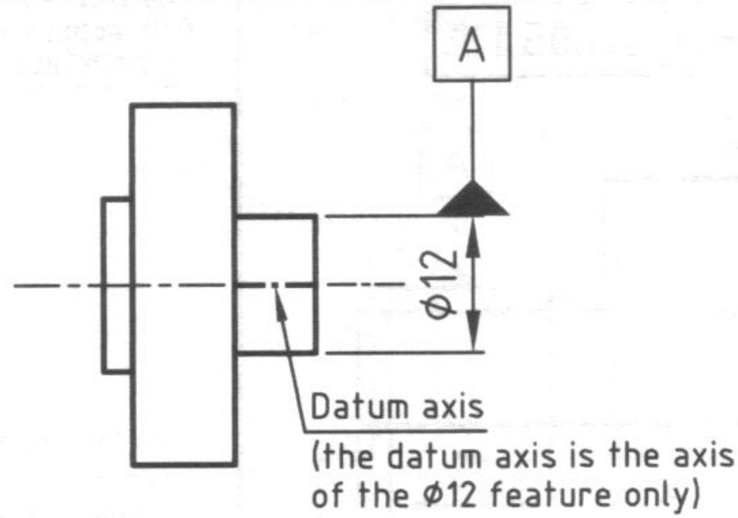



Fig. 17

## When the datum is an axis

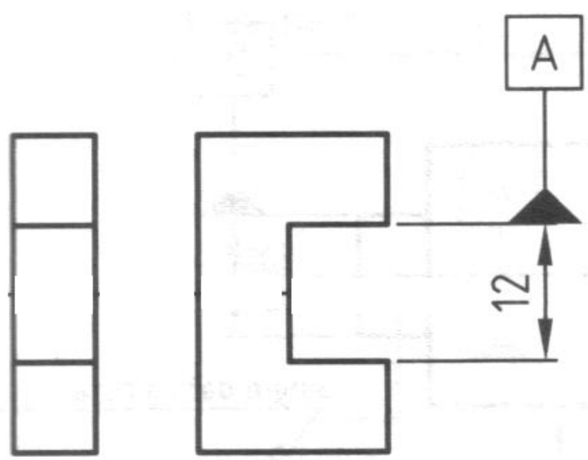
| Symbol on the drawing | Interpretation |
|---|---|

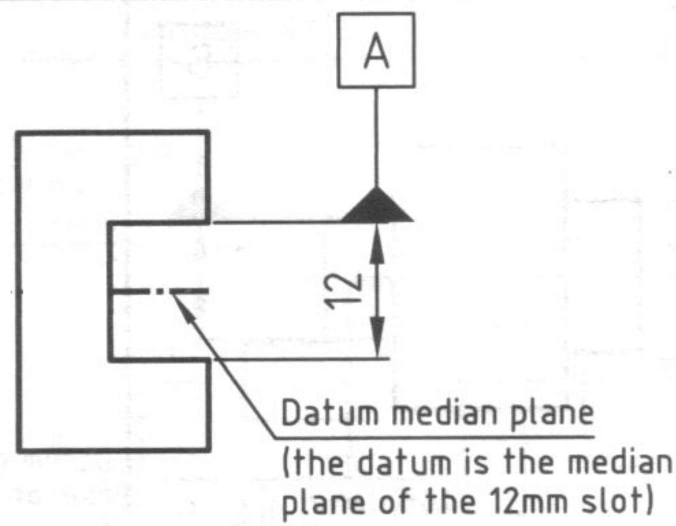



## When the datum is a median plane

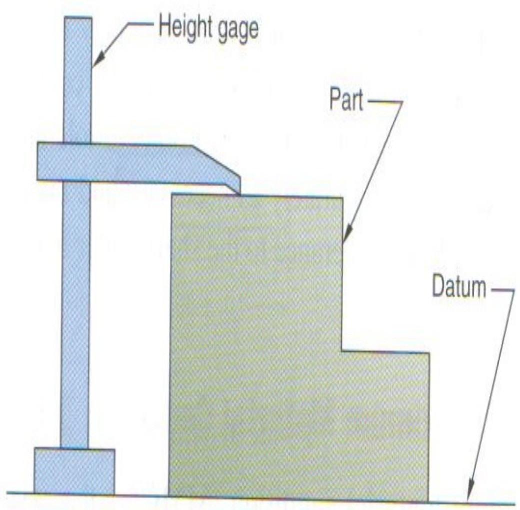
| Symbol on the drawing | Interpretation |
|---|---|



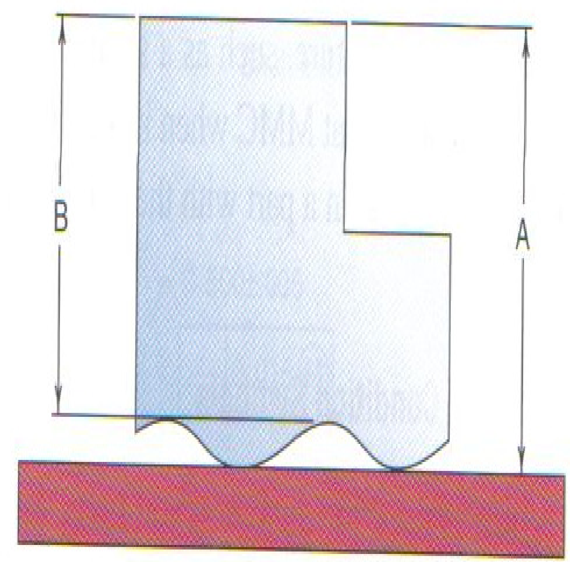



## Datums and Datum Features

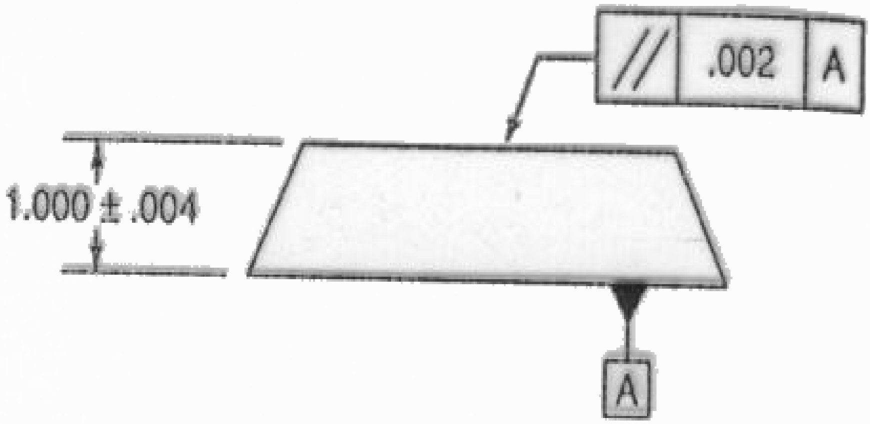
A **datum** is a starting place for a dimension. A datum may be a perfect plane, a center line, or a point. Datums are perfect, and they are not real. Examples are the center line of a shaft or the point at the center of a sphere. These are theoretical positions that either can be represented with inspection tools or can be derived. For example, a center line is represented by the center of an inspection pin or gage or by the center of an inspection spindle. A center line is derived by measuring to the edge of a gage pin and then adding half the diameter of the pin to locate the center of the gage pin from an edge or another feature. For a hole, the measurement is *not* to the edge of the feature hole but to the largest-gage pin that will fit into the hole.

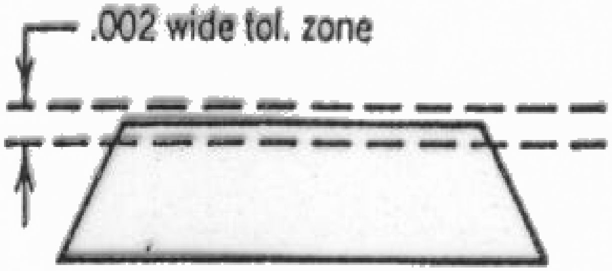



The bottom surface of the part is the datum feature, surface plate is the datum.

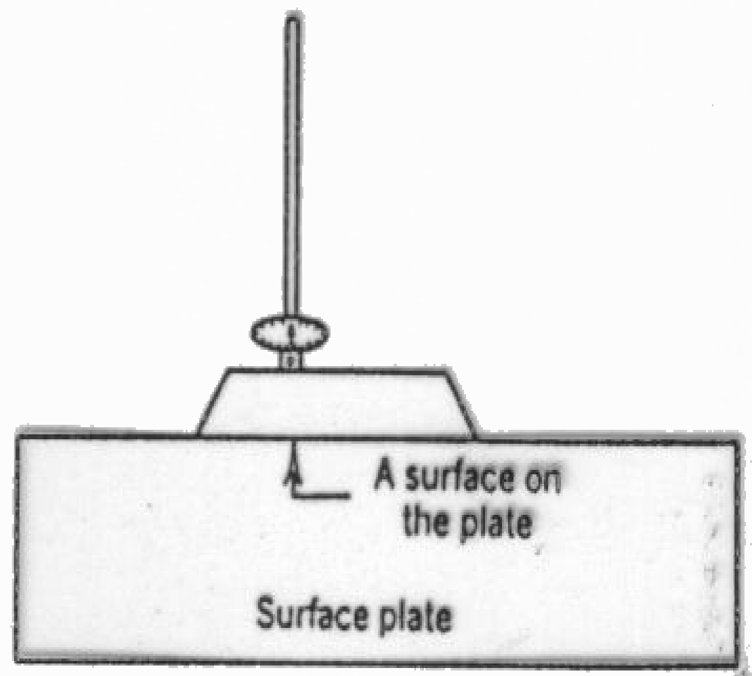


Parallelism callout.



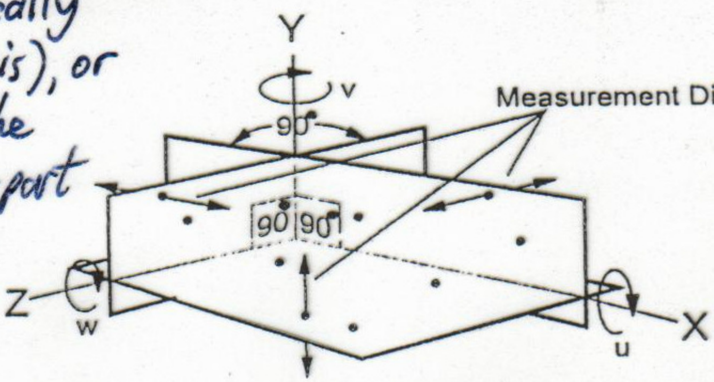




Parallelism tolerance zone.

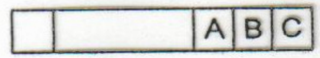


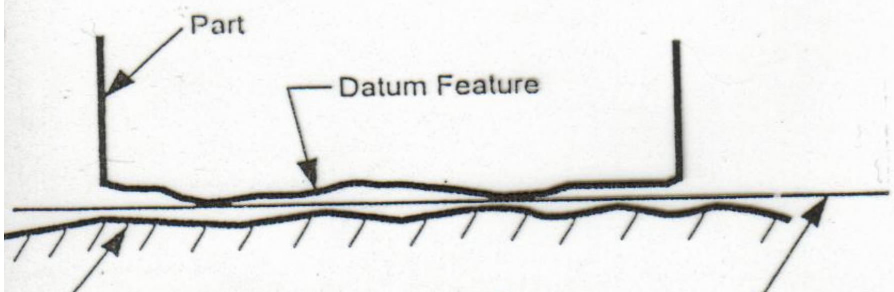

Datum is located.

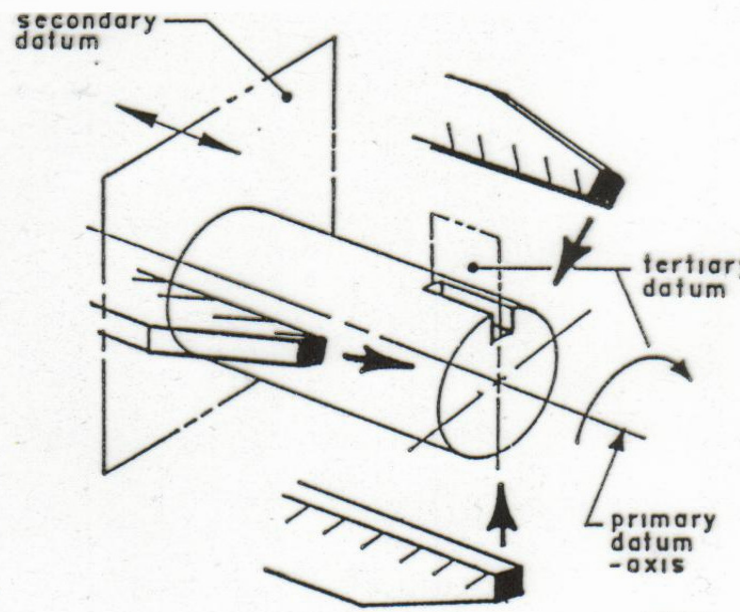
Datum is a theoretically exact point, line (axis), or plane derived from the true geometric counterpart of a specified datum feature.




Datum Reference Framework. Three Mutually Perpendicular Planes.




**Datum**

Theoretically perfect points, lines, or planes

**Simulated Datum (Datum Simulators)**

Surfaces and axes of processing or inspection equipment

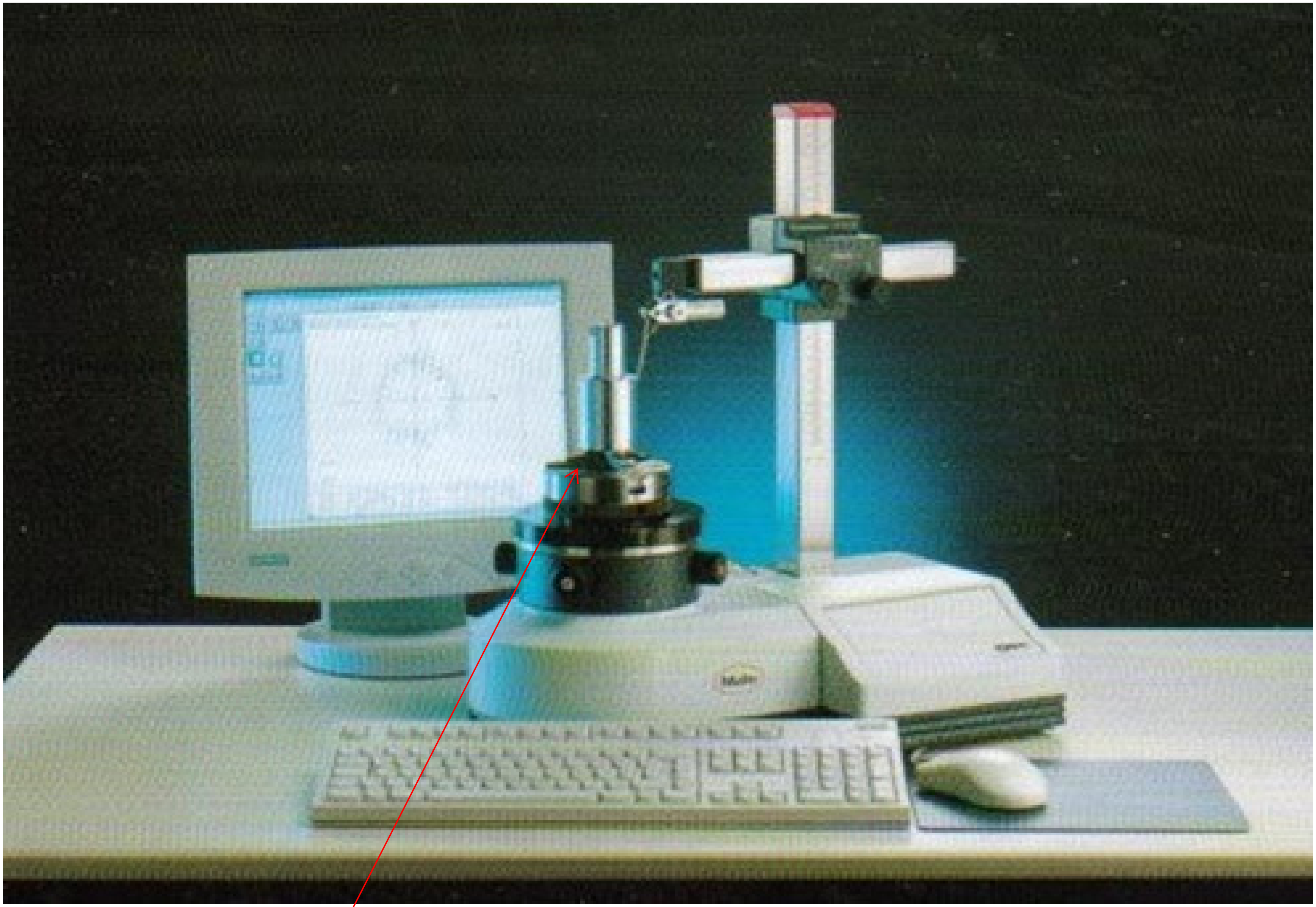
**Datum Feature**

Actual part feature surfaces

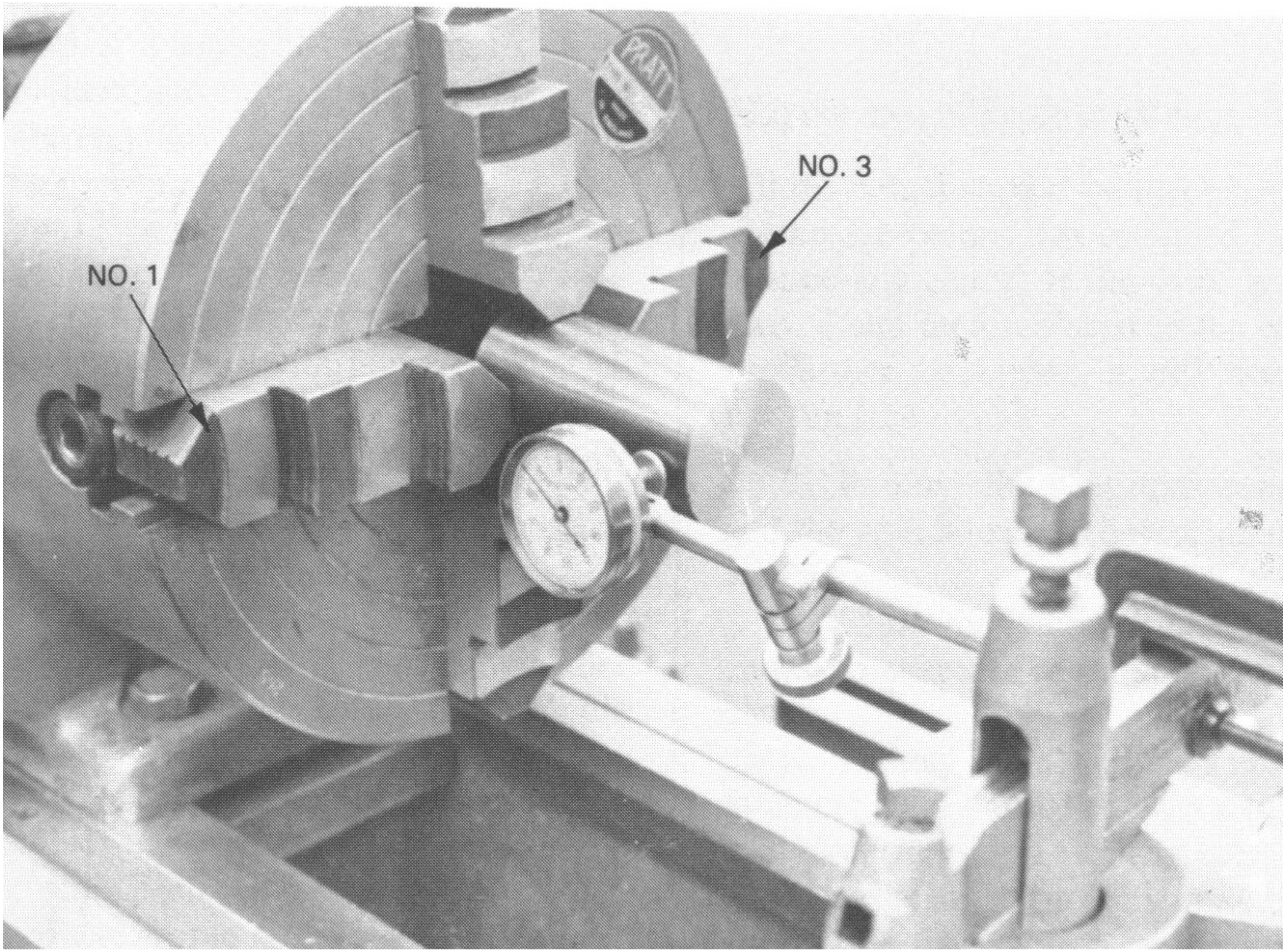
**Temporary Datum**

Introduced for processing or inspection purposes (may be removed)

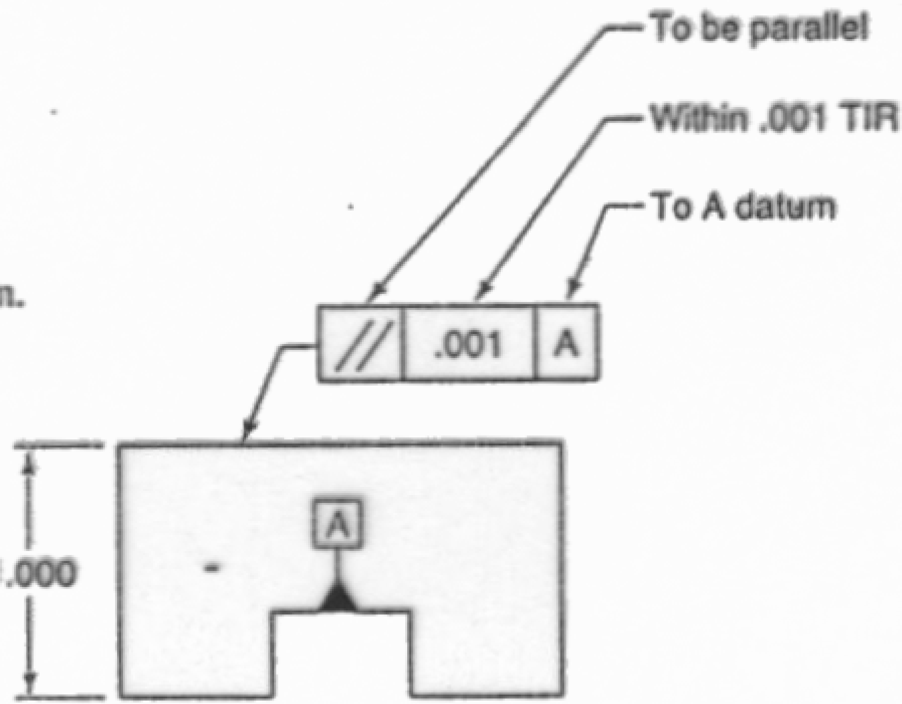

libero
libero



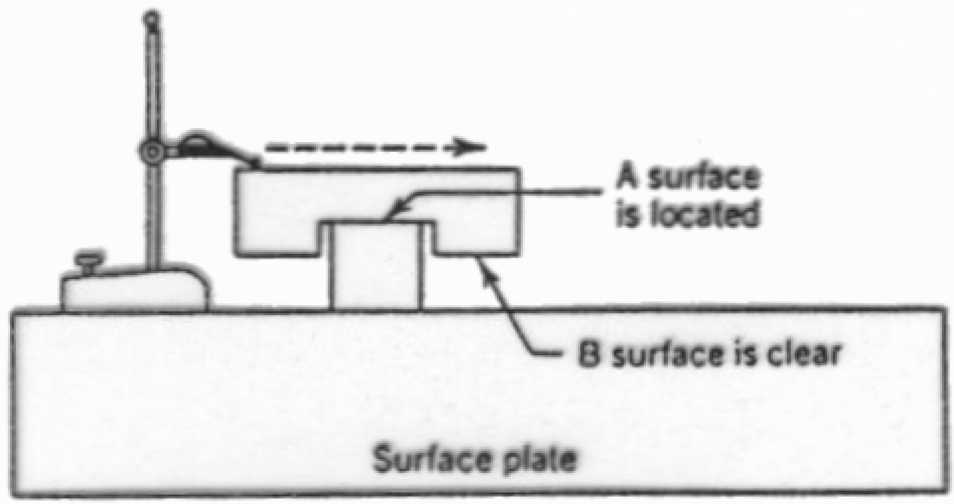
Datum simulator (jaws of the chuck)

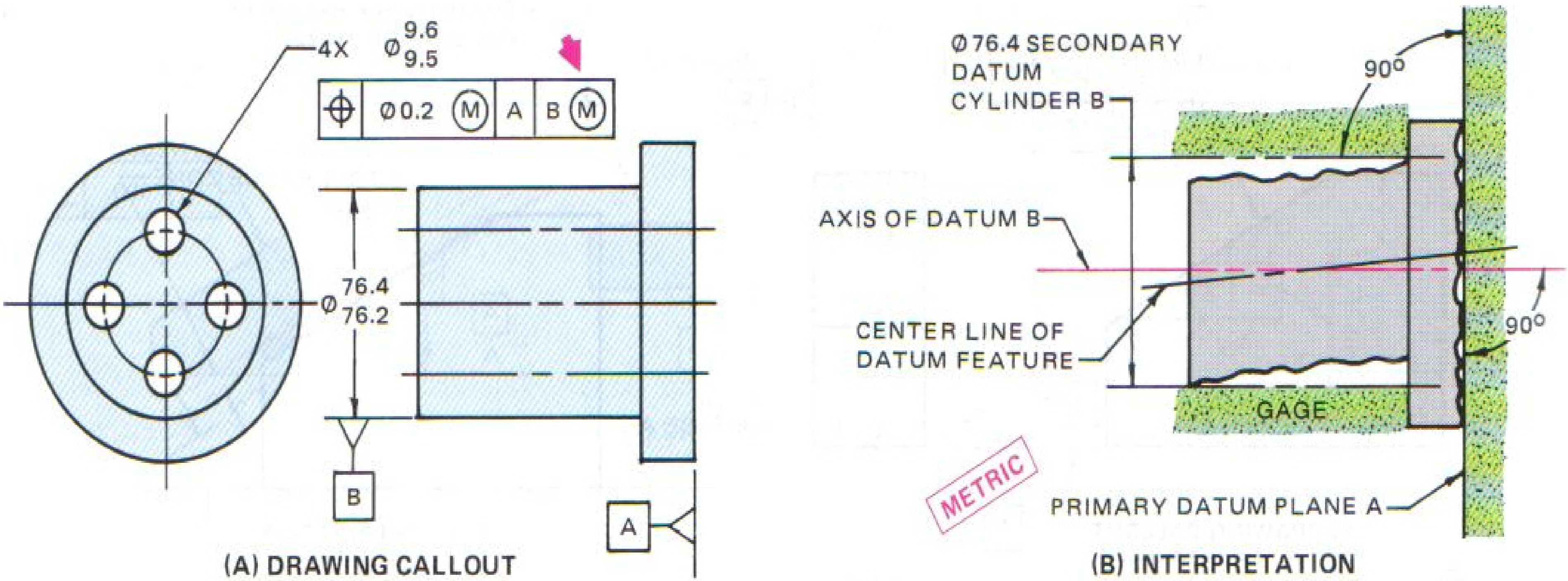

NO. 3
NO. 1

Parallelism to a hidden datum.

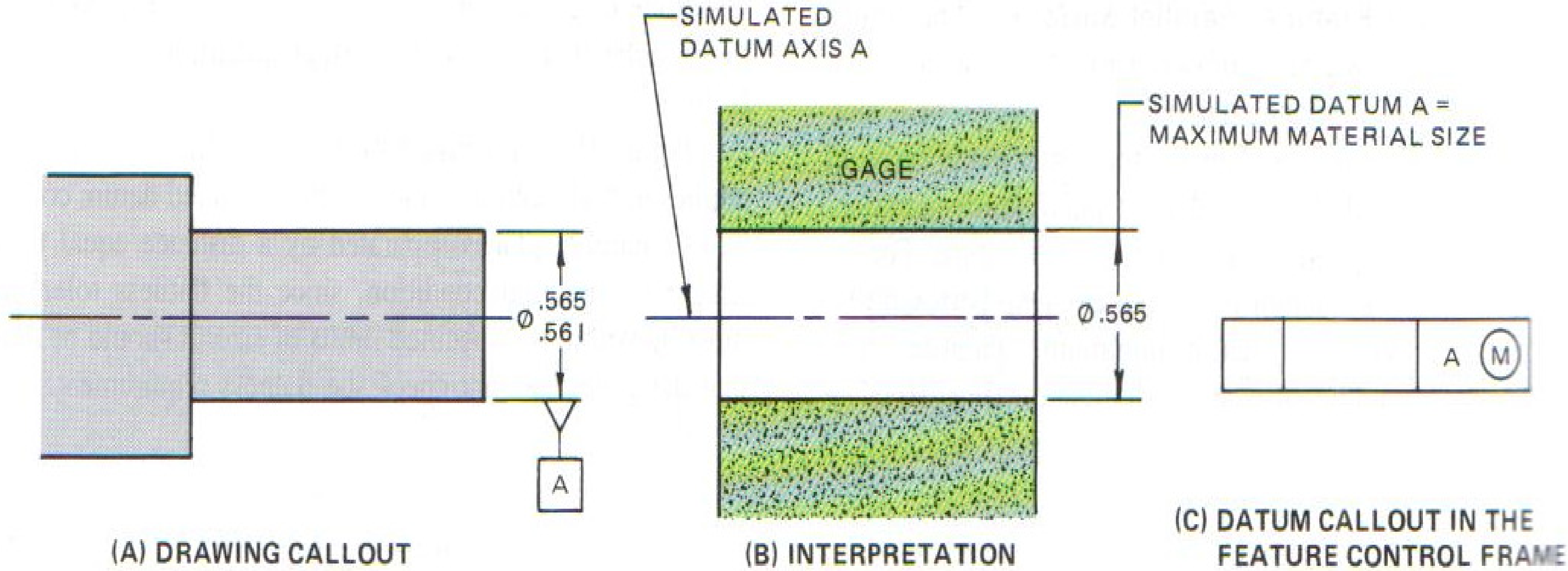



Fixed parallel locates the hidden datum.

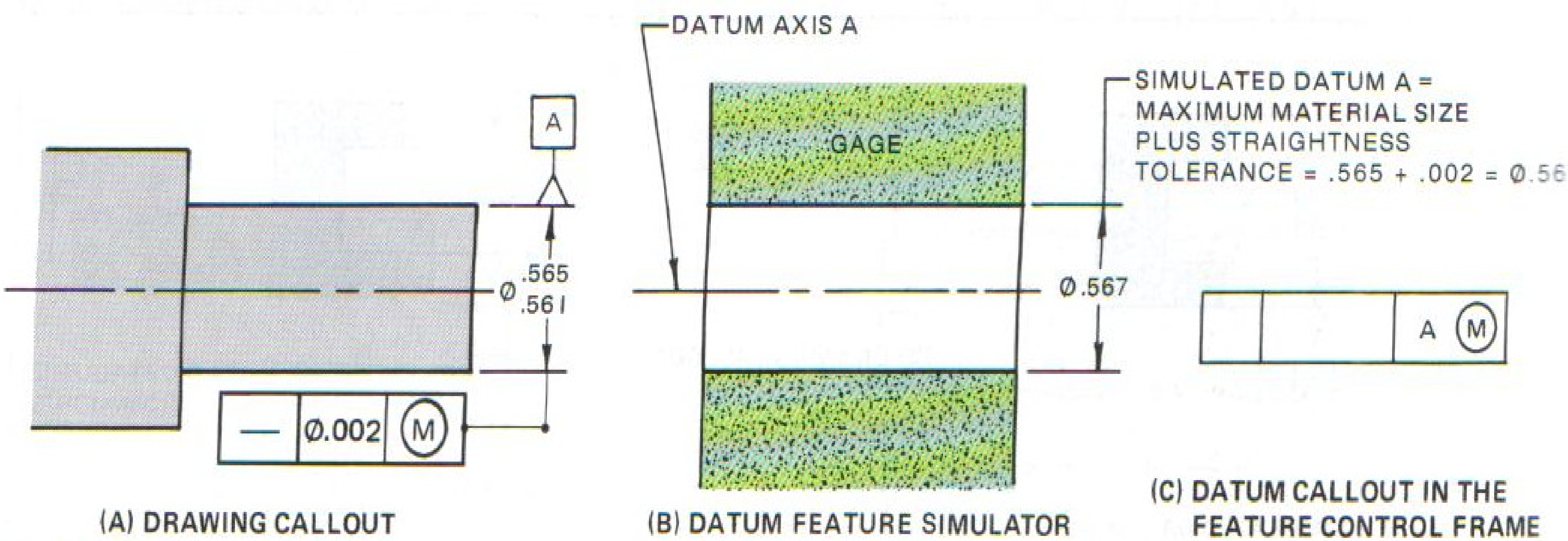





Cylindrical feature as secondary datum.

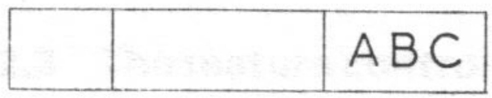



External primary datum without form tolerances—MMC.

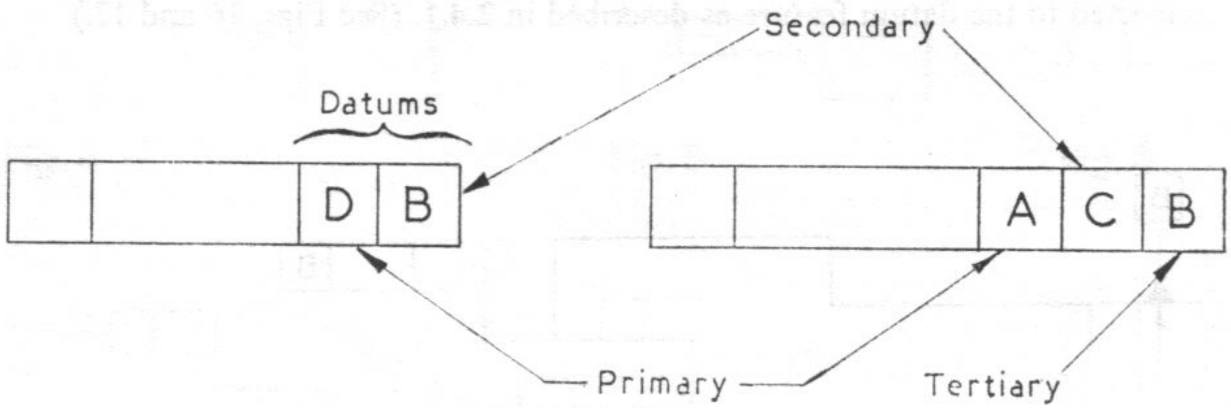



External primary datum with straightness tolerance—MMC.

Where a datum system consists of a number of datum features, i.e. a multiple datum system, the reference letters of the datum features are placed in the third compartment of the tolerance frame



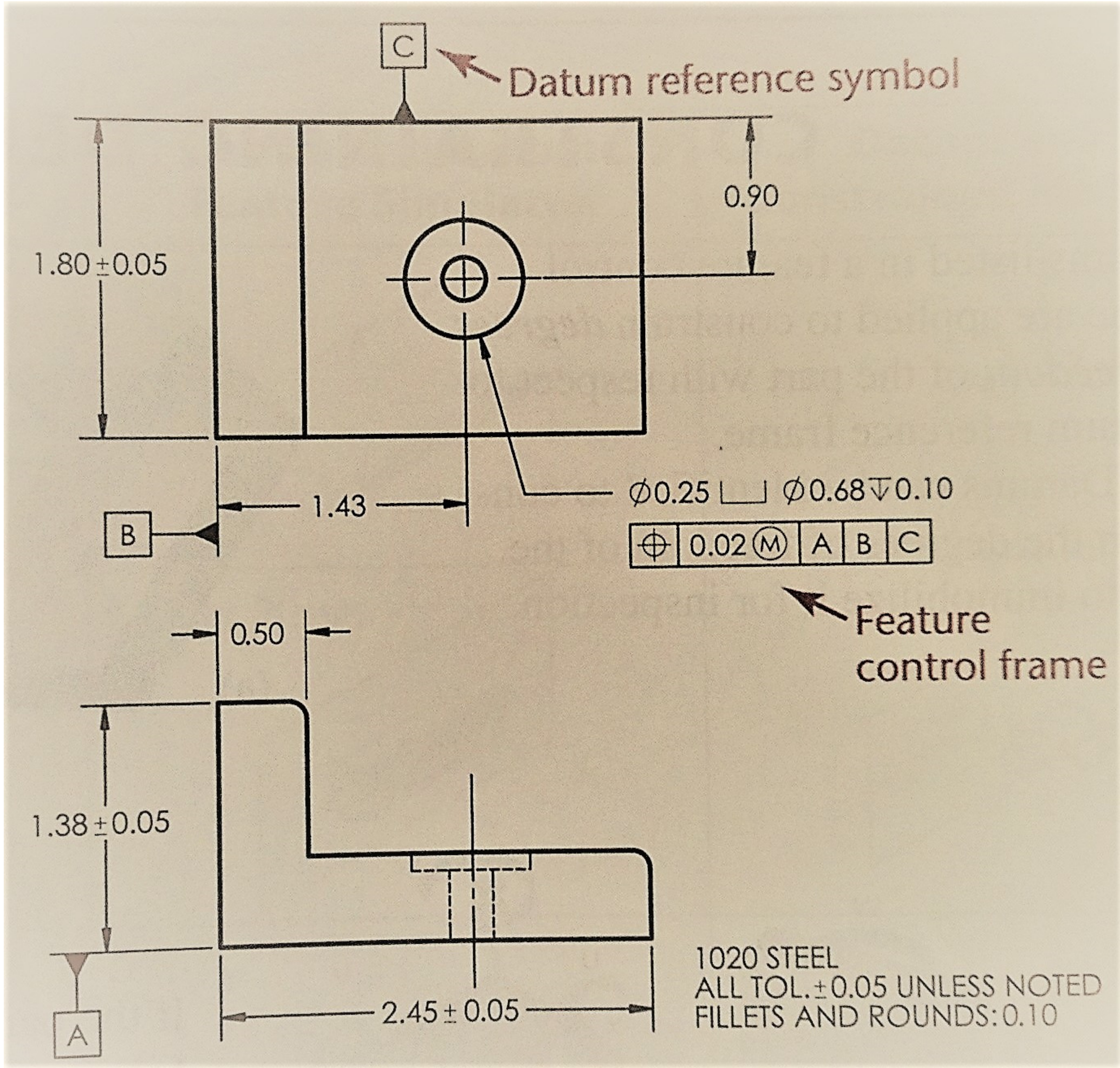

Where the desired relationship between the actual component and the datum system established by the multiple datums has to be obtained by the application of the datum features in a particular order, the designating letters should be quoted, in the needed order from left to right in the third and subsequent compartments of the tolerance frame



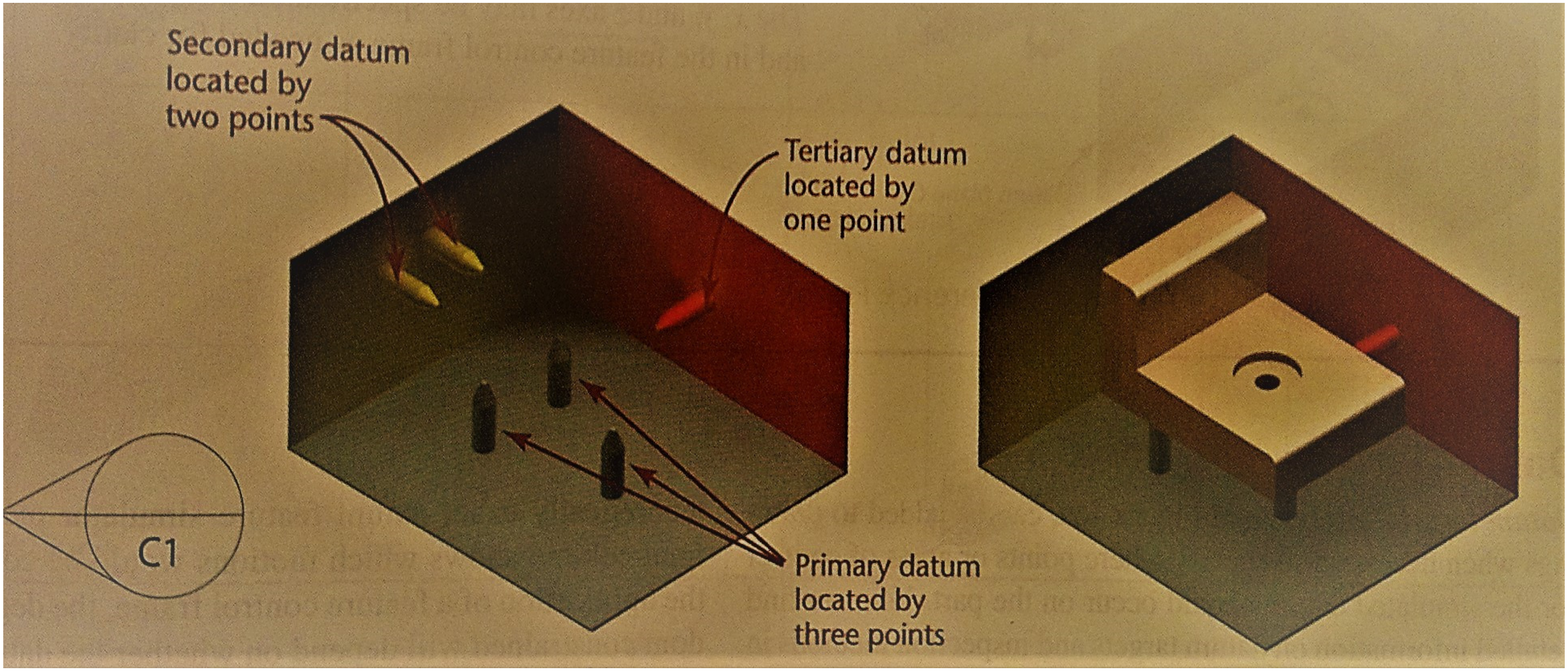
# DATUM TARGETS

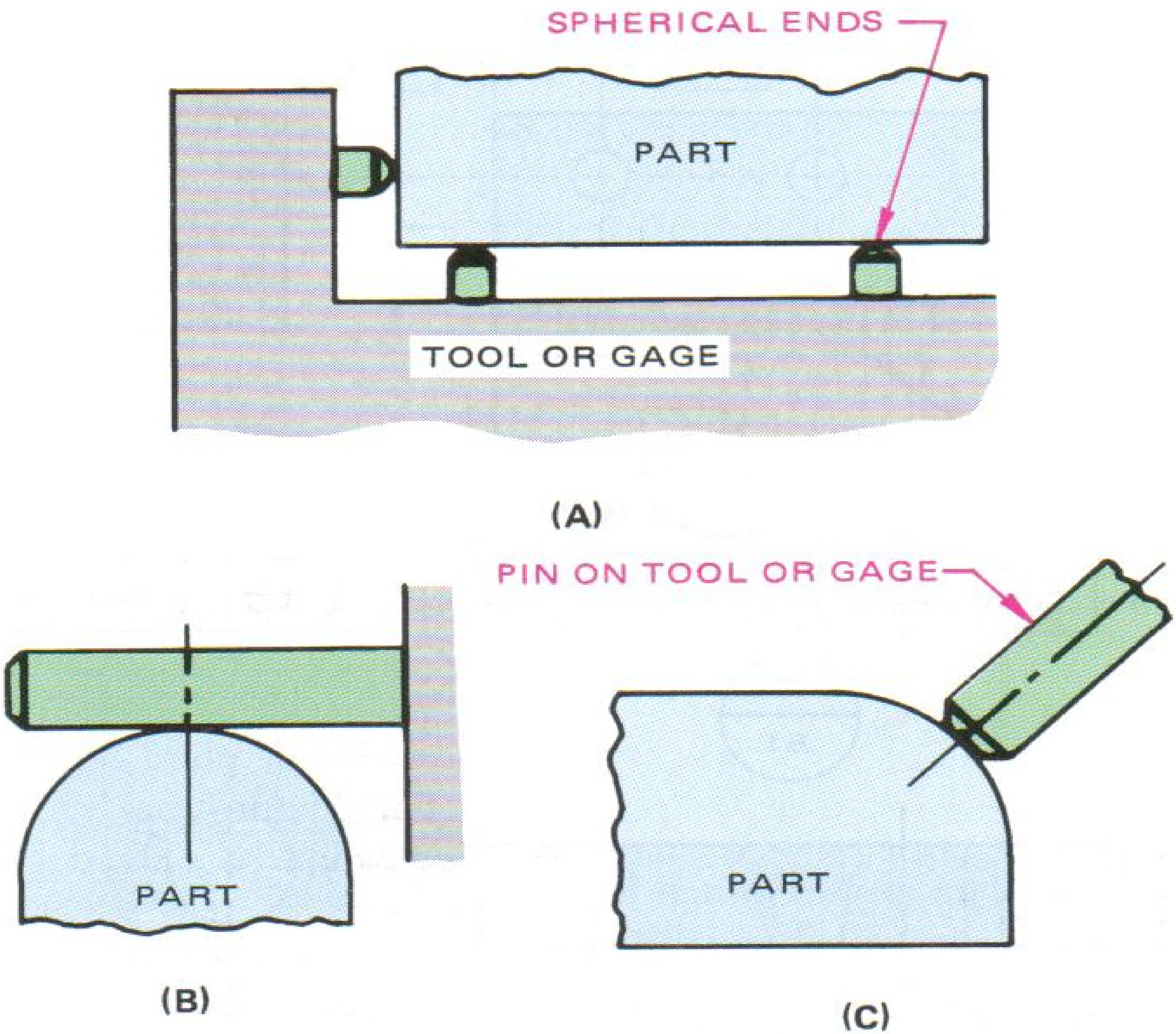
The full feature surface was used to establish a datum for the features so far designated as datum features. This may not always be practical for these three reasons:

1. The surface of a feature may be so large that a gage designed to make contact with the full surface may be too expensive or too cumbersome to use.
2. Functional requirements of the part may necessitate the use of only a portion of a surface as a datum feature, for example, the portion that contacts a mating part in assembly.
3. A surface selected as a datum feature may not be sufficiently true, and a flat datum feature may rock when placed on a datum plane, so that accurate and repeatable measurements from the surface would not be possible. This is particularly so for surfaces of castings, forgings, weldments, and some sheet-metal and formed parts.

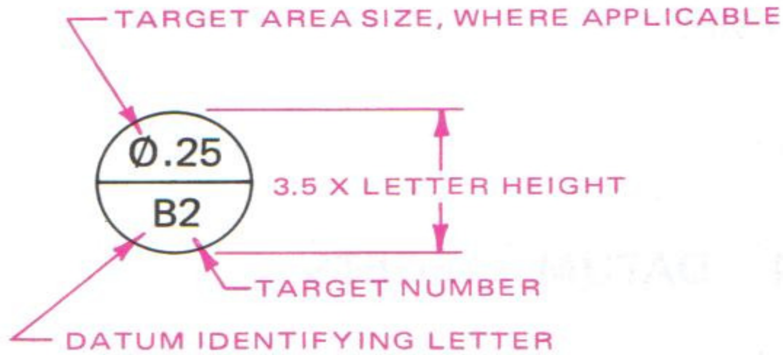


C
Datum reference symbol
0.90
1.80 ± 0.05
1.43
B
⌀0.25 ⌴ ⌀0.68 ↧ 0.10
⌖ 0.02 Ⓜ A B C
Feature control frame
0.50
1.38 ± 0.05
A
2.45 ± 0.05
1020 STEEL
ALL TOL. ± 0.05 UNLESS NOTED
FILLETS AND ROUNDS: 0.10

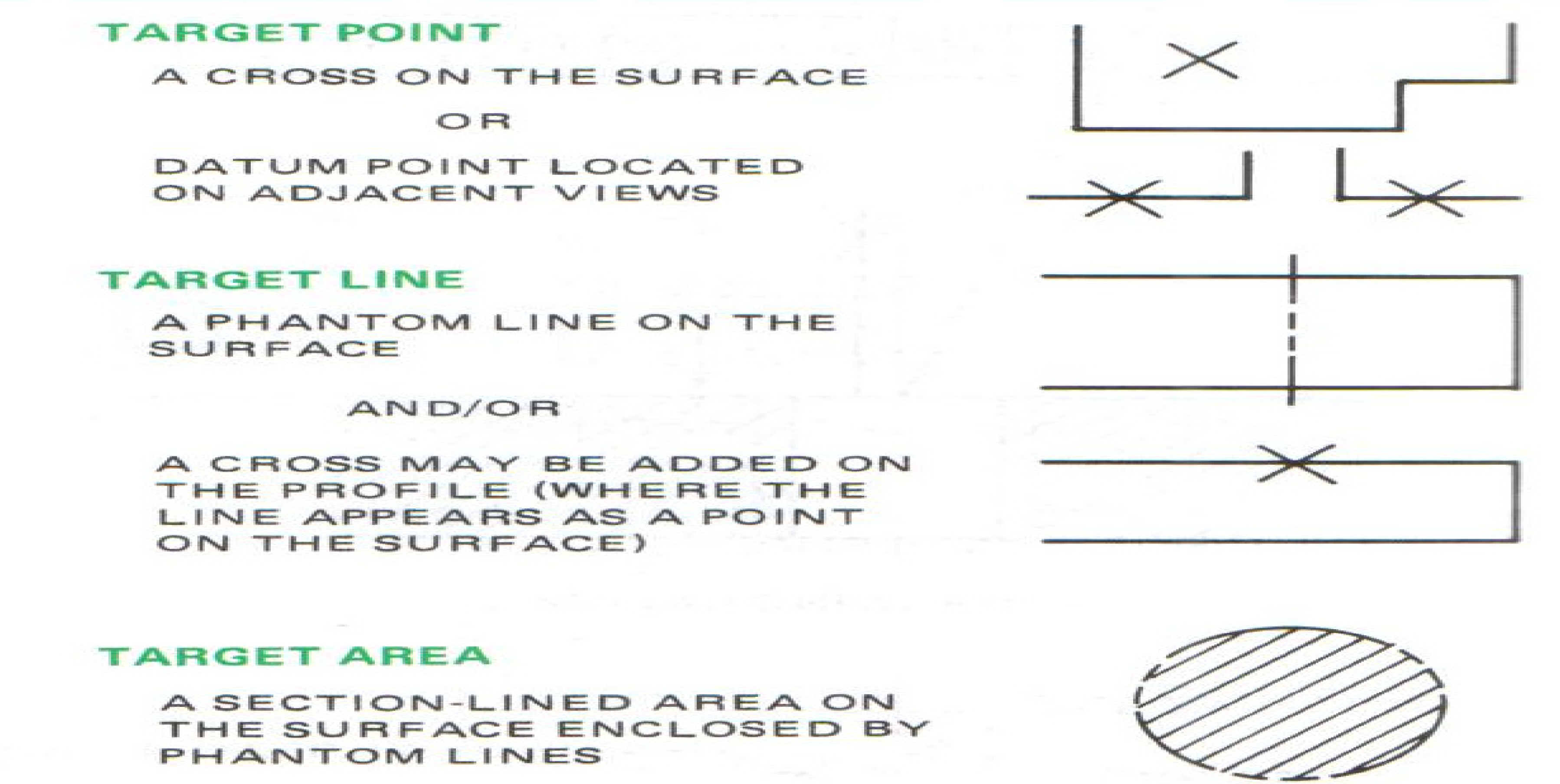

Secondary datum
located by
two points
Tertiary datum
located by
one point
C1
Primary datum
located by
three points



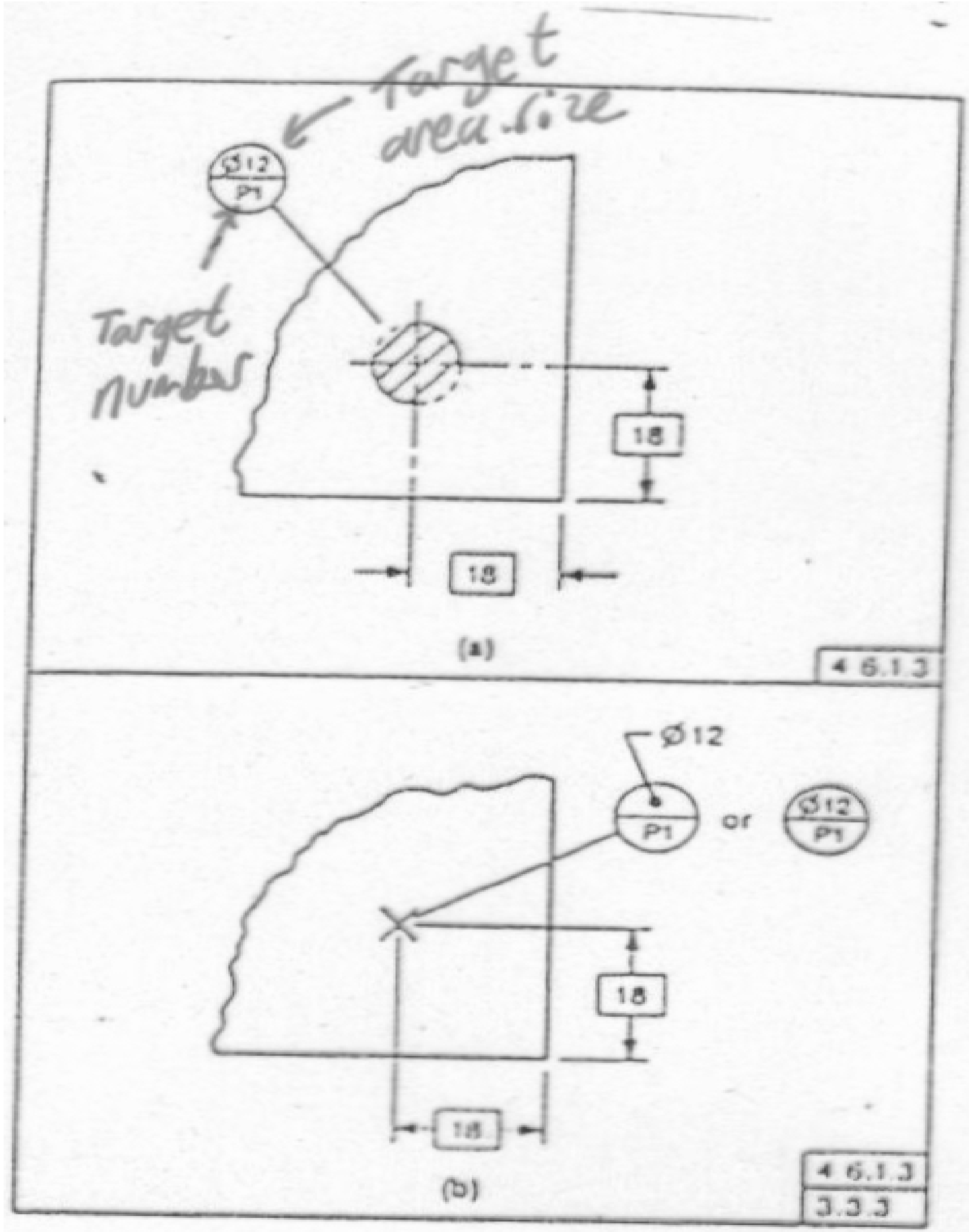

Location of part on datum target points.

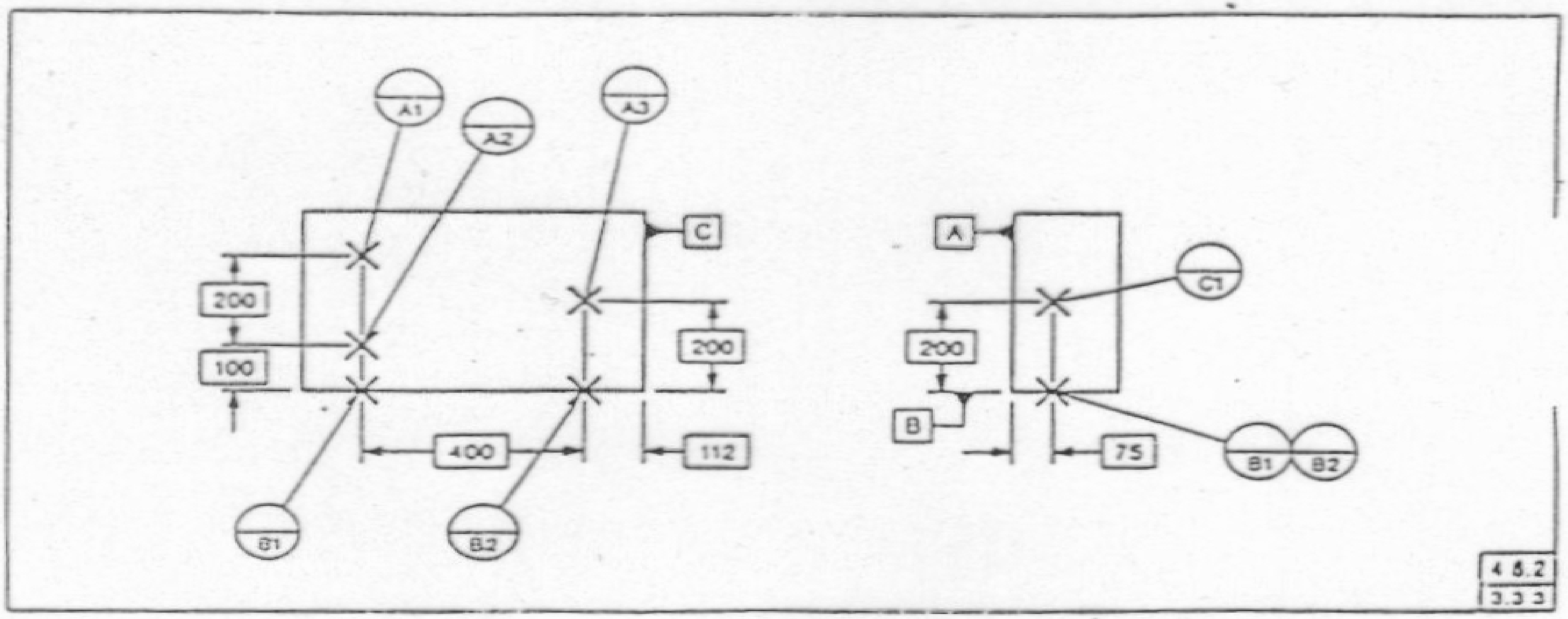



Datum target symbol.

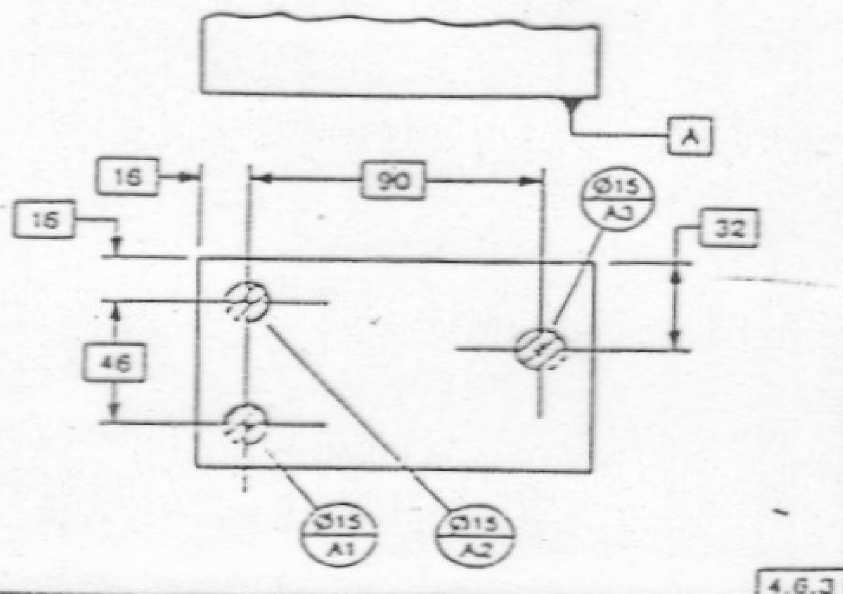



Symbol for a datum target point.

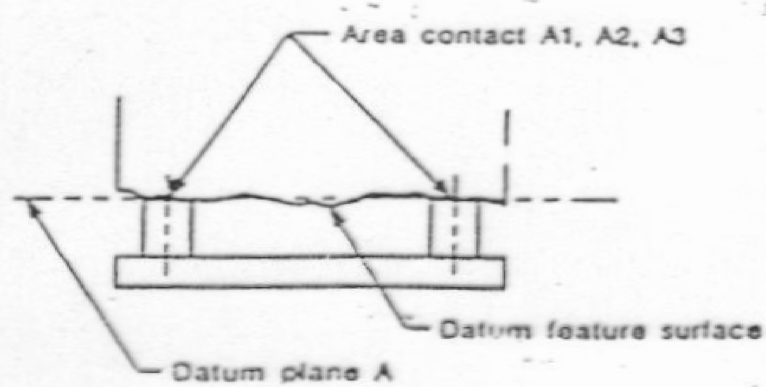

Target area size
Ø12
P1
Target number
18
18
(a)
4.6.1.3
Ø12
P1
or
Ø12
P1
18
18
(b)
4.6.1.3
3.3.3

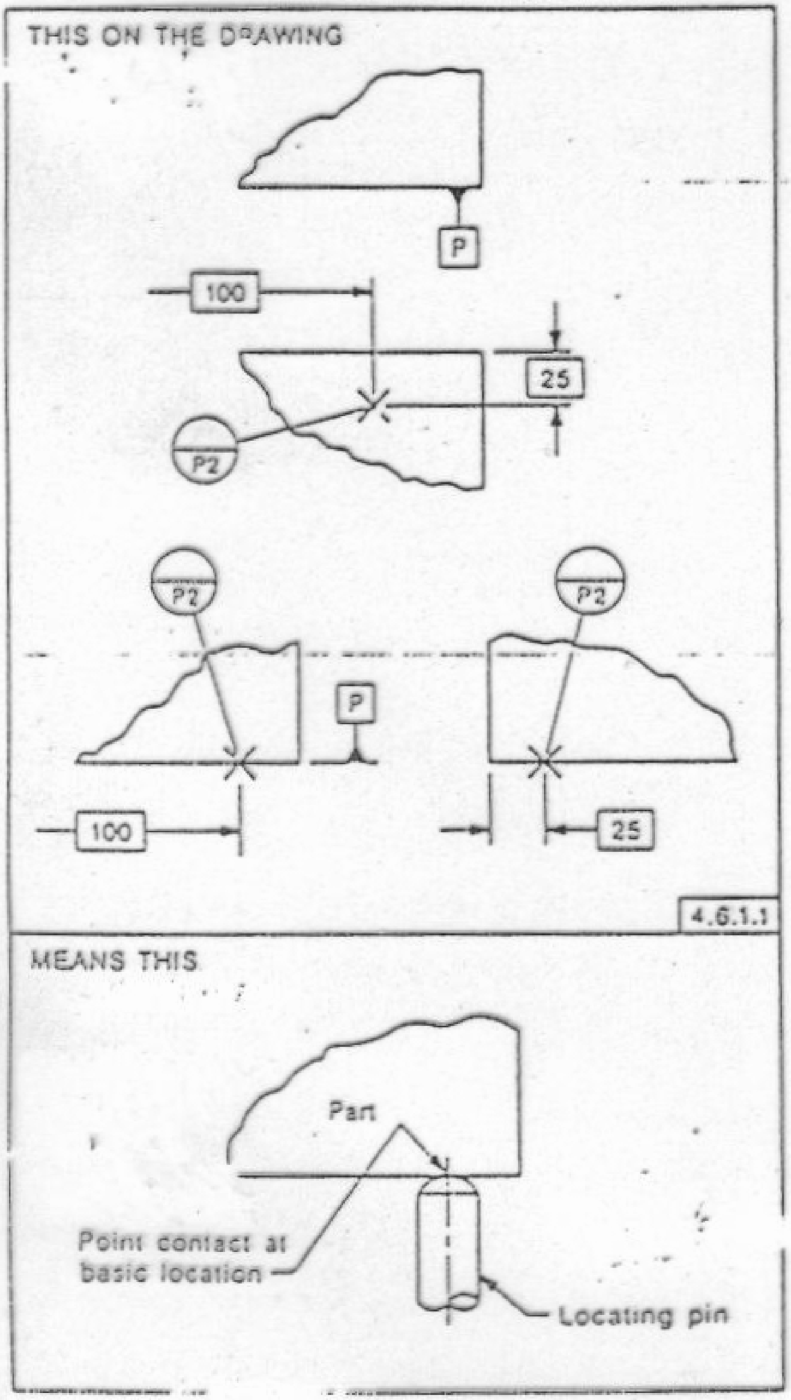


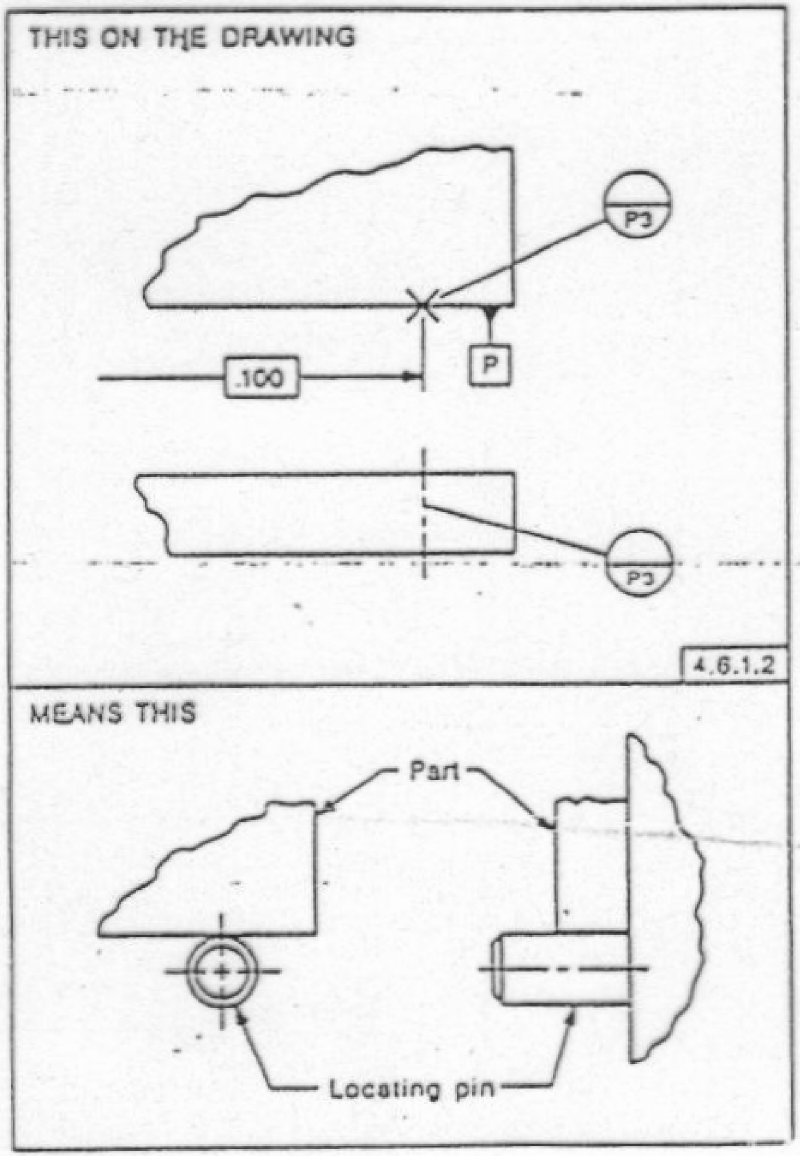

THIS ON THE DRAWING




MEANS THIS




THIS ON THE DRAWING
P
100
25
P2
P2
P
P2
100
25
4.6.1.1
MEANS THIS
Part
Point contact at
basic location
Locating pin


THIS ON THE DRAWING
P3
P
.100
P3
4.6.1.2
MEANS THIS
Part
Locating pin

**Tolerances of straightness.** A straightness tolerance may be used to control:

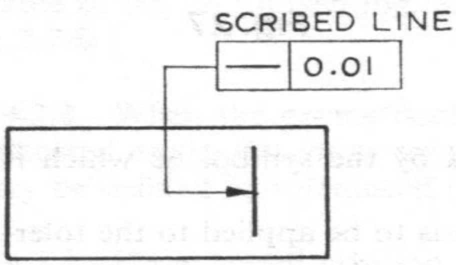
| | |
|---|---|
| (1) the straightness of a line on a surface, or (2) the straightness of a line in a single plane, | in which cases the tolerance zone is the area between two parallel straight lines in the plane containing the considered line or axis, and the tolerance value is the distance between the lines. |
| or (3) the straightness of a line which is the axis of a feature or features which are solids of revolution. | in which case the tolerance zone is a cylinder with a diameter equal to the tolerance value. |

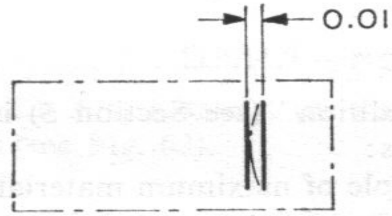
## Tolerances of straightness

| Examples | Interpretations |
|---|---|

**(1) Straightness of a line on a surface**

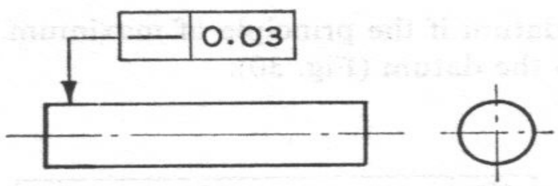
*a.*

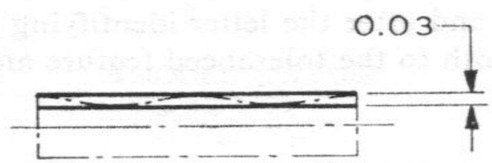



The particular line shown on the surface is required to lie between two parallel straight lines on the surface, 0.01 apart

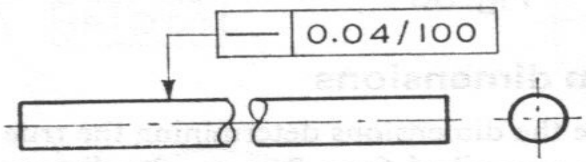
*b.*



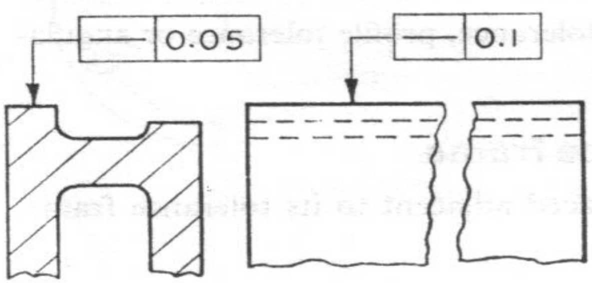

The surface of the feature in any of its positions, is required to lie between two parallel straight lines, 0.03 apart, lying in an axial plane

*c.*

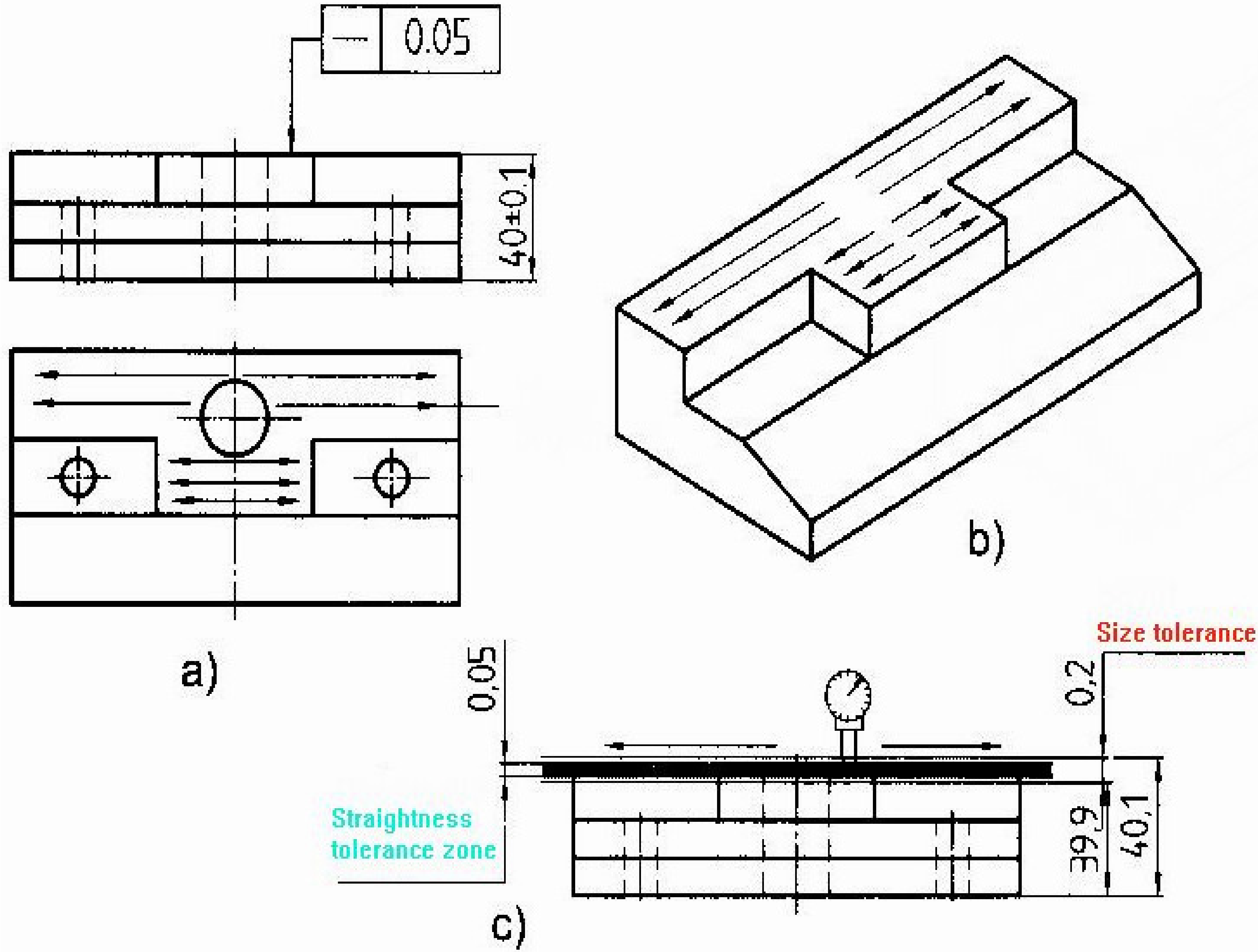



Any portion of length 100 of the generator of the cylinder is required to lie between two parallel straight lines, 0.04 apart, lying in an axial plane. See 2.5.3

*d.* **Straightness of lines in two directions on a surface**

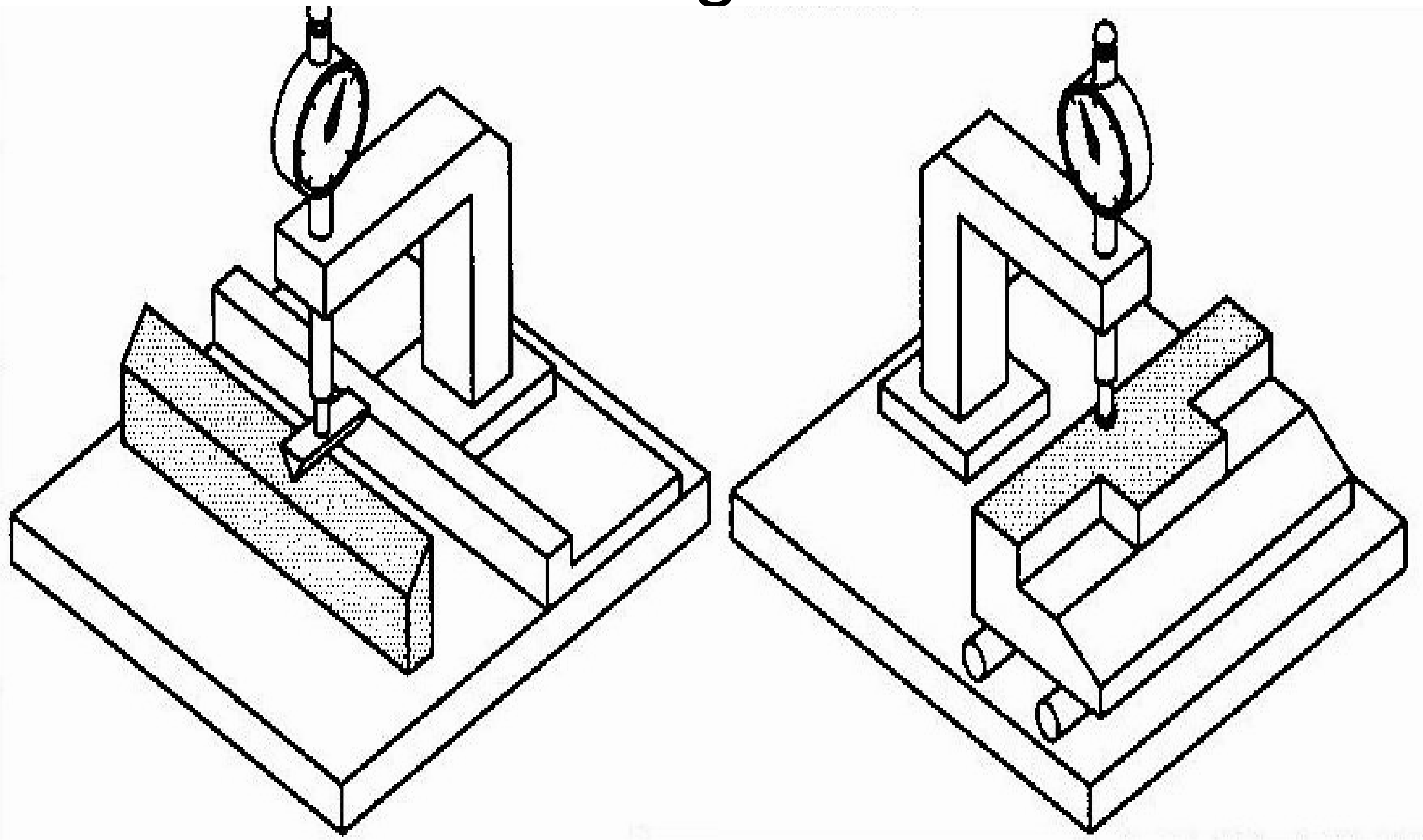



If two different straightness tolerances are applied in two directions on the same surface, the straightness tolerance zone of this surface is 0.05 in that direction shown on the left-hand view and 0.1 in that direction shown on the right-hand view



0.05
40±0.1
a)
b)
0,05
Size tolerance
0,2
Straightness
tolerance zone
39,9
40,1
c)

# Measurement of Straightness

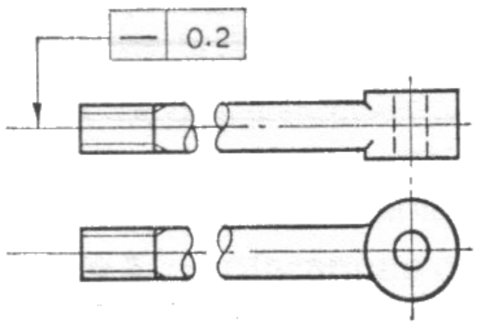


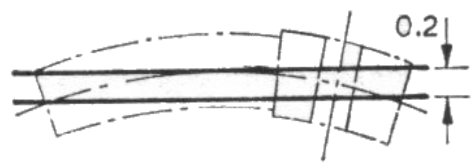
## Tolerances of straightness (*continued*)

| Examples | Interpretations |
| --- | --- |

(2) Straightness of a line in one plane

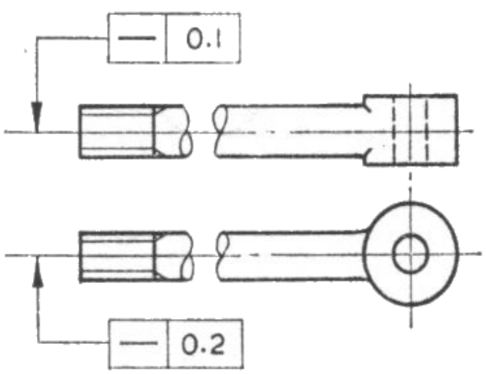
*a.*




In the plane of projection of the upper view, the axis of the whole tie rod is required to lie between two parallel straight lines 0.2 apart in that plane

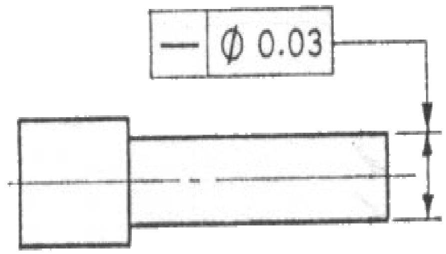
*b.* Straightness of a line in two planes

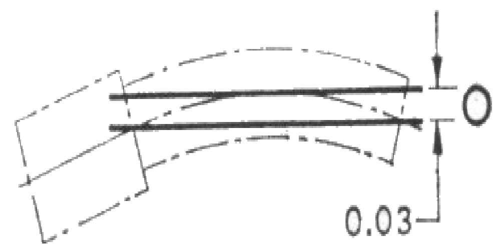



The axis of the bar is required to be contained within a parallelepipedic zone of width 0.1 in the vertical plane and 0.2 in the horizontal plane

(3) Straightness of a line (e.g. the axis of a solid of revolution)

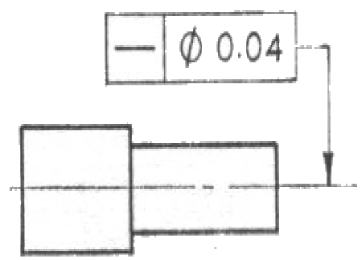
*a.*

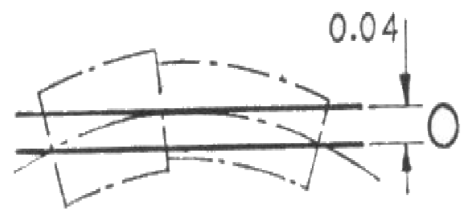


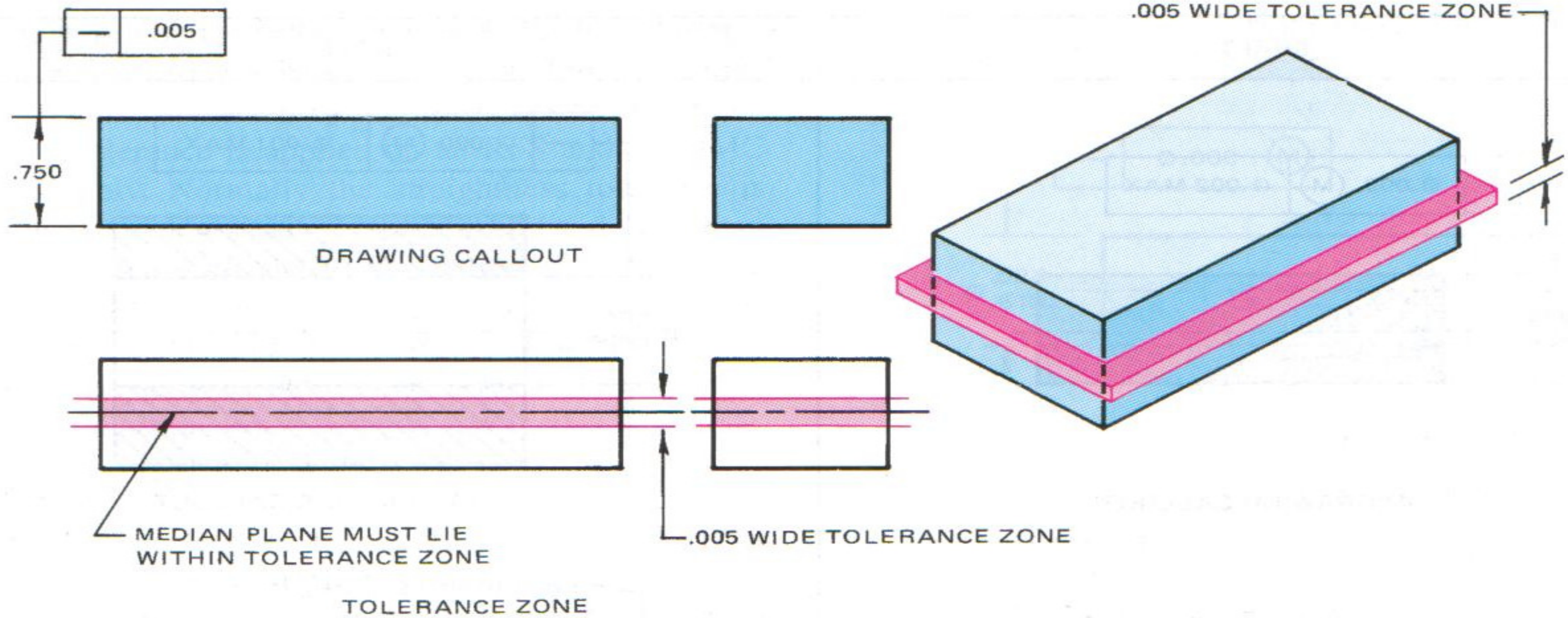

The axis of that part of the piece to be controlled is required to be contained in a cylindrical zone 0.03 diameter

*b.*

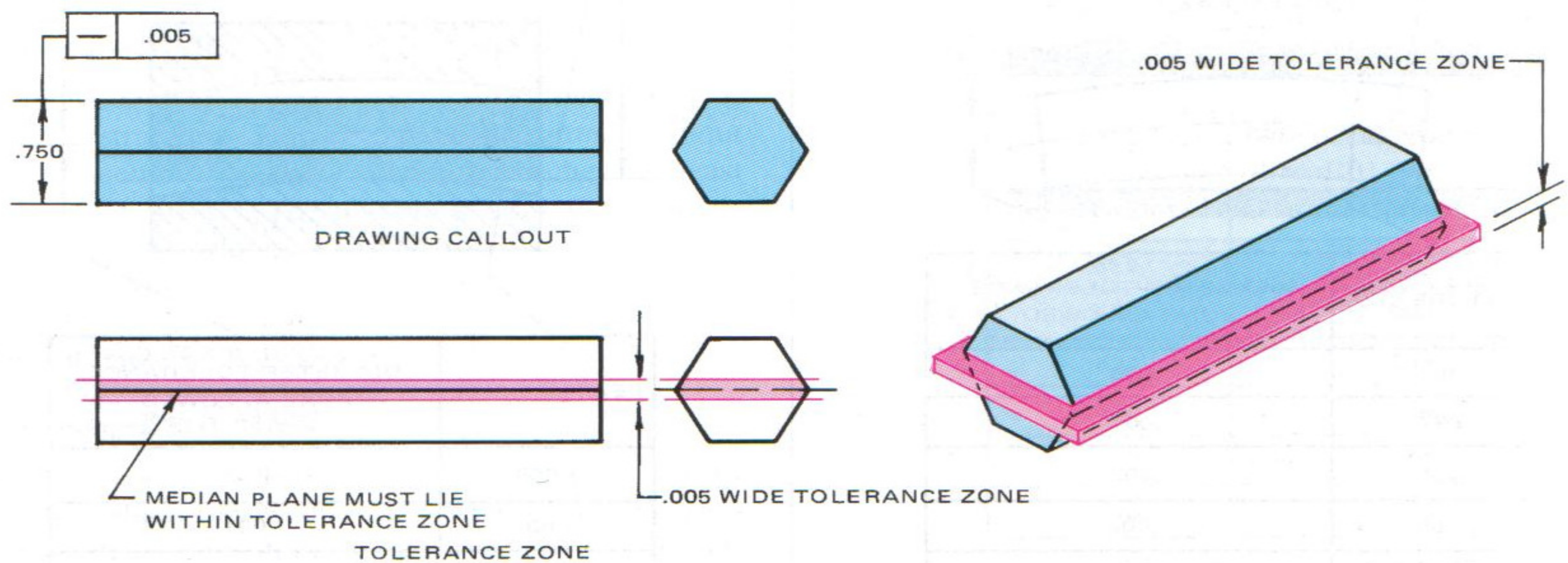



The axis of the whole piece is required to be contained in a cylindrical tolerance zone of 0.04 diameter

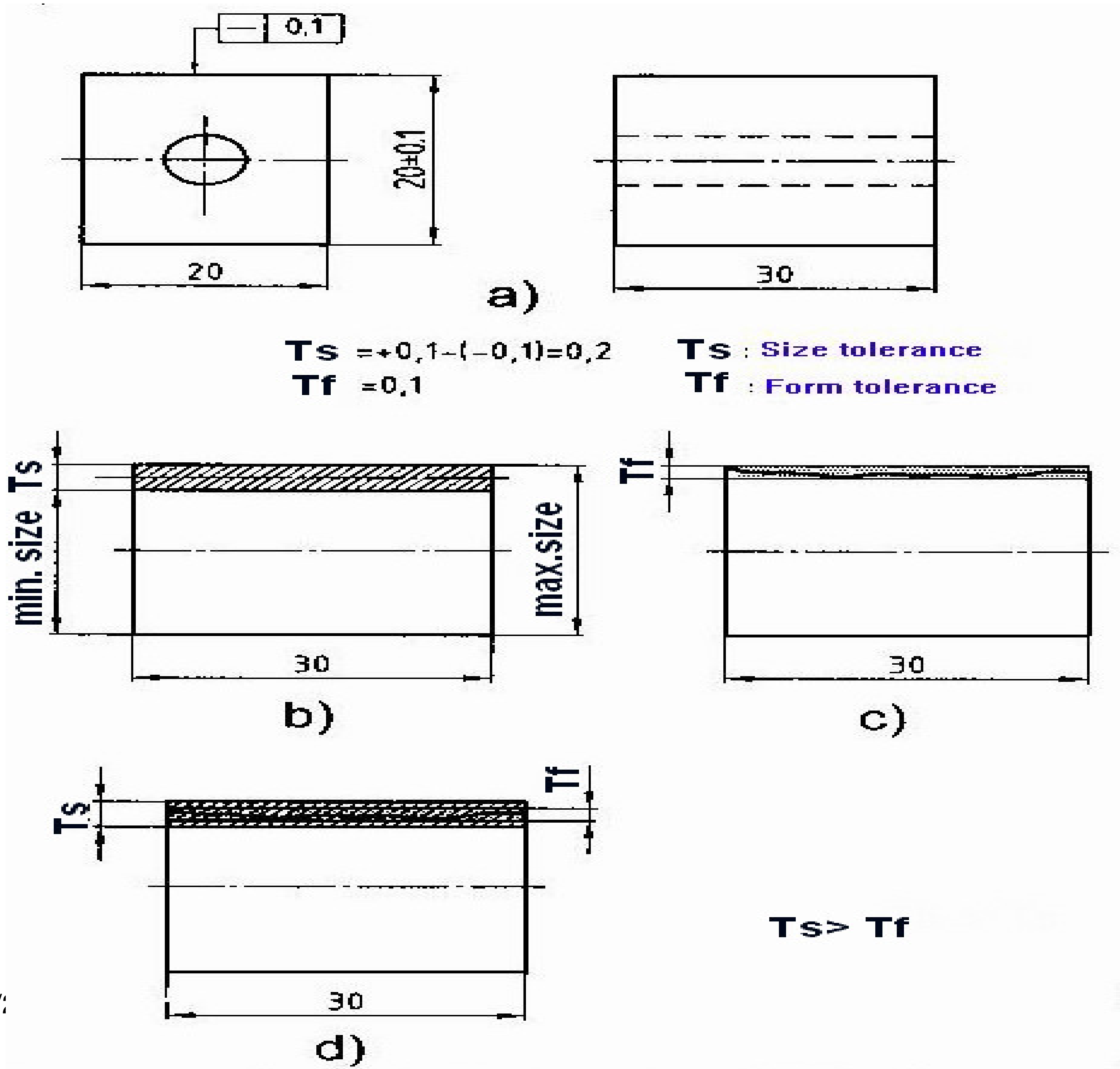



**(A) SQUARE AND RECTANGULAR PARTS**



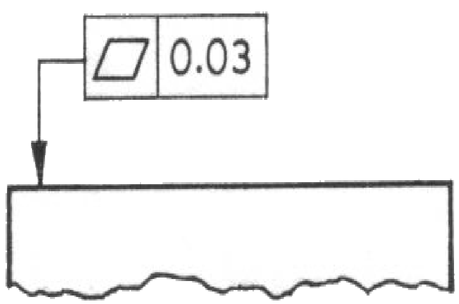
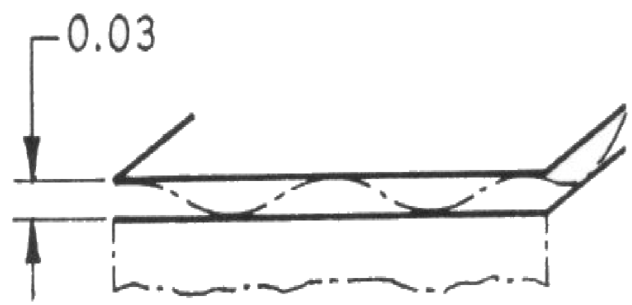

**(B) REGULAR POLYGONS**

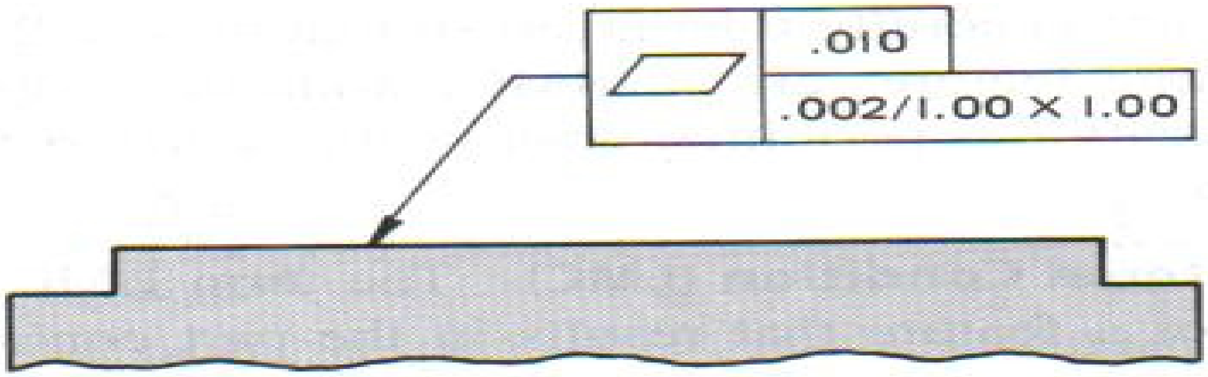
Straightness of a median plane—RFS.


0,1
20±0,1
20
30
a)
Ts =+0,1-(-0,1)=0,2
Tf =0,1
Ts : Size tolerance
Tf : Form tolerance
min. size Ts
max.size
30
b)
Tf
30
c)
Ts
Tf
30
d)
Ts> Tf

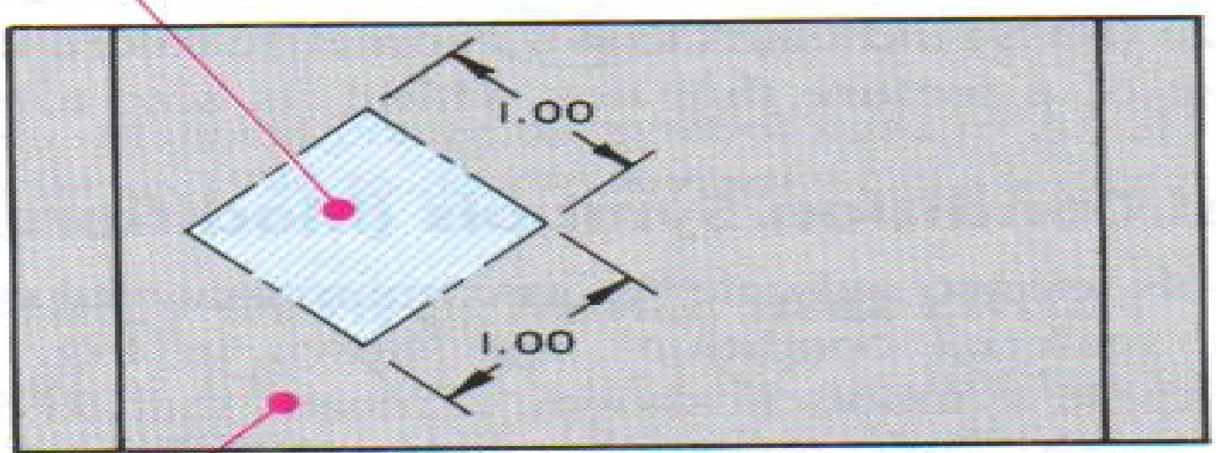
**Tolerances of flatness.** When a flatness tolerance is used to control the flatness of a surface the tolerance zone is the space between two parallel planes and the tolerance value is the distance between the planes.

## Tolerances of flatness

| Examples | Interpretations |
|---|---|
| *a.*   | The indicated surface is required to lie between two parallel planes 0.03 apart   |



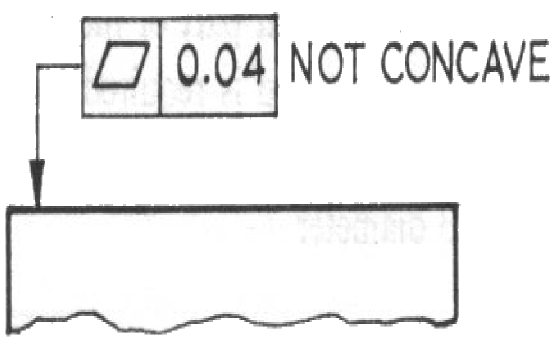

(A) DRAWING CALLOUT



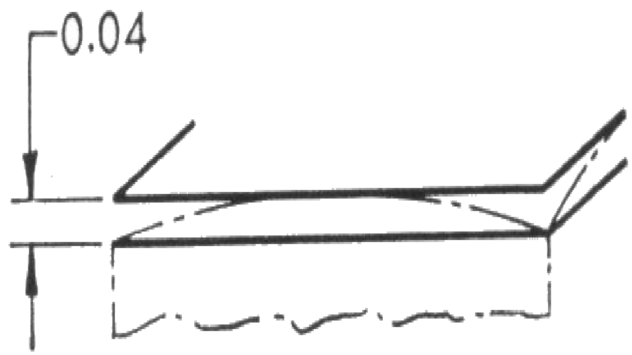

(B) INTERPRETATION

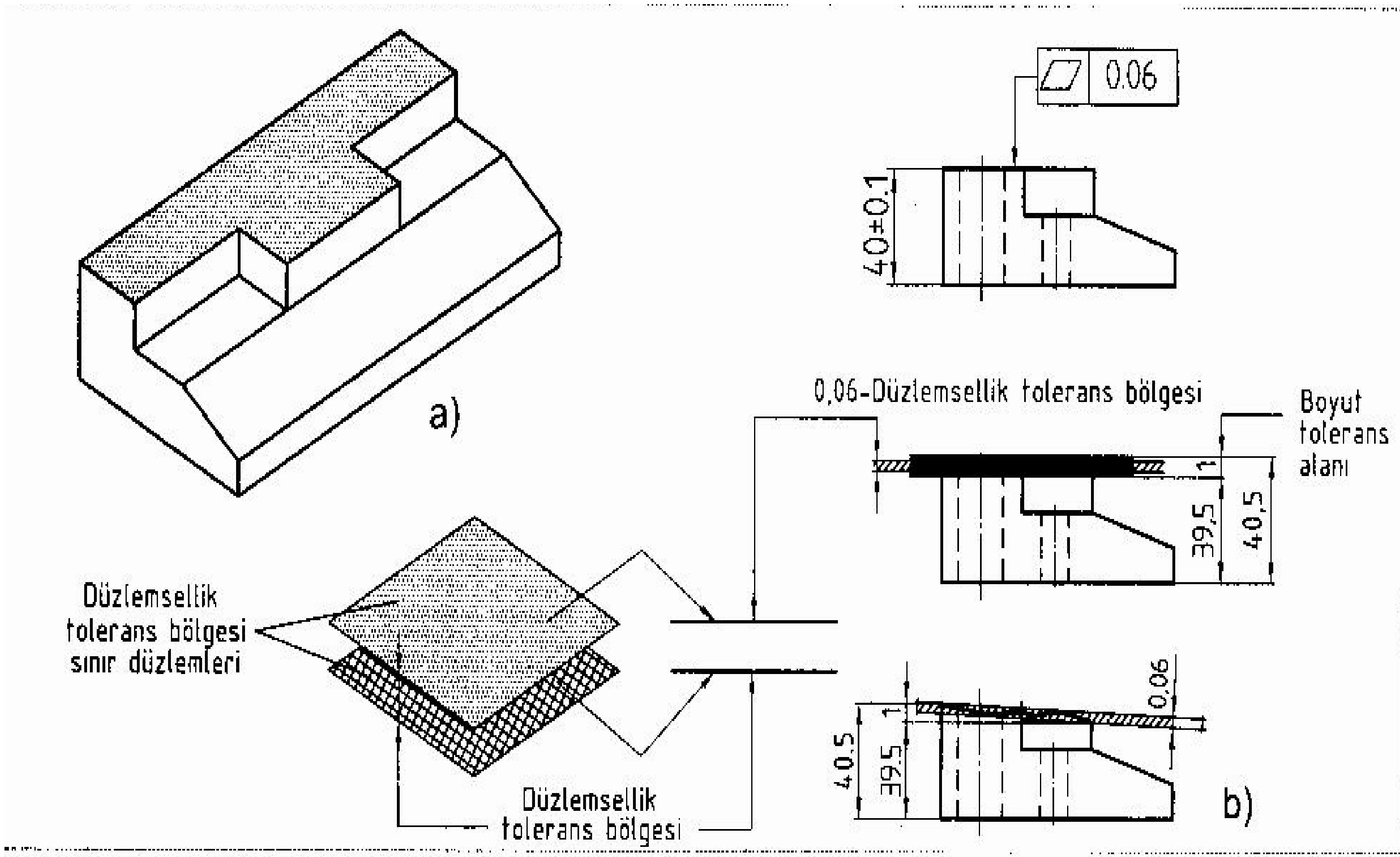
Overall flatness tolerance combined with a flatness tolerance of a unit area.

*b.*




The indicated surface is required to lie between two parallel planes 0.04 apart and is not to be concave





0.06
40±0,1
a)
0,06-Düzlemsellik tolerans bölgesi
Boyut tolerans alanı
1
39,5
40,5
Düzlemsellik tolerans bölgesi sınır düzlemleri
Düzlemsellik tolerans bölgesi
0,06
1
40,5
39,5
b)

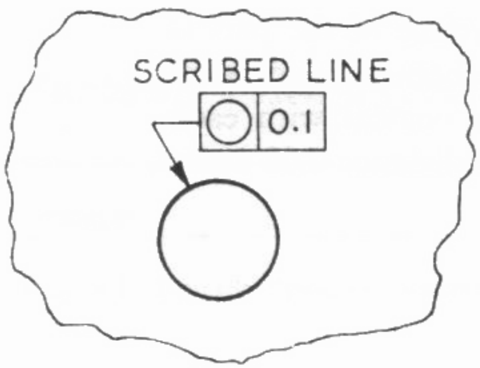
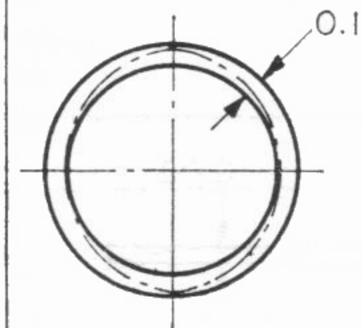
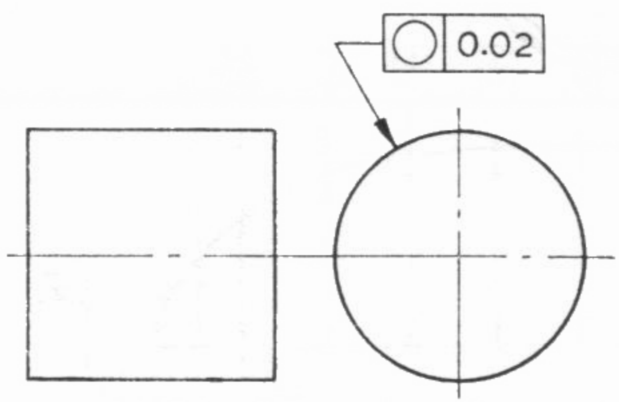
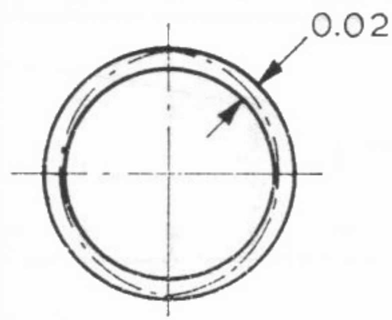
## Tolerances of roundness

A roundness tolerance may be used to control the errors of form of a circle in the plane in which it lies. In the case of a solid of revolution the tolerance controls the roundness of the circle formed by the intersection of the surface with a plane; for a cylinder or cone the plane is perpendicular to the axis and for a sphere it normally, or unless otherwise specified, intersects the sphere in a section of maximum diameter.

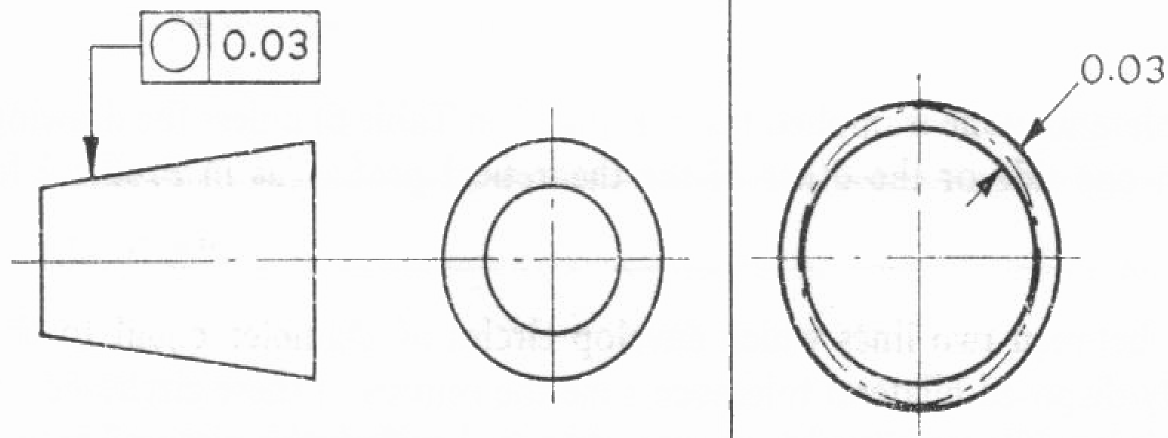
A roundness tolerance is not concerned with the position of the circle, e.g. its concentricity with a datum axis. In the case of a solid of revolution, the roundness of each cross section is an individual assessment.

A roundness tolerance zone is the annular space between two co-planar circles concentric with each other, having a radial separation equal to the specified tolerance value.

# Tolerances of roundness

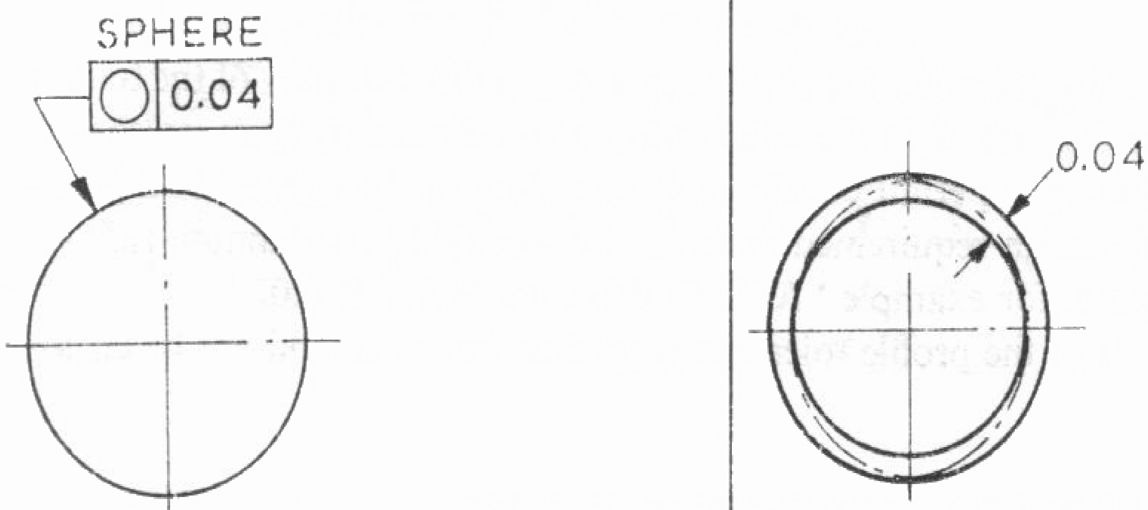
| Examples | Interpretations | |
|---|---|---|
| *a.*  |  | The considered line on the surface is required to lie between two circles concentric with each other, a radial distance 0.1 apart, on the surface |
| *b.*  |  | The periphery at any cross section perpendicular to the axis is required to lie between two circles concentric with each other, a radial distance 0.02 apart, in the plane of the section |

*c.*

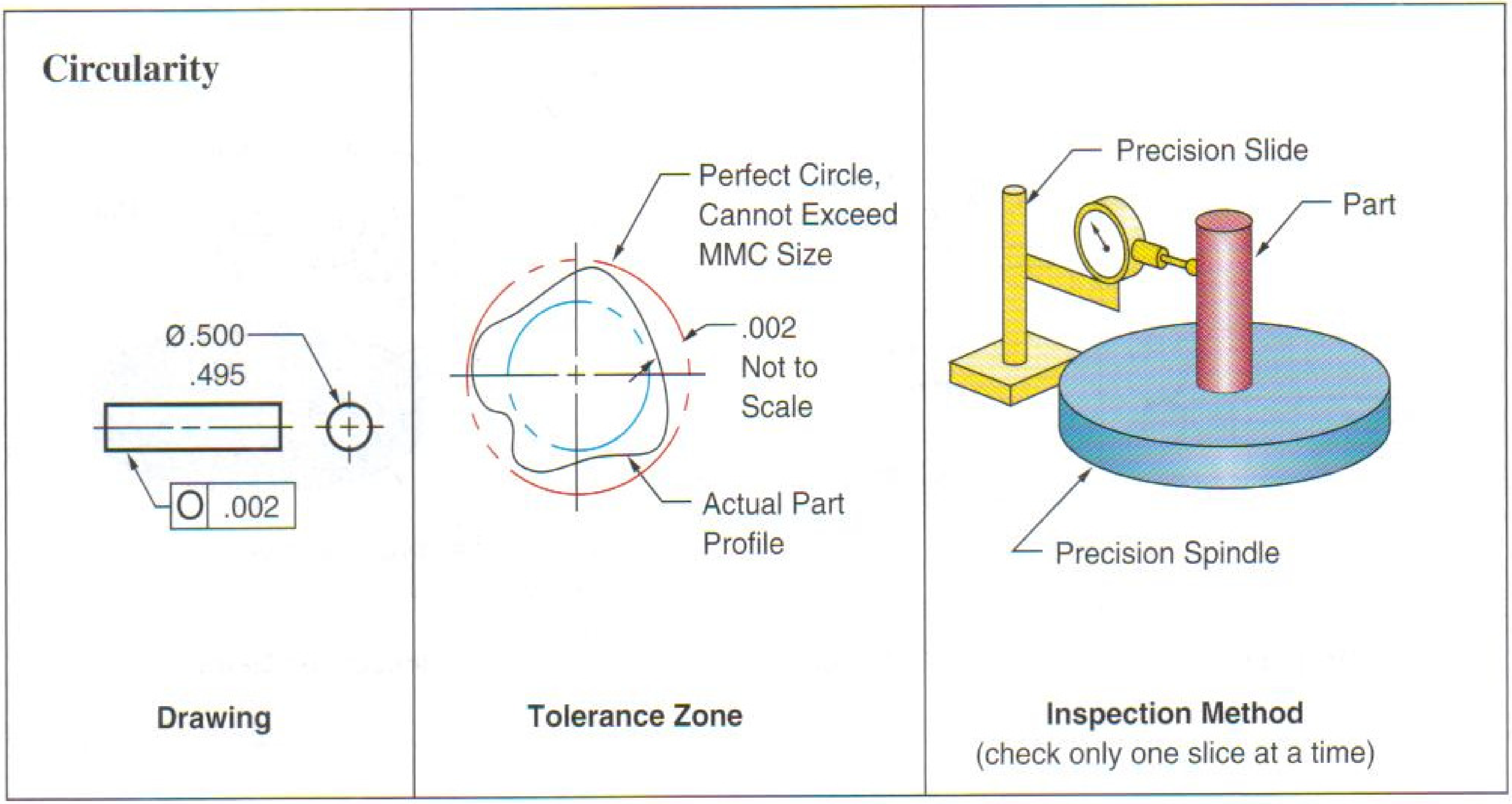



The periphery at any cross section perpendicular to the axis is required to lie between two circles concentric with each other, a radial distance 0.03 apart, in the plane of the section

*d.*




The periphery of any section of maximum diameter is required to lie between two circles concentric with each other, a radial distance 0.04 apart, in the plane of the section

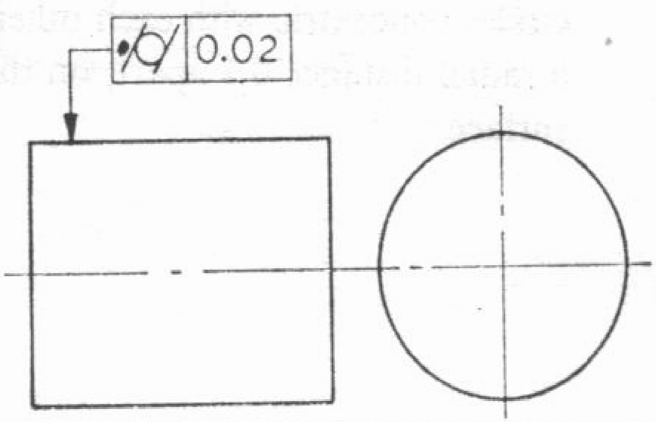
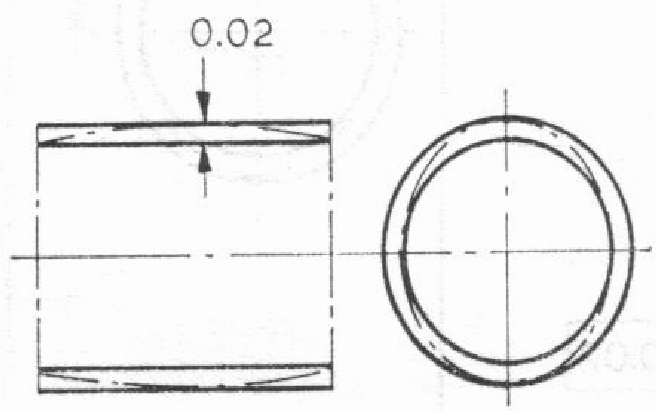

Circularity
Ø.500
.495
O .002
Drawing
Perfect Circle,
Cannot Exceed
MMC Size
.002
Not to
Scale
Actual Part
Profile
Tolerance Zone
Precision Slide
Part
Precision Spindle
Inspection Method
(check only one slice at a time)

## Tolerances of cylindricity

Cylindricity is a combination of roundness, straightness and parallelism applied to the surface of a cylinder. The plane (end) surfaces of a cylindrical part are not controlled by a cylindricity tolerance.

NOTE. Although the control of roundness, straightness and parallelism by means of a cylindricity tolerance may appear to be a convenient technique, the checking of cylindricity in accordance with its definition may present considerable difficulties. It is recommended that the individual characteristics comprising cylindricity be toleranced separately as appropriate to the part concerned.

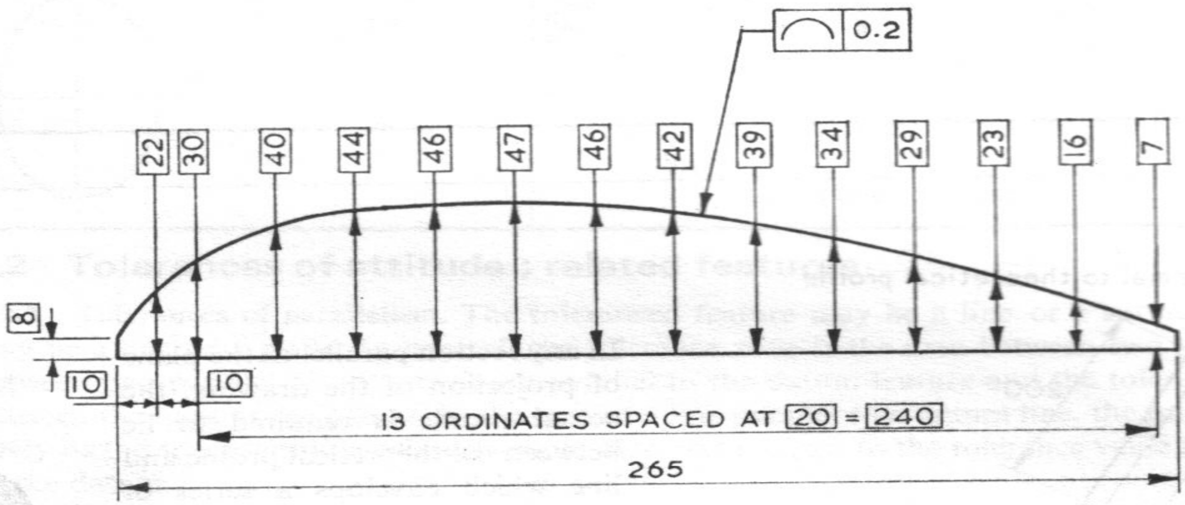
A cylindricity tolerance zone is the annular space between two cylinders coaxial with each other, having a radial separation equal to the specified tolerance value.

| Example | Interpretation |
| --- | --- |
|  | The curved surface of the part is required to lie between two cylindrical surfaces coaxial with each other, a radial distance 0.02 apart  |

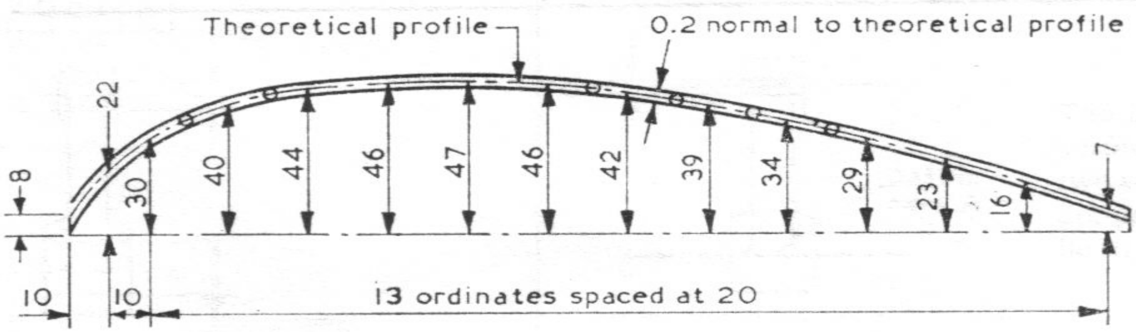
# Profile tolerance of a line

**Example**

*Profile* 1. Equally-disposed bilateral tolerance




**Interpretation**

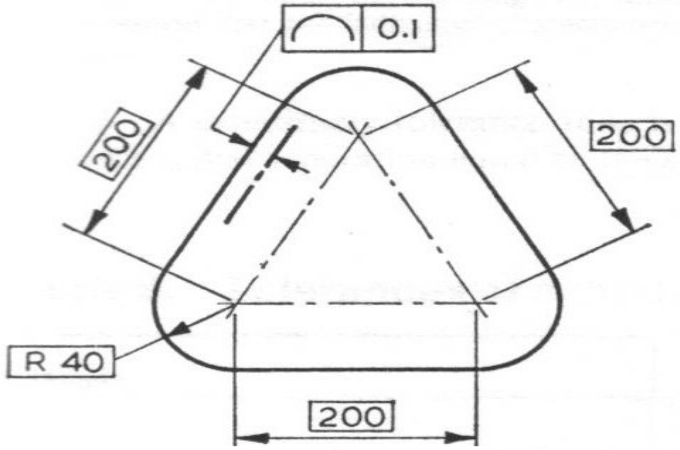



In any section parallel to the plane of projection of the drawing, the actual profile is required to lie between two lines which envelop a series of circles of diameter 0.2 with their centres on the theoretical profile

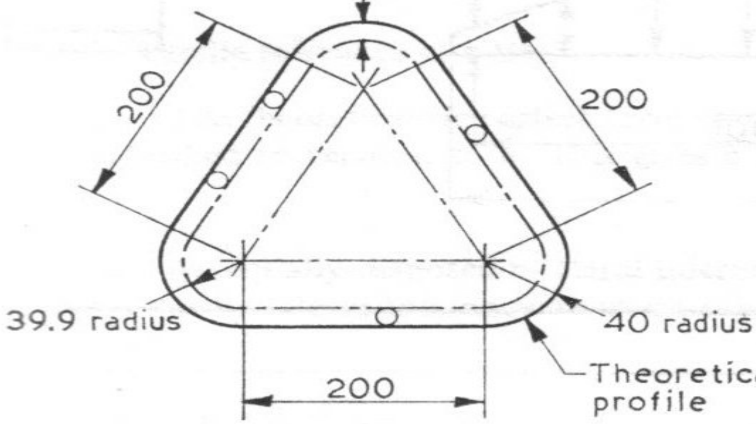
## Profile tolerance of a line

**Example**

*Profile* 2. Unilateral tolerance

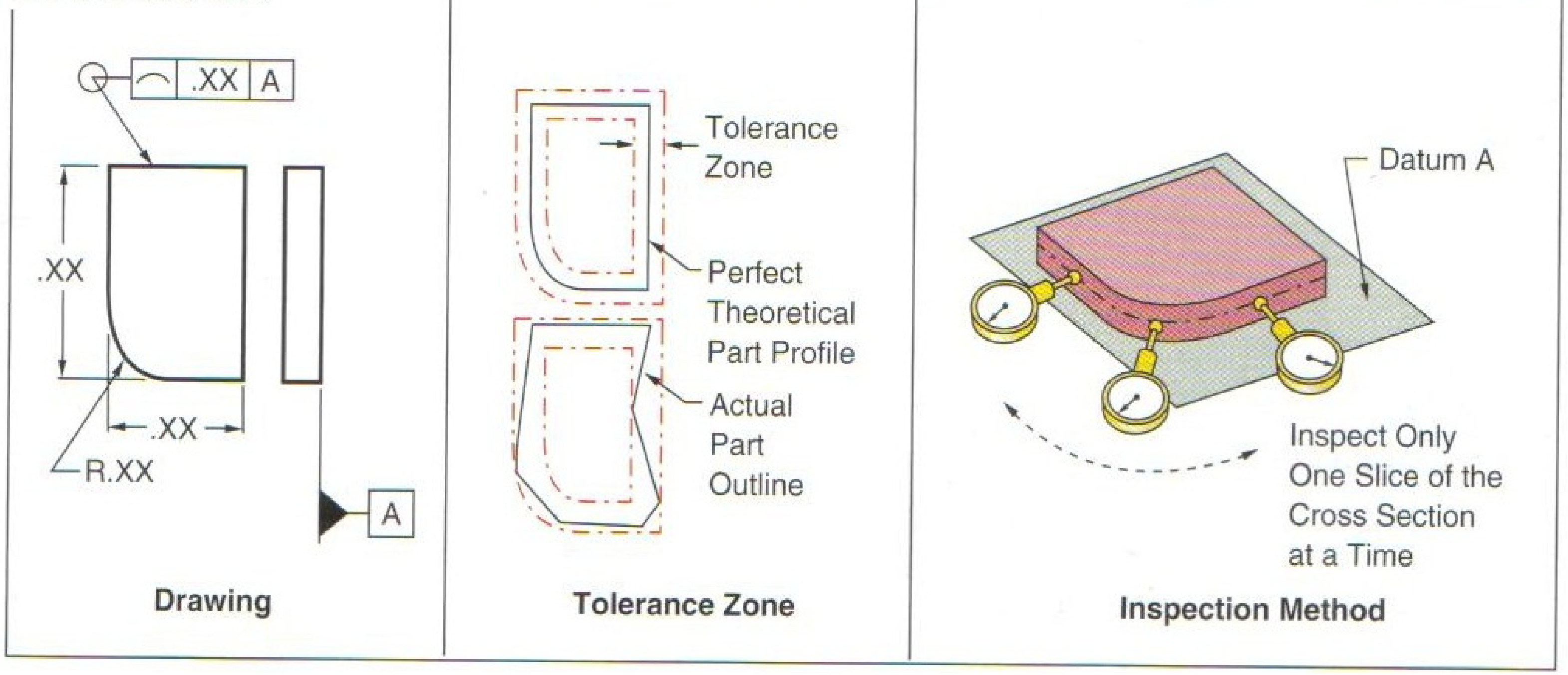



**Interpretation**

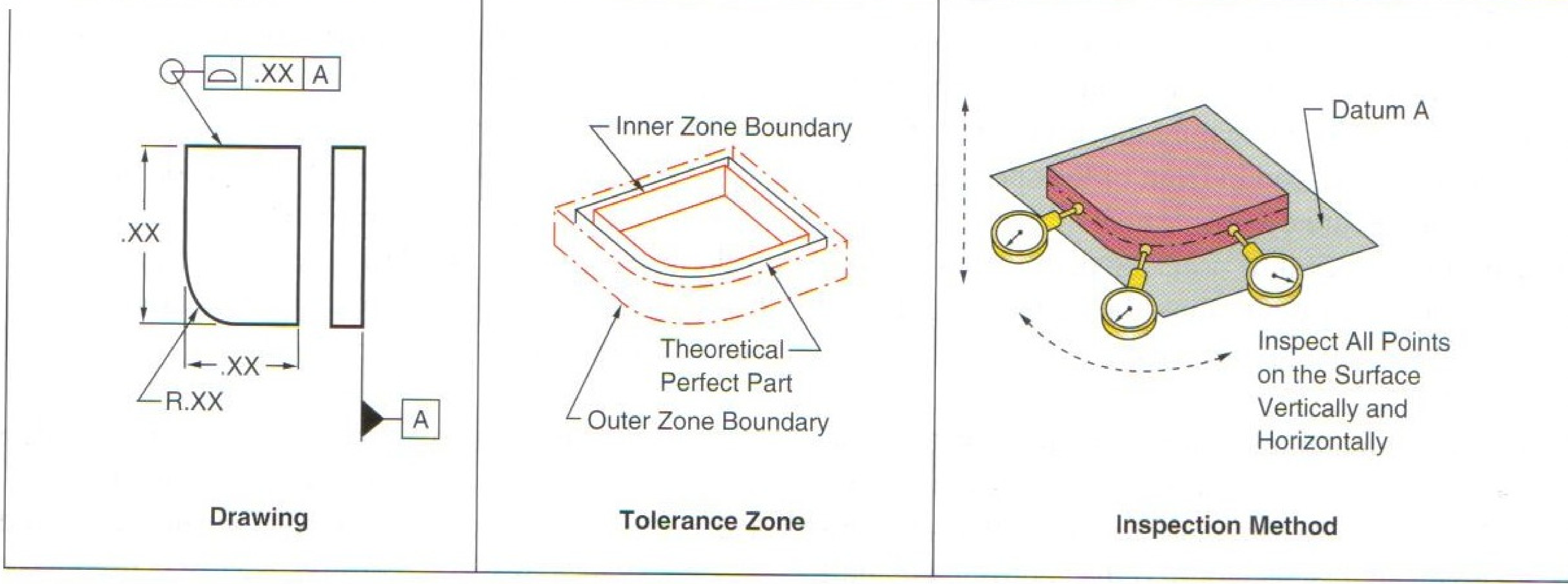



In any section parallel to the plane of projection of the drawing, the actual profile is required to lie between the theoretical profile and line which envelops a series of circles 0.1 diameter, touching and inside the theoretical profile

## Profile of a line



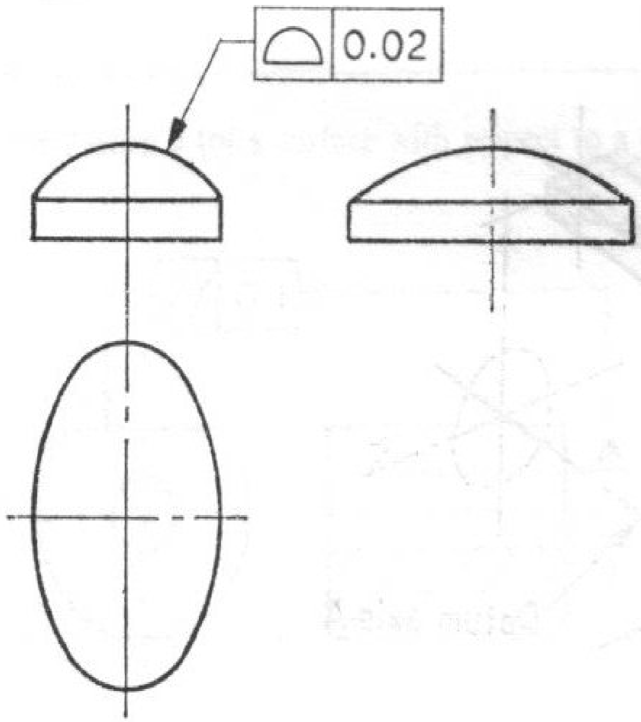

## Profile of a surface

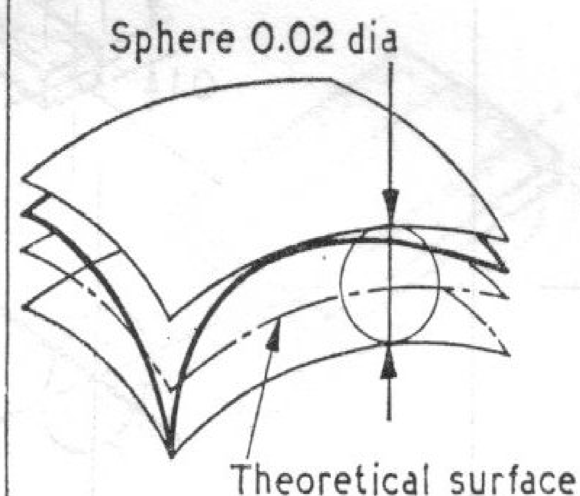


## Profile tolerance of a surface

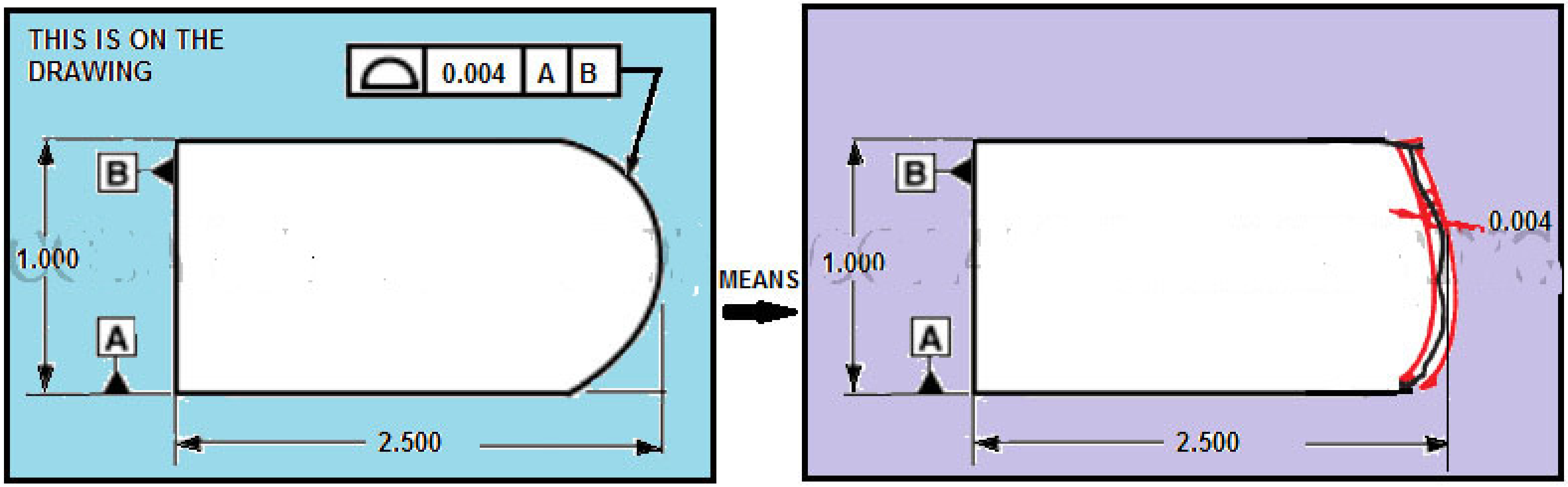
| Example | Interpretation |
|---|---|

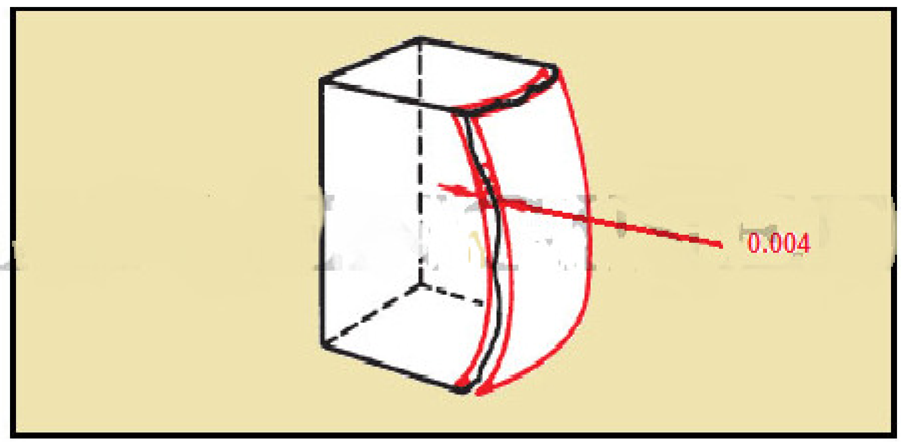
*Surface shape.* (an equally-disposed bilateral tolerance)

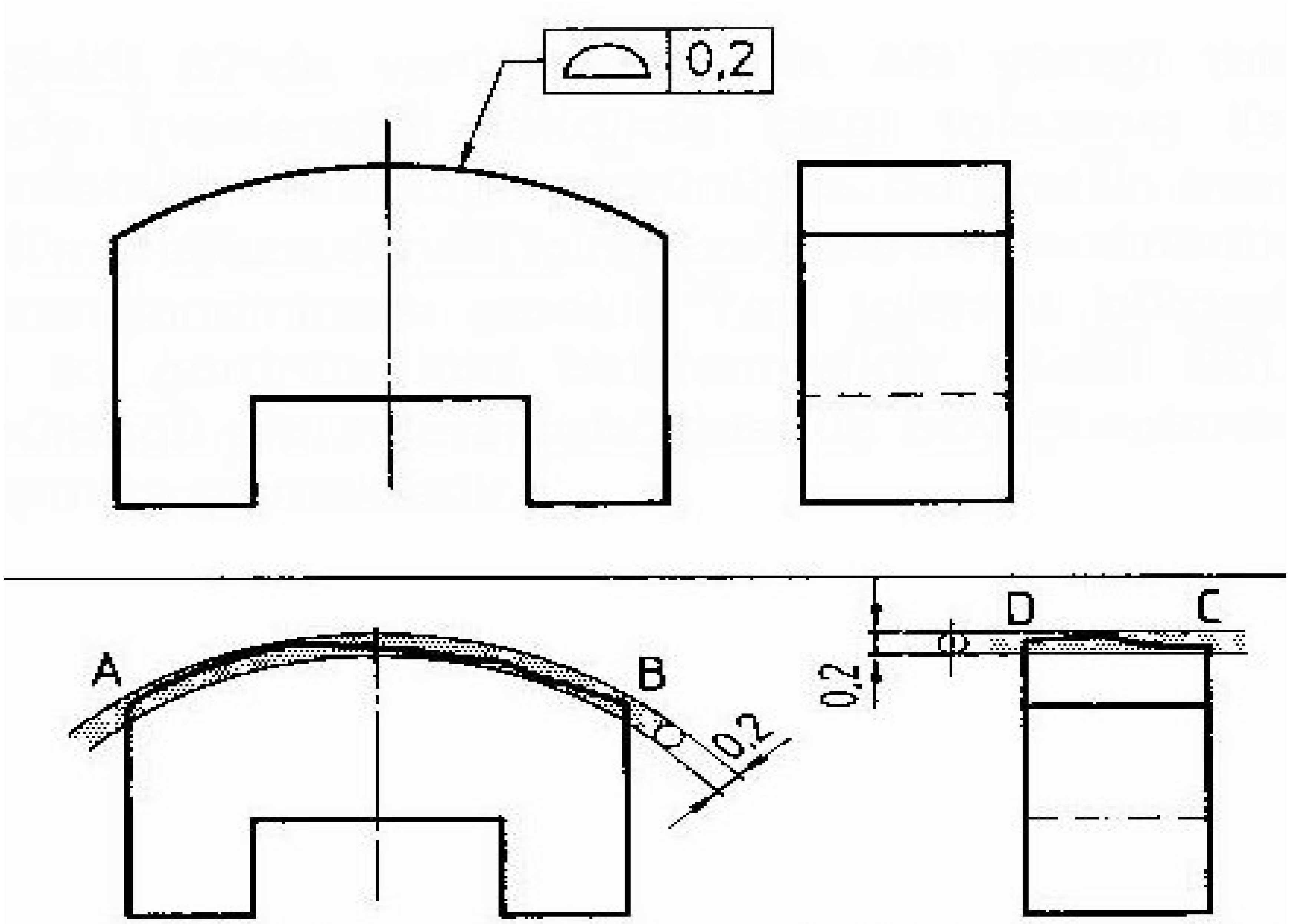



The curved surface of the part is required to lie between two surfaces which envelop a series of spheres, 0.02 diameter, with their centres on the surface having the correct geometrical shape


THIS IS ON THE DRAWING
0.004 A B
B
1.000
A
2.500
MEANS
B
1.000
A
0.004
2.500


0.004


0,2
A
B
0,2
D
C
0,2

## Tolerances of attitude ; related features

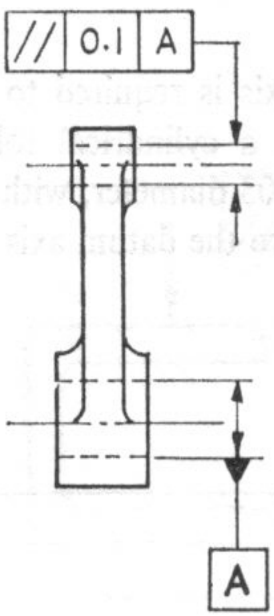
**Tolerances of parallelism.** The toleranced feature may be a line or a surface and the datum feature may be a line or a plane. In general, the tolerance zone is the area between two parallel lines or the space between two parallel planes which are parallel to the datum feature and the tolerance value is the distance between the lines or the planes. In the case of a line parallel to a datum line, the tolerance zone may alternatively be the space within a cylinder whose diameter is equal to the tolerance value and whose axis is parallel to the datum.

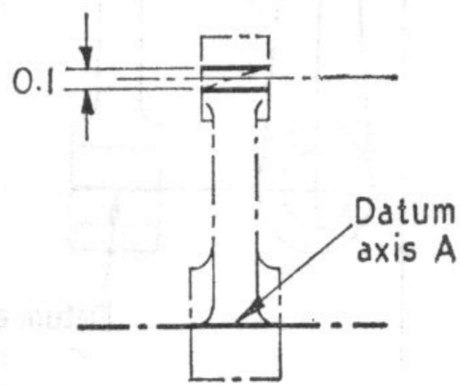
### Tolerances of parallelism

| Example | Interpretation |
|---|---|

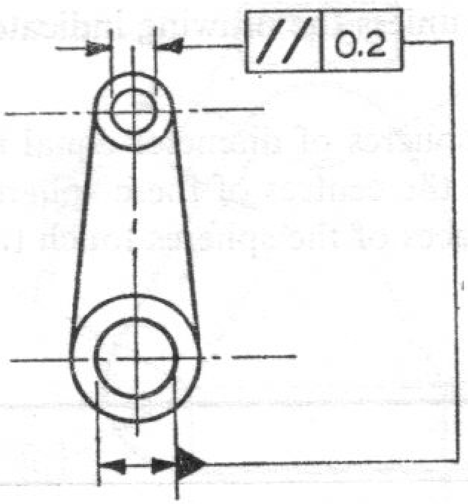
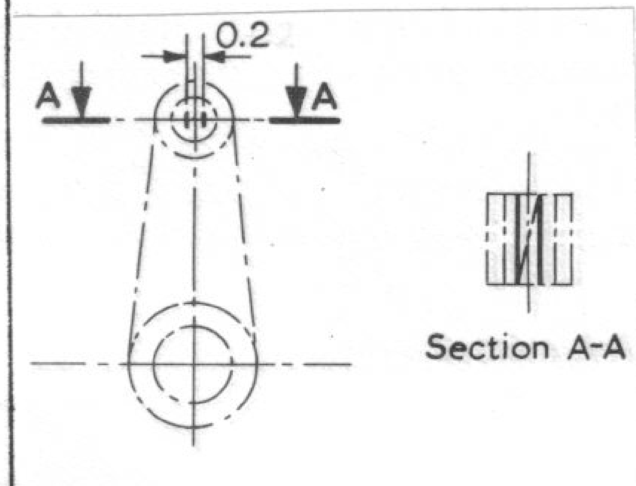
*Parallelism* 1 (of a line with respect to a datum line)

*a.*

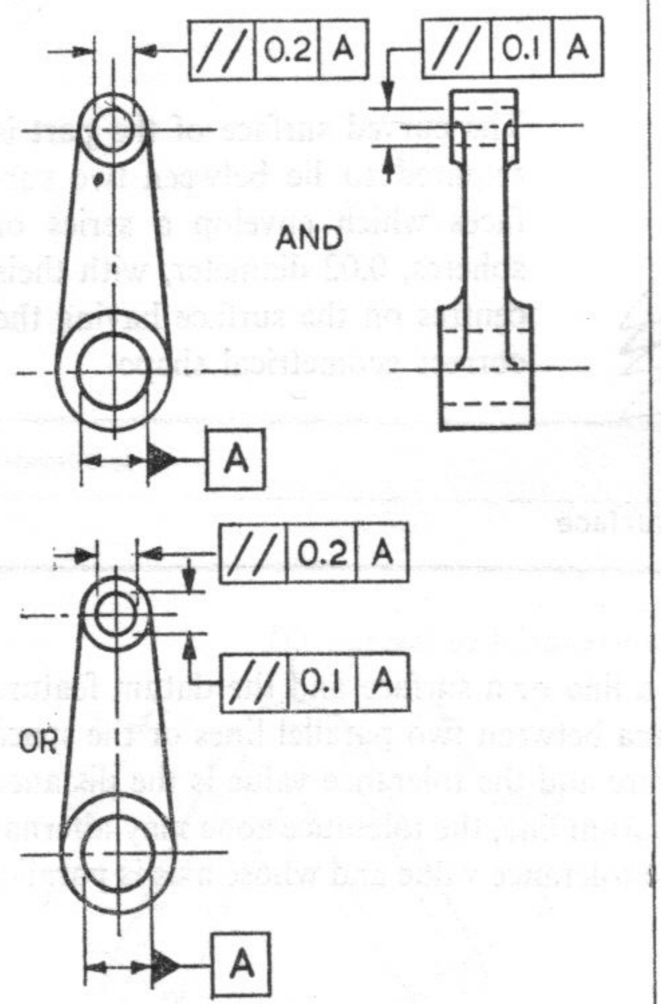


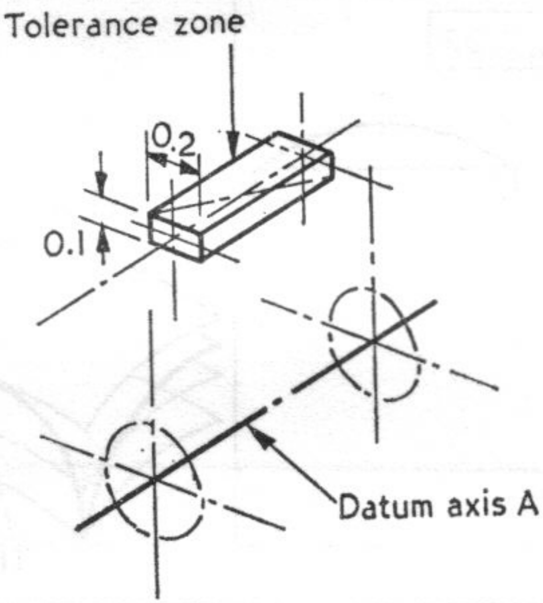




The axis of the upper cylindrical surface is required to lie between two straight lines, 0.1 apart, which are parallel to the datum axis and lie in the vertical plane through it

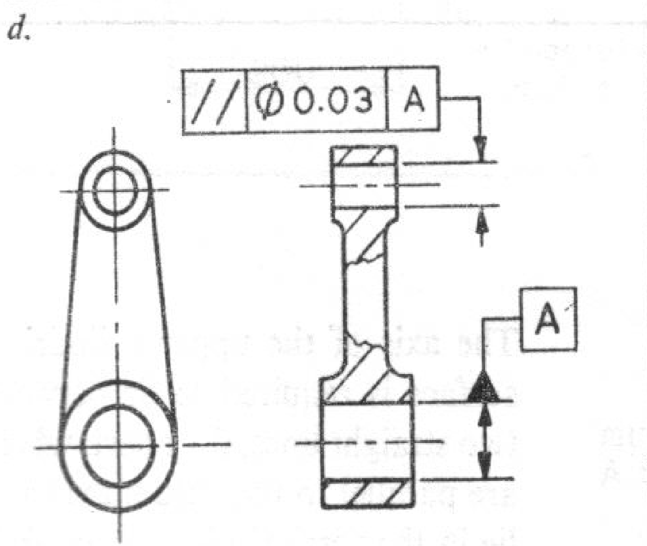
| Examples | Interpretations |
| --- | --- |
| *b.*   |   The upper axis is required to lie between two straight lines, 0.2 apart, which are parallel to the datum axis and lie in the horizontal plane |

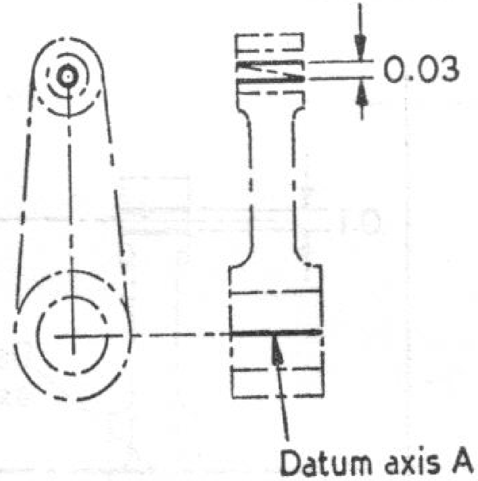
*c.*




The upper axis is required to be contained in a parallelepiped having a width of 0.2 in the horizontal plane and a width of 0.1 in the vertical plane, its sides being parallel to the datum axis A

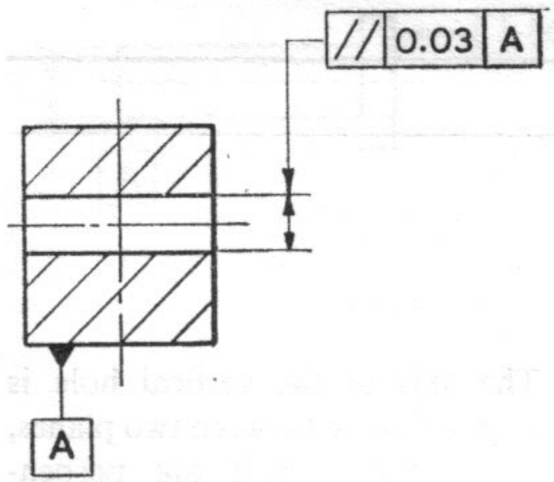


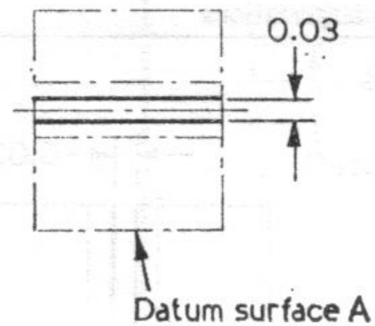




The upper axis is required to be contained in a cylindrical tolerance zone, 0.03 diameter, with its axis parallel to the datum axis

| Examples | Interpretations |
|---|---|

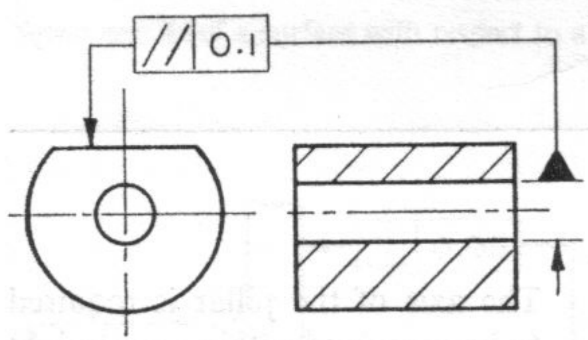
*Parallelism* 2 (of a line with respect to a datum plane)

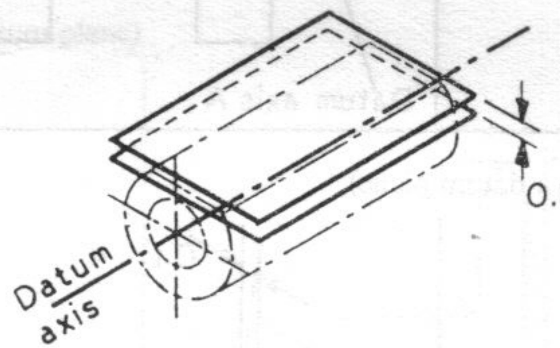



The axis of the hole is required to lie between two planes, 0.03 apart, parallel to the datum plane

*Parallelism* 3 (of a surface with respect to a datum line)

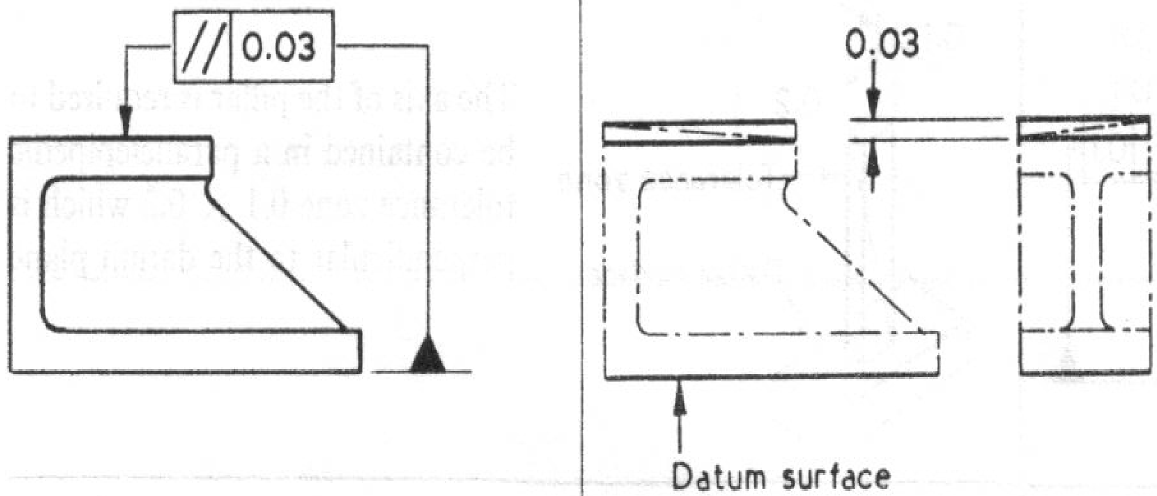



The surface of the part is required to lie between two parallel planes, 0.1 apart, parallel to the datum axis of the hole

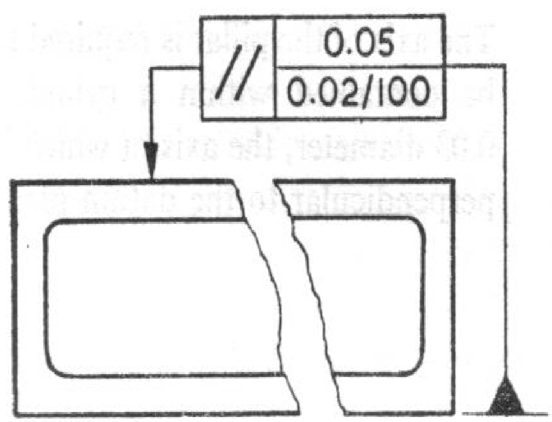
*Parallelism* 4 (of a surface with respect to a datum plane)

*a.*

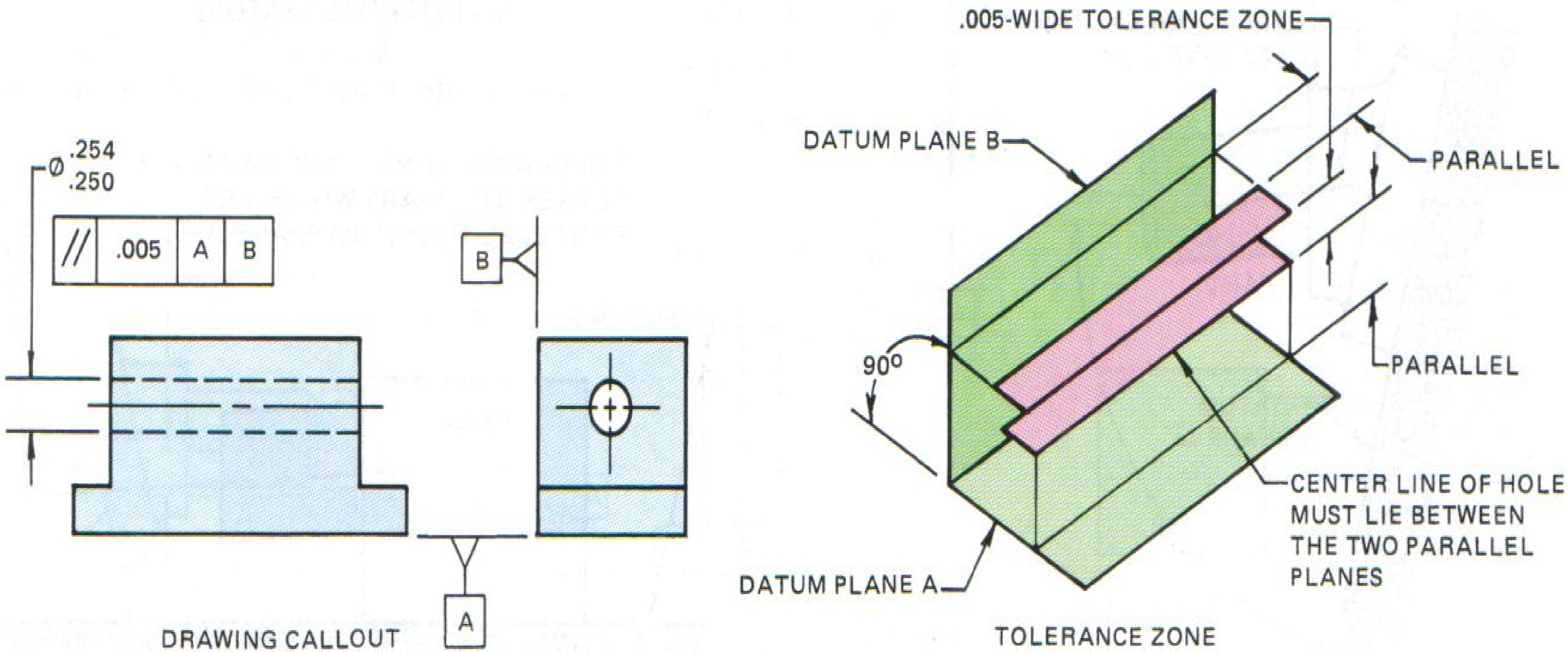



The top surface of the part is required to lie between two planes, 0.03 apart, parallel to the datum plane

*b.*




The top surface of the part is required to lie between two planes, 0.05 apart, parallel to the datum plane and all the points in a line of length 100 anywhere in the top surface of the part are required to lie between two planes, 0.02 apart, parallel to the datum plane.




**PARALLELISM TOLERANCE**

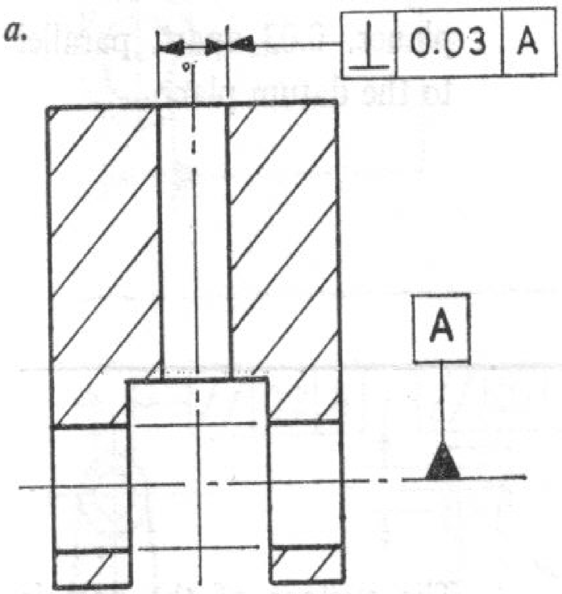
**Tolerances of squareness.** The toleranced feature may be a line or a surface and the datum feature may be a line or a plane. In general, the tolerance zone is the area between two parallel lines or the space between two parallel planes which are perpendicular to the datum feature and the tolerance value is the distance between the lines or the planes. In the case of a line with respect to a datum plane, the tolerance zone may alternatively be the space within a cylinder of diameter equal to the tolerance value; in this case the tolerance value is preceded by the symbol Ø.

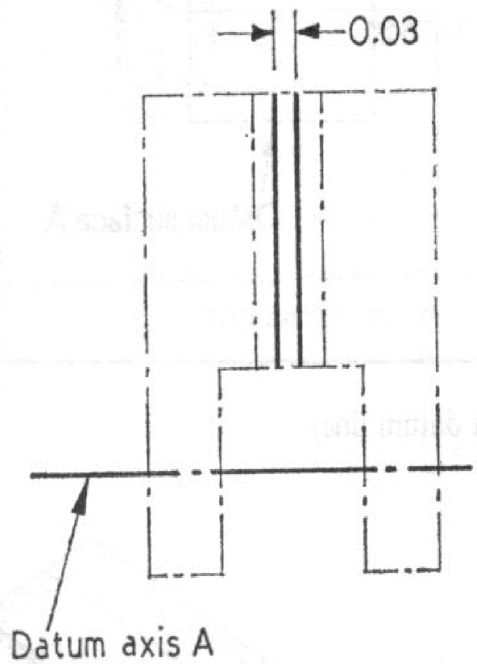
## Tolerances of squareness

| Examples | Interpretations |
|---|---|

*Squareness* 1 (of a line with respect to a datum line)

*a.*

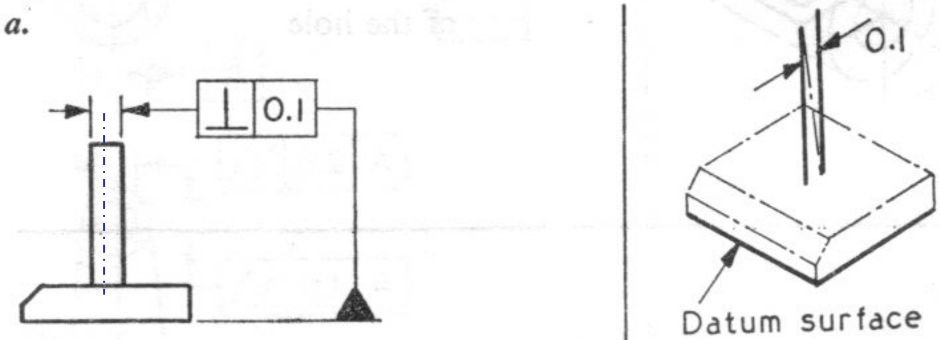



The axis of the vertical hole is required to lie between two planes, 0.03 apart, which are perpendicular to the common axis of the two horizontal holes

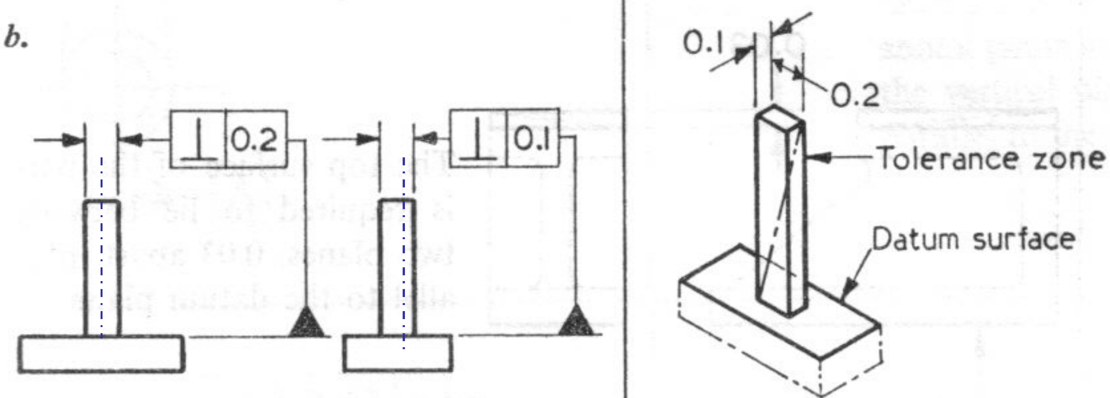
*Squareness* 2 (of a line with respect to a datum plane)

*a.*



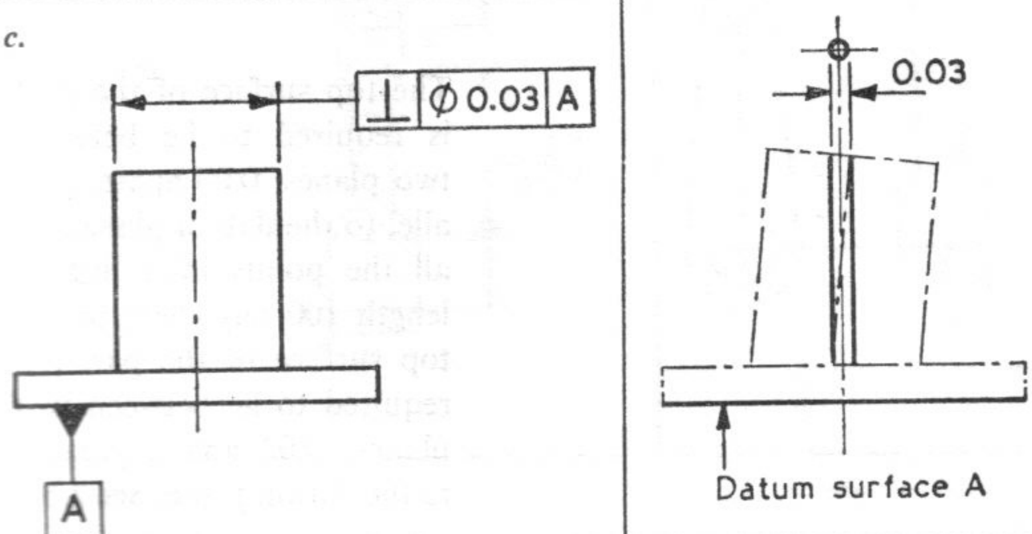

The axis of the pillar is required to be contained within two straight lines 0.1 apart, perpendicular to the datum surface and lying in the plane shown on the drawing

*b.*




The axis of the pillar is required to be contained in a parallelepipedic tolerance zone 0.1 × 0.2 which is perpendicular to the datum plane

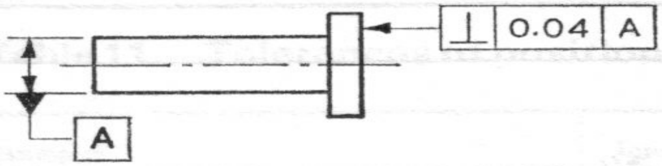
*c.*

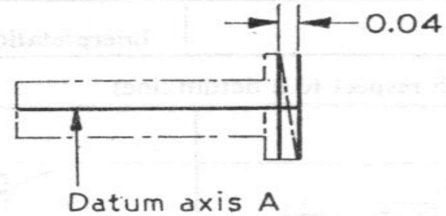



The axis of the pillar is required to be contained within a cylinder, 0.03 diameter, the axis of which is perpendicular to the datum plane

| Examples | Interpretations |
|---|---|

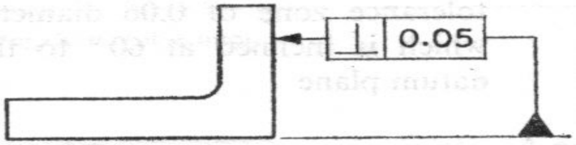
*Squareness* 3 (of a surface with respect to a datum line)

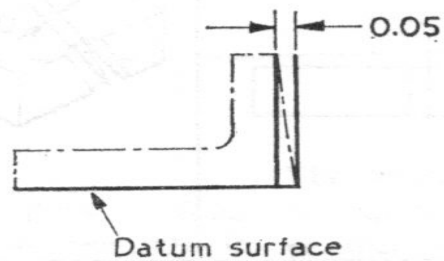



The right-hand end-face of the part is required to lie between two planes, 0.04 apart, which are perpendicular to the axis of the cylindrical portion on the left of the drawing

*Squareness* 4 (of a surface with respect to a datum plane)

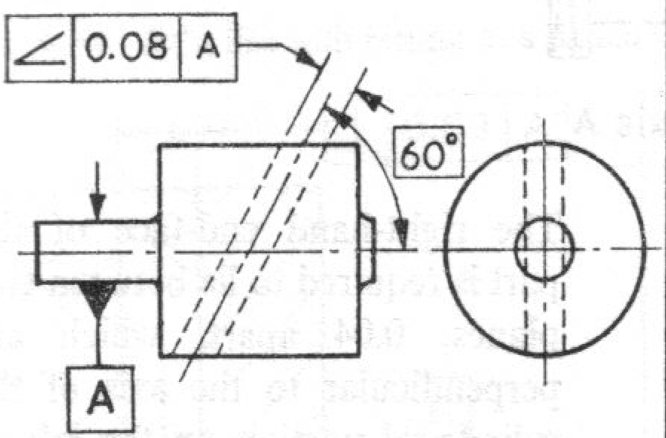
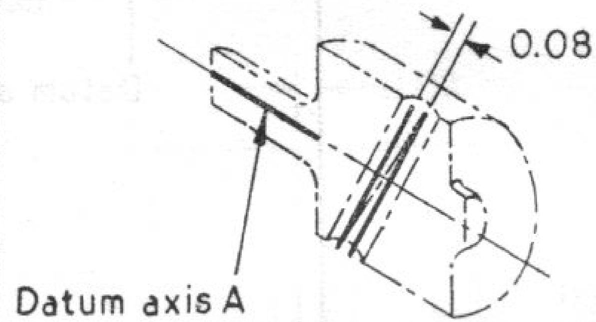


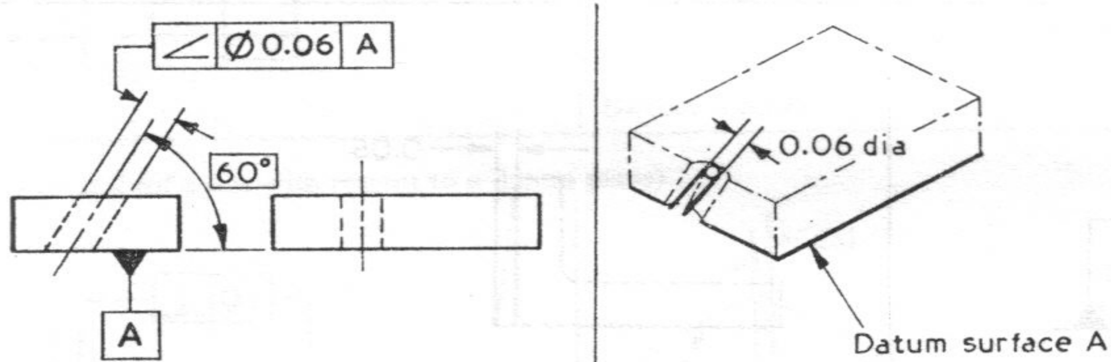

The right-hand face of the part is required to lie between two parallel planes, 0.05 apart, which are perpendicular to the datum plane

**Tolerances of angularity.** The toleranced feature may be a line or a surface and the datum feature may be a line or a plane. The tolerance zone is the area between two parallel lines or the space between two parallel planes which are inclined at the specified angle to the datum feature and the tolerance value is the distance between the lines or the planes.

## Table 10. Tolerances of angularity

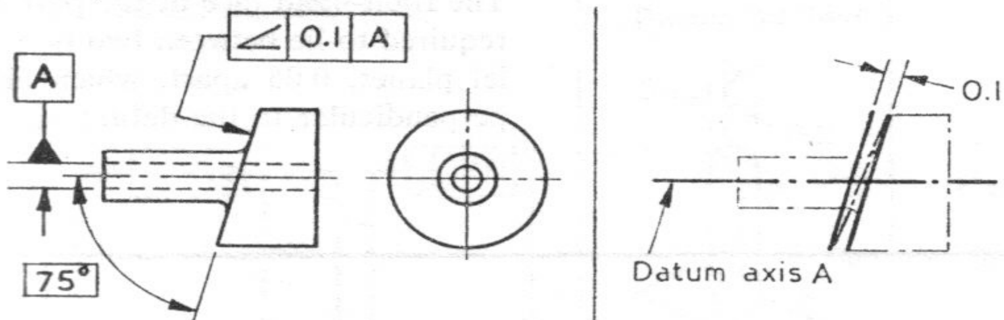
| Examples | Interpretations |
|---|---|
| *Angularity* 1 (of a line with respect to a datum line) | |
|  |  The axis of the inclined hole is required to lie between two parallel straight lines, 0.08 apart, which are inclined at 60 ° to the datum axis |

*Angularity* 2 (of a line with respect to a datum plane)



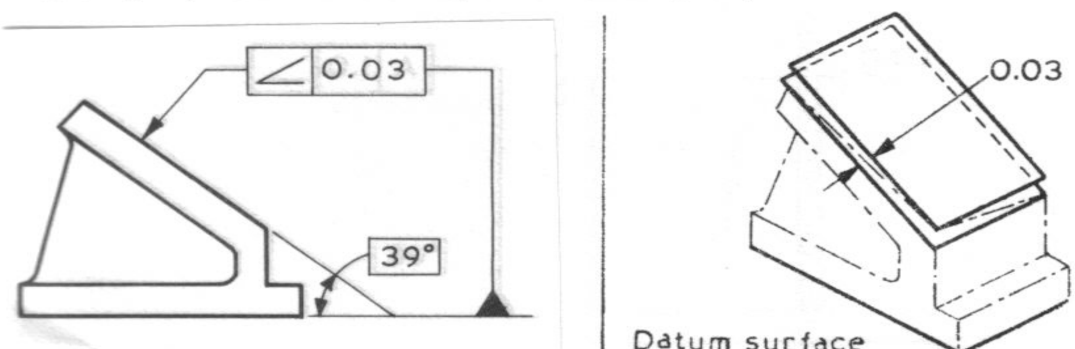

The axis of the inclined hole is required to lie within a cylindrical tolerance zone of 0.06 diameter which is inclined at 60 ° to the datum plane

*Angularity* 3 (of a surface with respect to a datum line)

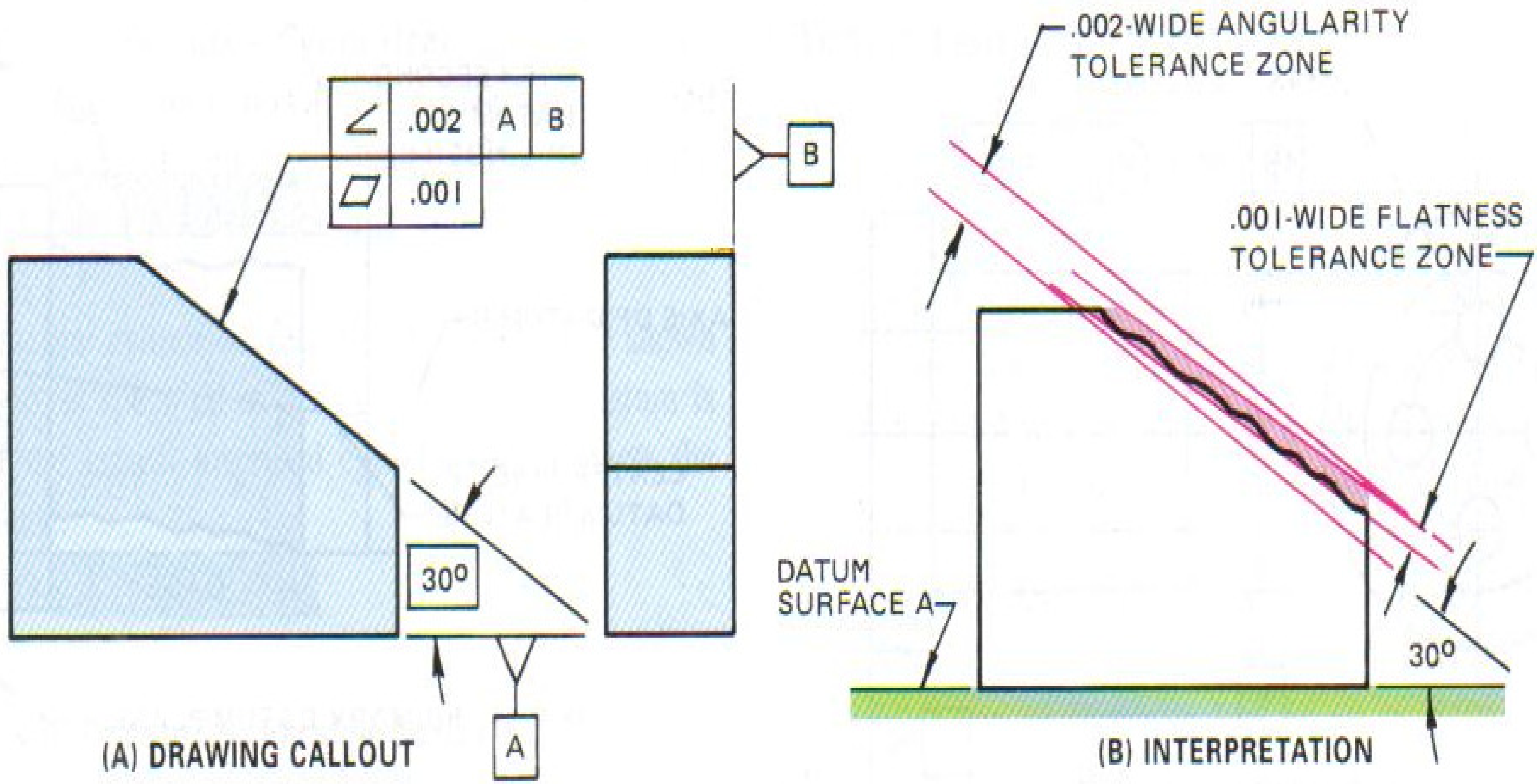



The inclined surface of the part is required to lie between two planes, 0.1 apart, which are inclined at 75 ° to the datum axis

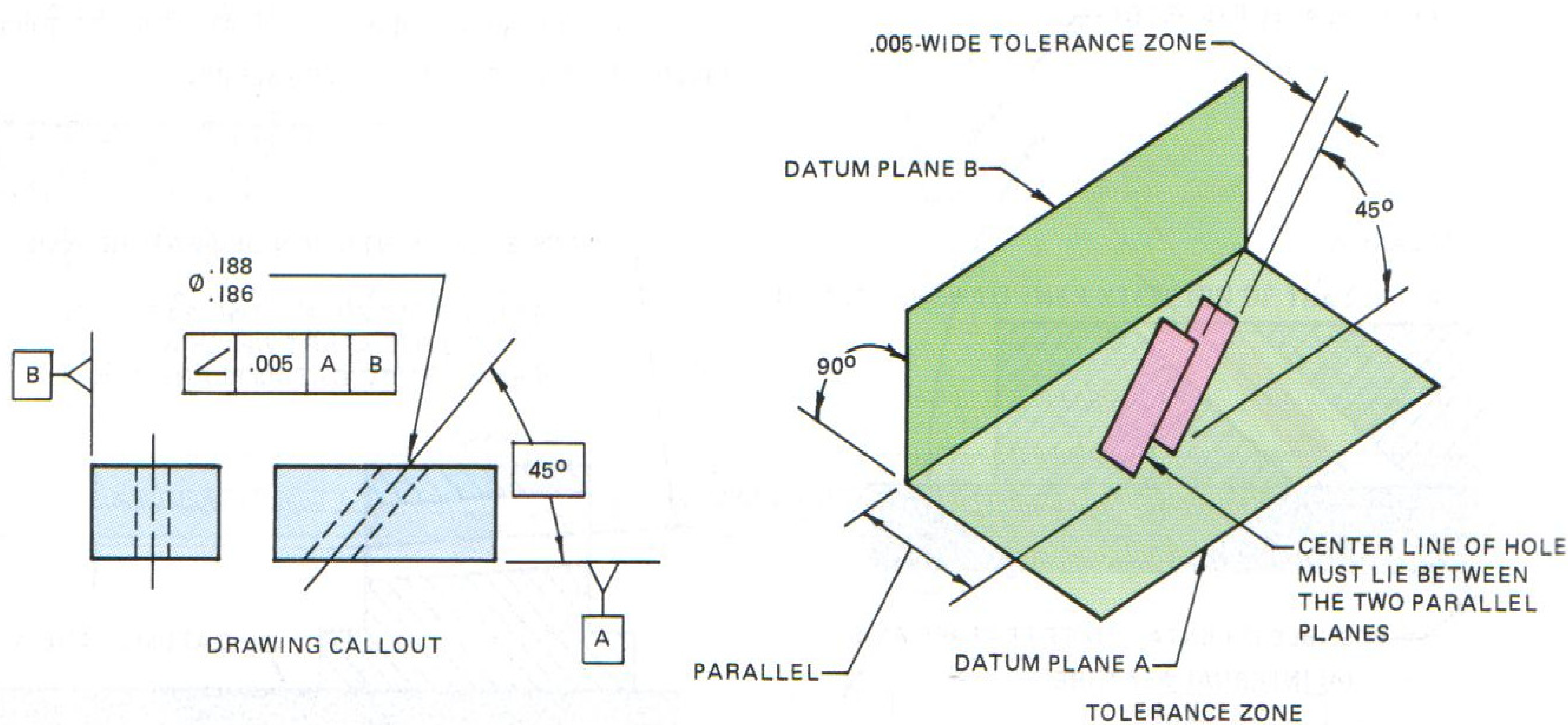
*Angularity* 4 (of a surface with respect to a datum plane)




The inclined surface of the part is required to lie between two planes 0.03 apart, which are inclined at 39 ° to the datum plane



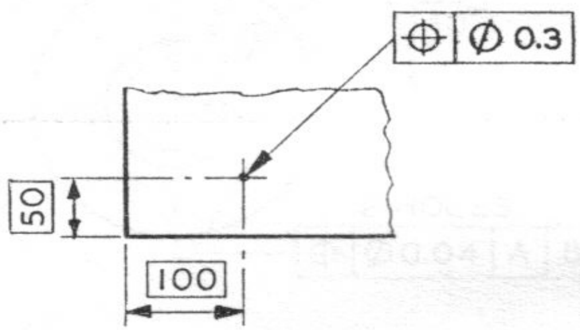
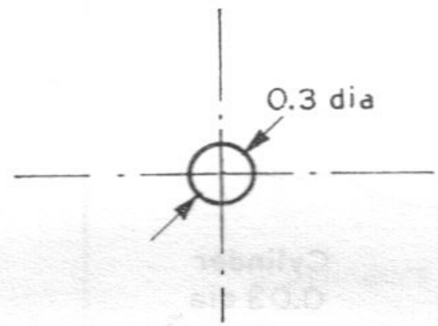
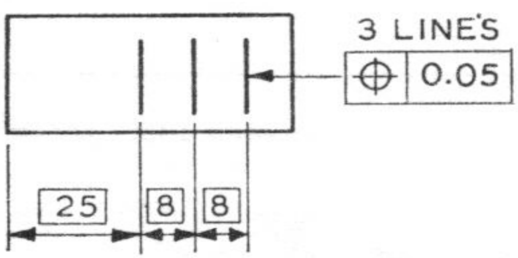
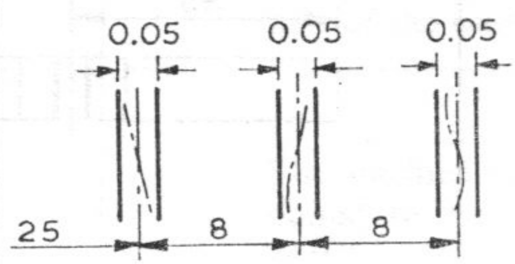

Applying both an angularity and a flatness tolerance to a flat surface.

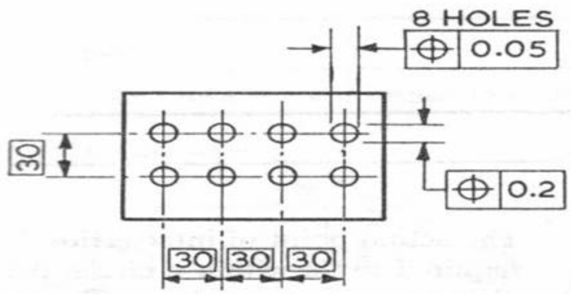
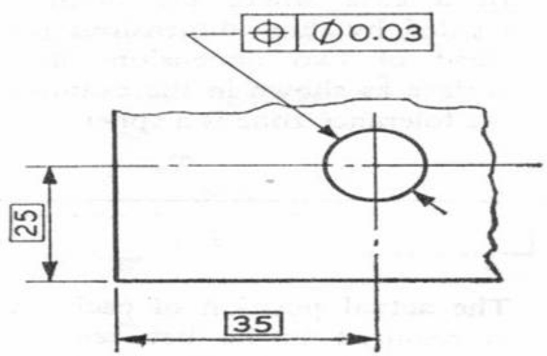
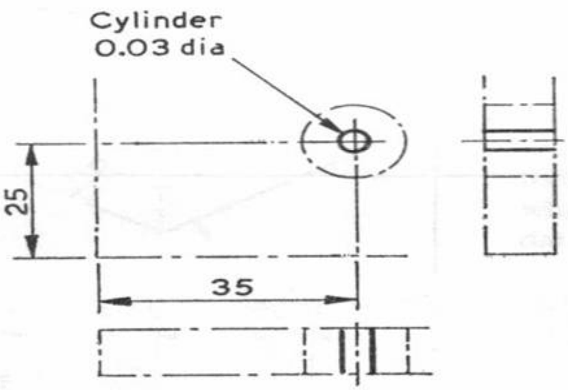
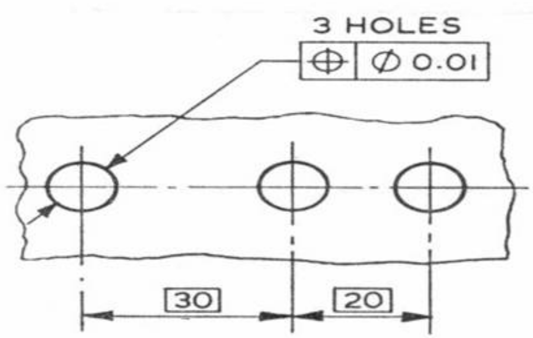
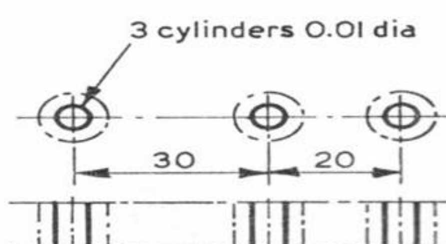



ANGULARITY TOLERANCE

**Tolerances of position.** A tolerance of position limits the deviation of the position of a feature from its specified true position.

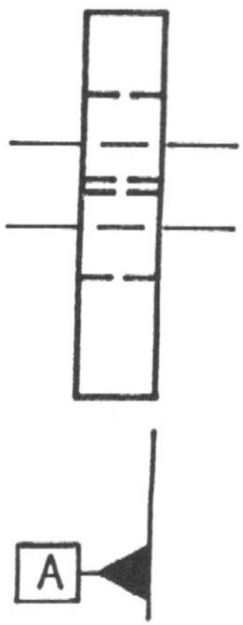
## Tolerances of position

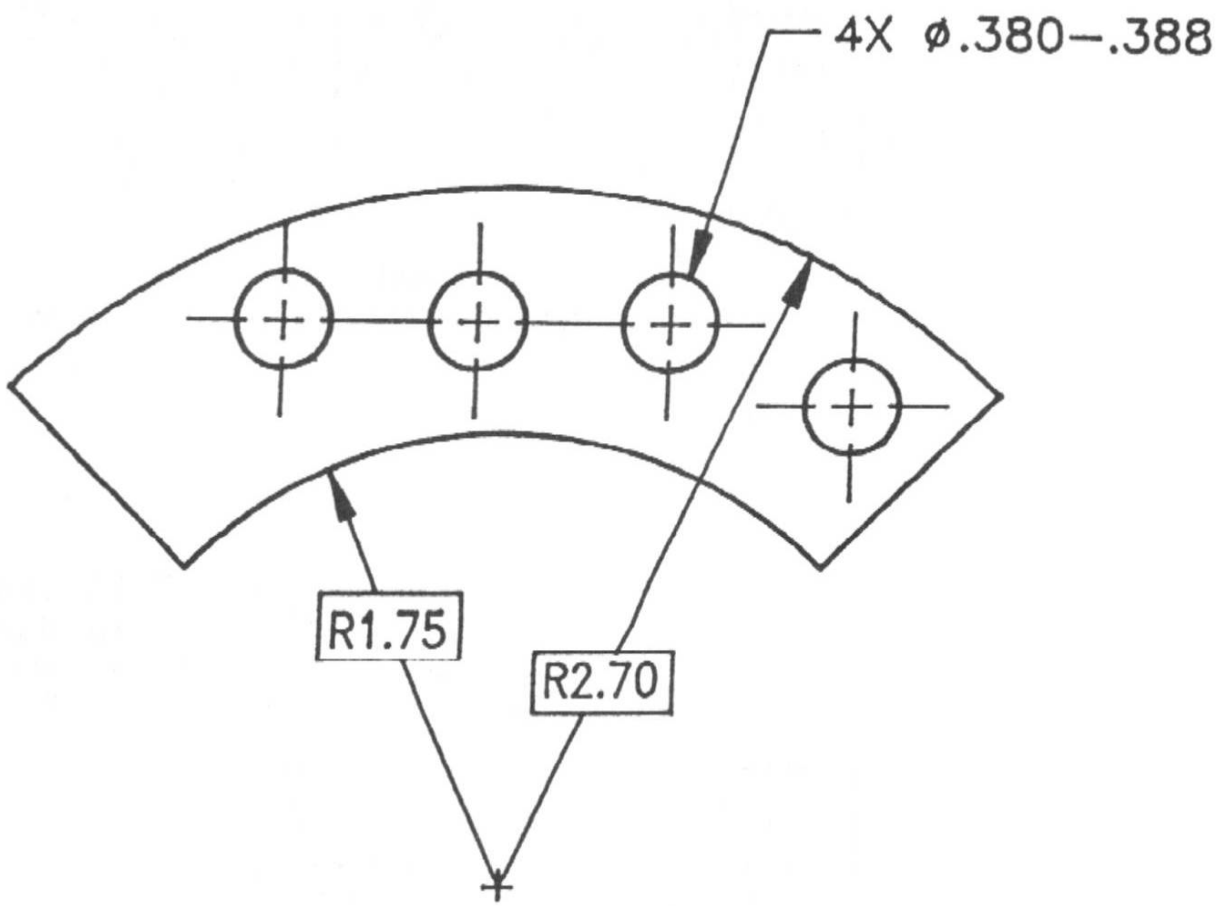
| Examples | Interpretations | |
|---|---|---|
| *Position* 1 (of a point) | | |
|  |  | The actual point of intersection is required to lie within a circle, 0.3 diameter, in the plane of the surface, which has its centre at the specified true point of intersection. In a case where the point is located by three dimensions (instead of two dimensions in a surface as shown in the example) the tolerance zone is a sphere |
| *Position* 2 (of a line)<br>*a.* | | |
|  |  | The actual position of each line is required to lie between two parallel straight lines on the surface, 0.05 apart, which are symmetrically disposed about the specified true position of the line |

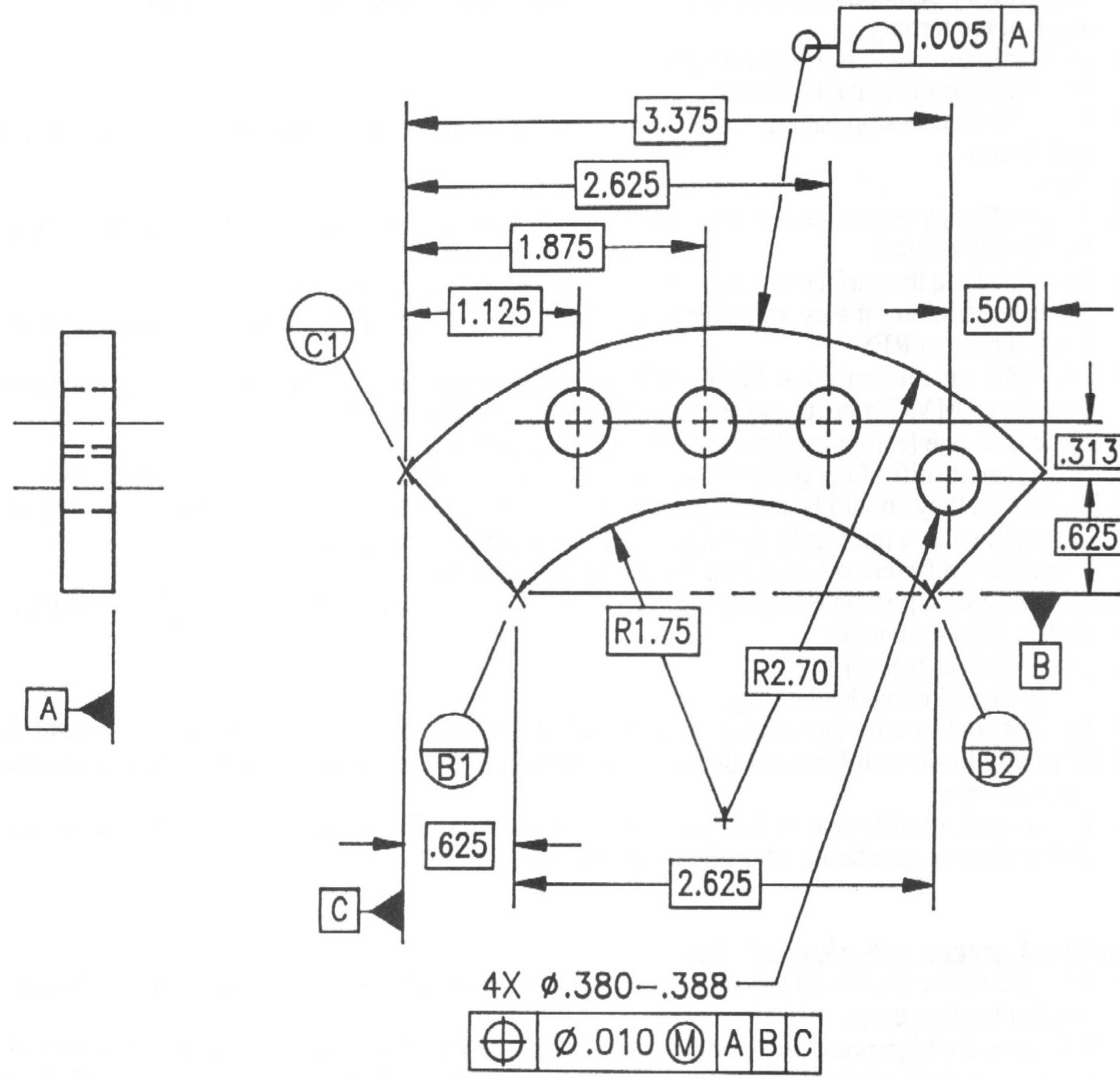
| Examples | Interpretations |
| --- | --- |
| *b.*   | The axis of each of the 8 holes is required to be contained within a parallelepiped, of size 0.2 and 0.05, symmetrically disposed about the specified true position of the axis of the hole*. |
| *c.*   |   The axis of the hole is required to be contained within a cylinder, 0.03 diameter, with its axis in the specified true position of the axis of the hole* |
| *d.*   |   The axis of each hole is required to be contained within a cylinder, 0.01 diameter, with its axis in the specified true position of the axis of the hole* |

# Use of Profile and Position on Irregular-Shaped Parts

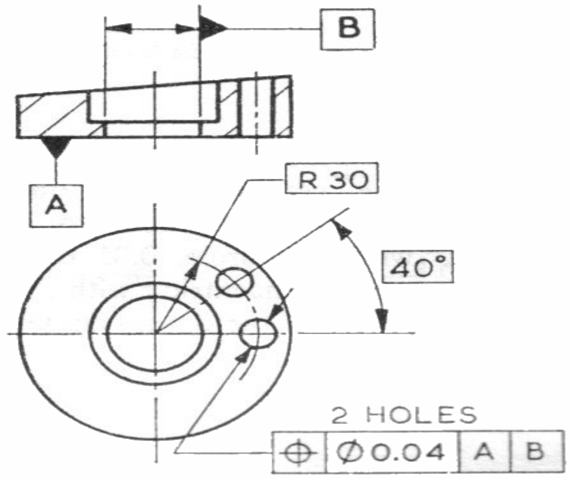
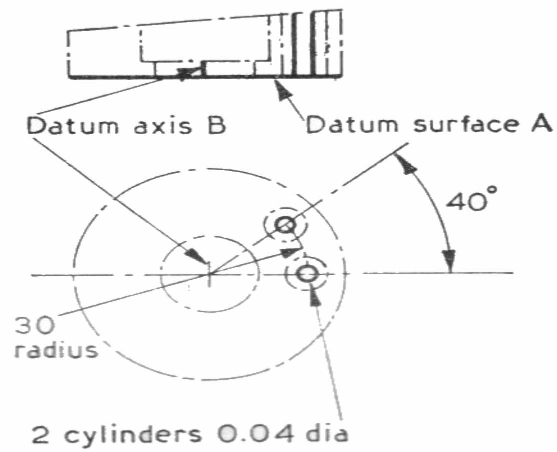
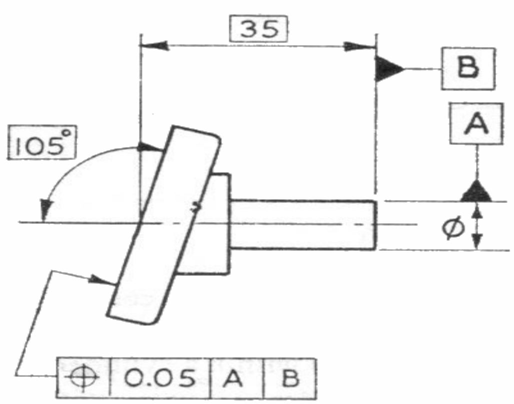
a) Profile the outside of this part all around to within .005.
b) Locate the four holes to a datum reference frame that uses a planar surface for a perpendicularity datum and targets for location.
[**Note**: Dimensions should be included but do not have to be accurate or per scale. The virtual condition of the mating shafts for the four .380 - .388 holes is .370.]



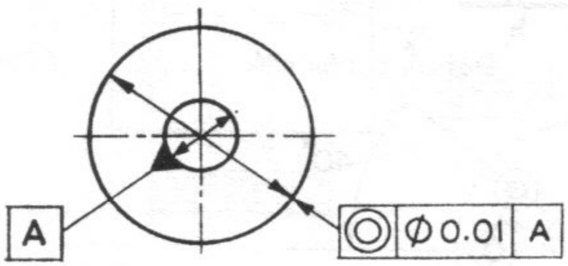
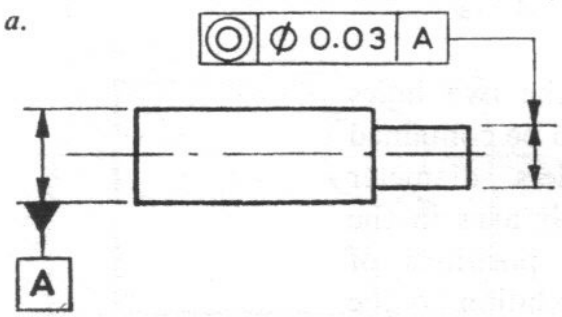
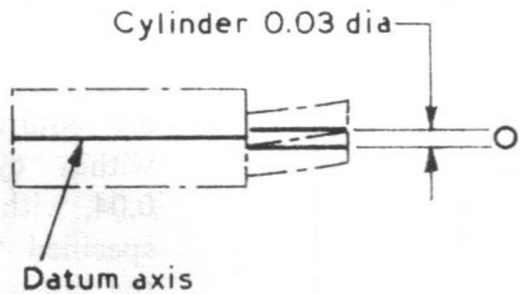


.005
A
3.375
2.625
1.875
1.125
.500
C1
.313
.625
R1.75
R2.70
B
B1
B2
A
.625
2.625
C
4X ⌀.380–.388
⌀.010 Ⓜ A B C

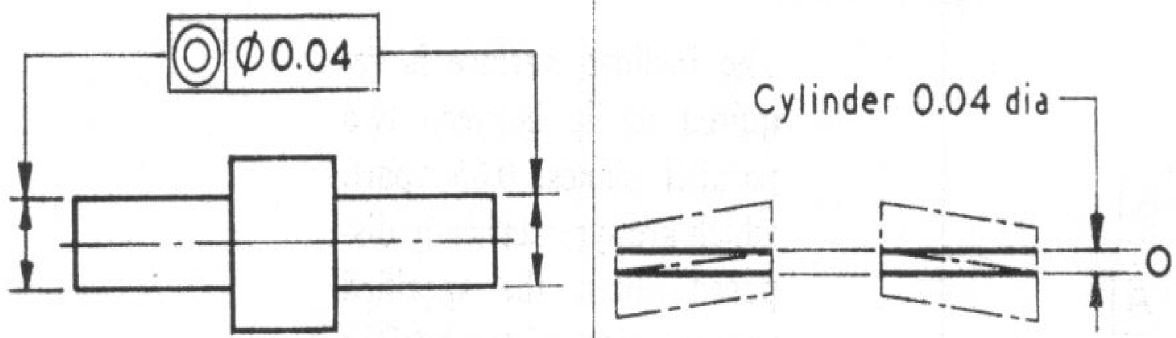
| **Examples** | **Interpretations** |
|---|---|
| *e.*   |   The axes of the two holes are required to be contained within cylinders diameter 0.04, with their axes in the specified true positions of the holes in relation to the datum surface A, and the axis of the central hole B |
| *Position* 3 (of a surface)   | The inclined surface is required to lie between two parallel planes, 0.05 apart, which are symmetrically disposed about the specified true position of the surface in relation to the axis of the right-hand portion and the end |

**Tolerances of concentricity.** A concentricity tolerance is a particular case of a positional tolerance in which the toleranced feature and the datum feature are circles or cylinders. The tolerance limits the deviation of the position of the centre or axis of the toleranced feature from its true position, i.e. the centre or axis of the datum feature. The tolerance value is the diameter of the tolerance zone.

## Tolerances of concentricity

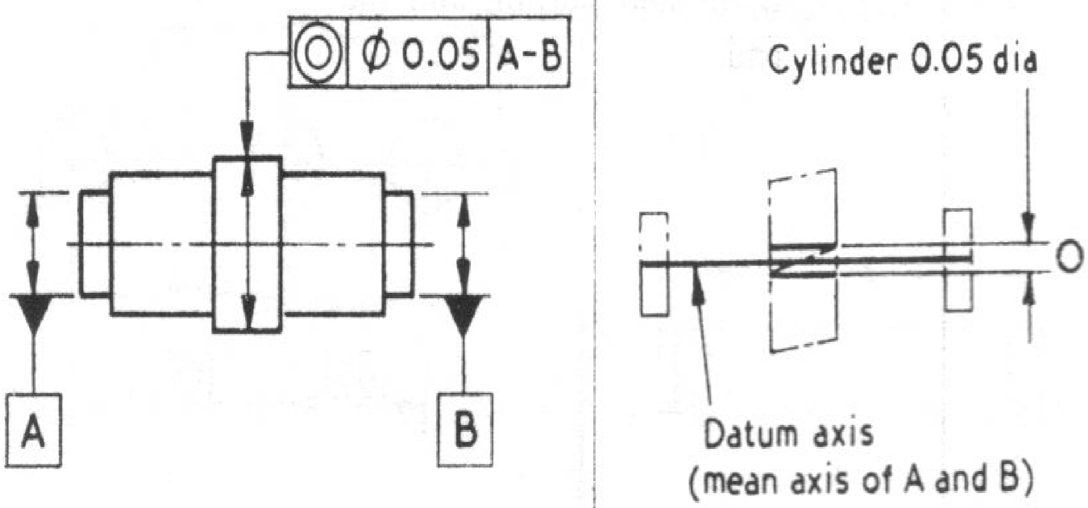
| **Examples** | **Interpretations** |
|---|---|
| *Concentricity* 1 (of a point)   | The centre of the outer circle is required to lie within a circle, 0.01 diameter, concentric with the centre of the datum circle A |
| *Concentricity* 2 (of a line)<br>*a.*   |   The axis of the right-hand cylindrical portion is required to be contained within a cylinder, 0.03 diameter, co-axial with the left-hand portion |

*b.*




The axes of the left-hand and right-hand cylindrical portions are required to be contained within one cylinder, 0.04 diameter

*c.*




The axis of the central portion is required to be contained within a cylinder, 0.05 diameter, co-axial with the mean axis of the end projections of the part

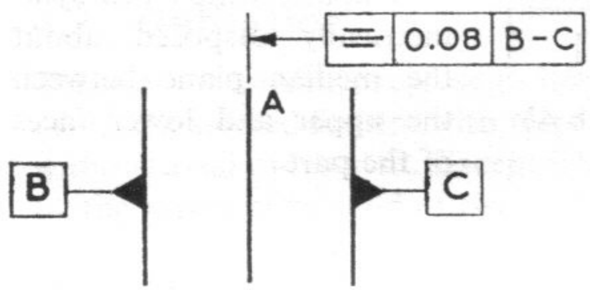
**Tolerances of symmetry.** A symmetry tolerance is a particular case of a positional tolerance in which the position of the feature is specified by its symmetrical relationship to a datum

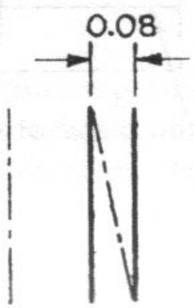
## Tolerances of symmetry

| Examples | Interpretations |
|---|---|

*Symmetry* 1 (of a line)

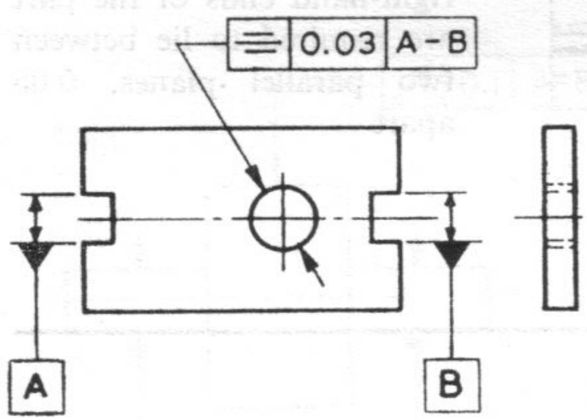
***a.***

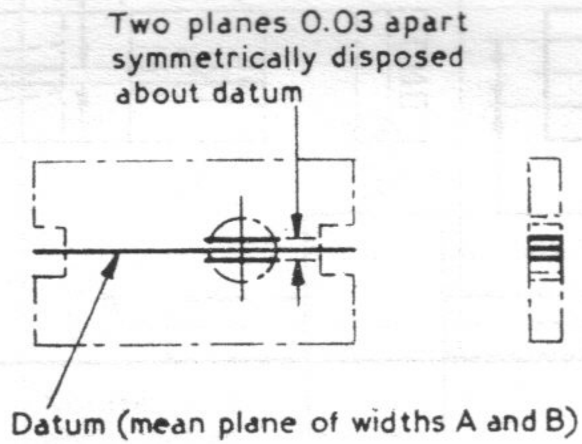


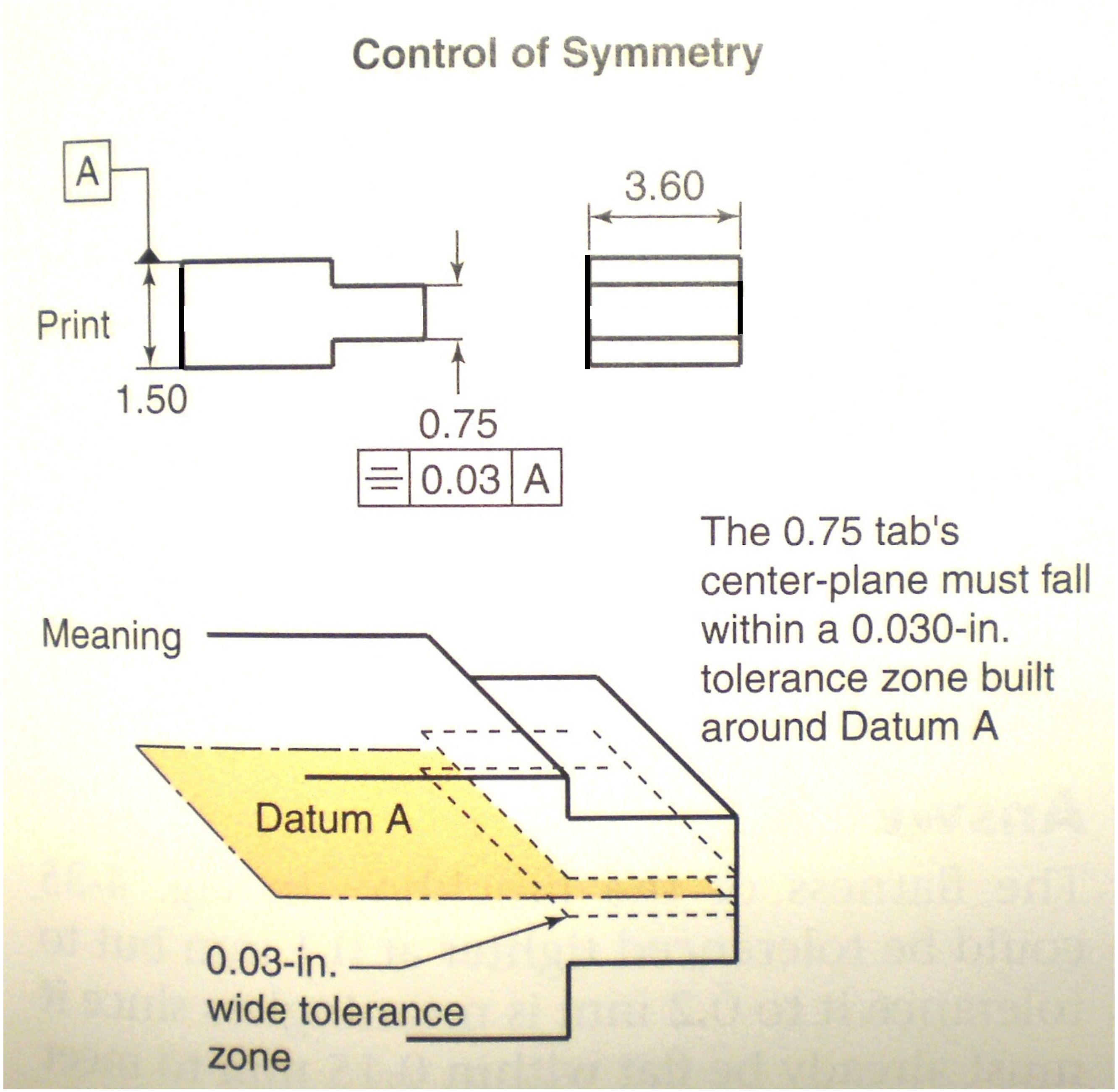

The line A is required to lie between two parallel straight lines, 0.08 apart, which are symmetrically disposed about the line situated symmetrically between the datum lines B and C

***b.***

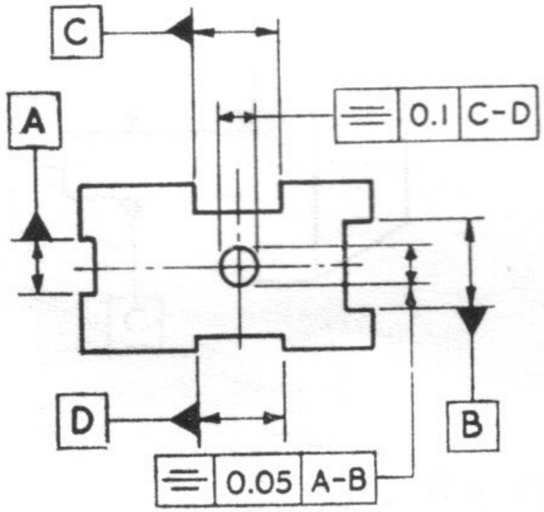


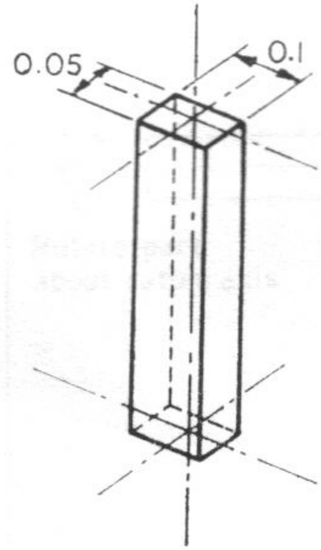




The axis of the hole is required to lie between two parallel planes, 0.03 apart, which are symmetrically disposed about the common median plane of the slots in the ends of the part


Control of Symmetry
A
3.60
Print
1.50
0.75
0.03 A
The 0.75 tab's center-plane must fall within a 0.030-in. tolerance zone built around Datum A
Meaning
Datum A
0.03-in. wide tolerance zone

c.



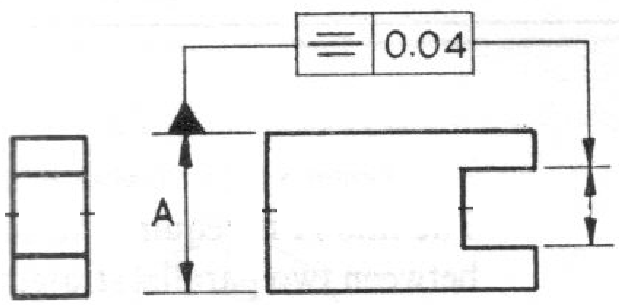



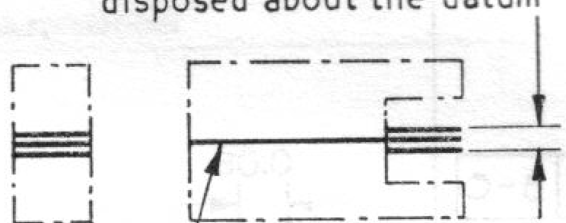

The axis of the hole is required to lie in a parallelepiped, its sides parallel to the longer sides of the part, 0.05 apart, and symmetrically disposed about the common median plane of the slots in the left-hand and right-hand ends of the part and its sides parallel to the shorter sides of the part, 0.1 apart, and symmetrically disposed about the common median plane of the slots in the top and bottom sides

| Examples | Interpretations |
|---|---|

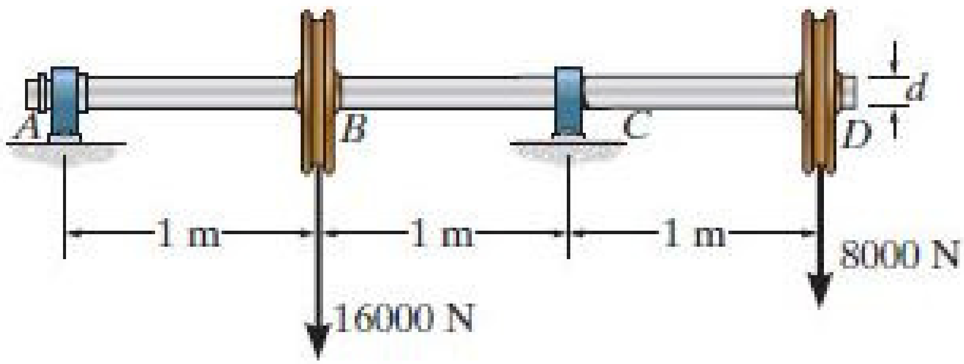
*Symmetry* 2 (of a median plane)

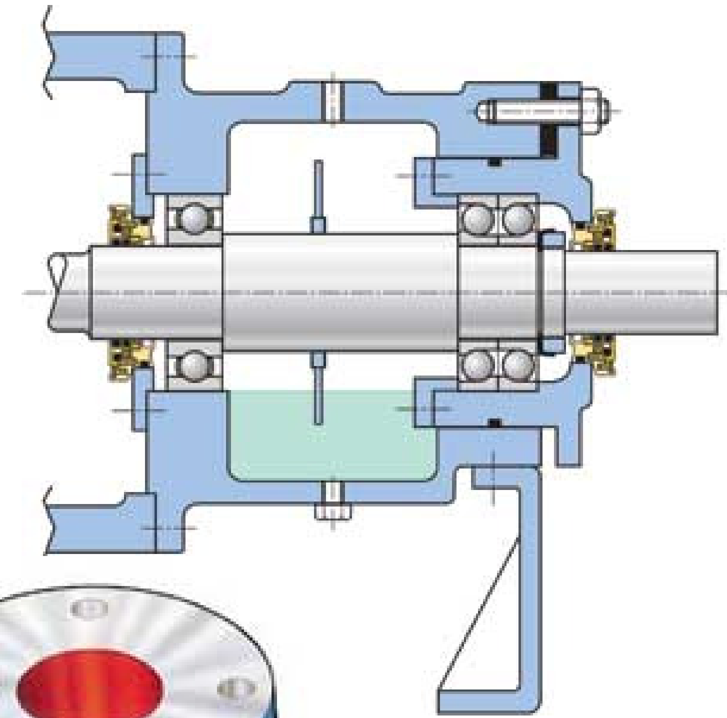
*a.*

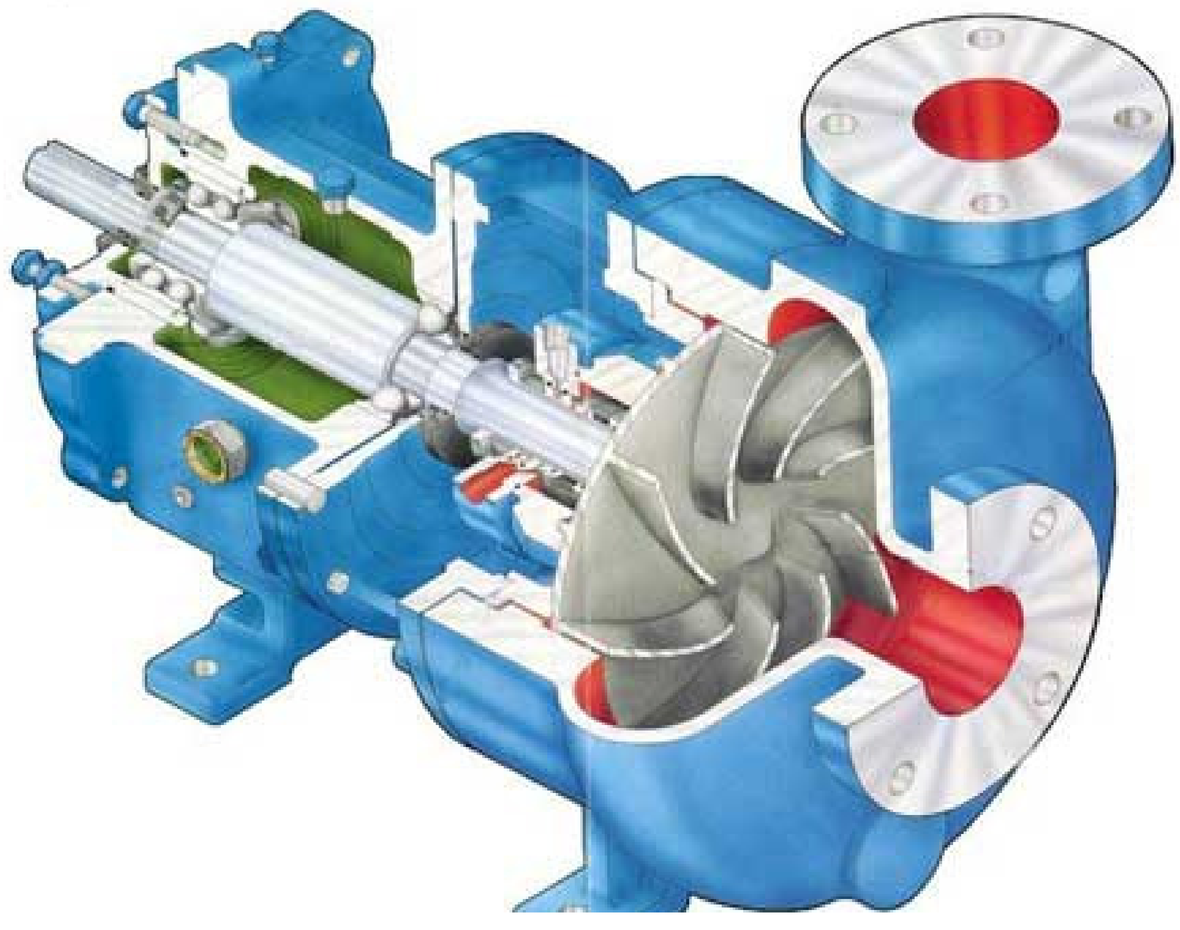



The median plane of the slot in the right-hand end of the part is required to lie between two parallel planes, 0.04 apart, which are symmetrically disposed about the median plane between the upper and lower faces of the part


A
B
C
D
d
1 m
1 m
1 m
16000 N
8000 N





## Run-out

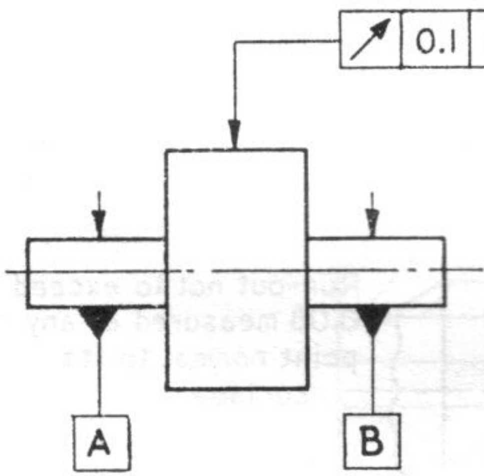
### Definition

The run-out tolerance represents the maximum permissible variation of position (i.e. full indicator movement) of the considered feature with respect to a fixed point during one complete revolution about the datum axis without axial movement. Except when otherwise stated this variation is measured in the direction indicated by the arrow at the end of the leader line which points to the toleranced feature.

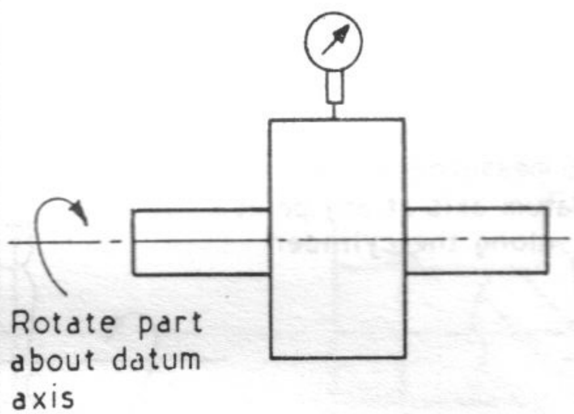
### Tolerances of run-out

**Examples**

*a.*



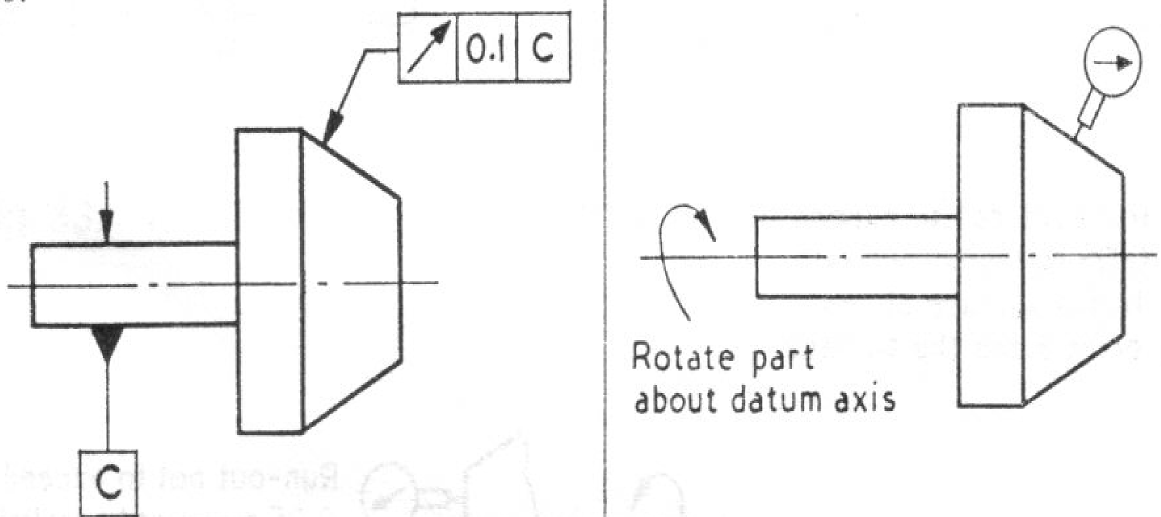

**Interpretations**




Run-out not to exceed 0.1 measured perpendicular to datum axis at any point along the cylinder

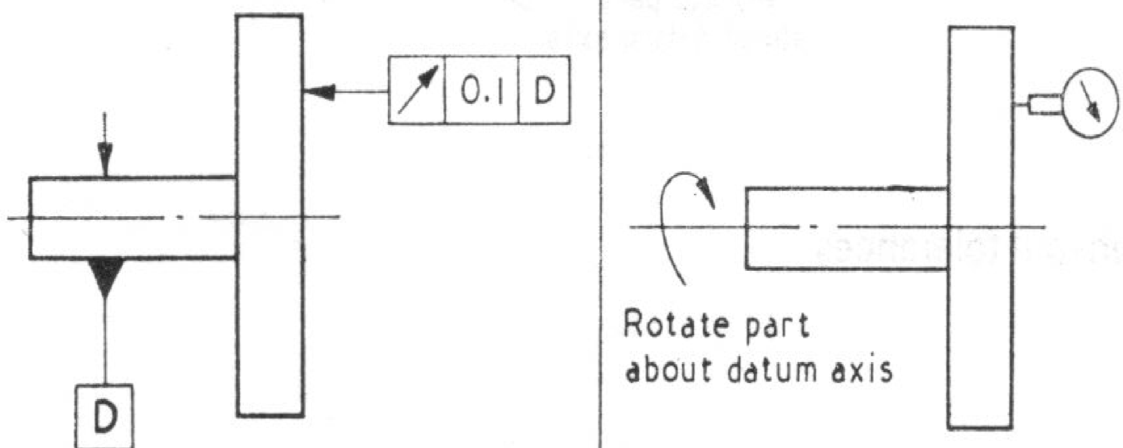
Mitutoyo
0.0000 in
OFF
in
mm
ON
ZERO

*b.*

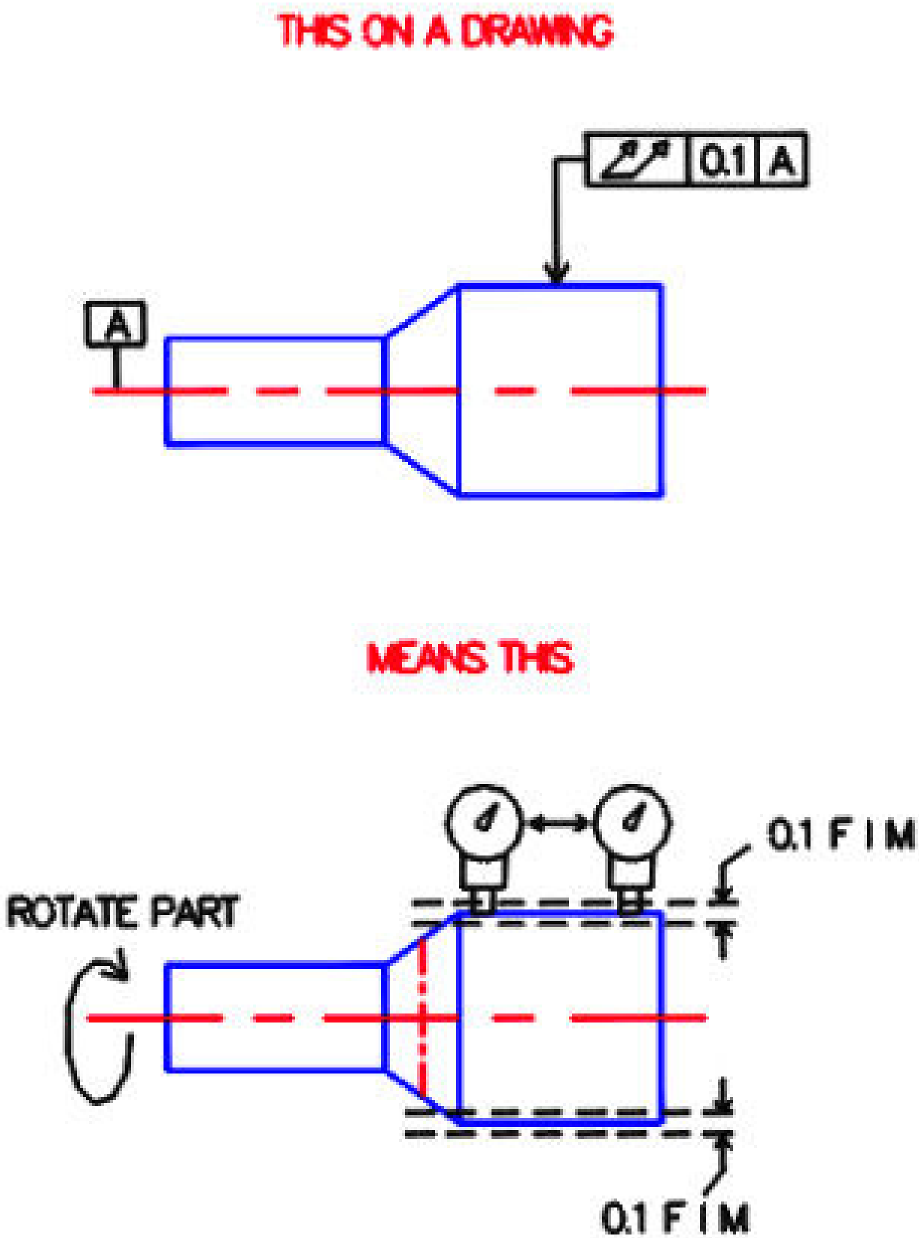


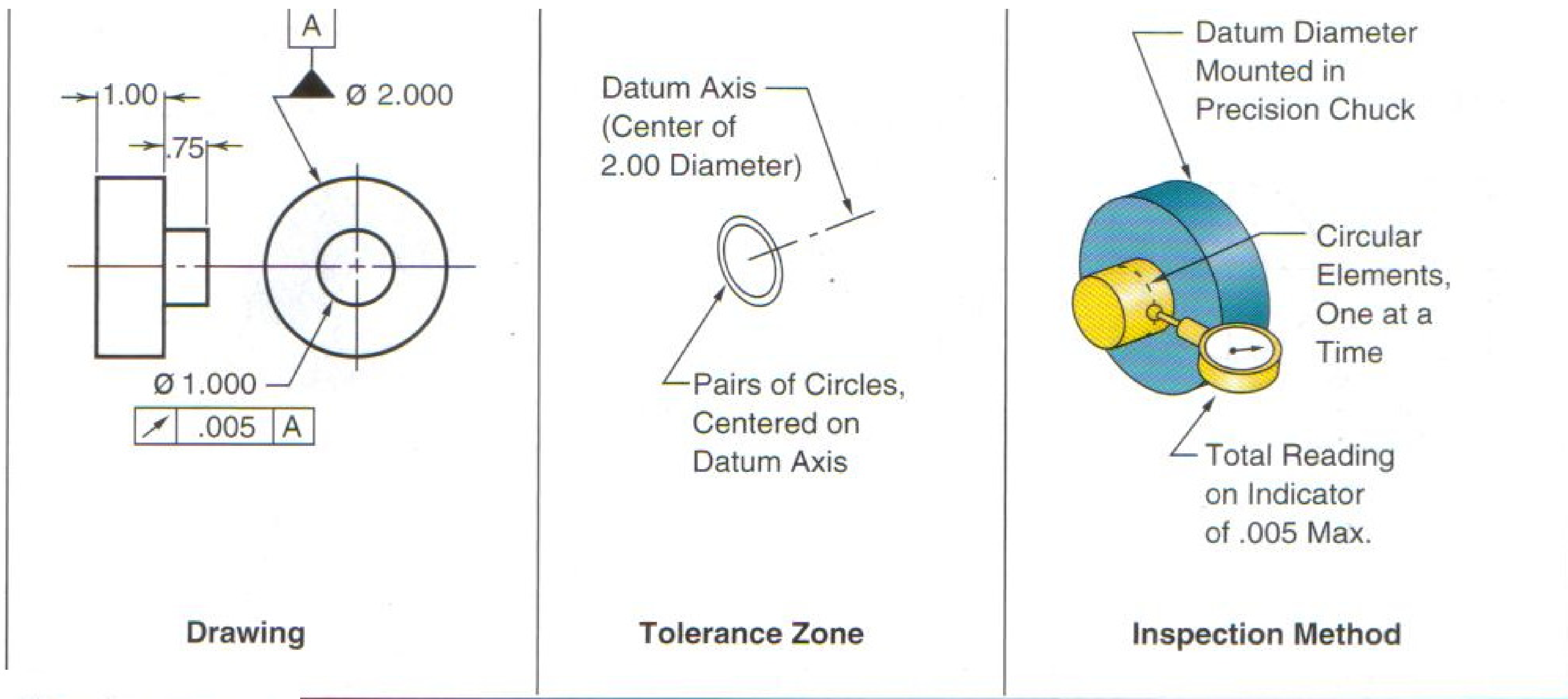

Run-out not to exceed 0.1 measured at any point normal to the surface

*c.*

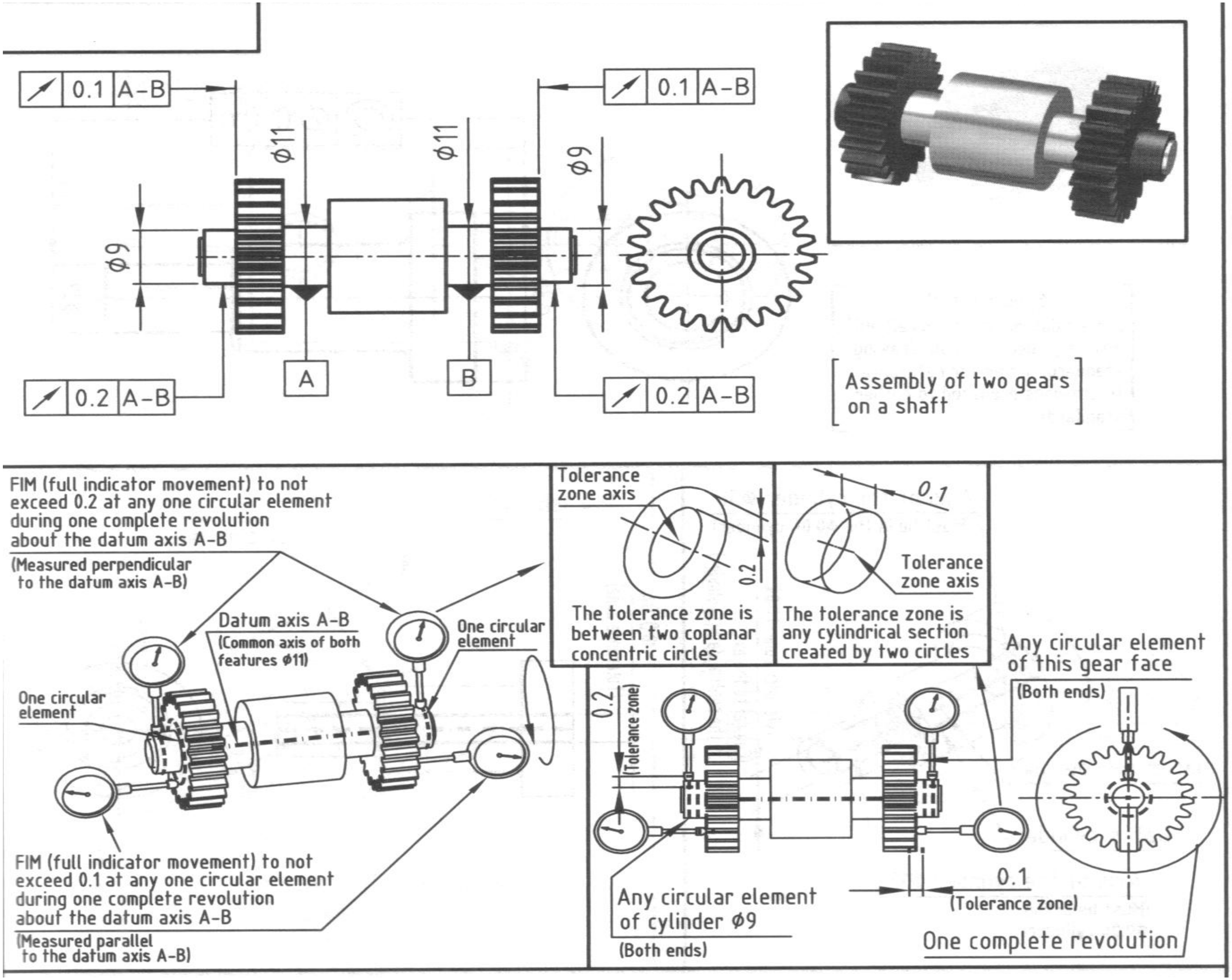



Run-out not to exceed 0.1 measured parallel to the datum axis at any radius


THIS ON A DRAWING
0.1
A
A
MEANS THIS
ROTATE PART
0.1 F I M
0.1 F I M




**Circular runout**


0.1 A–B
0.1 A–B
Ø11
Ø11
Ø9
Ø9
0.2 A–B
A
B
0.2 A–B
Assembly of two gears on a shaft
FIM (full indicator movement) to not exceed 0.2 at any one circular element during one complete revolution about the datum axis A–B
(Measured perpendicular to the datum axis A–B)
Datum axis A–B
(Common axis of both features Ø11)
One circular element
One circular element
FIM (full indicator movement) to not exceed 0.1 at any one circular element during one complete revolution about the datum axis A–B
(Measured parallel to the datum axis A–B)
Tolerance zone axis
0.2
The tolerance zone is between two coplanar concentric circles
0.1
Tolerance zone axis
The tolerance zone is any cylindrical section created by two circles
Any circular element of this gear face
(Both ends)
0.2 (Tolerance zone)
0.1
(Tolerance zone)
Any circular element of cylinder Ø9
(Both ends)
One complete revolution

## Run-out as a composite tolerance

Run-out may sometimes be applied as a composite tolerance in place of separate specifications of other geometrical tolerances, e.g. roundness or concentricity; however it should not be so used where the design requirement demands that these characteristics be separately controlled. When required, run-out tolerances may be specified for a part or feature as well as other geometrical tolerances.

## Tolerance zone orientation

the width of the run-out tolerance zone lies in the direction of the arrow terminating the leader line. This will often, but not necessarily, be normal to the surface.

## Run-out tolerance applied to an assembly

Fig. illustrates the application of run-out tolerance to an assembly of parts.

During one complete revolution of the assembly about the axis defined by the two journals E and F, the radial run-out of the peripheries of the couplings shall not exceed 0.1 and the axial run-out of the faces of the couplings shall not exceed 0.2.

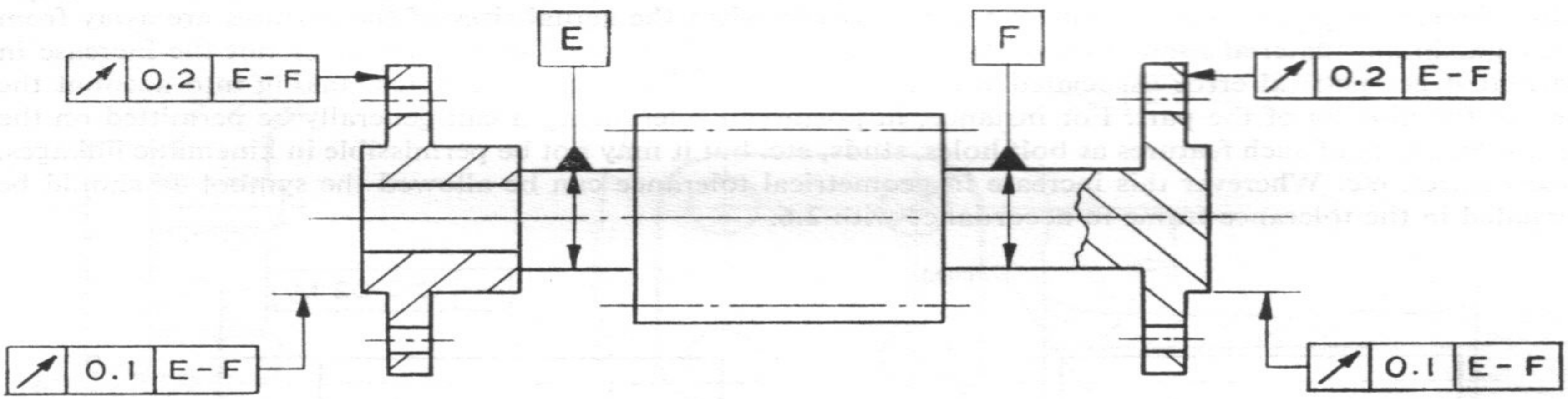
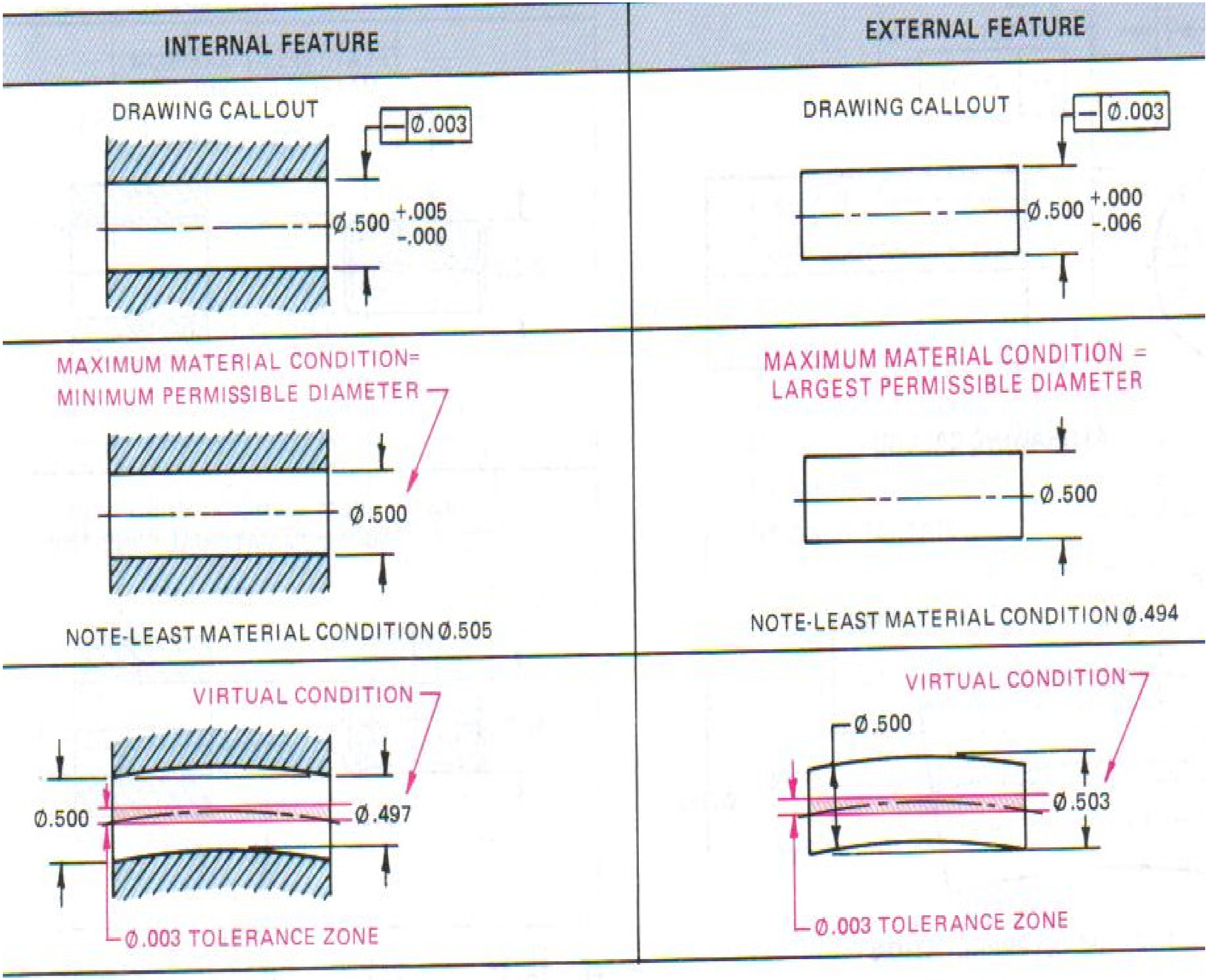
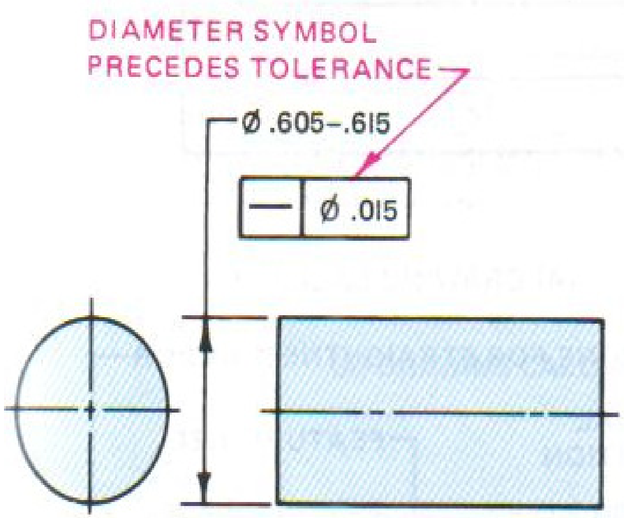


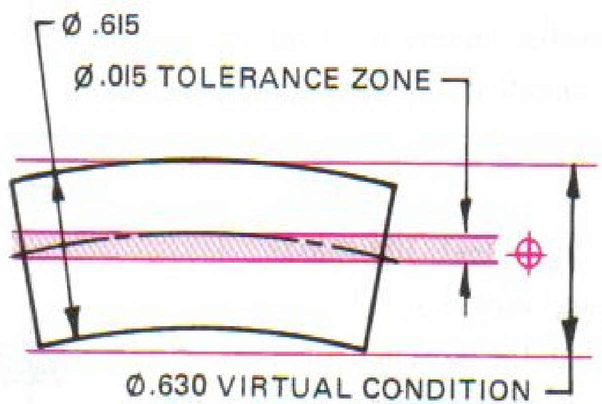

Fig. Run-out tolerance applied to an assembly




Maximum material and virtual conditions.



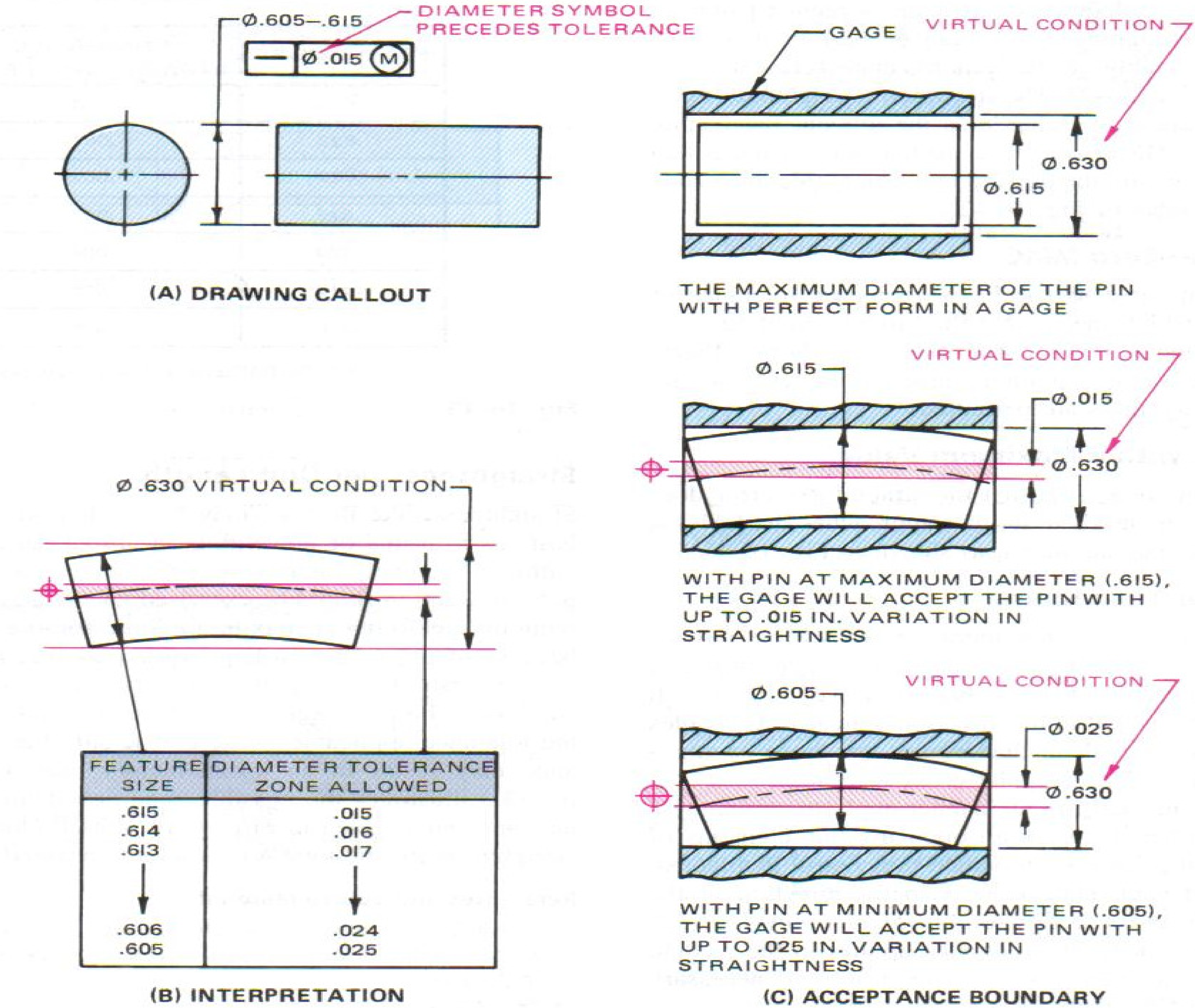

(A) DRAWING CALLOUT




| FEATURE SIZE | DIAMETER TOLERANCE ZONE ALLOWED |
|---|---|
| .615 | .015 |
| .614 | .015 |
| .613 | .015 |
| ↓ | ↓ |
| .606 | .015 |
| .605 | .015 |

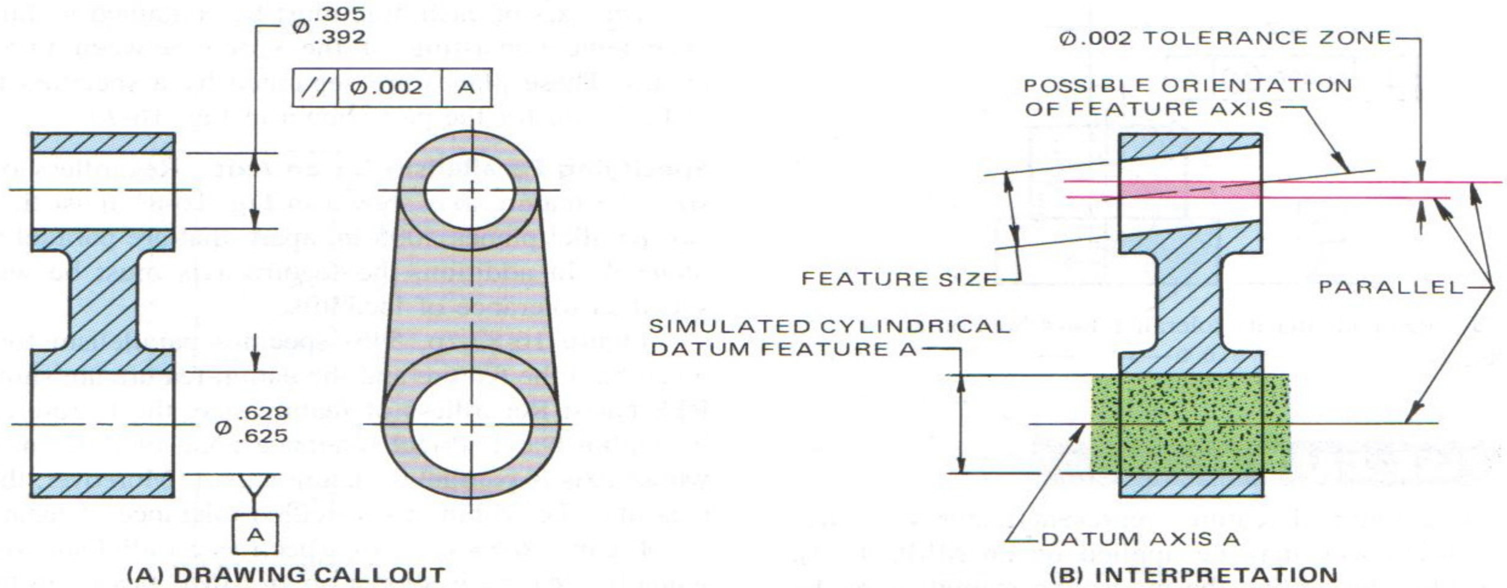
(B) INTERPRETATION

Specifying straightness—RFS.

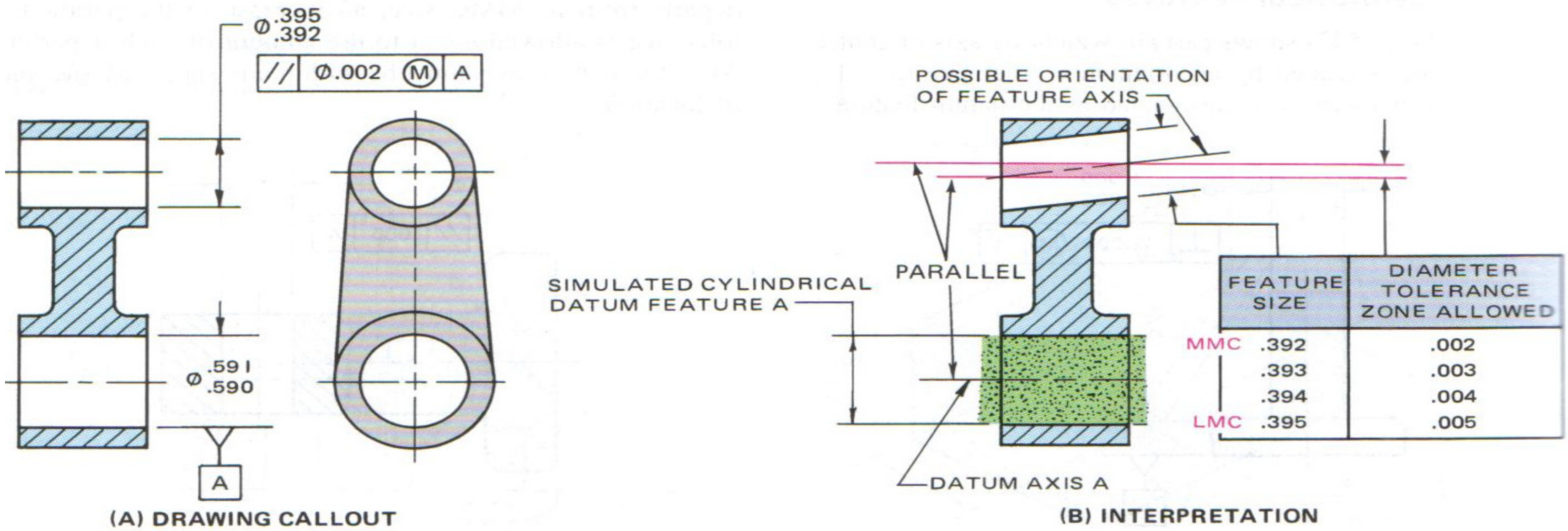



| FEATURE SIZE | DIAMETER TOLERANCE ZONE ALLOWED |
|---|---|
| .615 | .015 |
| .614 | .016 |
| .613 | .017 |
| ↓ | ↓ |
| .606 | .024 |
| .605 | .025 |

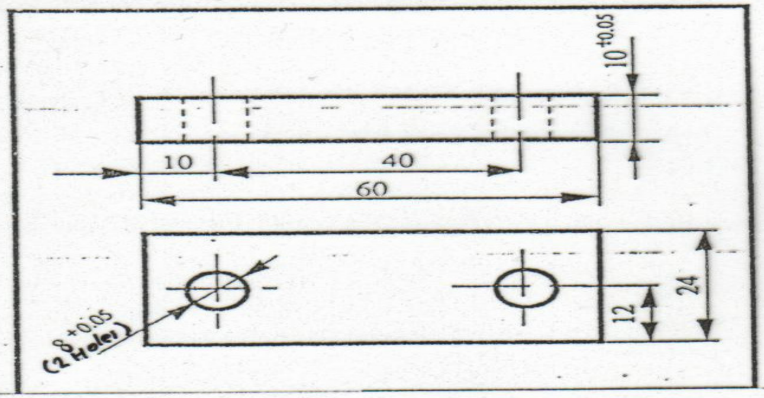
Specifying straightness—MMC.

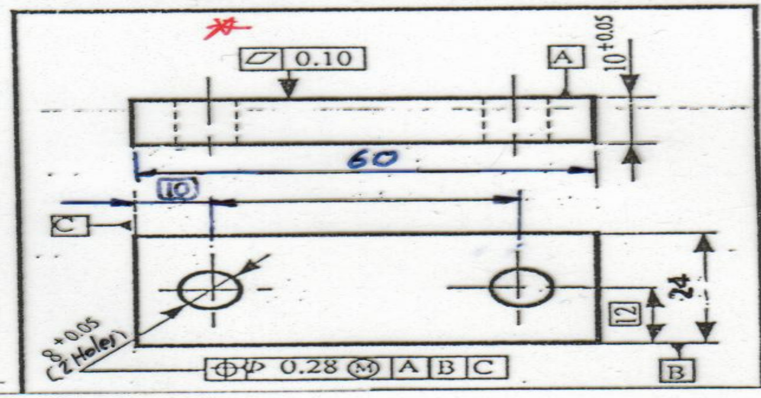



Specifying parallelism for an axis (both feature and datum feature RFS).




Specifying parallelism for an axis (feature at MMC and datum feature RFS).




If dia. = 8.00 material becomes maximum. [mm]

the given position tolerance is to be satisfied in this case

If the actual size is greater than the minimum limit, the position tolerance must be determined as follows:

eg. actual size 8.03

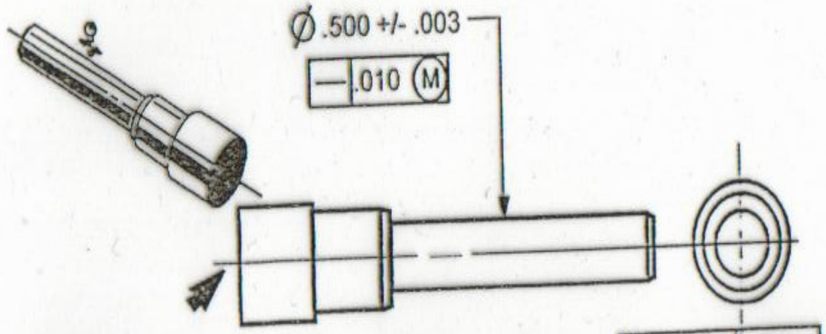
$$|8.03 - 8.00| = 0.03$$

tolerance : $0.28 + 0.03 = 0.31$ [mm]

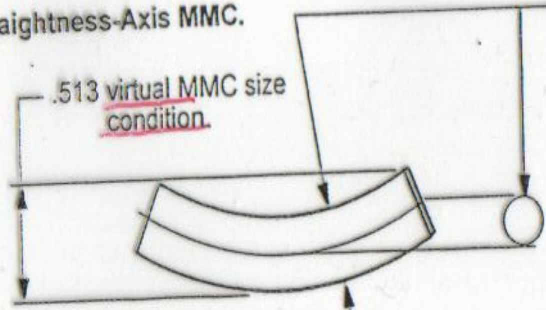
eg. actual size 8.05

$$|8.05 - 8.00| = 0.05$$

tolerance : $0.28 + 0.05 = 0.33$ [mm]

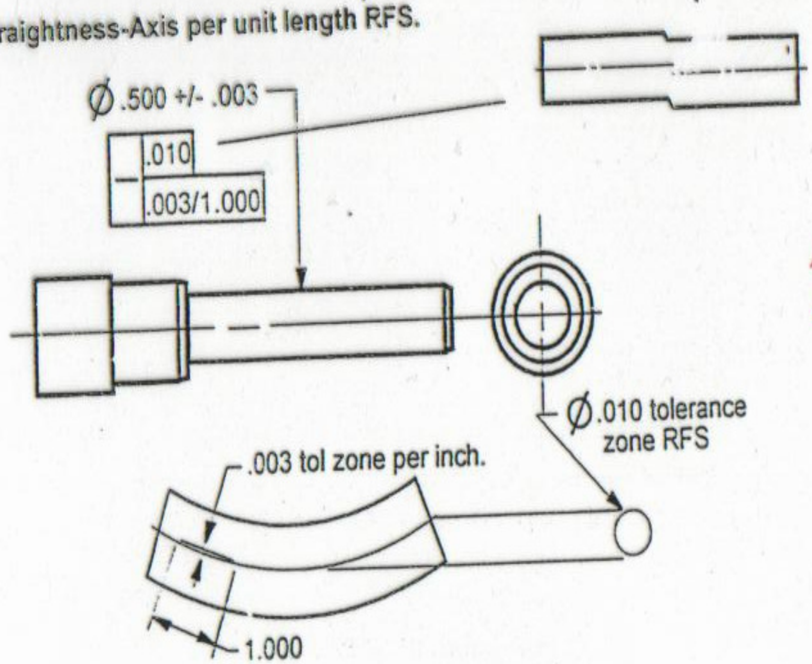



Straightness-Axis MMC.




| Size | Tol Zone |
|---|---|
| .503 MMC | .010 |
| .502 | .011 |
| .501 | .012 |
| .500 | .013 |
| .499 | .014 |
| .497 | .016 |

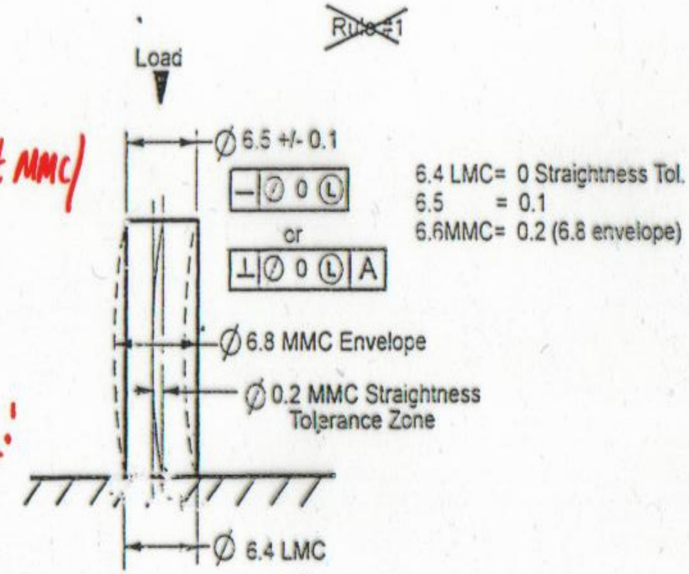
Straightness-Axis per unit length RFS.




Straightness - axis MMC and per unit length.

|Actual size − Size at MMC| + Geo. tol.

Virtual condition: is the boundary generated by the collective effects of MMC (or LMC) size and geo. tol.

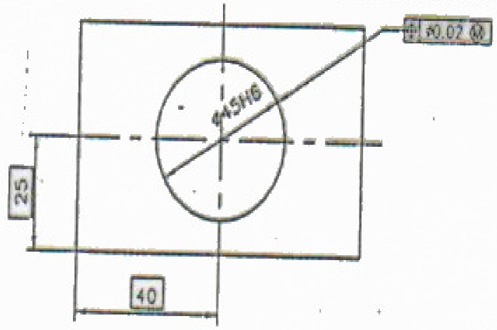



Note: Where a straightness tolerance is used in conjunction with an orientation or position tolerance, the straightness tolerance value shall be no greater than the specified orientation (or position) tolerance value.

Straightness applied LMC.

EX: a) Measured diameter of the hole is 45.010. Determine the position tolerance

b). Determine the maximum value of the position tolerance permitted



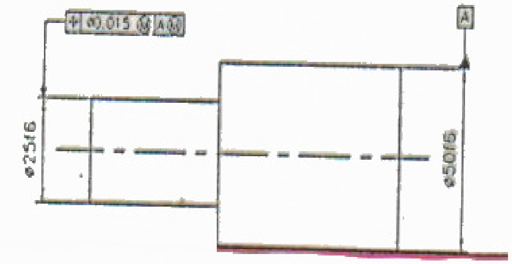

a). $\phi 45\,H6 = \phi 45^{0.016}_{0}$

when the hole has been produced to minimum diameter, 45.000 [mm], the object will have maximum amount of material

$|45.010 - 45.000| = 0.01$ [mm] is to be added to the given position tolerance for the measured size of the hole the corresponding tolerance $= 0.02 + 0.01 = 0.03$ [mm]

b). The maximum value of the position tolerance is to be applied when the object has been produced to minimum amount of material (i.e. when the hole is in its maximum size) Therefore $tolerance_{max} = 0.02 + |45.016 - 45.000| = 0.02 + 0.016 = 0.036$ [mm]

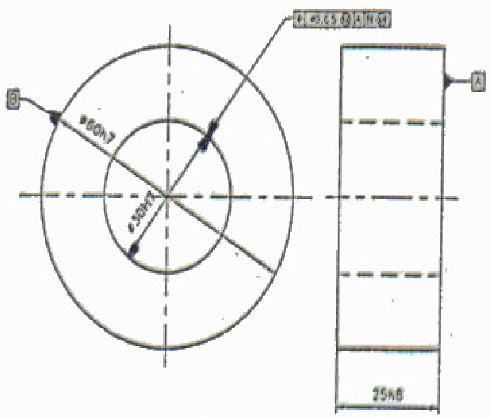
Ex: If the measured diameters are 26.970 & 49.965, determine the position tolerance to be applied.



Solution $\phi 25f6 = \phi 25^{-0.020}_{-0.033} \Rightarrow \begin{matrix} 24.980 \\ 24.967 \end{matrix}$ , $\phi 50f6 = \phi 50^{-0.025}_{-0.041} \Rightarrow \begin{matrix} 49.975 \\ 49.959 \end{matrix}$

Position Tolerance $= 0.015 + |24.970 - 24.980| + |49.965 - 49.975| = 0.035$ (mm)

Ex:



Determine the maximum value of the position tolerance which may be in consideration.

Solution: $\phi 30H7 = \phi 30^{0.021}_{0.000} \Rightarrow \begin{matrix} 30.021 \\ 30.000 \end{matrix}$

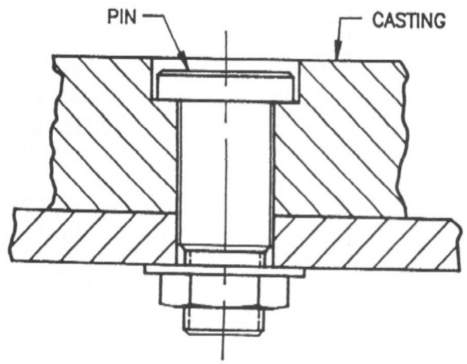
$\phi 60h7 = \phi 60^{0.000}_{-0.030} \Rightarrow \begin{matrix} 60.000 \\ 59.970 \end{matrix}$

max. value of the position tolerance,

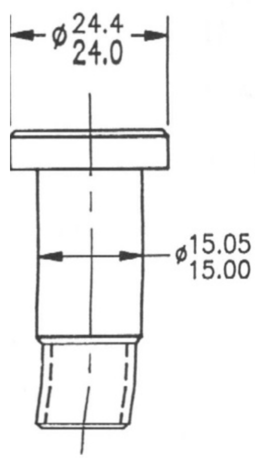
$= 0.05 + |30.021 - 30.000| + |59.970 - 60.000| = 0.101$

# • Coaxiality
# • Mating Part Design

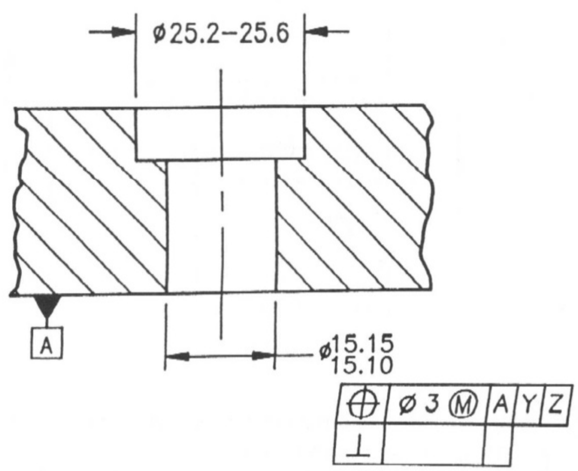
Assembly




Detail drawing callouts



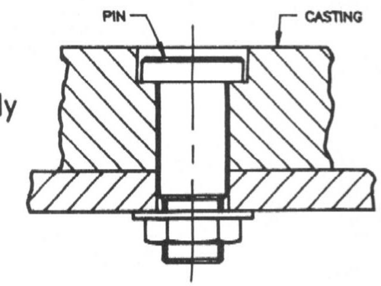

PIN



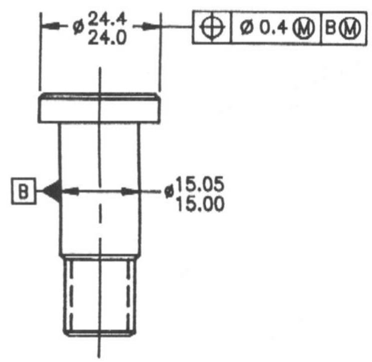

CASTING

a) Assign appropriate coaxiality controls to assure these two parts will mate at assembly.
b) Fill in the perpendicularity control.

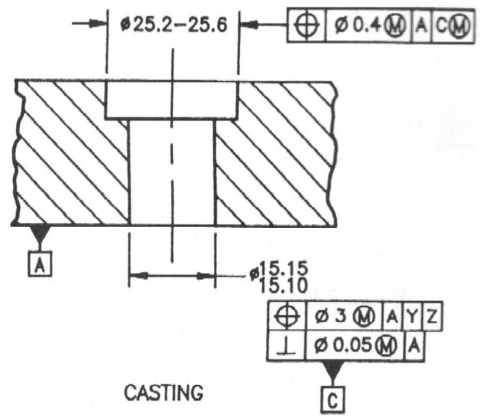
Assembly




## Example 1: Direct Tolerancing on Casting



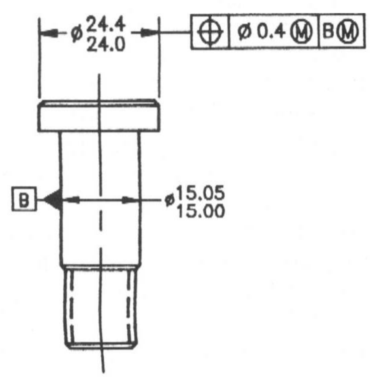

PIN



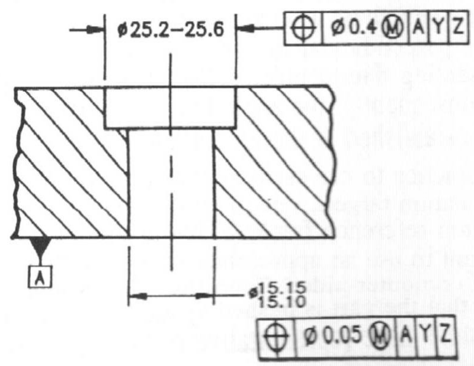

CASTING

VS

## Example 2: Indirect Tolerancing on Casting




PIN



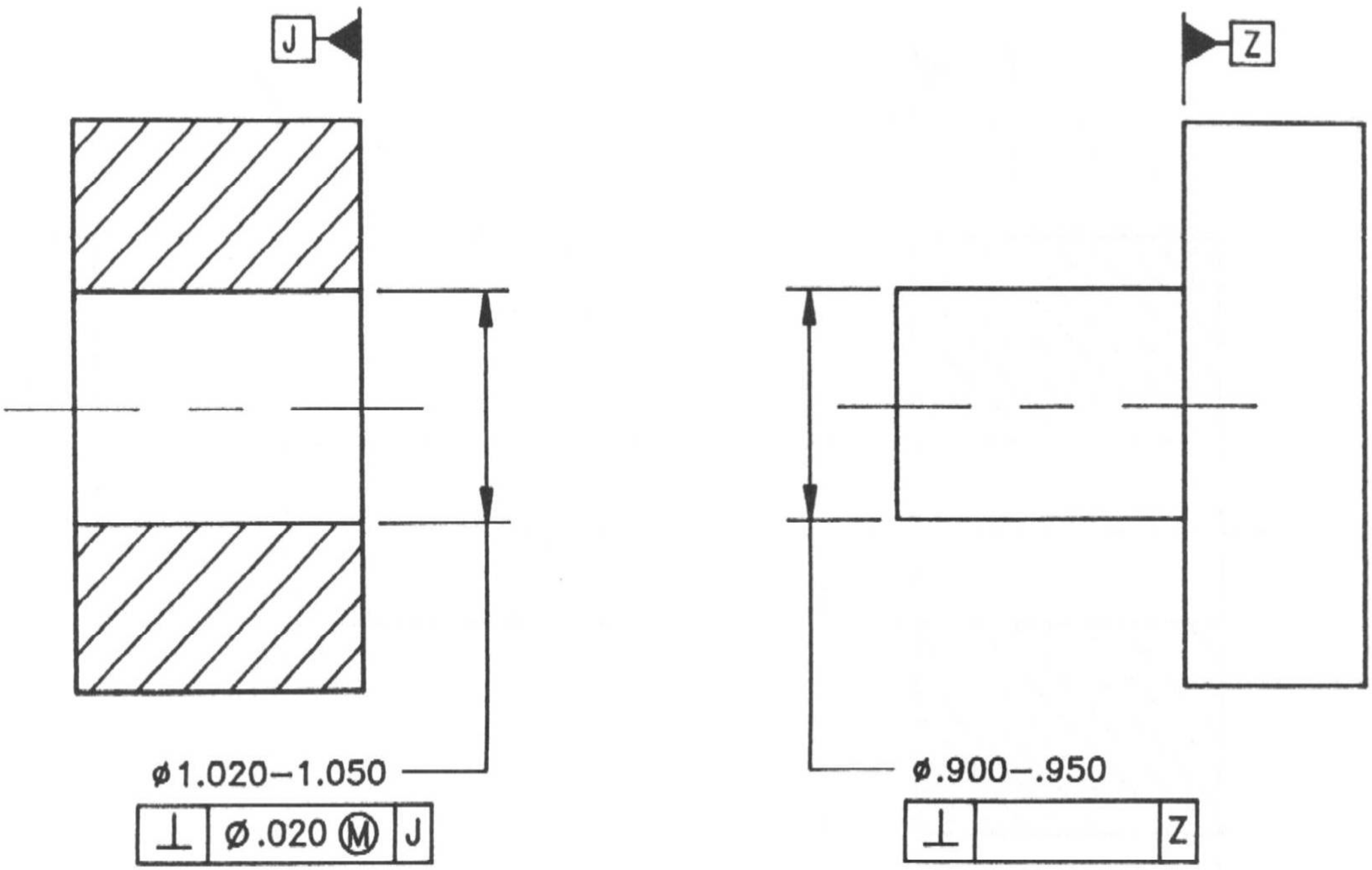

CASTING

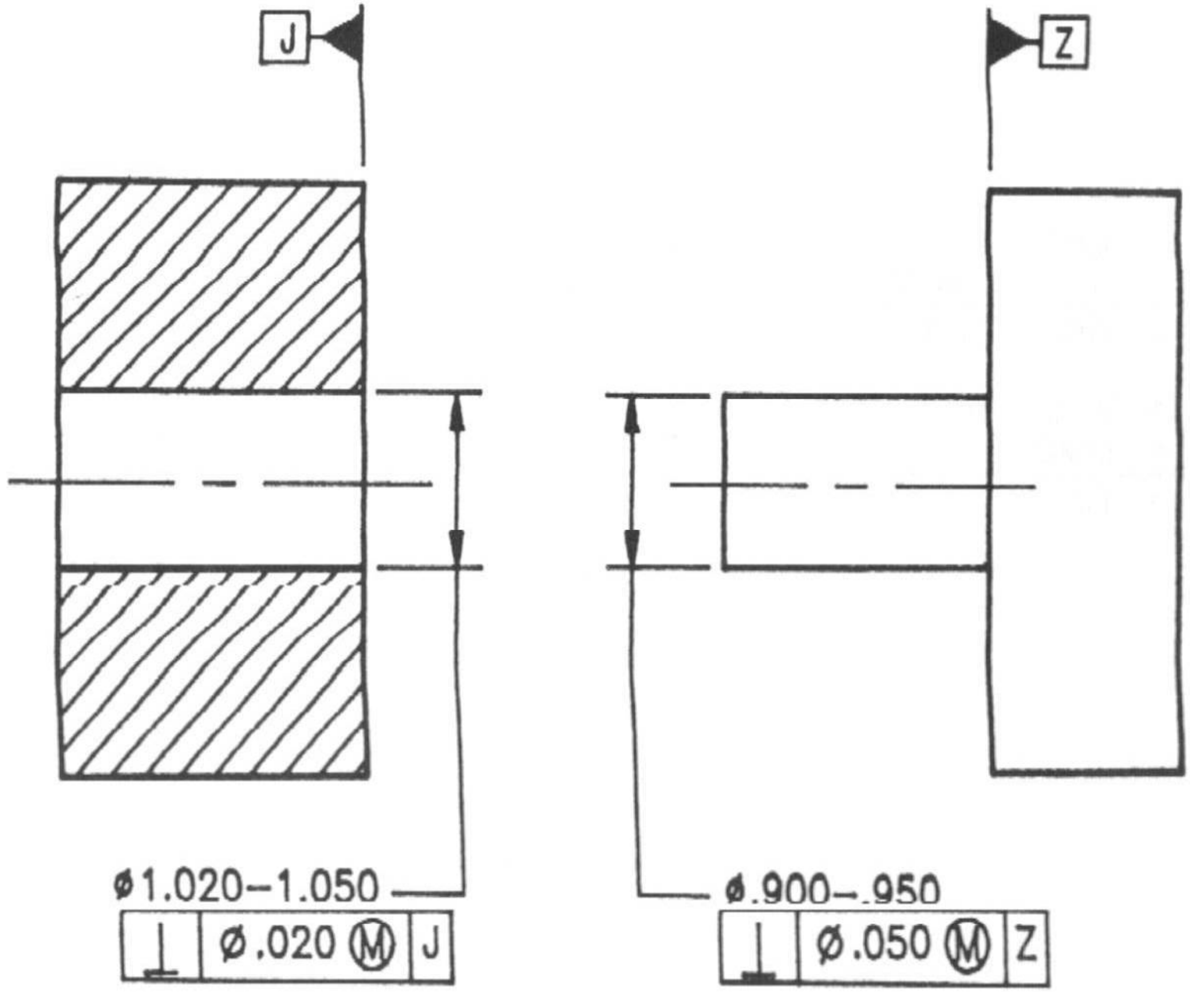
Fill in the rest of the shaft's feature control frame to assure:

a) that it will mate with the hole.

b) the most cost-effective design.

## Mating Part Design






ø1.020 = MMC
− .020 = Tol. at MMC
1.000 = Virt. Cond. hole

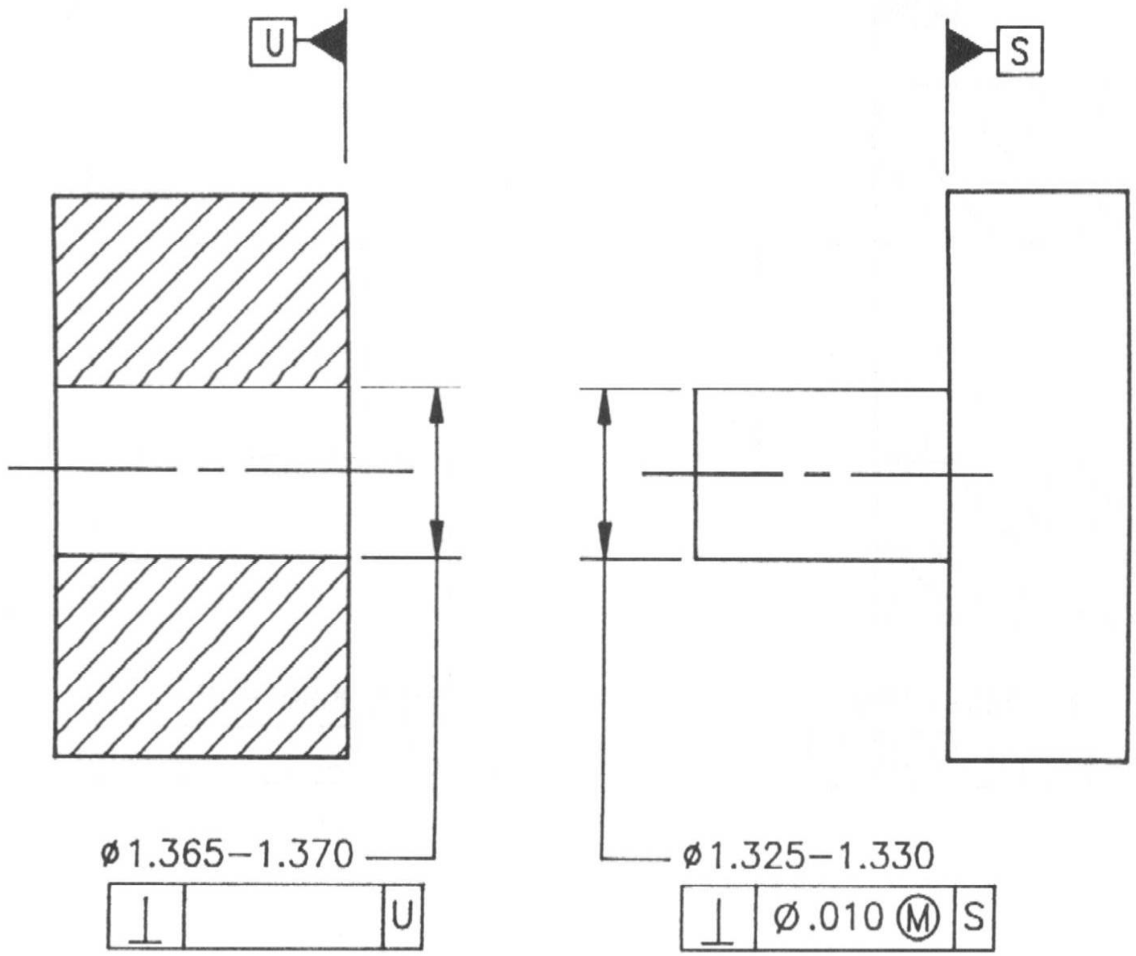
ø1.000 = Virt. Cond. hole
− .950 = Shaft MMC
ø.050 = Tol.

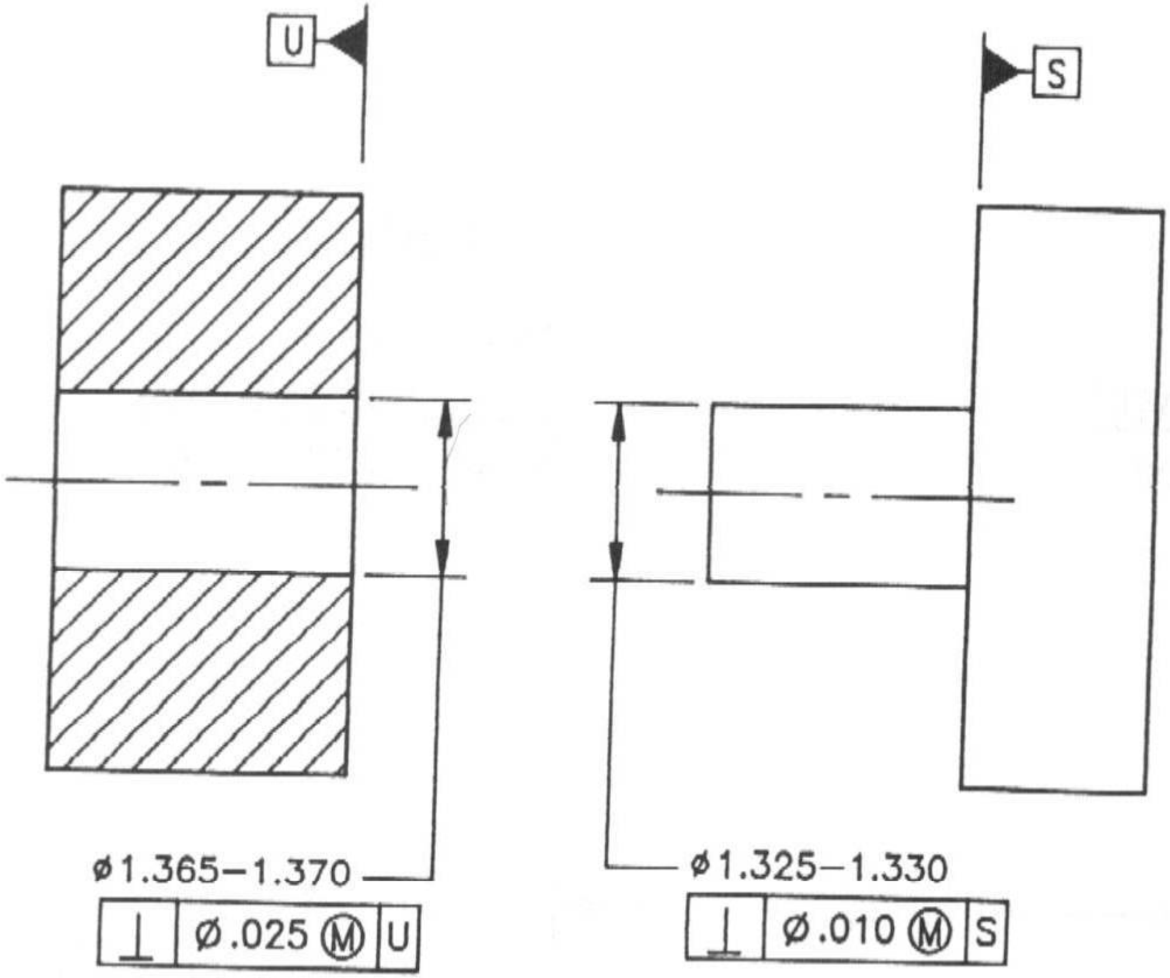
Using the illustration below, fill in the rest of the hole's feature control frame to assure:

a) that it will mate with the shaft.

b) the most cost-effective design.

## Mating Part Design





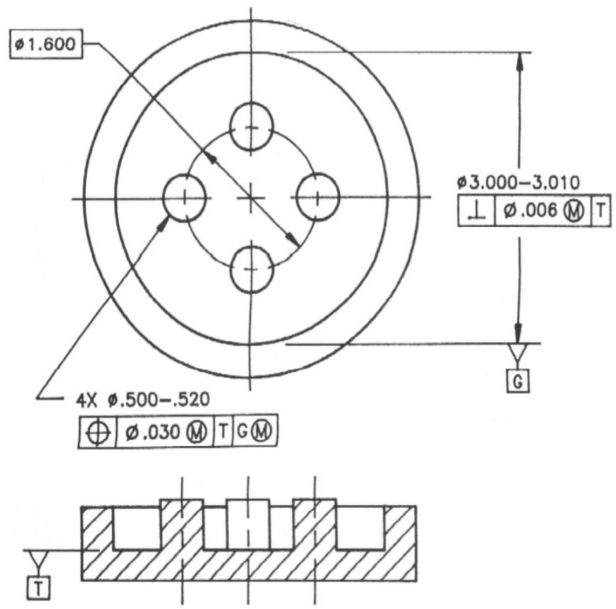

1.330 = MMC
+ .010 = Tol. at MMC
1.340 = Virt. Cond. Shaft

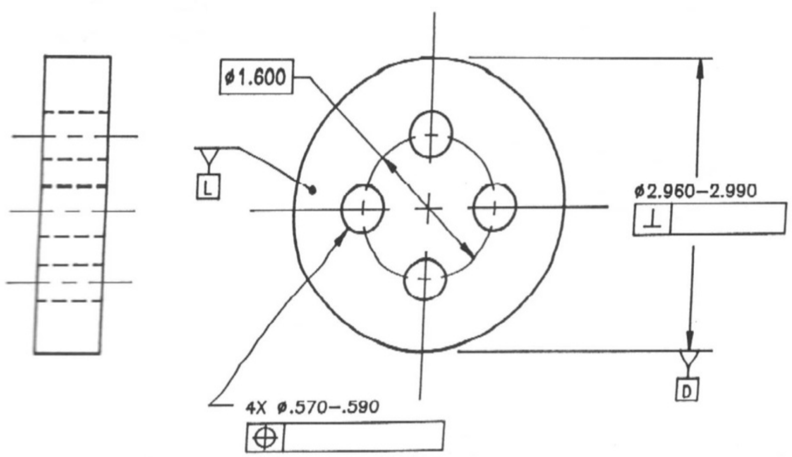
1.365 = Hole MMC
− 1.340 = Virt. Cond. Shaft
ø.025 = Tol.

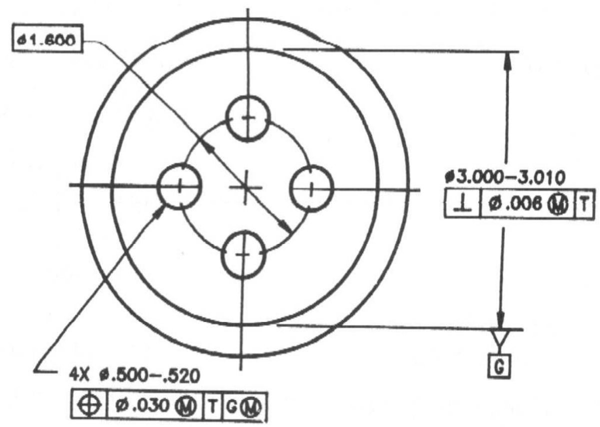
Using the illustration below, fill in the feature control frame so:

a. the hockey puck (below the thick horizontal line) fits down inside of the cylindrical cavity (above the thick horizontal line).
b. the holes in the hockey puck fit over the pins in the cavity.
c. load is not born by the pins but rather by L and T.

## Mating Part Design

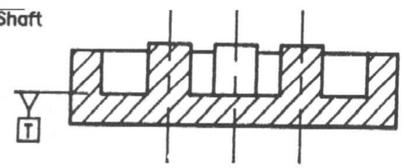


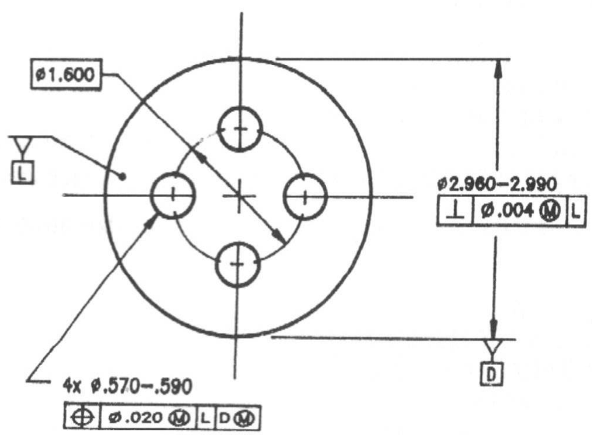






ø.520 = MMC
+.030 = Tol. at MMC
.550 = Virt. Cond. Shaft

ø3.000 = MMC
− .006 = Tol. at MMC
ø2.994 = Virt. Cond. Hole



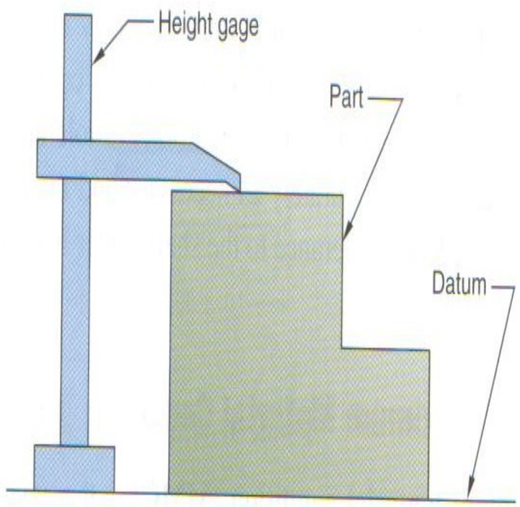




ø.570 = MMC
−.550 = Virt. Cond. Shaft
ø.020 = Tol.

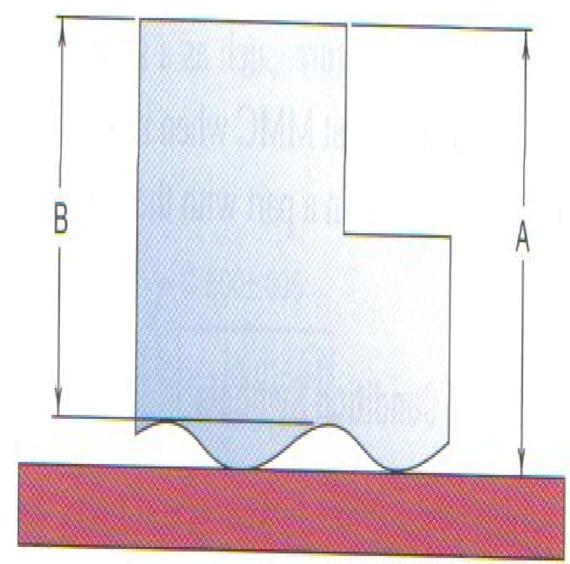
ø2.994 = Virt. Cond. Hole
− 2.990 = MMC
ø.004 = Tol.

# Datums and Datum Features

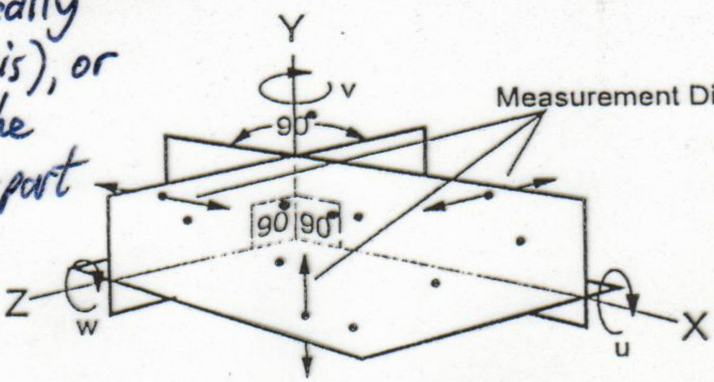
A **datum** is a starting place for a dimension. A datum may be a perfect plane, a center line, or a point. Datums are perfect, and they are not real. Examples are the center line of a shaft or the point at the center of a sphere. These are theoretical positions that either can be represented with inspection tools or can be derived. For example, a center line is represented by the center of an inspection pin or gage or by the center of an inspection spindle. A center line is derived by measuring to the edge of a gage pin and then adding half the diameter of the pin to locate the center of the gage pin from an edge or another feature. For a hole, the measurement is *not* to the edge of the feature hole but to the largest-gage pin that will fit into the hole.




The bottom surface of the part is the datum feature, surface plate is the datum.

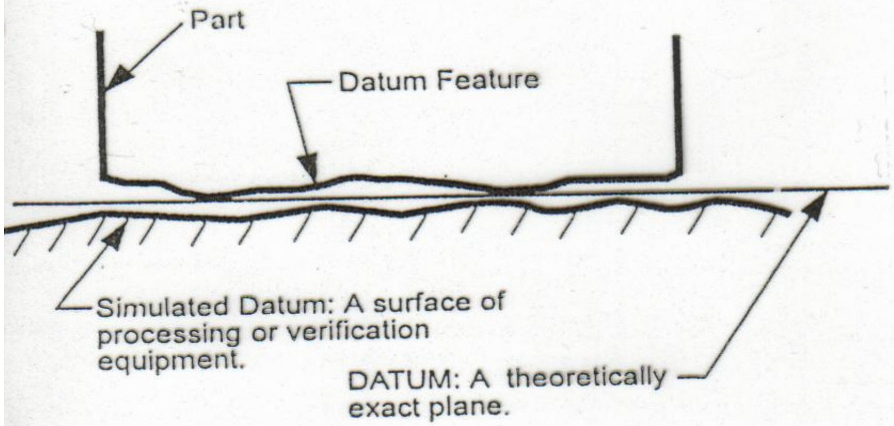


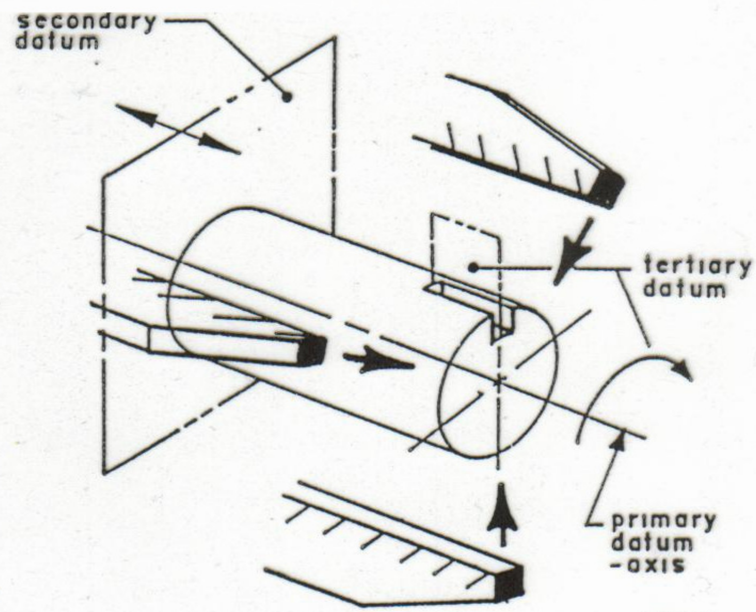
Datum is a theoretically exact point, line (axis), or plane derived from the true geometric counterpart of a specified datum feature.




Datum Reference Framework. Three Mutually Perpendicular Planes.

| | | A | B | C |
|---|---|---|---|---|




**Datum**

Theoretically perfect points, lines, or planes

**Simulated Datum (Datum Simulators)**

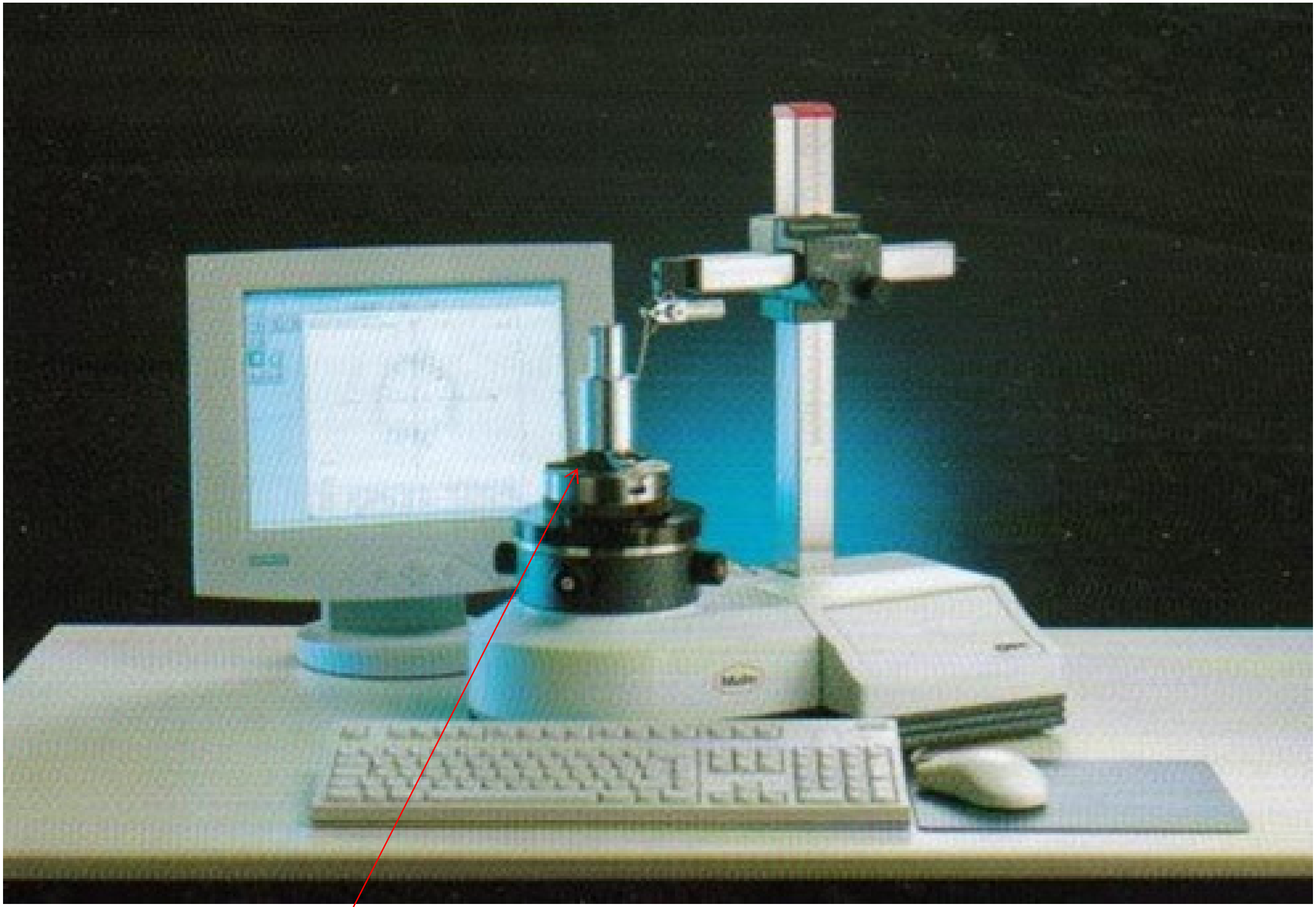
Surfaces and axes of processing or inspection equipment

**Datum Feature**

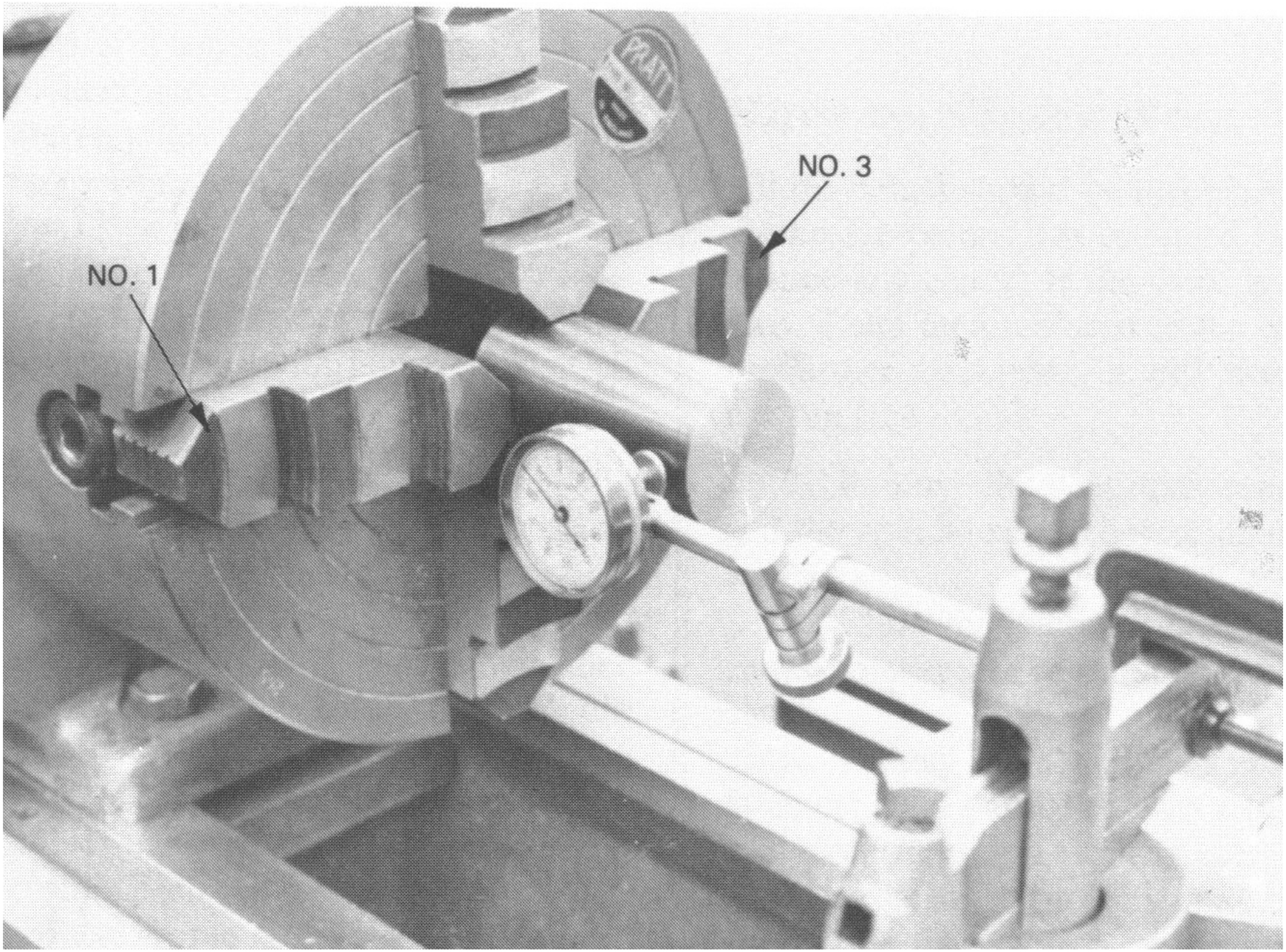
Actual part feature surfaces

**Temporary Datum**

Introduced for processing or inspection purposes (may be removed)

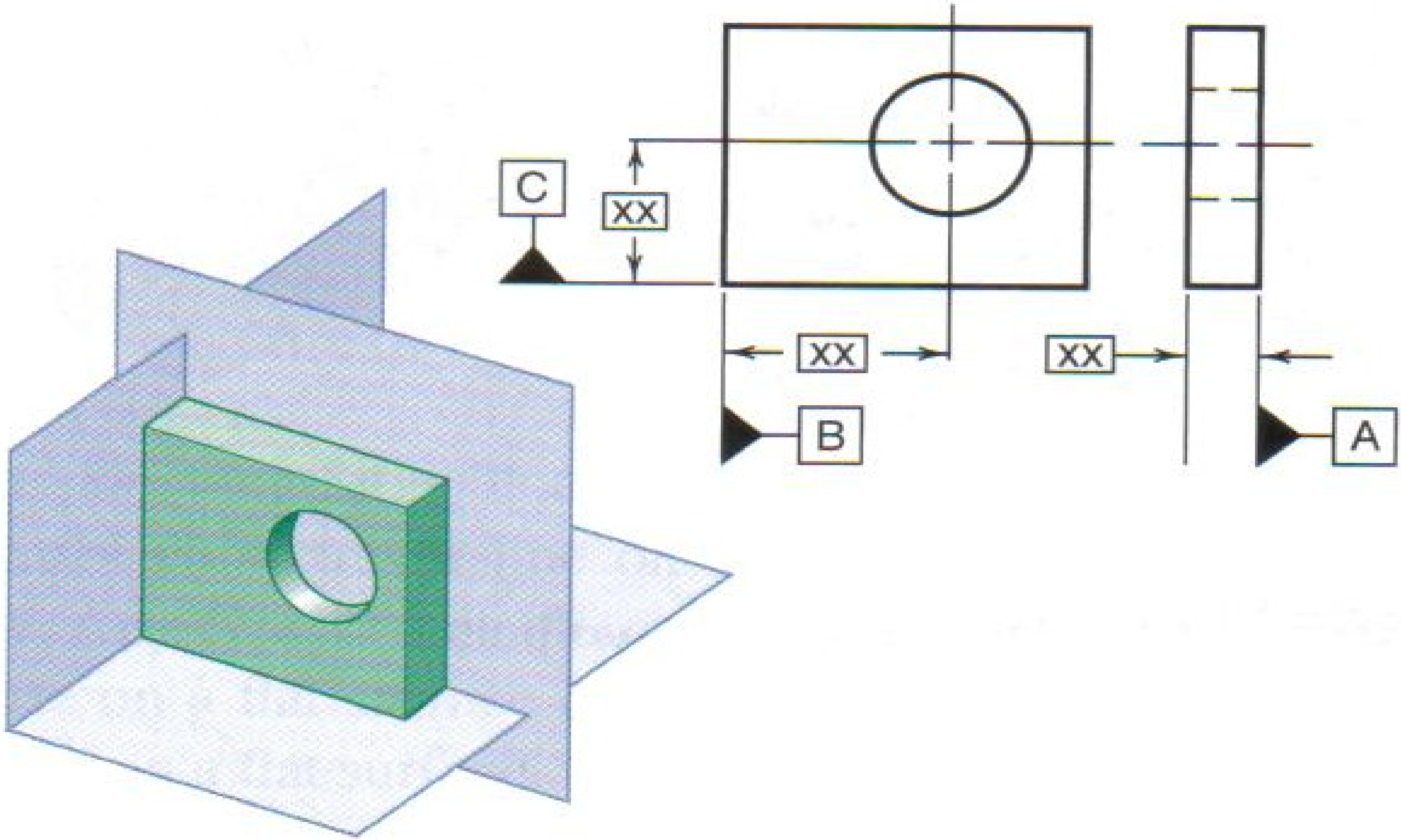


Datum simulator (jaws of the chuck)

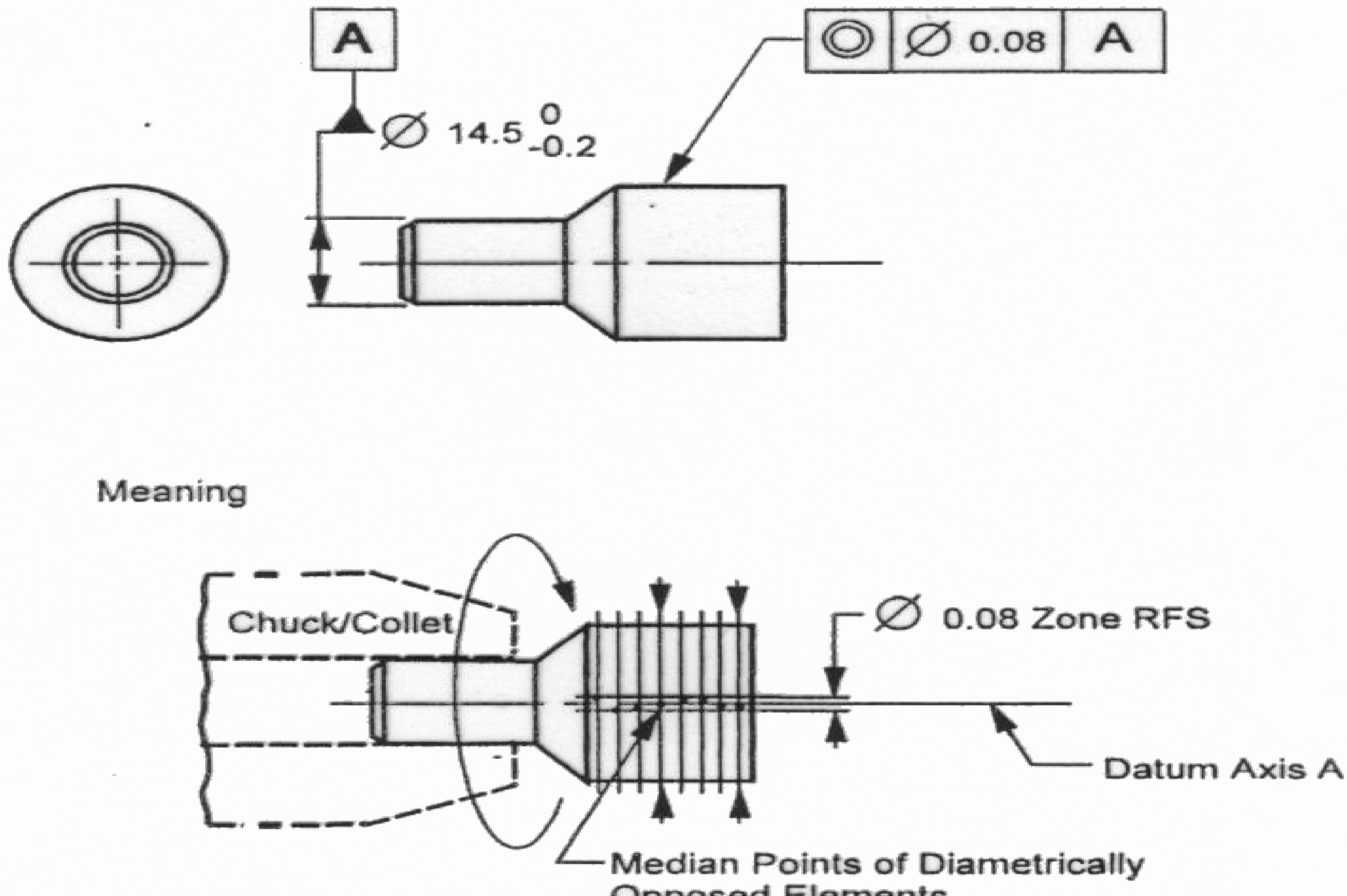

NO. 1
NO. 3

# Datum Selection

- *Function and functional relationships*: This is the most critical issue, impacting not only the design but the manufacturing and quality plans as well.
- *Reality*: Datums should be real, identifiable, and verifiable. Imaginary features, points in space, or features that are impossible (or at least difficult) to locate are acceptable in a lab or mathematical environment, but are difficult for the manufacturing world to deal with. Equally difficult are datum features that are inaccessible or hidden within the product.
- *Accuracy*: Datums should be accurate and should offer the best repeatability, because datum error can impact assessment and verification of other features. If a datum feature is within its specification control (flatness for example), it becomes theoretically perfect (or zero) for measurement of other features.

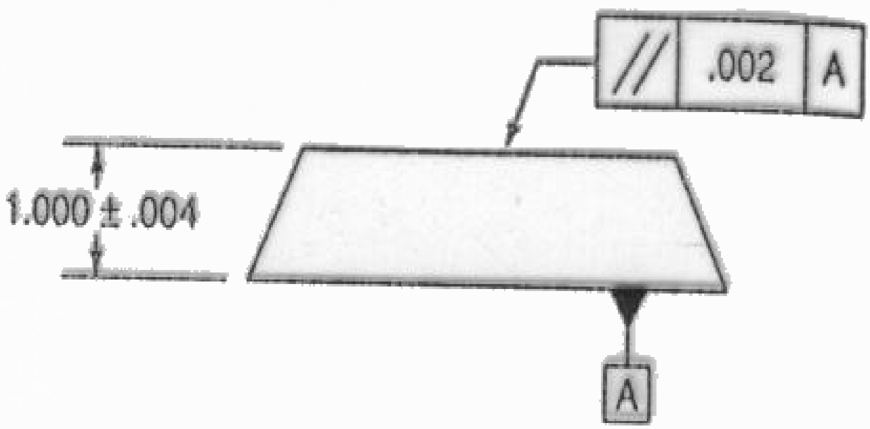


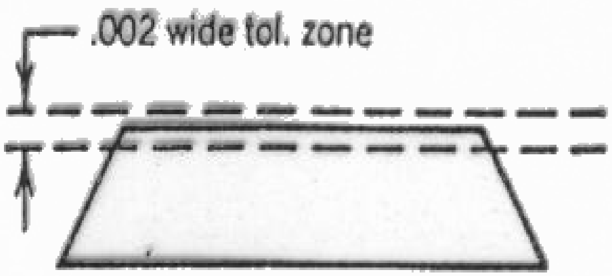

**Three-plane datum reference plane**



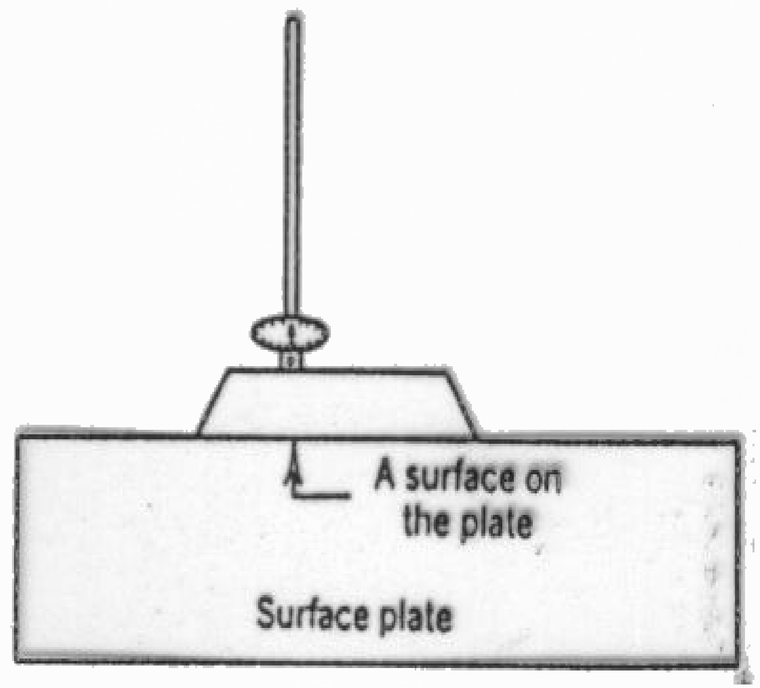

Concentricity.

Parallelism callout.



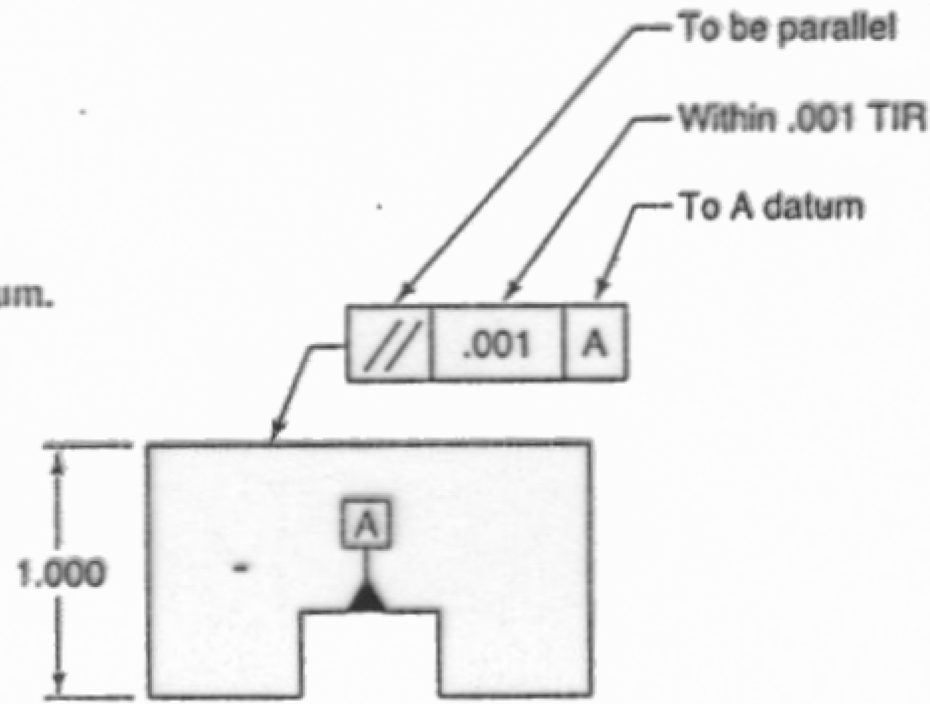




Parallelism tolerance zone.

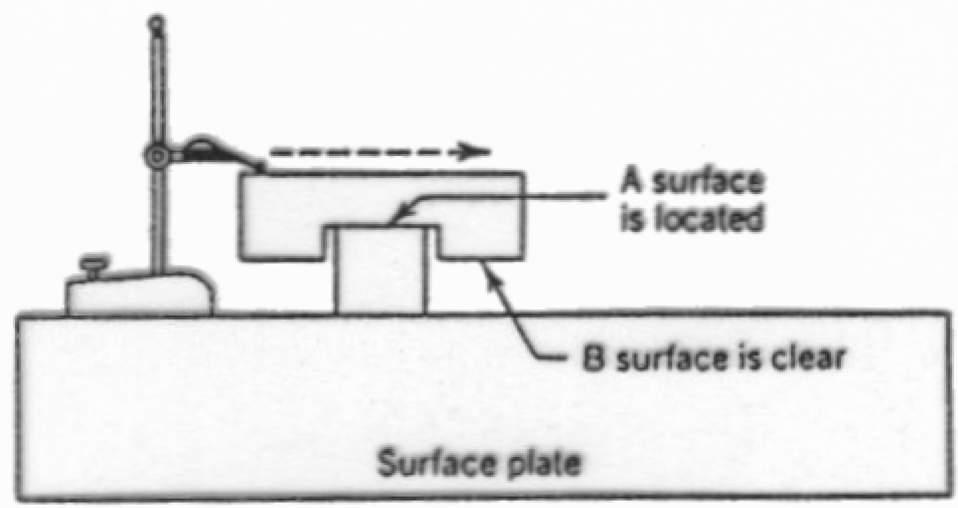


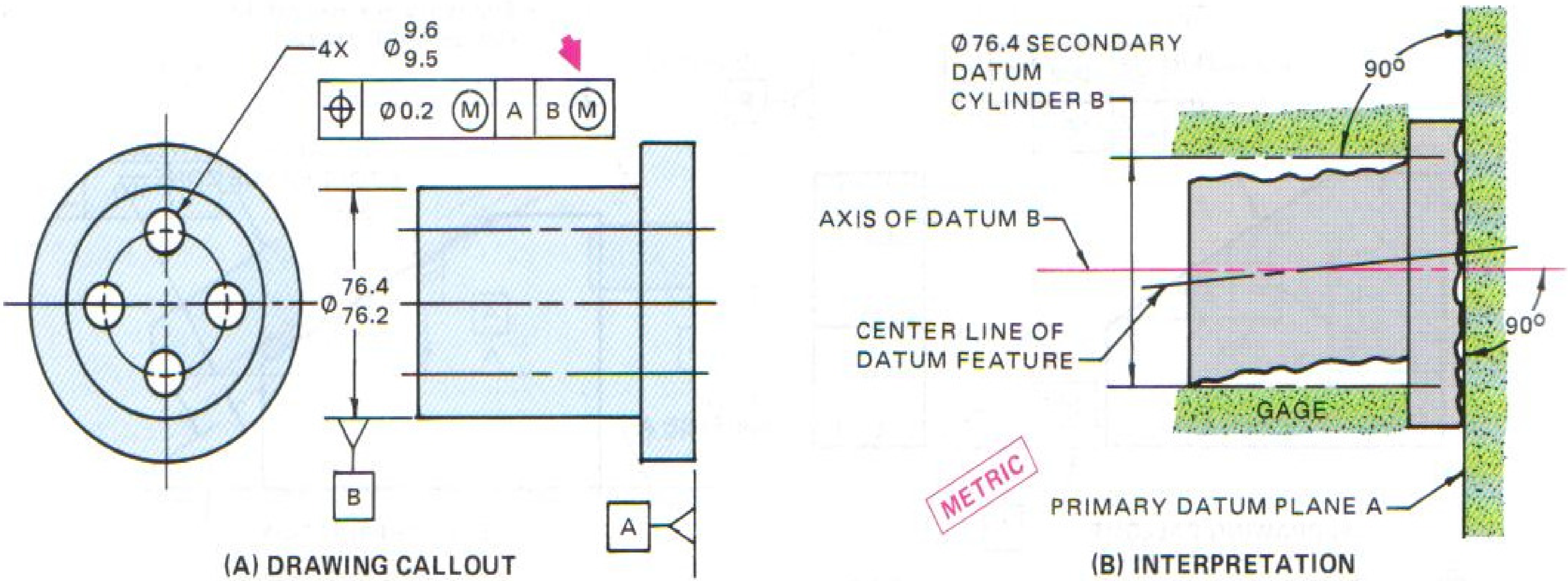

Datum is located.

Parallelism to a hidden datum.

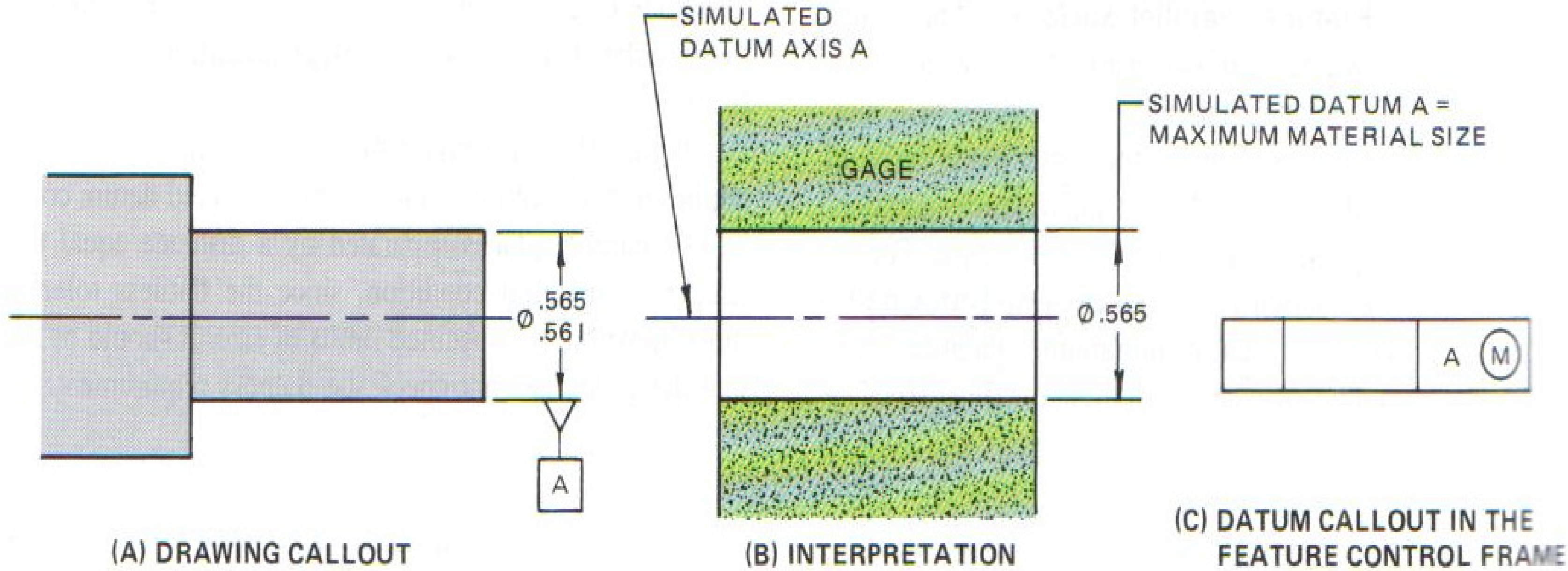



Fixed parallel locates the hidden datum.

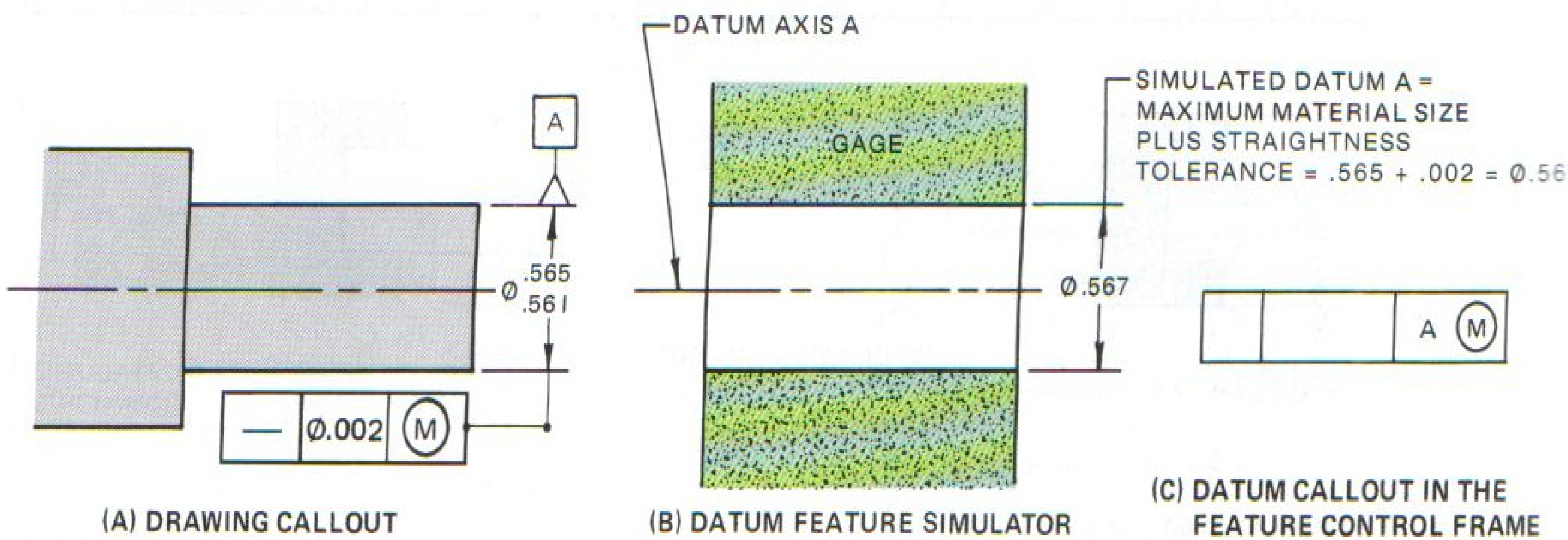




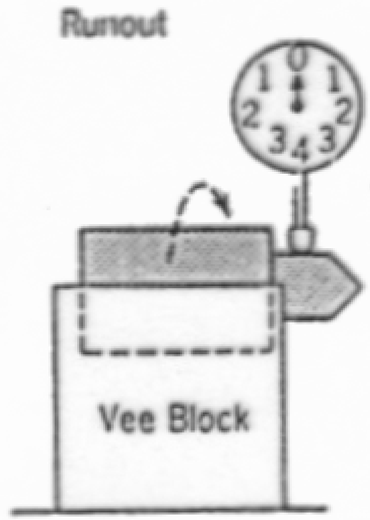

Cylindrical feature as secondary datum.



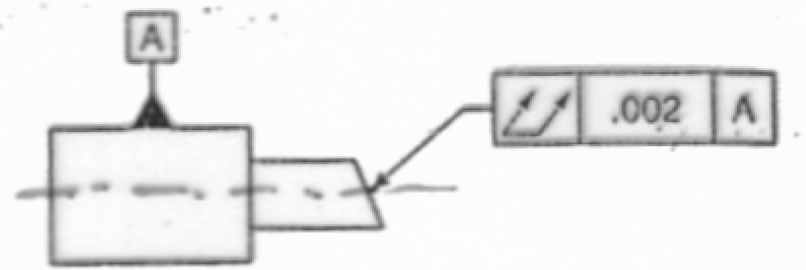

External primary datum without form tolerances—MMC.




External primary datum with straightness tolerance—MMC.

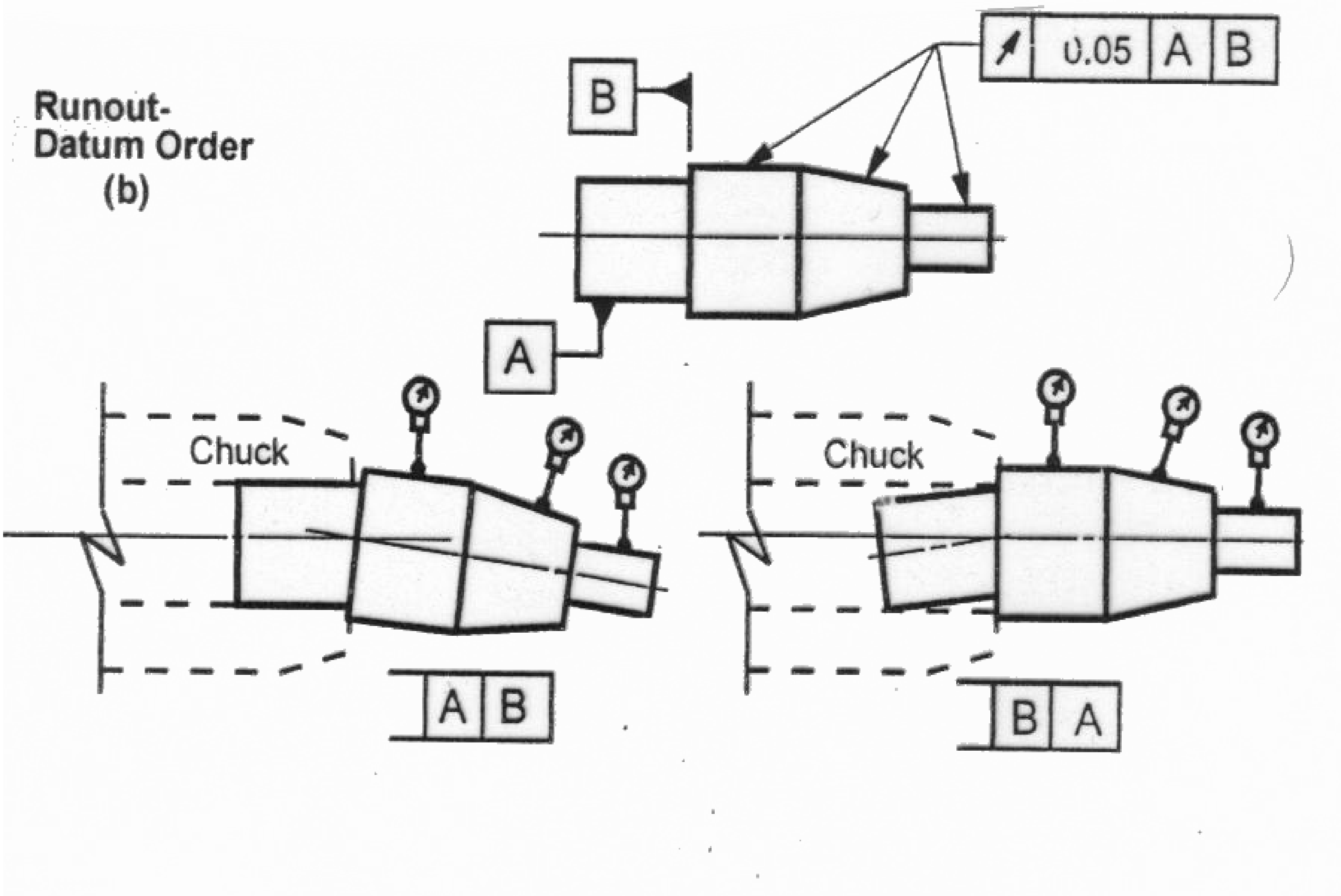



Measuring circular runout.

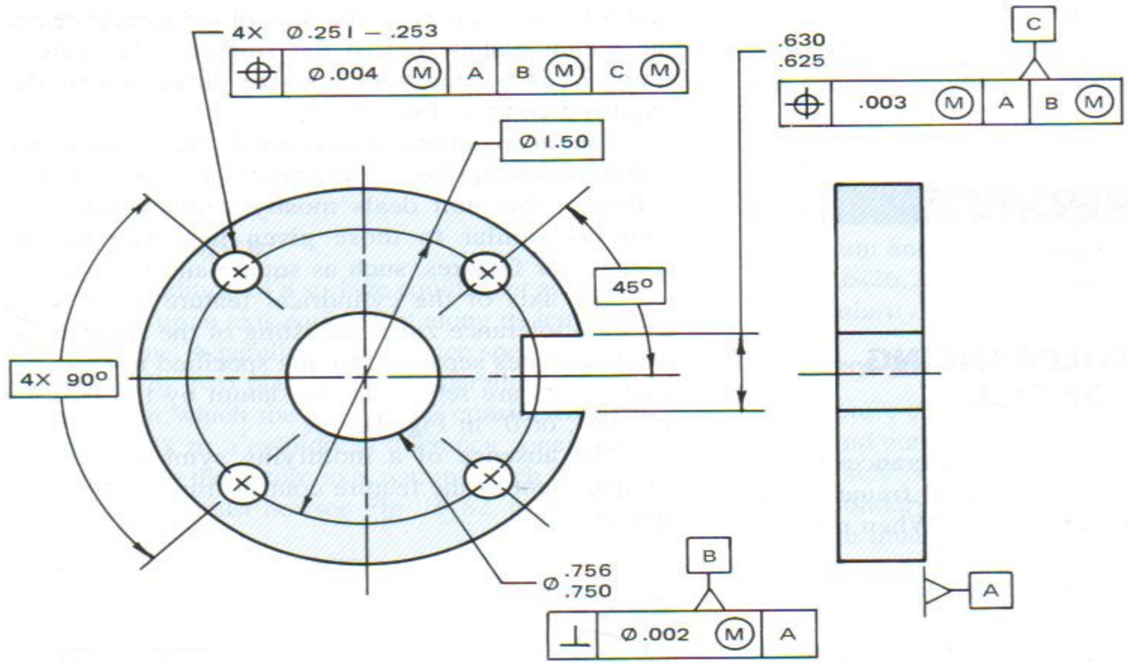



End surface total runout.

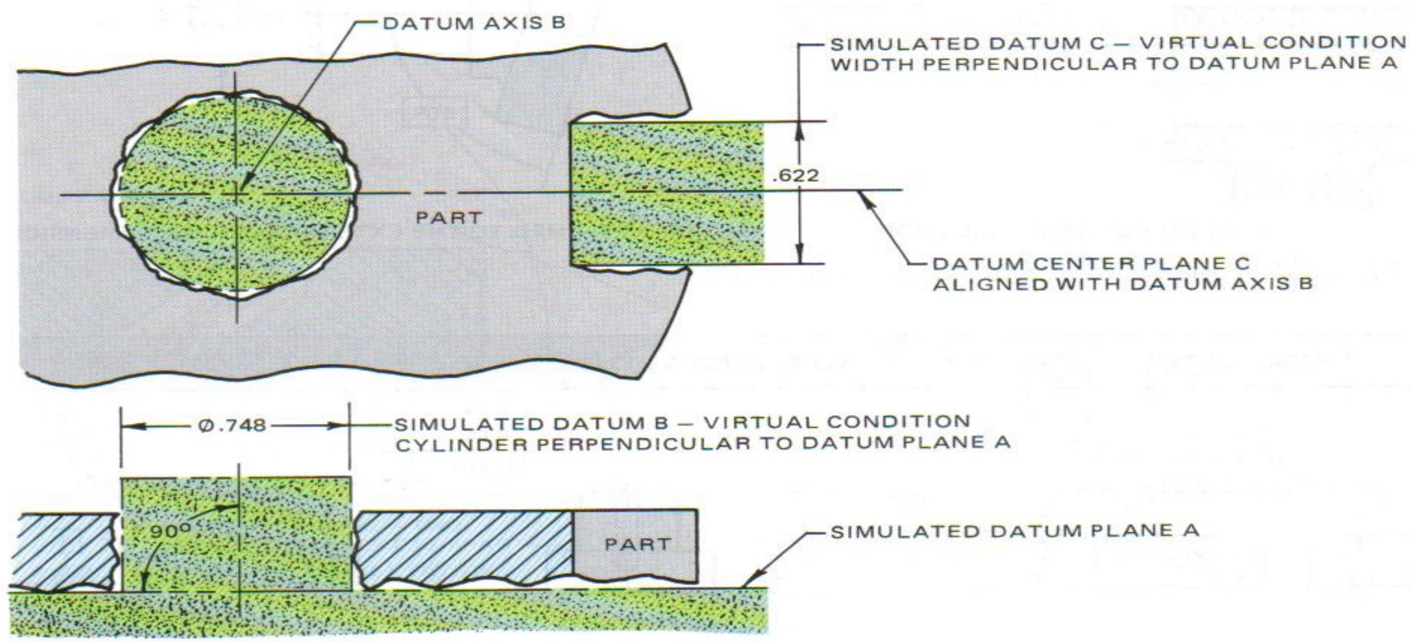
Steps for Total Runout Measurement



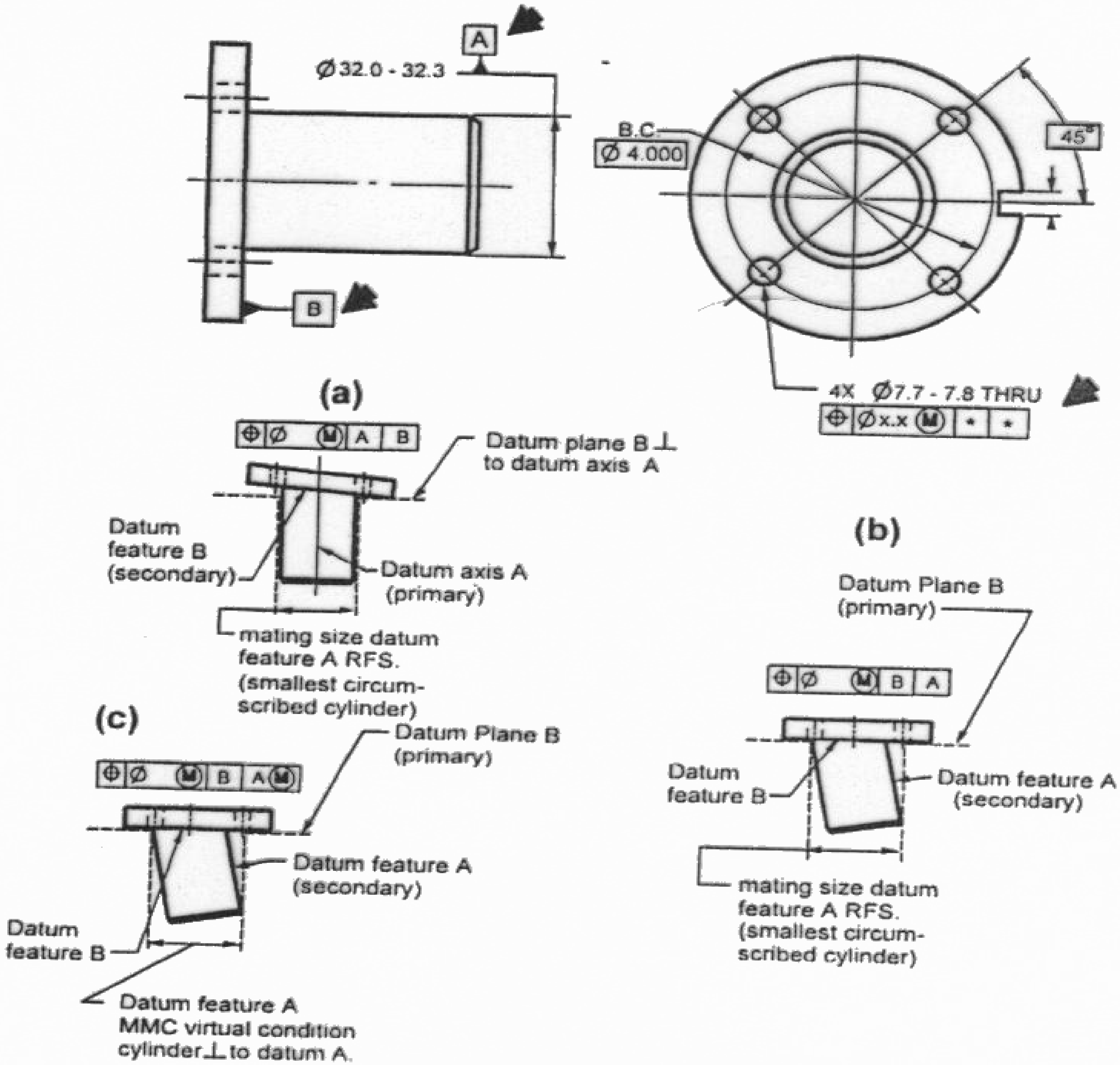
Circular runout and datum order.




(A) DRAWING CALLOUT

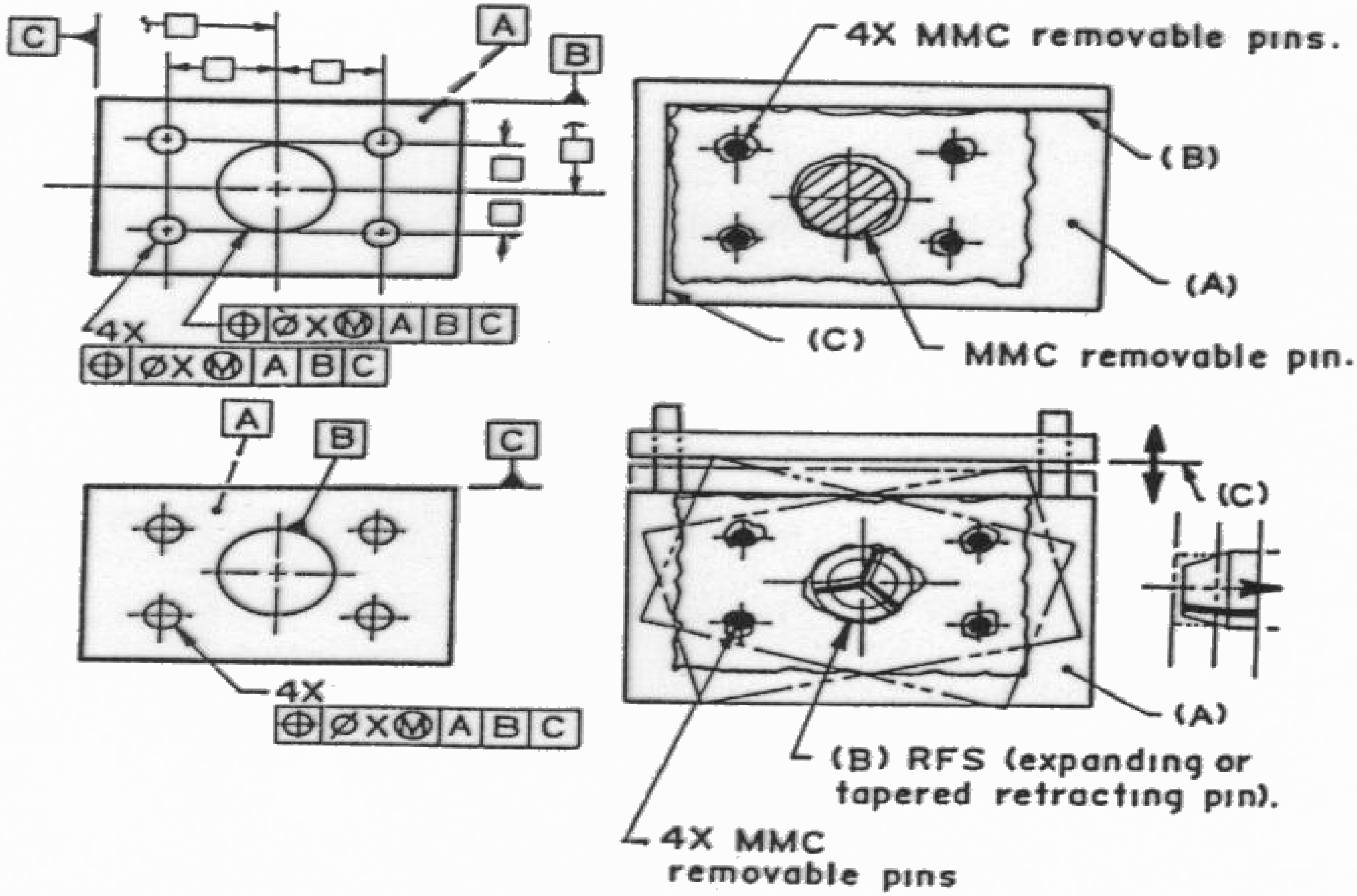



(B) DATUM FEATURE SIMULATORS

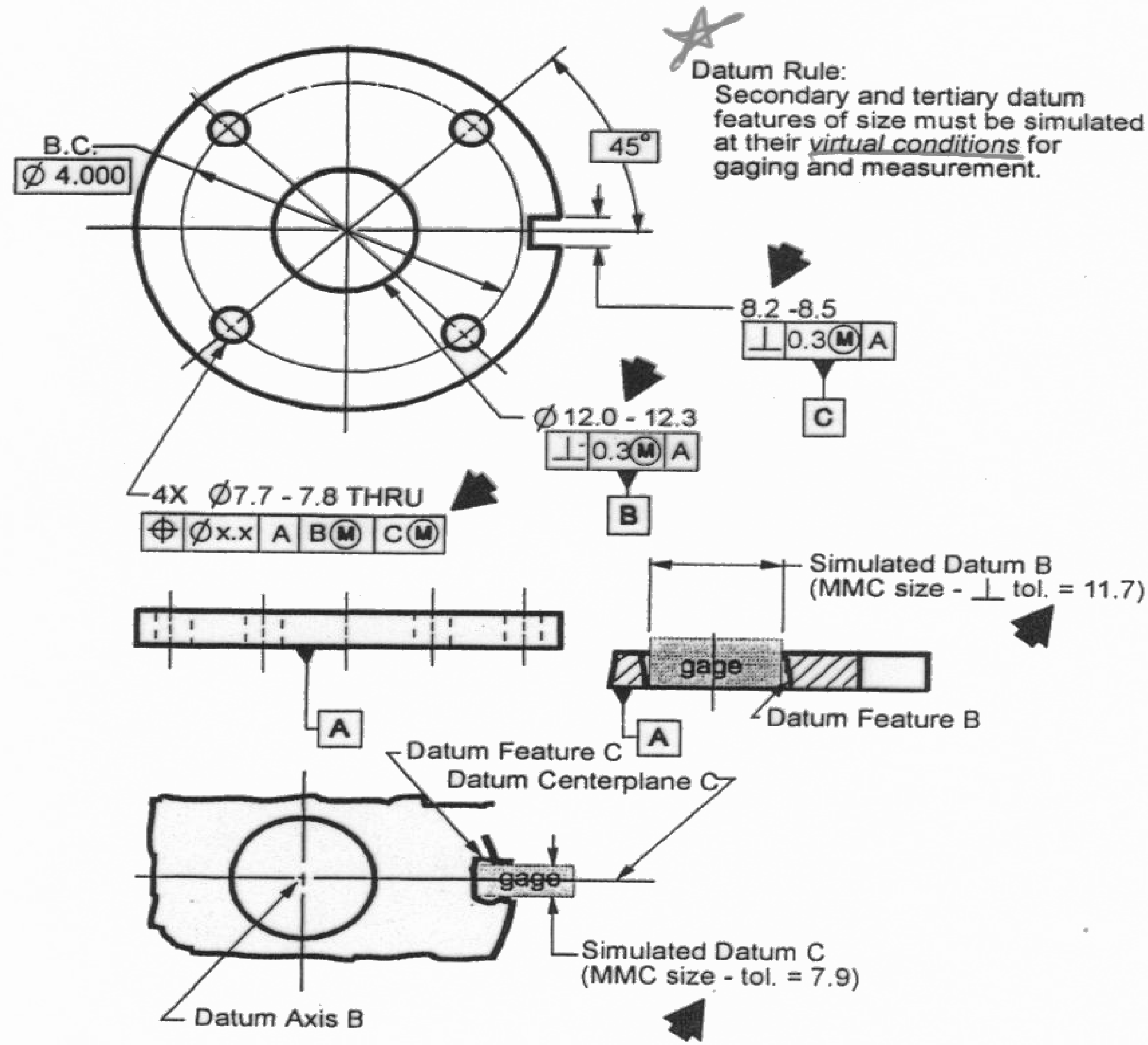



Datum order (precedence).

When the secondary datum is an axis, the tertiary datum serves to orient the part (stop rotation).

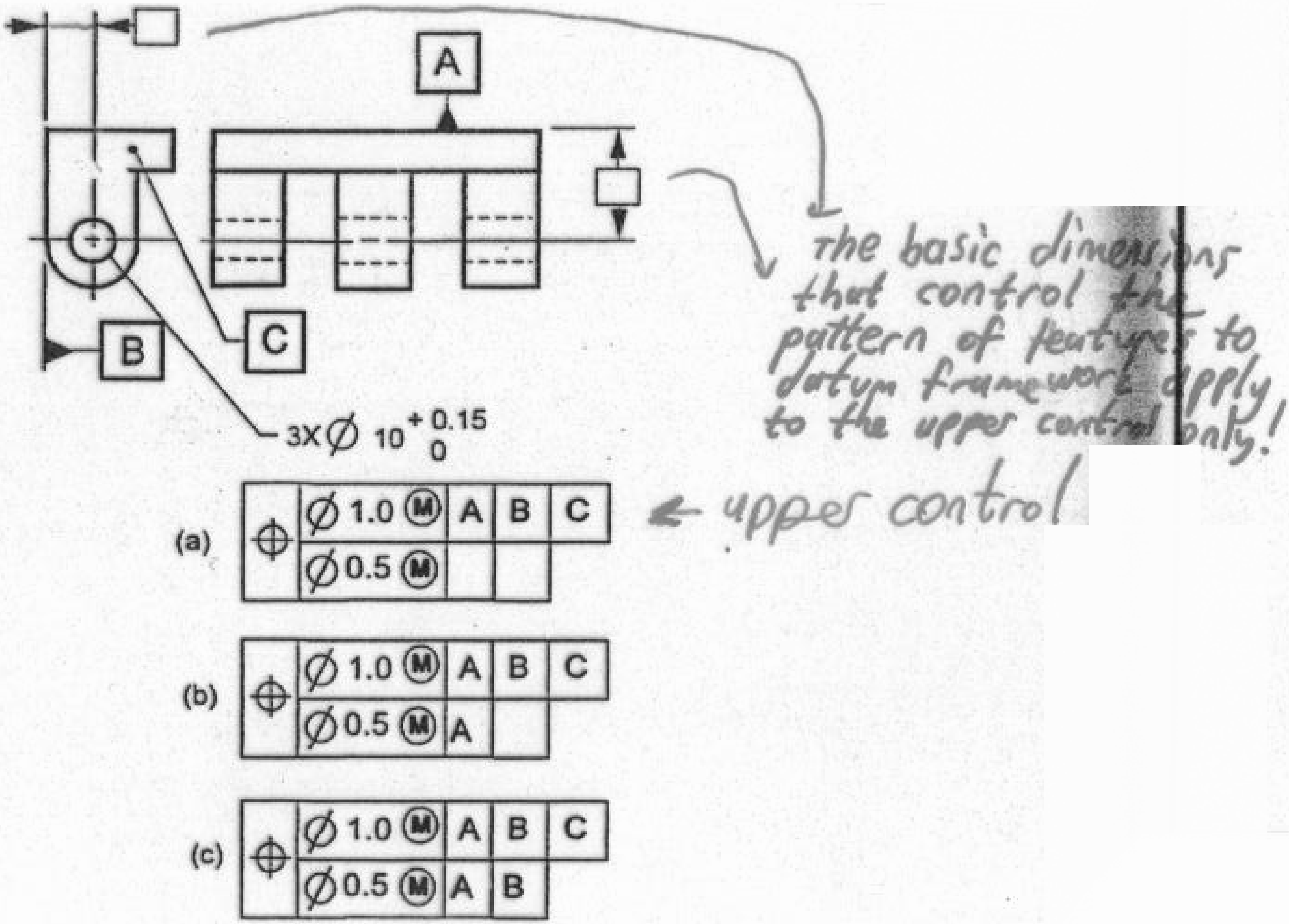



Secondary and tertiary datum features.

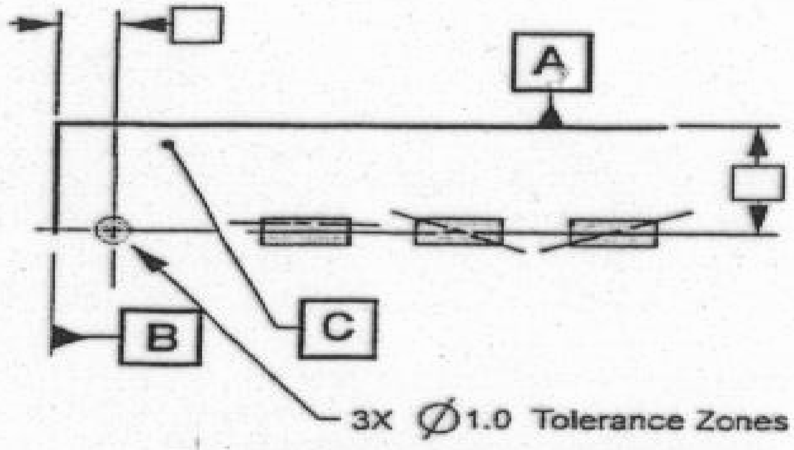



(a)

(a) Secondary/tertiary datum rule;
(b) datums MMC and bonus tolerance.




**Composite position controls: aligned holes.**




**For the Upper Control Callout**

The 1.0 diameter tolerance zones must be aligned and basically located from the datum framework A, B, C.

**For the Lower Control Callouts**

(a)

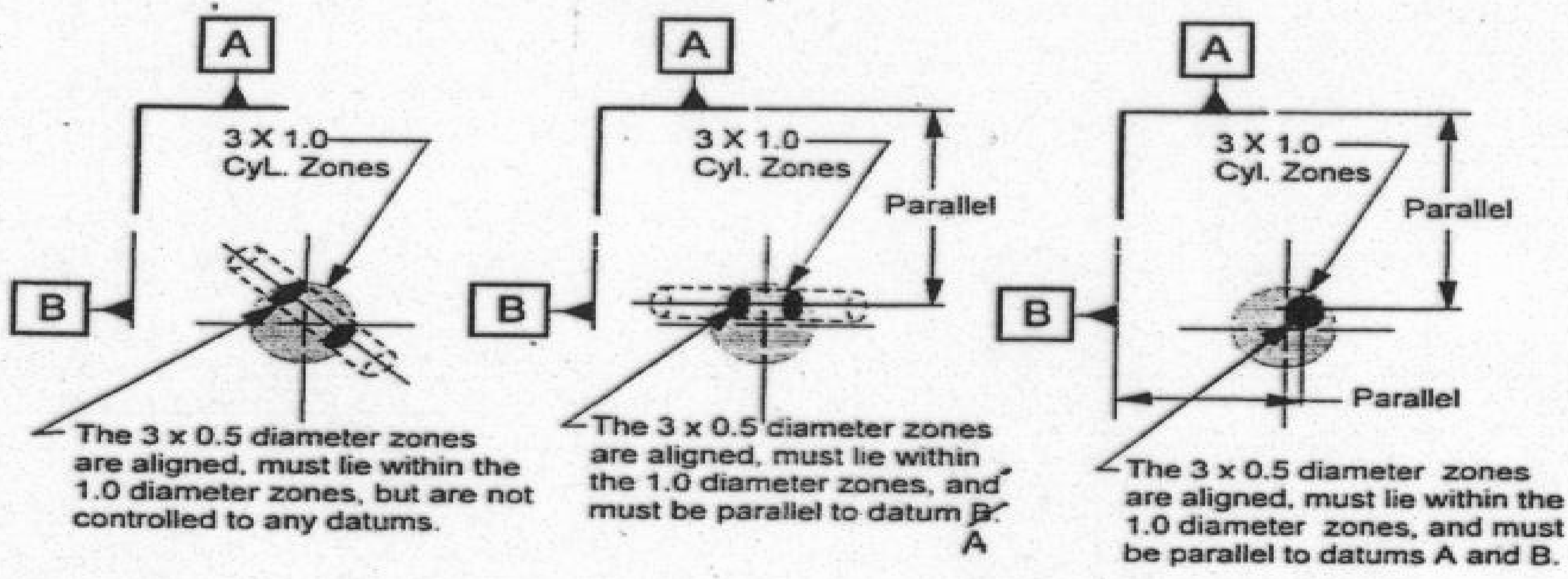
| ⌖ | ⌀1.0 Ⓜ | A | B | C |
|---|---|---|---|---|
| | ⌀0.5 Ⓜ | | | |

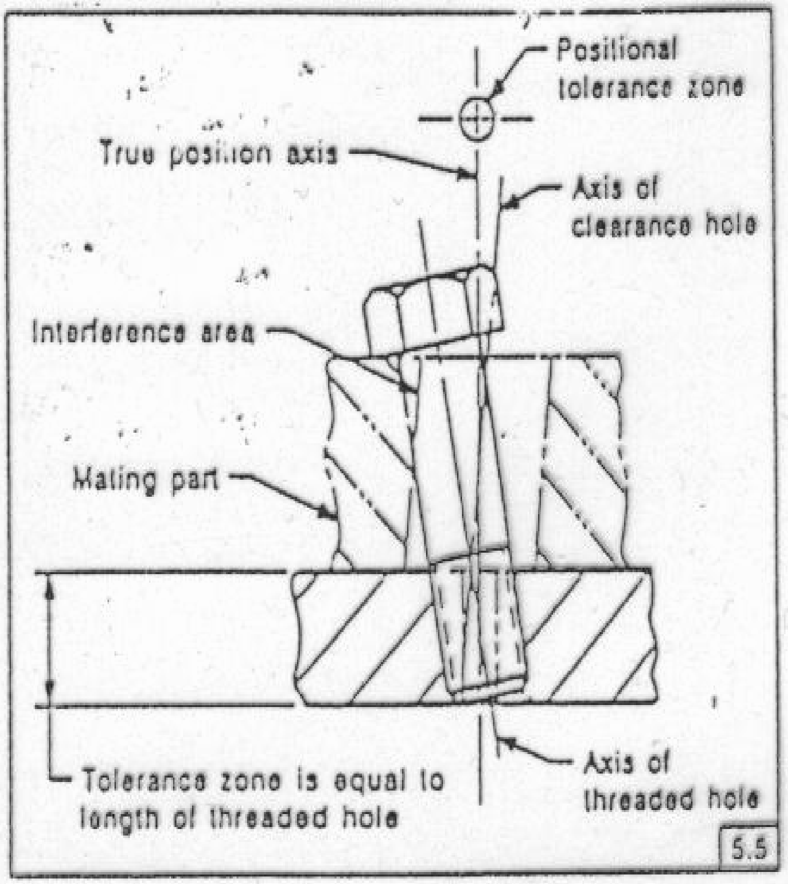
(b)

| ⌖ | ⌀1.0 Ⓜ | A | B | C |
|---|---|---|---|---|
| | ⌀0.5 Ⓜ | A | | |

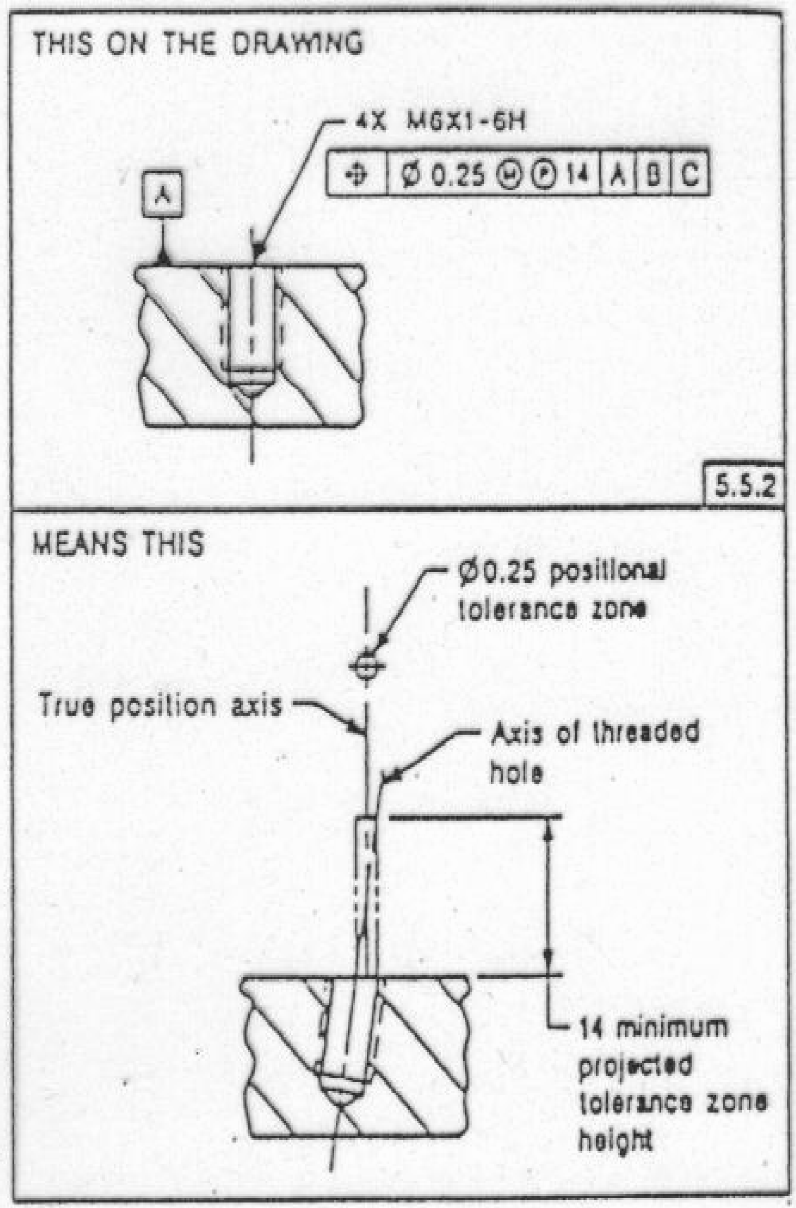
(c)

| ⌖ | ⌀1.0 Ⓜ | A | B | C |
|---|---|---|---|---|
| | ⌀0.5 Ⓜ | A | B | |

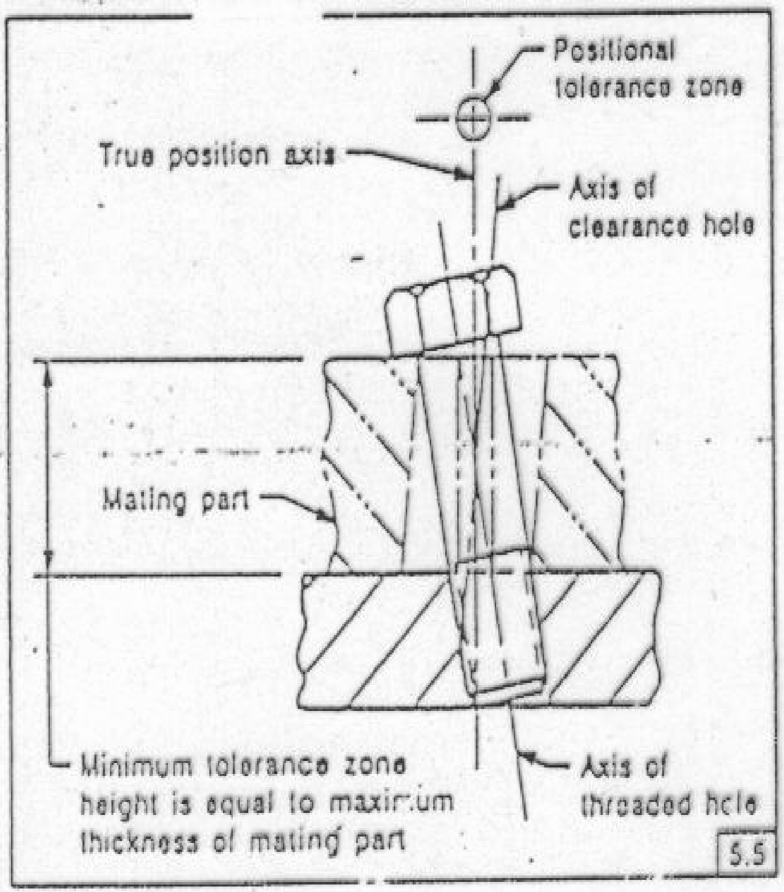




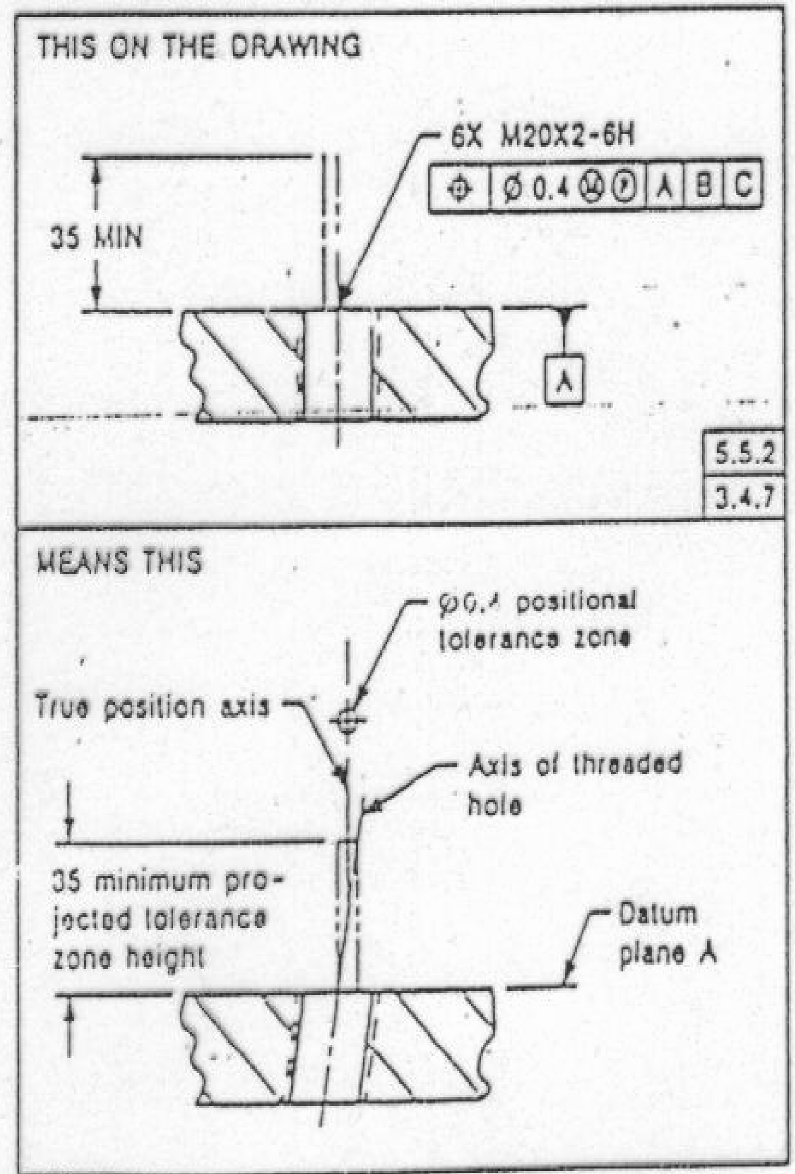

INTERFERENCE DIAGRAM, FASTENER AND HOLE




PROJECTED TOLERANCE ZONE SPECIFIED




BASIS FOR PROJECTED TOLERANCE ZONE




PROJECTED TOLERANCE ZONE INDICATED WITH CHAIN LINE

# General tolerances — *(Geometrical)*

## Part 2:
Geometrical tolerances for features without individual tolerance indications

### 5.1.1 Straightness and flatness

The general tolerances on straightness and flatness are given in table 1. When a tolerance is selected from table 1, it shall be based, in the case of straightness, on the length of the corresponding line and, in the case of flatness, on the longer lateral length of the surface, or the diameter of the circular surface.

**Table 1 — General tolerances on straightness and flatness**

Values in millimetres

| Tolerance class | Straightness and flatness tolerances for ranges of nominal lengths | | | | | |
|---|---|---|---|---|---|---|
| | up to 10 | over 10 up to 30 | over 30 up to 100 | over 100 up to 300 | over 300 up to 1 000 | over 1 000 up to 3 000 |
| H | 0,02 | 0,05 | 0,1 | 0,2 | 0,3 | 0,4 |
| K | 0,05 | 0,1 | 0,2 | 0,4 | 0,6 | 0,8 |
| L | 0,1 | 0,2 | 0,4 | 0,8 | 1,2 | 1,6 |

### 5.1.2 Circularity

The general tolerance on circularity is equal to the numerical value of the diameter tolerance, but in no case shall it be greater than the respective tolerance value for circular radial run-out given in table 4 (see examples in clause B.2).

### 5.1.3 Cylindricity

General tolerances on cylindricity are not specified.

### 5.2.3 Perpendicularity

The general tolerances on perpendicularity are given in table 2. The longer of the two sides forming the right angle shall be taken as the datum; if the sides are of equal nominal length, either may be taken as the datum.

**Table 2 — General tolerances on perpendicularity**

Values in millimetres

| Tolerance class | Perpendicularity tolerances for ranges of nominal lengths of the shorter side | | | |
|---|---|---|---|---|
| | up to 100 | over 100 up to 300 | over 300 up to 1 000 | over 1 000 up to 3 000 |
| H | 0,2 | 0,3 | 0,4 | 0,5 |
| K | 0,4 | 0,6 | 0,8 | 1 |
| L | 0,6 | 1 | 1,5 | 2 |

### 5.2.4 Symmetry

The general tolerances on symmetry are given in table 3. The longer of the two features shall be taken as the datum ; if the features are of equal nominal length, either may be taken as the datum.

NOTE — The general tolerances on symmetry apply where

— at least one of the two features has a median plane, or

— the axes of the two features are perpendicular to each other.

See examples in clause B.5.

**Table 3 — General tolerances on symmetry**

Values in millimetres

| Tolerance class | Symmetry tolerances for ranges of nominal lengths | | | |
|---|---|---|---|---|
| | up to 100 | over 100 up to 300 | over 300 up to 1 000 | over 1 000 up to 3 000 |
| H | 0,5 | | | |
| K | 0,6 | | 0,8 | 1 |
| L | 0,6 | 1 | 1,5 | 2 |

### 5.2.5 Coaxiality

General tolerances on coaxiality are not specified.

NOTE — The deviation in coaxiality may, in an extreme case, be as great as the tolerance value for circular radial run-out given in table 4, since the deviation in radial run-out comprises the deviation in coaxiality and the deviation in circularity.

### 5.2.6 Circular run-out

The general tolerances on circular run-out (radial, axial and any surface of revolution) are given in table 4.

For general tolerances on circular run-out, the bearing surfaces shall be taken as the datum if they are designated as such. Otherwise, for circular radial run-out, the longer of the two features shall be taken as the datum ; if the features are of equal nominal length, either may be taken as the datum.

**Table 4 — General tolerances on circular run-out**

Values in millimetres

| Tolerance class | Circular run-out tolerances |
|---|---|
| H | 0,1 |
| K | 0,2 |
| L | 0,5 |

## 6 Indications on drawings

**6.1** If general tolerances in accordance with this part of ISO 2768 shall apply in conjunction with the general tolerances in accordance with ISO 2768-1, the following information shall be indicated in or near the title block:

a) "ISO 2768";

b) the tolerance class in accordance with ISO 2768-1;

c) the tolerance class in accordance with this part of ISO 2768.

EXAMPLE

**ISO 2768-m** K