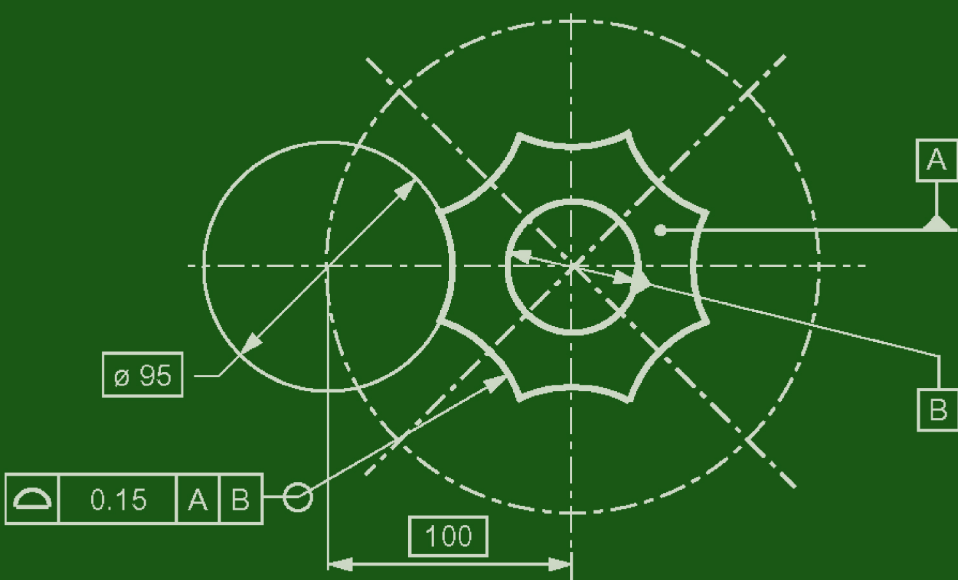
THIRD EDITION

Geometrical Dimensioning and Tolerancing for Design, Manufacturing and Inspection

A Handbook for Geometrical Product Specification Using ISO and ASME Standards



Georg Henzold

Geometrical Dimensioning and Tolerancing for Design, Manufacturing and Inspection

This page intentionally left blank

Geometrical Dimensioning and Tolerancing for Design, Manufacturing and Inspection

A Handbook for Geometrical Product Specification Using ISO and ASME Standards

Third Edition

Georg Henzold

ELSEVIER

Butterworth-Heinemann
An imprint of Elsevier

Butterworth-Heinemann is an imprint of Elsevier
The Boulevard, Langford Lane, Kidlington, Oxford OX5 1GB, United Kingdom
50 Hampshire Street, 5th Floor, Cambridge, MA 02139, United States

Library of Congress Cataloging-in-Publication Data
A catalog record for this book is available from the Library of Congress

British Library Cataloguing-in-Publication Data
A catalogue record for this book is available from the British Library

ISBN: 978-0-12-824061-8

Publisher: Matthew Deans
Acquisitions Editor: Brian Guerin
Editorial Project Manager: Mariana Kuhl
Production Project Manager: Poulouse Joseph
Cover Designer: Victoria Pearson

Typeset by SPi Global, India

Contents

Author Biography

Georg Henzold was manager of the Department for Standardization of a manufacturer of power plant machinery. He was a longstanding chairman of the committee dealing with standardization in the field of geometrical dimensioning and tolerancing in the German Standardization Institute, DIN, and in the European Committee for Standardization, CEN. He is a long-time delegate in the pertinent committees of the International Standardization Organization (ISO).

This page intentionally left blank

Preface

Technical developments leading to better performance, higher efficiency, fewer errors and greater reliability, along with the constraints imposed by the need for economy, rationalization and greater cooperation between licensees and subcontractors, have necessitated the use of completely toleranced drawings suitable to the function for manufacturing and inspection. "Completely" toleranced means that the geometries (form, size, orientation and location) of geometrical elements of the workpiece are completely defined and toleranced, with nothing left to the individual judgement of the manufacturer or inspector. It is only with this approach that proper functioning can be verified and the possibilities available for economization and rationalization in manufacturing and inspection can be fully utilized, and reliable quality assurance can be achieved.

This book presents the state of the art in geometrical dimensioning and tolerancing. It describes the international standardization in this field, which is laid down in ISO standards. It also indicates the deviations between ISO and the US standard ASME Y14.5. Crucial manufacturing issues within geometrical dimensioning and tolerancing are thoroughly explained. Principles for the inspection of geometrical deviations are given, together with a basis for tolerancing suitable for inspection. Also, examples of tolerancing appropriate to various functional requirements are presented, providing a guide for geometrical tolerancing.

In the past, ±tolerancing was used for nonsizes (centre distances, steps, radii and dimensions for contours). These tolerances have large ambiguities and the drawings often permitted parts that were nonfunctional. However, good quality was achieved by optimization of the manufacturing process by the cooperation of the concerned parties.

Today, outsourcing and globalization require complete and correct tolerancing. This can be achieved by using TEDs (e.g., CAD models), profile tolerancing for integral features (solid surfaces) and position tolerancing for derived features (median lines, median surfaces), related to a general datum system locking all degrees of freedom of the part. The general datum features are to be toleranced either by surface profile tolerances for integral datum features or by position tolerances for derived features, the latter together with size tolerances (±tolerances) with Ⓔ, Ⓜ or Ⓛ. The primary datum feature is to be toleranced using a form tolerance (without relation to a datum), the secondary datum feature by a tolerance related to the primary datum, and the tertiary datum feature by a tolerance related to the primary and secondary datums.

For surfaces without tolerance indication (tolerance for the integral feature, the derived feature, and size), a general profile surface tolerance is to be indicated. With that, the part is completely toleranced. If necessary for functional reasons, other tolerances related to the general datum system (profile surface tolerances for integral features or position tolerances for derived features, the latter together with size tolerances (±tolerances) with Ⓔ, Ⓜ or Ⓛ) may be indicated. They overrule the general tolerance.

If necessary for functional reasons, other additional tolerances (profile, position, run-out) may be indicated as further constraints. They are to be indicated together with a tolerance related to the general datum system (profile surface tolerance for the integral feature or position tolerance for the derived feature, the latter together with a size tolerance (±tolerance) with Ⓔ, Ⓜ or Ⓛ). This tolerance may be equal to the general tolerance (but is to be indicated because the general tolerance applies only to surfaces without tolerance indication).

The symbols for special tolerances (straightness, roundness, flatness, cylindricity, coaxiality/concentricity, symmetry) are then superfluous and can be avoided. However, they are explained in this book in order to enable the reading of older drawings.

Some tolerancings can **only** be expressed by using profile or position tolerances, e.g., tolerancing of the form of a torus or orientation tolerancing partly related to a datum system (with the symbol ><).

This book follows this concept and facilitates complete and correct tolerancing.

The GPS standards are rule based. They are voluminous and somewhat difficult to read. This book, however, explains the application of the ISO and ASME standards concisely and in plain language.

This book can serve as an introduction to geometrical dimensioning and tolerancing for students, and it is also useful to practitioners in the fields of design, manufacturing and inspection. The author has relied on his experience gathered during the elaboration of the ISO standards and during his lectures and discussions on geometrical dimensioning and tolerancing in industry and education. However, proposals for improvements are appreciated and should be directed to the publisher.

The book represents the author's understanding of the various standards and is written for educational purposes. In cases of dispute, the original standards should be considered. The functional cases of geometrical requirements vary over a wide range. Therefore, before application in practice, the reader should carefully determine whether the presentation in the book fits the particular purpose.

All dimensions and tolerances in this book are in millimetres (mm).

Notation

A reading, measured value
C mean size
D diameter, hole
I actual size
L length
N number
P mating size or content or statistical probability or pitch
R calculated value from readings
T size tolerance
U expanded measurement uncertainty
Z zone
α angle
β angle
γ angle
Δ difference
δ deviation
λ cut-off, wavelength

c distance or safety factor with statistical tolerancing
d diameter, shaft
e deviation at a certain location
h height
k correction factor
n number
p coordinate
q coordinate
r radius
s clearance
t geometrical tolerance
u measurement uncertainty
x coordinate
y coordinate
z coordinate

Subscripts

c section
g geometrical
i individual
m form
o orientation
p projected tolerance or number of points or peak
r run-out
s situation, location
th theoretically exact
v valley

Abbreviations

AVG	average
BASIC	theoretical exact dimension
CMM	coordinate measuring machine
DIA	diameter
D&T	dimensioning and tolerancing
DOF	degree of freedom
FIM	full indicator movement
FIR	full indicator reading
FRTZF	feature-relating tolerance zone framework
GD&T	geometrical dimensioning and tolerancing
GPS	geometrical product specification
IB	inner boundary
LD	least minimum diameter
LMB	least material boundary
LMC	least material condition
LMR	least material requirement
LMS	least material size
LMVC	least material virtual condition
LMVS	least material virtual size
LSC	least-squares circle
MCC	minimum circumscribed circle
MD	major diameter
MIC	maximum inscribed circle
MMB	maximum material boundary
MMC	maximum material condition
MMR	maximum material requirement
MMS	maximum material size
MMVC	maximum material virtual condition
MMVS	maximum material virtual size
MPE	maximum permissible error (of a measuring device)
MZC	minimum zone circle
OB	outer boundary
PD	pitch diameter
PLTZF	pattern-locating tolerance zone framework

PUMA	procedure for uncertainty management
RFS	regardless of feature size
SEP REQT	separate requirement
SIM	simultaneous requirement
TED	theoretically exact dimension
TEF	theoretically exact feature
TIR	total indicator reading
TP	true position, theoretically exact position or location
VD&T	vectorial dimensioning and tolerancing

ISO Drawing Rules

The ISO drawing rules are laid down in ISO 128 and ISO 129. As previously noted in Chapter 1, ISO drawings use a comma instead of the decimal point. Therefore in the drawings of this book the comma is used.

Text Equivalents

An ISO Standard (ISO 14 995-1) for text equivalents was planned, but not finalized. The following text equivalents (to be used in text, not in drawings) were planned:

⌒	Line profile	PFL
⏤	Straightness	STR
○	Roundness	RON
⌓	Surface profile	PFS
⏥	Flatness	FLT
⌭	Cylindricity	CYL
∠	Angularity	ANG
//	Parallelism	PAR
⊥	Perpendicularity	PER
⌖	Position	POS
◎	Coaxiality	CAX
⌯	Symmetry	SYM
↗	Circular run-out	CRO
⌰	Total run-out	TRO

Chapter 1

General

1.1 Properties of the surface

The suitability of a workpiece for its purpose depends on the inner properties (material properties, inner discontinuities such as shrink holes or inner imperfections such as segregations) and on its surface condition.

The surface condition comprises the properties of the surface border zone. These are chemical, mechanical and geometrical properties, as shown in Fig. 1.1.

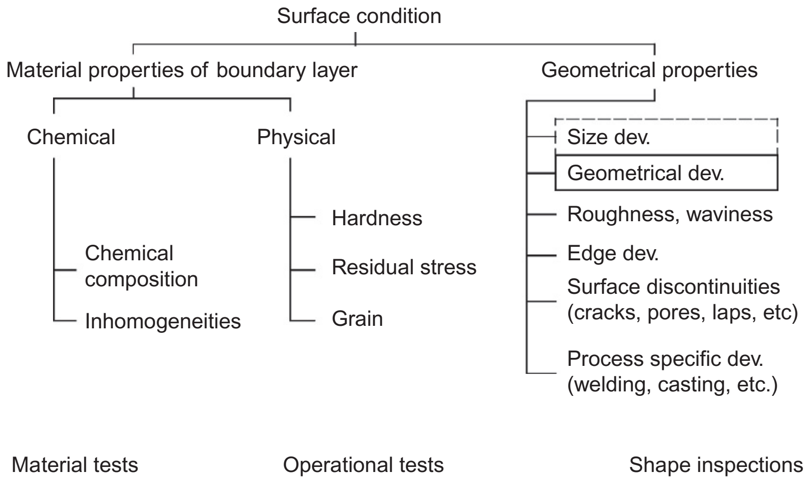


FIG. 1.1 Properties of surfaces and their tests and inspections

The chemical and mechanical properties comprise chemical composition, grain, hardness, strength and inhomogeneities. The properties of the surface border zone may be different from those in the core zone.

The geometrical properties of the workpiece are defined by the geometrical dimensioning and tolerancing (GD&T) of the drawing or model. In this book the term drawing is used for all documentation of GD&T (including the model).

The geometrical properties are defined as deviations from geometrical ideal elements (features) of the workpiece. Geometrical ideal elements (features) are parts of the entire workpiece surface, which have geometrical, unique

Geometrical Dimensioning and Tolerancing for Design, Manufacturing and Inspection
https://doi.org/10.1016/B978-0-12-824061-8.00008-6

and nominal form as, e.g. planes, cylinders, spheres, tori, cones and wedges. They can also be derived as, e.g. median lines (axes) or median surfaces. They can also be edges. See Fig. 1.2.

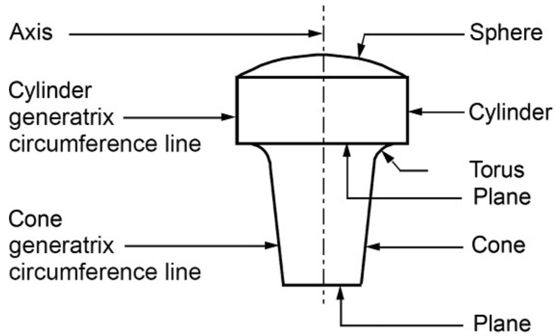


FIG. 1.2 Examples of geometrical elements

Geometrical deviations are

– Size deviations	– Waviness
– Form deviations	– Roughness
– Orientation deviations	– Surface discontinuities
– Location deviations	– Edge deviations

Size deviation is the difference between actual size and nominal size. It must be distinguished between

– deviation of the actual local size from the nominal linear size or from the nominal angular size;
– deviation of the actual global size from the nominal global size;
– deviation of the actual statistical size from the nominal statistical size (see 4).

The linear sizes are defined in ISO 14 405-1. The angular sizes are defined in ISO 14 405-3 (see 4).

Size deviations are assessed over the entire geometrical element. They originate mainly from imprecise adjustment of the machine tool and from variations during the manufacturing process, e.g. due to tool wear.

The two-point size, according to ISO 14 405-1, is the default (± tolerance without a modifier) and is used in this book (in contrast to ASME Y14.5 rule #1, where size together with the envelope requirement is the default). See 4.

Form deviation is the deviation of a feature (geometrical element, surface or line) from its nominal form, shown in Fig. 1.3. If not otherwise specified, form deviations are assessed over (or along) the entire feature. Form deviations result, e.g. from the looseness or error in guidances and bearings of the machine tool, deflections of the machine tool or the workpiece, error in the fixture of the

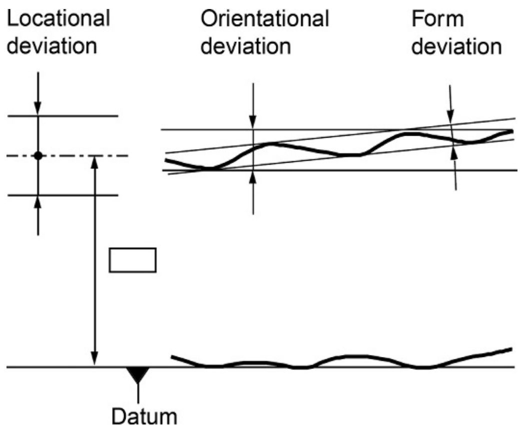


FIG. 1.3 Form deviation, orientation deviation, location deviation

workpiece, hardness distortion and wear. The ratio between width and depth of local form deviations is generally more than 1000:1 (VDI/VDE 2601).

Orientation deviation is the deviation of a feature (geometrical element, surface or line) from its nominal form and orientation. The orientation is related to one or more (other) datum feature(s). The orientation deviation includes the form deviation (Fig. 1.3). If not otherwise specified, orientation deviations are assessed over the entire feature. The causes of orientation deviations are similar to those of form deviations. They can also result, for example, from erroneous fixture of the workpiece after re-mounting on the machine tool.

Location deviation is the deviation of a feature (geometrical element, surface or line) from its nominal location. The location is related to one or more (other) datum feature(s). The location deviation also includes the form deviation and the orientation deviation (of the surface, median line or median surface), as shown in Fig. 1.3. If not otherwise specified, location deviations are assessed over the entire feature. The causes of location deviations are similar to those of size, form and orientation deviations.

Waviness refers to the more-or-less periodic irregularities of a workpiece surface with spacings greater than the spacings of its roughness (DIN 4774). The ratio between spacing and depth of the waviness is generally between 1000:1 and 100:1. In general, more than one wave can be recognized (VDI/VDE 2601). Waviness is assessed from one or more representative parts of the surface, as seen in Fig. 1.4. It is caused by eccentric fixtures during the manufacturing process or by form deviations of the cutter or vibrations of the machine tool and/or the cutting tool and/or the workpiece (DIN 4760). Waviness is not dealt with in the following.

Roughness refers to periodic or non-periodic irregularities of a workpiece surface with small spacings inherent to the forming process. The ratio between

spacing and depth of the roughness is generally between 150:1 and 5:1 (VDI/VDE 2601). Roughness is assessed from one or more representative parts of the surface, as seen in Fig. 1.4.

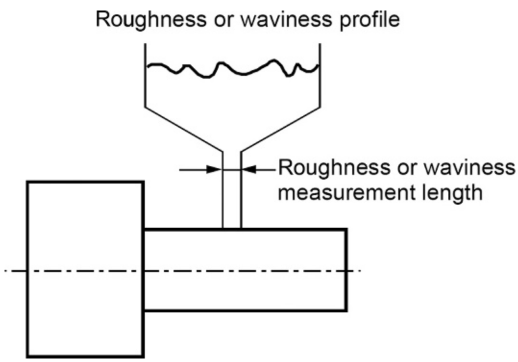


FIG. 1.4 Assessment of roughness or waviness

Roughness is caused by the direct effect of the cutting edges (VDI/VDE 2601), i.e. by imprinting the cutting edges on the surface. Due to the cutting process (tear chip, shear chip) the print is modified. Other origins are deformations from blasting, gemmation with galvanizing, crystallization or chemical effects (such as from mordants or corrosion). Roughness ranges down to the crystal structure (DIN 4760). Roughness is not dealt with in the following.

A surface discontinuity is an isolated imperfection of the surface, such as a crack, pore or lap. In general, it is not taken into account when assessing deviations of size, form, orientation, location, waviness and roughness. The definitions, sizes and permissibilities of surface discontinuities have to be dealt with separately.

At this time, there are very few standards on this subject available, e.g. for fasteners in ISO 6157 and for hot milled steel products in ISO 9443. Surface discontinuities are not dealt with in the following.

Edge deviations are deviations of the workpiece edge zone from the geometrical ideal shape, such as burrs or abraded edges instead of sharp edges. ISO 13 715 defines tolerances for edges and gives the drawing indications. These deviations are dealt with in clause 9.1 of this book.

Transitions of defined shape are edges of special forms. They are discussed in clause 9.2 of this book.

The **classification of surface irregularities** as described here is useful for the following reasons:

a) The different types of surface irregularities have different origins in the manufacturing process. In order to control the manufacturing process, they must be assessed separately.

b) The different types of surface irregularities often have different effects on the suitability of the surface for its purpose. For example, on raceways of ball bearings, waviness has a strong influence on lifetime and noise, while roughness has little influence. In order to specify the permissible deviations as related to function, the different types must be specified separately.

c) The depths of the irregularities vary over large ranges, between about 0,1 μm (and sometimes smaller) with roughness and about 100 μm (and sometimes more) with form deviations. The ratio between spacing and depth of the irregularities also varies greatly. The smallest ratio between spacing and depth occurs with cracks and is generally smaller than 5:1, whereas the ratio between spacing and depth with form deviations is generally greater than 1000:1. Because of these wide ranges, the requirements for measuring devices and for diagrams are quite different. For the assessment of different kinds of irregularities (deviations), different types of measuring instruments are used, having different magnifications, profile diagrams and ratios of horizontal to vertical magnification.

The definitions of the different types of irregularities (deviations) are somewhat uncertain, with no distinct borderlines. Therefore it was discussed within ISO whether to define borderlines in terms of defined spacing of irregularities, defined ratios between spacing and depths of irregularities, or defined ratios between spacing of irregularities and feature lengths. However, it was decided to retain the definitions according to the causes of the irregularities (ISO 4287, ANSI B46.1, BS 1134, DIN 4760).

There is another distinction between **micro and macro deviations**. Macro deviations are those for which the usual measuring devices can be used for the assessment of size, form, orientation and location (e.g. dial indicator). Micro deviations, on the other hand, are assessed with roughness or waviness measuring instruments. Macro deviations are assessed over the entire feature length; micro deviations are assessed from a representative part of the surface. Also, there is no distinct boundary, because sometimes parts of the waviness can contribute to the result of the measured macro deviations, and sometimes parts of the form deviations can contribute to the result of the measured micro deviations (waviness).

Figure 1.5 gives an idea of the combination (superposition) of the types of surface irregularities (deviations) on a surface.

1.2 Principles for tolerancing

It is impossible to manufacture workpieces with no deviations from the nominal shape. Workpieces always have deviations of size, form, orientation and location.

When these deviations are too large, the usability of the workpiece for its purpose will be impaired. However, if the deviations are kept as small as

Geometrical deviation Profile diagram	Description Examples of origin
1st order: Form 	Errors in guidance of machine tool Deflections of machine tool or workpiece, error in fixture of workpiece Warping, wear
2nd order: Waviness 	Eccentric fixture, form deviation of tool, vibration
3rd order: Roughness 	Grooves, form of tool cutting edge, horizontal and vertical feed
4th order: Roughness 	Cutting process (tear chip, shear chip), deformation form blasting, gemmation with galvanizing
5th order: Roughness not presentable	Crystallization process, mordant, corrosion
6th order: Roughness not presentable	Crystal structure
Superposition 	Actual surface

FIG. 1.5 Superposition of surface deviations (DIN 4760)

possible during manufacturing in order to avoid any impairment of usability, the production costs will generally be too expensive.

In general competition forces to use all possibilities that yield more economical production costs, including possibilities which occur with current developments. Therefore it is necessary that the drawing tolerances define the workpiece completely: i.e. each property (size, form, orientation, location) must be toleranced. Only then is the manufacturer able to choose the most economical production method, depending on the number of pieces to be produced and on the production methods available.

Incompletely toleranced drawings result in:

– questions for the production-planning engineer
– questions for the manufacturing engineer

- questions for the inspection engineer
- reworking
- defects and damages

Only completely toleranced drawings enable the production of workpieces to be as precise as necessary and as economical as possible, which is necessary for competition. However, when all tolerances necessary to define the workpiece completely are indicated individually, the drawing becomes overloaded with indications and is hard to read. Therefore general tolerances should be applied.

General tolerances must be equal to or larger than the customary workshop accuracy, which is equal to those tolerances the workshop does not exceed with normal effort using normal workshop machinery. Larger tolerances bring no gain in manufacturing economy. The normal workshop accuracy depends on the workshop machinery that produces the largest deviations (disregarding exceptions that must be dealt with in certain unusual cases). The customary workshop accuracy is, in general, the same within one field of industry.

The general tolerances should be applied by an indication in or near the title block of the drawing. Tolerances, which must be smaller, have to be indicated individually; see 3 to 11. If not otherwise specified (see Fig. 3.8), the tolerances apply over the entire lengths of the features.

(**Note**: ISO drawings use a comma instead of the decimal point, so the drawings in this book use the comma. In the text, however, the UK/US style of decimal point is used.)

This page intentionally left blank

Chapter 2

ISO Geometrical Product Specifications (GPS), New Approach

2.1 ISO GPS standards, overview, GPS matrix

2.1.1 ISO GPS standards overview

General principles ISO 8015

Operations general: ISO 17450-1, -2
Operations specific: ISO 14406 Extractions ISO 16610-1 ff Filterings ... in preparation

Definitions of features: ISO 17450-3 ISO 22432
Definitions of characteristics ISO 17450-4 ISO 25378

Sizes	Geometrical tolerancings			Surface textures		Edges
ISO 14405-1	ISO 1101	ISO 1660	ISO 14405-2	ISO 1302	ISO 8785	ISO 13175
ISO 286	ISO 5459	ISO 3040	ISO 20170	ISO 4287		ISO 21204
ISO 14405-3	ISO 2692	ISO 2538	ISO 22081	ISO 4288		
	ISO 5458	ISO 10579		ISO 25178-1 ff		

Measuring instruments and calibrations

Simple length measuring instruments			CMMs	Roughness measuring instruments	
ISO 14978	ISO 9493	ISO 13225	ISO 10360-1 ff	ISO 3274	ISO 25178-6 ff
ISO 7863	ISO 13102	ISO 13385		ISO 5436-1 ff	

Gauges: ISO 1938

Measurement uncertainty: ISO 14253-1 ff (general) ISO 15530-1 ff (CMMs)

2.1.2 GPS matrix

Each GPS standard contains at the end the GPS matrix, with the indication of the concerned contents.

Geometrical Dimensioning and Tolerancing for Design, Manufacturing and Inspection
https://doi.org/10.1016/B978-0-12-824061-8.00006-2

	Chain Links						
	A	B	C	D	E	F	G
	Symbols and indications	Feature requirements	Feature properties	Conformance and non-conformance	Measurement	Measurement equipment	Calibrations
Size							
Distance							
Form							
Orientation							
Location							
Run-out							
Profile surface texture							
Areal surface texture							
Surface imperfections							

In the past, the matrix contained global and complementary GPS standards. They are now to be marked (as appropriate) in the new matrix.

2.2 Terms and definitions

ISO aims to give mathematically exact definitions of the constituents of the geometrical specifications in order to provide a sound foundation for

– unambiguous drawing specifications
– programming of measurement instruments
– estimation of measurement uncertainties.

All geometrical specifications deal with **features**.
According to ISO TS 17 450-1 and ISO 22 432-1:

(Single) (geometrical) feature: a geometric entity which is a single point, a single line, or a single surface, e.g. a cylinder.
Compound feature: a geometric entity which is a collection of features, e.g. two parallel opposite planes connected by a size dimension (plane pair).

According to ASME Y14.5:

Feature: physical portion of a part, such as a surface, pin, tab or slot.

Practically, there is no difference in the meaning of a drawing indication due to the different definitions of a feature.
ISO 17 450-1 defines the following:

– **nominal feature** (ideal geometry according to the drawing)
– **real feature** (non-ideal geometry as existing on the workpiece)

– **extracted feature** (non-ideal geometry as detected from the workpiece)
– **associated feature** (ideal geometry and fitted to the extracted feature according to an objective function (fitting rule, e.g. Chebyshev)).

See Fig. 2.1.

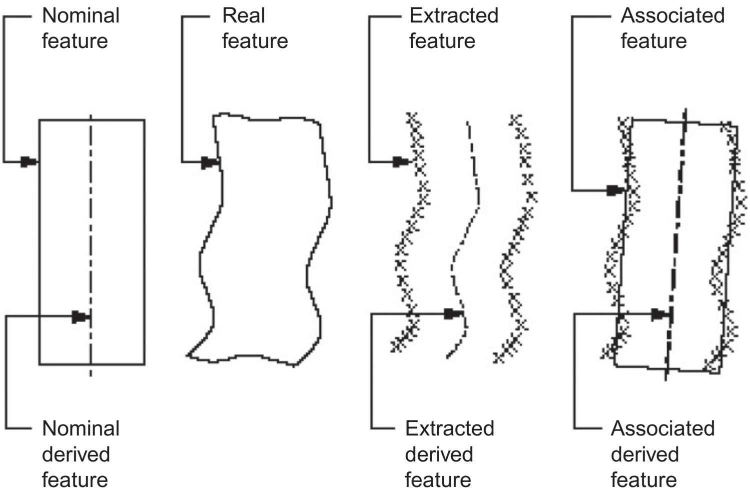


FIG. 2.1 Features

ISO 22 432 describes various types of features:

– **actual feature:** extracted feature
– **ideal feature** (nominal feature according to the drawing outline or CAD model)
– **non-ideal feature** (with deviations from the ideal feature)
– **integral feature** (ideal or non-ideal, surface or line on a feature)
– **derived feature** (ideal or non-ideal, centre point, median line or median surface derived from one or more integral features)
– **feature of linear size** (cylindrical hole or shaft (single feature), tab or slot, **plane pair** = two opposite parallel planes connected by a size (compound feature), circle, pair of opposite straight lines connected by a size)
– **feature of angular size** (cone, wedge)
– **extracted integral feature:** approximated representation of the real (integral) feature, obtained by extracting a finite number of points from the real (integral) feature; this extraction is performed in accordance with specified conventions (several such representations may exist for each (integral) feature) (also: **actual surface, actual surface line**)

- **extracted derived feature:** centre point, median line or median surface derived from one or more extracted integral features (extracted axis, extracted median surface) (also: **actual centre point, actual axis, actual median surface**)
- **associated integral feature**: integral feature of perfect form associated to the extracted integral feature in accordance with specified conventions (also: **substitute element, least-squares element, minimum zone element, contacting element, maximum inscribed element, minimum circumscribed element, element = feature**)
- **associated derived feature:** centre point, median straight line or median plane derived from one or more associated integral features
- **specification feature** (non-ideal, defined by the specification operator)
- **verification feature** (non-ideal, defined by the verification operator)
- **filtered feature** (e.g. feature after filtering; e.g. according to Gauss)
- **candidate features** (set of ideal features which satisfies geometrical constraints like outside the material used to model the function of a fit)
- **substitute feature** (unique ideal feature associated with a non-ideal feature, e.g. as used in CMM techniques)
- **limited feature** (portion of an ideal feature, e.g. restricted area)
- **enabling feature** (ideal feature, identified from an ideal feature and used solely to build other features, e.g. section plane)
- **reference feature** (ISO 25 378, ideal feature used as reference for the determination of deviations or characteristics)
- **situation feature** (ideal feature (point, straight line, plane) to define the orientation or location of a feature)
- **criterion for situation features:** definition of the situation feature from the non-ideal extracted feature
  - **Chebyshev:** minimax requirement, minimum zone requirement, distance to the ideal feature (reference feature) is minimized, ISO default
  - **Gauss (least squares):** the sum of the squares of the distances to the ideal feature (reference feature) is minimized
  - **Maximum inscribed:** with sizes of cylinders and plane pairs
  - **Minimum circumscribed:** with sizes of cylinders and plane pairs

ISO 17 450-1 defines the **duality principle** by defining the **non-ideal surface model** of the workpiece (skin model, imagination of the designer, design intent) and the **verification model** (defined by the verification process executed by the inspector on the manufactured workpiece). The difference between the two models leads to the **measurement uncertainty**. See Fig. 2.2.

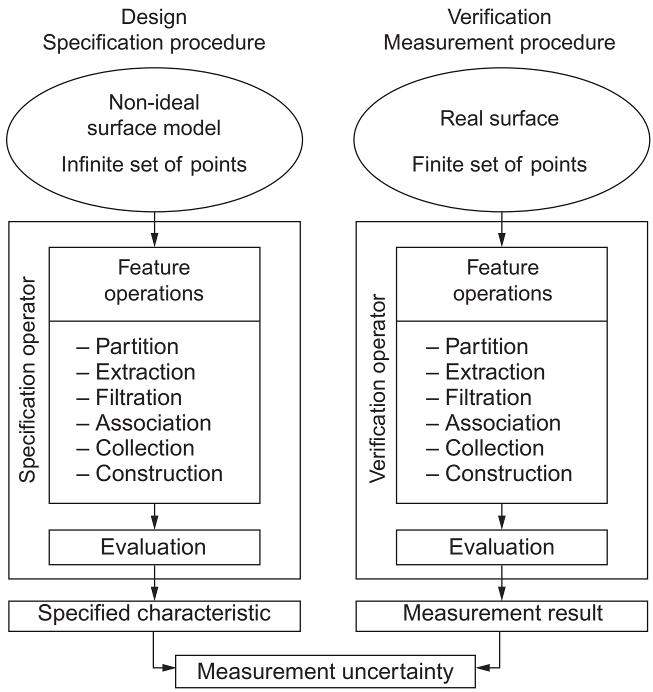


FIG. 2.2 Duality principle

The models incorporate the following **operations** according to ISO 17 450-1:

- **partition** (application part, area)
- **extraction** (assessment of points of the workpiece)
- **filtration** (e.g. probing by a sphere, using a Gaussian filter)
- **association** (fitting an ideal feature to the non-ideal extracted and filtered feature by using a particular method (objective function), e.g. Gauss or Chebyshev or maximum inscribed feature or minimum circumscribed feature)
- **collection** (taking together, e.g. two coaxial cylinders as a common datum)
- **construction** (e.g. building a straight line as a datum (situation feature) by the intersection of two associated planes in the case of a wedge)
- **evaluation** (determining the value of the characteristic (e.g. diameter) or the deviation).

Extraction is not yet standardized. In specification, the entire feature is completely assessed. In verification, a finite number of points are assessed. The difference goes into the measurement uncertainty. For the tip radius, see Fig. 3.1.

The **operation** is a tool required to obtain a feature or the value of a characteristic. Each operation that may influence the result (feature or value of a characteristic) should be specified by an indicated specification or by the use of an ISO default.

For example, the measurement result of a diameter of a shaft measured by a coordinate measuring machine depends on the following operations:

- number of points probed (extraction)
- stylus tip radius (extraction, filtration)
- filter, if any (filtration)
- adjustment of the workpiece to the reference, e.g. minimum circumscribed cylinder (association)
- algorithm used for calculation, e.g. circumference divided by π (evaluation).

Leading are the specification operators. When the verification operators deviate from the specification operators (and they always deviate), the difference goes into the measurement uncertainty. For more information on measurement uncertainty, see 13.11.

Figures 2.3 to 2.6 show examples of partitioning, filtering, collection and construction.

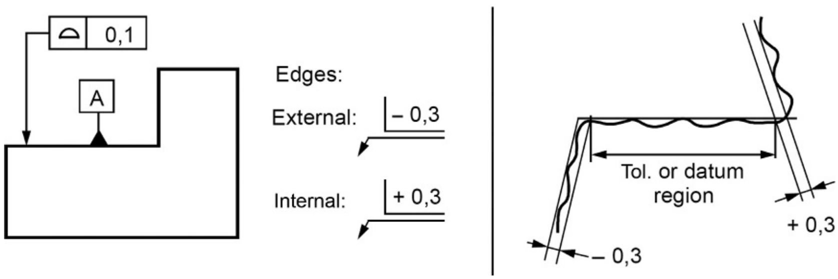


FIG. 2.3 Example of partitioning

There is an ISO standard in preparation to define the borders of the features on a mathematical basis. Until the issuing of this standard, common sense is needed.

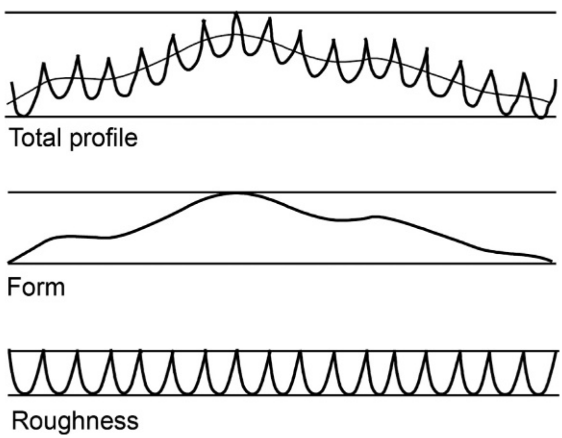


FIG. 2.4 Example of filtering

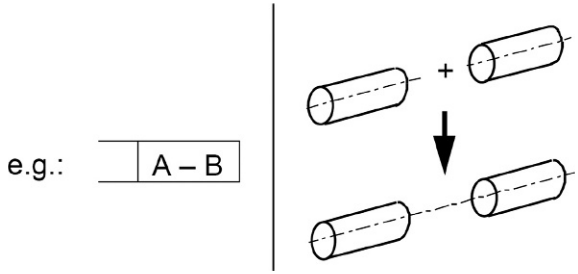


FIG. 2.5 Example of collection

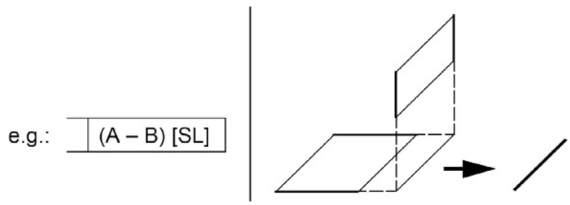


FIG. 2.6 Example of construction

Defaults are specifications of operators to be used when no specification is indicated (applies if not otherwise specified).

When GPS symbols are used and no reference to other standards (e.g. to a national standard) is indicated on the drawing, the ISO GPS standards apply, according to the World Trade Organization (WTO) rules. Then the ISO defaults apply, if any. An example of an ISO default is the Chebyshev requirement (minimax) for tolerances according to ISO 1101. Defaults defined by company standards or national standards must be referred to on the drawing, and they then overrule the pertinent ISO defaults.

The (non-ideal surface) **specification model** refers to an infinite number of extracted points.

The **verification model** refers to a finite number of points.

The **operator** defines a part of the specification.

The **specification operator** is an ordered set of (mathematically formulated) operations, i.e. a theoretical concept to identify a requirement of a feature (feature attributes), using the feature operations as shown in Fig. 2.2.

The **verification operator** is an ordered set of operations implemented physically in a measurement and/or measurement apparatus of the corresponding specification operator.

The **perfect verification operator** is based on a full set of perfect verification operations performed in the prescribed order (see Fig. 2.2). The only measurement uncertainty contributions from a perfect verification operator (perfect measuring instrument) are from physical deviations in the implementation of

the operator. The purpose of calibration is generally to reduce or eliminate these measurement uncertainty contributors.

A **simplified verification operator** includes one or more simplified verification operations and/or deviations from the prescribed order of operations. It causes additional measurement uncertainty contributions, the magnitude of which is dependent on the geometrical characteristics (geometrical deviations) of the actual workpiece. For example, the procedure for measurement of roundness deviations using a V-block and dial gauge is a simplified verification operator. When the workpiece has lobed form deviations, this contributes to the measurement uncertainty and the measurement uncertainty is relatively large.

ISO 17 450-2 defines the following **types of uncertainty and ambiguity** (uncertainty is quantifiable, whereas ambiguity is only qualifiable):

Specification ambiguity due to a specification which is not correctly defined (inherent uncertainty by incomplete and/or mathematically incorrectly defined operations); e.g. step dimension with ± tolerance.

Correlation ambiguity arising from the difference between the specification (operator) and the (real) functional requirement.

Method uncertainty arising from the difference between the (actual) specification operator and the (actual) verification operator, disregarding the physical deviations of the actual verification operator.

Implementation uncertainty arising from the physical deviations of the verification operator.

Measurement uncertainty arising from the difference between the specification operator and the verification operator (including those contributions caused by physical deviations in the implementation of the operator). It is equal to the sum of the method uncertainty and the implementation uncertainty. According to GUM, it is a parameter, associated with the result of a measurement, that characterizes the dispersion of the values that could reasonably be attributed to the measurand (reasonable range for the real (exact) value).

ISO 17 450-2 refers to GUM and VIM. Accordingly, the measurement uncertainty is expressed statistically. According to ISO 14 253-1, the expanded uncertainty U (of a measurement) with a coverage factor 2 of the standard deviation u, if not otherwise indicated, is $\pm U = \pm 2u$.

How to deal with the measurement uncertainty is given by the conformance rules in ISO 14 253-1; see 2.6.

Drawing specification (tolerance) gives the limits which are to be respected.

Manufacturing tolerance gives the limits for measurements (readings) for manufacturing. This is the drawing specification diminished by the

measurement uncertainty U for one-sided tolerances and 2U for two-sided tolerances; see 13.11.2. This is to verify that the drawing tolerance is respected.

2.3 Filters

Filters are used to assess particular components of the surface; e.g. when roundness (circularity) deviations are to be assessed, the roughness shall be eliminated by filtering. Otherwise, the measurement result may be significantly falsified by the roughness, i.e. a perfectly turned circular cylinder would show a roundness deviation of the amount of the roughness depth (see Fig. 2.7).

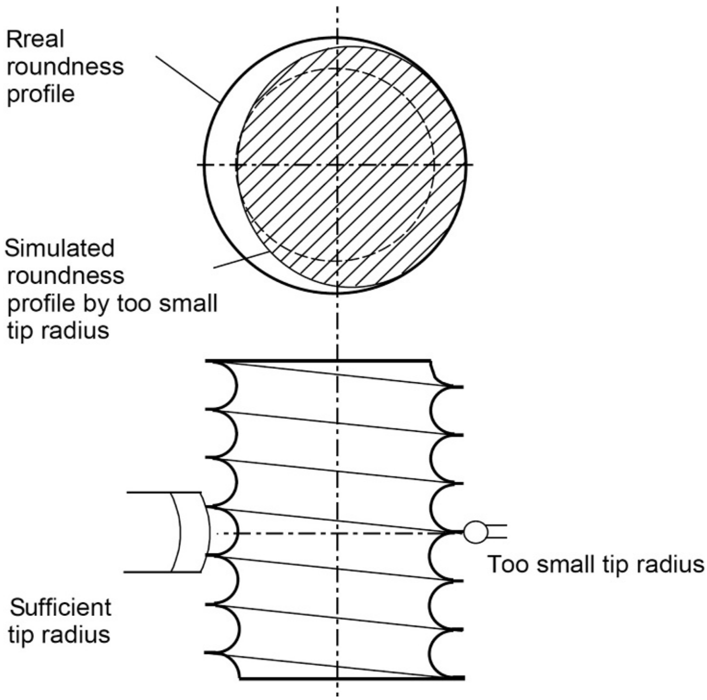


FIG. 2.7 Roughness falsifies form deviation

There are various filter types available; see ISO 16 610-1 ff and Table 3.11. The filter symbols in Table 3.11 are intended for drawing indications.

No default filter has been standardized for GD&T. When not otherwise indicated, morphological filters (sphere filtering) are mainly used, with no other filtration. When using dial gauges, the usual sphere diameter is 3 mm. For very

rough surfaces, a larger sphere diameter may be necessary. For smaller holes and slots, smaller sphere diameters or auxiliary devices (like gauge blocks or mandrels) may be used.

The **nesting index** defines the filter size. It is similar to the grid size of a mesh sieve. For morphological filters, the nesting index is given by the sphere radius, and for Gaussian and spline filters, by the cut-off (e.g. in the case of surface roughness). A discrete surface model consists of a finite number of points. When the nesting index is low, the number of required points of the model is high. When the nesting index tends to zero, the discrete surface model tends towards the skin model of an infinite number of points without filtering.

Morphological filters (other names are sphere filters or ball filters) probe the surface by using a sphere. They retain the high points of the surface and eliminate narrow valleys into which the sphere cannot penetrate; see Fig. 2.8. These filters may be used for datums and fits.

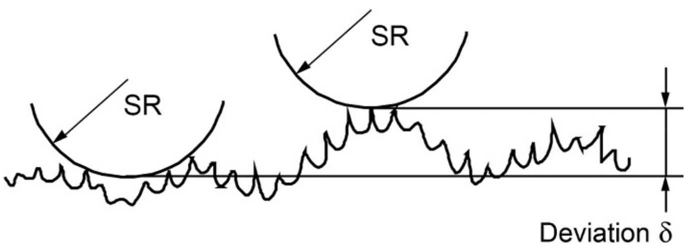


FIG. 2.8 Morphological filter: roughness filtered out and form deviations retained; waviness may be only partially filtered out

Gaussian filters and spline filters separate high- and low-frequency components (short and long wavelength components) of the surface. Figure 2.4 shows the components of the surface separated by different filter sizes. (In the case of a roller bearing, the deviations of median wavelength are more detrimental than in the others and should be toleranced separately and very tightly.)

Gaussian and spline filters may eliminate the high points of the surface (see Fig. 2.10) and are therefore not suitable for datums and fits. Spline filters work similarly to Gaussian filters, but do not have some of the disadvantages of Gaussian filters (end effects, sometimes is the mean line out of the material; see ISO 16 610-1). Spline filters have similar properties as Gauss filters but they seem to be better than the Gaussian filters.

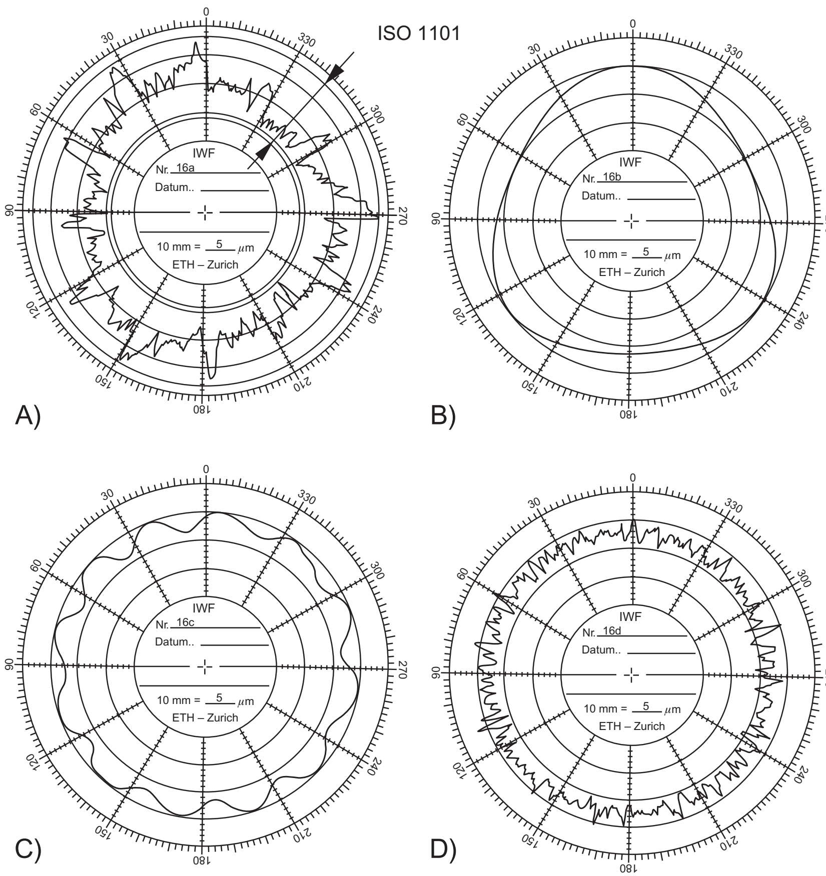


FIG. 2.9 Roundness deviation (same trace) a) without, b) to d) with different filters *(Source: Wirtz ETH Zuric)*

Robust filters do not create distortions at outliers. RC filters and non-robust Gaussian and spline filters do and can thus falsify the measurement result.

Spline wavelet filters allow detection of outliers.

An overview of the advantages and disadvantages of the various filters is given in ISO 16 610-1.

Figure 2.9 shows filtering of the same workpiece feature with a Gaussian filter G by different nesting indexes. Figure 2.10 shows the filter effect of a Gaussian filter for a narrow form deviation.

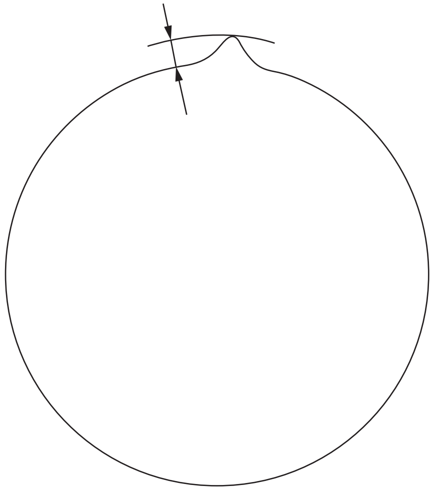


FIG. 2.10 Gaussian and spline filters do not assess the full height of narrow form deviations

Filters for datums: The minimum rock requirement is a filter that eliminates voids on the surface.

ISO 5459:2011 specifies that the filtration shall retain the highest (material outward) points in the integral feature (shall eliminate voids). The minimum rock requirement does this.

For the next edition of ISO 5459, it is planned to specify for flat and convex datum features to use the convex hull. This can be explained as a rubber sheet spanned over the surface.

For concave features, α hull filtering is planned: that is, filtering by a sphere. The radius of the sphere shall be half of the smallest radius of the nominal datum feature, or one-fourth of the diameter of a hole.

2.4 Datums

For the definition of datums, ISO 17 450-1 and ISO 5459 refer to the **invariance classes** of ideal features (geometrical elements). All ideal features belong to one of the following seven invariance classes. Each feature has six degrees of freedom (translations along the axes x, y, z and rotations around the axes x, y, z). The invariance class corresponds to the degrees of freedom (which are remaining and not locked). It describes the displacement of the feature (translation, rotation) for which the feature is kept identical in space.

Invariance class	Invariance degrees (remaining degrees of freedom)
Complex	None
Prismatic	1 Translation
Revolute	1 Rotation
Helical	Combined translation and rotation
Cylindrical	1 Translation and 1 rotation
Planar	1 Rotation and 2 translations
Spherical	3 Rotations

The **situation feature** is a point, straight line, plane or helix, which defines the location and/or orientation of an ideal feature (datum). Examples follow.

Invariance class	Example	Situation feature	Invariance degrees (degrees of freedom)
Complex	Saddle	Plane straight line point	None
Prismatic	Prism wedge	Plane straight line	Translation along axis
Revolute	Cone torus	Straight line point	Rotation around axis
Helical	Helix	Helix	Translation and rotation around axis
Cylindrical	Cylinder	Straight line	Translation, rotation around axis
Planar	Plane	Plane	1 Rotation $\perp$ to plane 2 translations in plane
Spherical	Sphere	Point	3 Rotations around centre

Screw threads are considered as belonging to the invariance class cylindrical (because the pitch, i.e. the movement along the axis, is of no interest for the datum).

The **datum specification** refers to a situation feature, derived from an ideal feature associated with the extracted feature. If not otherwise specified, the objective function for the association is minimax (Chebyshev) and the constraint is tangent (external) to the material. For filtration, see the following paragraphs. See ISO 5459.

Figure 2.12 shows the history of the development of the definitions for datums. The minimum rock requirement according to ISO 5459:1981 and

ISO TR 5460 is explained in Fig. 3.81. It is an easily understandable and, in the most functional cases, a near and sufficient approximation to the correct and standardized definition of the datum.

According to ISO 5459:2011 cl. A.2.1, the minimax criterion (Chebyshev) between the filtered datum feature and the associated (ideal) situation feature applies. The filter method should remove the voids in the datum feature surface. The minimum rock requirement does this; see Fig. 3.81. The differences between these two standards are in most cases negligible. In this book, the simpler minimum rock method is used. In cases of dispute, the correct method according to the valid standard ISO 5459 should be used.

Using the minimum rock requirement, it is usual to disregard the small border portions of the datum feature when they would cause an obviously unreasonable orientation of the datum; see Fig. 2.11. For programming coordinate measuring systems, a mathematical solution for this correction is necessary.

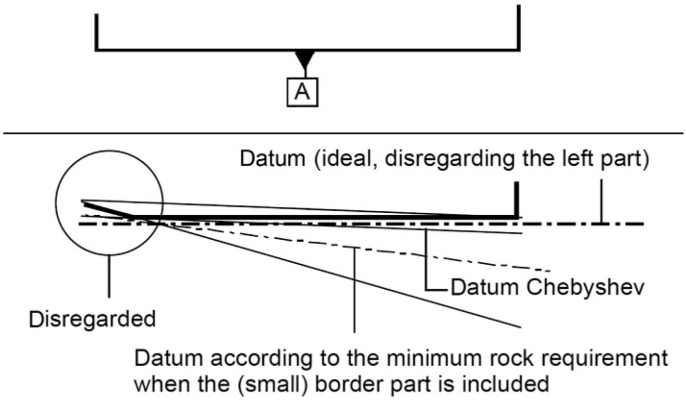


FIG. 2.11 Minimum rock requirement, disregarding short portions of the datum feature that would result in an unreasonable datum (manual correction)

It is planned for the next issue of ISO 5459 to supersede the minimax criterion for datums by the criterion of least squares external (outside) material. The result is considered to be closer to the minimum rock requirement disregarding portions of the datum feature which would result in unreasonable datums (Fig. 2.12).

When the drawing indication for the datum in the tolerance indicator contains the datum letter only, all situation features are applied. When only one situation feature is to be applied, e.g. the apex of the cone and not the axis, then the applied situation feature is to be indicated in square brackets, e.g. [PT] for the apex of the cone and not the straight line of the cone (see Fig. 3.94).

Datum: (ideal) situation feature (point, straight line, plane) derived from contacting (ideal) association feature

Location of datum:

Plane:	Outer contacting point of datum plane
Feature of size:	Centre of the outer contacting max. inscribed or min. circumscribed ideal feature

Orientation of datum:

Up to 2011-08:	Min.-rock.-requirement
Since 2011-08:	Minimax of contacting ideal feature with filtered datum feature

Fiiltering of datum feature:

Up to 2011-08:	Min.-rock.-requirement
Since 2011-08:	Filtering with removing voids of the datum feature (not precise defined) (e.g. by min.-rock-requirement)

FIG. 2.12 History of the development of the definition of datums; [5] gives a description of the development of the definition of datums

2.5 Principle of independency and Ⓜ or Ⓛ

With the new approach, a rigorous application of the principle of independency is planned (for the future).

In order to keep the rule "what is indicated applies", it shall always be indicated, with multiple features and position tolerances with Ⓜ or Ⓛ, whether SZ or CZ applies.

Ⓜ or Ⓛ without SZ or CZ is former practice with the meaning of CZ. When there is a datum system that locks all degrees of freedom, the indication of CZ may be omitted.

In the case shown in Fig. 2.13, separate gauging may be appropriate to a function where a lever with two shafts turns around a centre hole and must fit with the other shaft into each of the other holes separately.

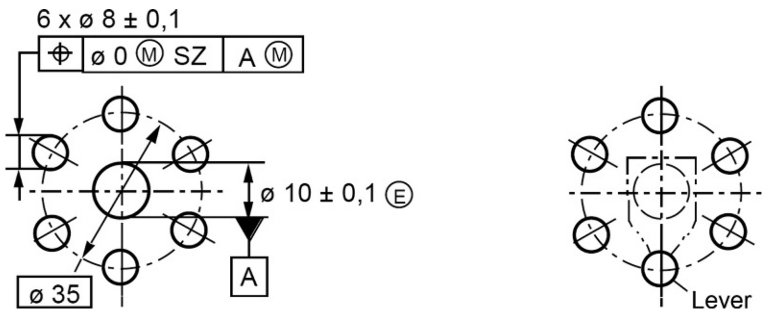


FIG. 2.13 Pattern of holes related to a datum hole with Ⓜ, SZ indicated, (left) indication for the holes, (right) assembling with lever

Figure 2.14 shows an example where the gauge with seven shafts must fit into the holes.

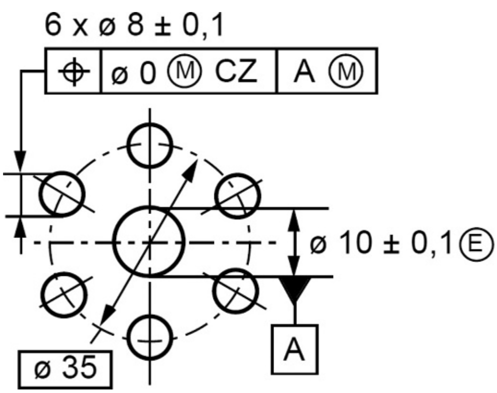


FIG. 2.14 Pattern of holes related to a datum hole with Ⓜ, CZ indicated

Figures 2.15 and 2.16 show examples with both possibilities: a) simultaneous gauging and b) separate gauging.

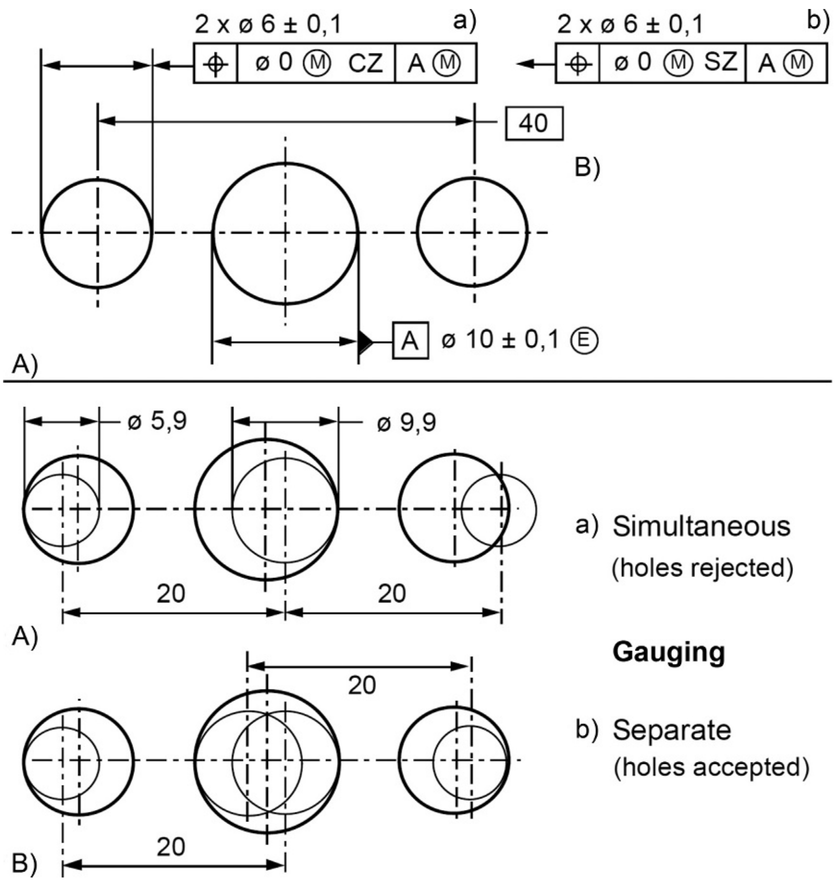


FIG. 2.15 Pattern of holes related to a datum hole with Ⓜ, a) simultaneous gauging, b) separate gauging

A similar situation appears when multiple features are toleranced with Ⓜ but without datum. Figure 2.17 shows an example. With SZ, separate gauging is allowed so that the open end wrench fits but the box wrench does not.

The examples in Figs 2.13 to 2.15 concern only the relationship of the features to each other and to a single datum. It is to be observed that the features additionally are to be toleranced in a complete datum system, locking all degrees of freedom; see, e.g. Fig. 5.6.

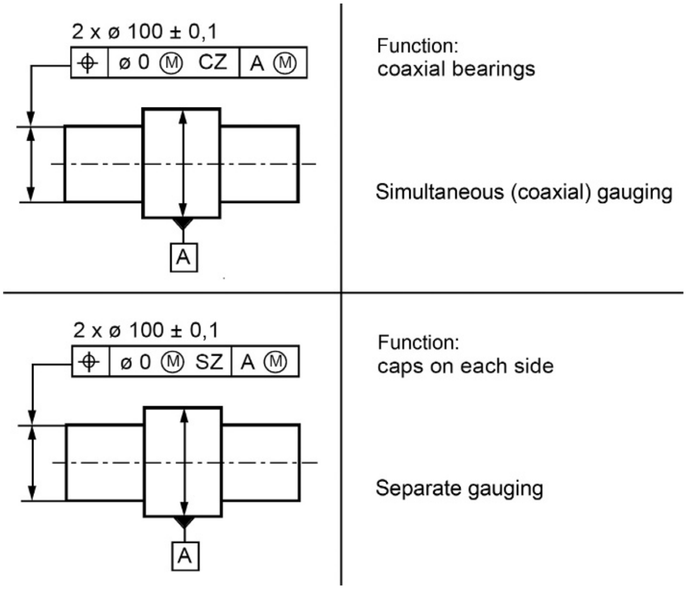


FIG. 2.16 Coaxial shafts related to a datum shaft with Ⓜ, a) simultaneous gauging, b) separate gauging

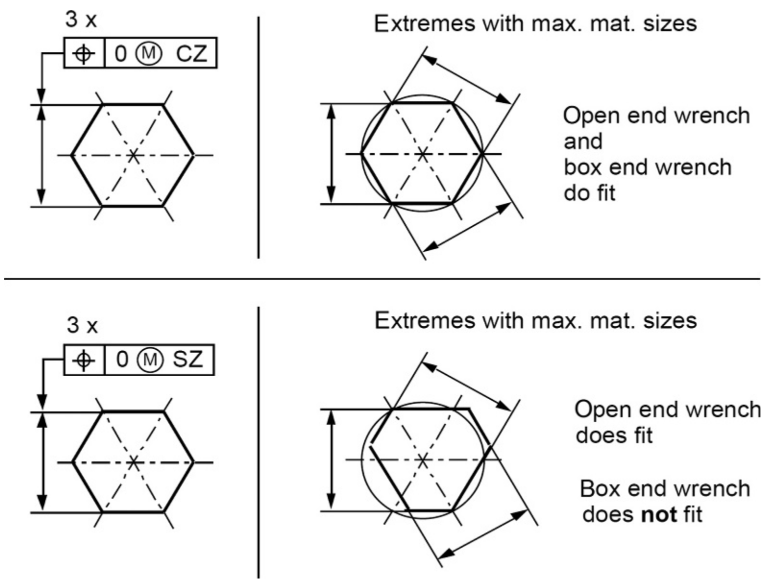


FIG. 2.17 Hexagon, widths across flats with Ⓜ, CZ simultaneous gauging, SZ separate gauging

In the past, multiple features were position toleranced with Ⓜ or Ⓛ without CZ or SZ. In these cases, CZ applied; see Figs 2.18 and 2.19. In order to avoid misunderstandings in these cases, the manufacturer should always work according to CZ, if not otherwise agreed to with the purchaser. This is in conformance with the new definition of Ⓜ and Ⓛ in ISO 2692.

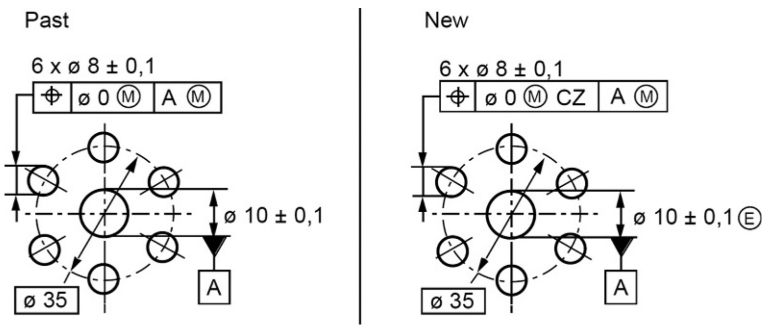


FIG. 2.18 Pattern of holes related to a datum hole with Ⓜ, former practice

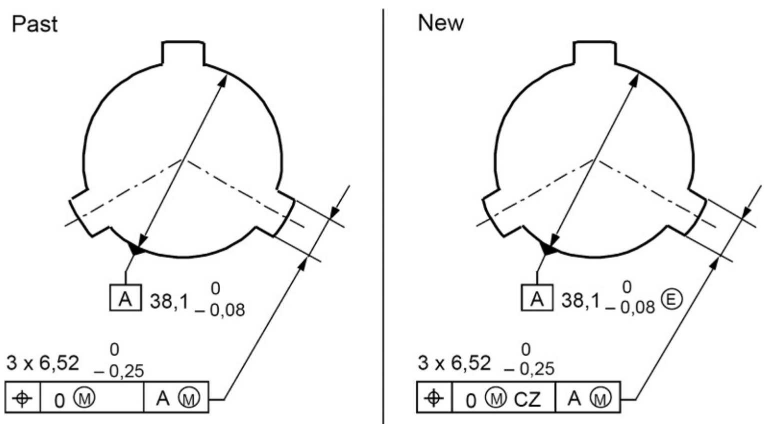


FIG. 2.19 Indication of tolerancing of splines, past and new

2.6 Conformance rule

ISO 14 253-1 provides conformance rules: that is, how to decide whether a workpiece conforms to the specification or not. According to this standard, the default is that the measurement uncertainty goes to the debit of the measuring party; see Fig. 2.20.[1)] That is, if the indication is within the uncertainty range,

[1)]Intermediates (merchants) shall not inspect; they shall use the manufacturer's inspection documentation.

for an outgoing inspection the workpiece shall not be delivered and for an ingoing inspection the workpiece shall not be rejected; see Fig. 2.20 and 13.118. This means the tolerance may in reality be exceeded, but the workpiece cannot be rejected. This may be acceptable in many cases, but in some cases it is not acceptable. Then another rule must be agreed upon, such as the following:

In cases of the envelope requirement Ⓔ, the maximum material requirement Ⓜ and the least material requirement Ⓛ, the drawing specifies the absolute functional limit. For the contract, a contractual tolerance is to be applied, e.g. that the tolerance at this limit is to be diminished by 10%. Then an inspection is to be applied, in which the maximum measurement uncertainty is 10% of the tolerance.

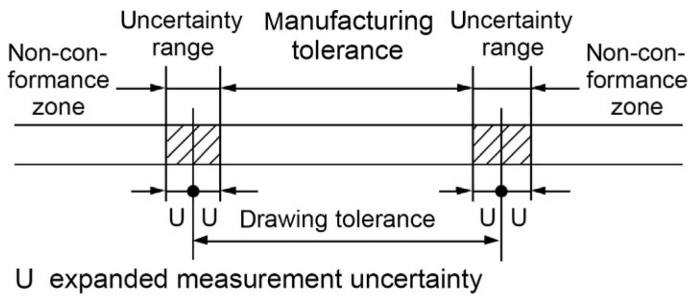


FIG. 2.20 Measurement uncertainty and decision rule

The smaller the measurement uncertainty, the more remains as the tolerance for the manufacturer. See also 13.11.

2.7 New approach principles and rules

The following principles and rules apply with the new approach:

Invocation principle

If not otherwise indicated, the ISO standards for GD&T apply.

Independency principle, ISO 8015

Each requirement must be indicated and each requirement must be respected (no hidden rule, such as the envelope requirement must be respected for each feature of size without respective indication). The requirements are to be respected separately, if not other-wise indicated, e.g. by CZ or SIM.

If not otherwise indicated, each requirement is to be respected separately. If any indication shall be disregarded, a disclaimer is to be indicated: for example, OZ when a theoretically exact dimension (TED) of a cylinder diameter is indicated, but the profile surface tolerance shall be applied disregarding this TED.

Drawing definitive principle, ISO 8015

What is not specified is not required. The specification may be directly indicated or be given by the reference to a default (ISO standard default, national standard default or company standard default). However, the ISO standard defaults apply without indication.

Operator principle, ISO 17 450-1&2

Definition of specifications by an ordered set of operations; see Fig. 2.2.

Duality principle, ISO 17 450-1&2

From the specification operator the theoretical verification operator is derived; see Fig. 2.2. The practical specification operator may deviate and increases the measurement uncertainty.

Uncertainty principle, ISO 17 450-2

From the difference between the specification operator and the verification operator, the measurement uncertainty is derived. The specification operator may (should not) have a specification ambiguity and a correlation ambiguity; see 2.2.

Conformance rules, ISO 14 253-1

Measurement uncertainty goes to the debit of the measuring party, if not otherwise specified.

Decimal rule, ISO 8015

After the last indicated digits of dimensions and tolerances, zero applies.

Reference temperature, ISO 1

20°C applies.

Rigid workpiece principle, ISO 8015

By default, a workpiece shall be considered as having infinite stiffness.

Responsibility principle, ISO 8015

Ambiguities of the drawing indications are the designer's responsibility.

2.8 Application of the new approach

The new approach with its sophisticated terms and definitions is relevant for

– programming of measuring instruments (like coordinate measuring machines)
– definition of very small tolerances
– precise evaluation of the measurement uncertainty.

For normal designer use (geometrical tolerances larger than 20 μm), it is considered as sufficient (and easier to understand) to use what is described in 3.2 as ISO defaults. When very small tolerances are required, the precise definition of the operators (filter, etc.) according to the new approach may be necessary, such as for the piston of the injection pump of a diesel engine, where tolerances of 1 μm or less are required for roughness, waviness, form and size.

In the existing ISO standards, there is **no default** (= if not otherwise stated) **for filtration** (type and nesting index of the filter, e.g. morphological filter with value of tip radius or Gaussian filter with cut-off value).

If necessary, the conditions are to be specified. ISO 1101 gives the rules for the indication. See 2.3 and 3.2.1.

For datums, the association default is Chebyshev external. For filtering, see 2.4. However, for a designer's normal use, the concept of the minimum rock requirement is still sufficient.

For multiple features with Ⓜ without datum or related to a datum with Ⓜ, the rules given in 2.5 shall be followed in order to avoid misunderstandings in the future.

In general, it can be assumed that the new approach provides the possibility for more precise specifications, but for the rest there is no dramatic change to the former ISO standards (ISO 1101, ISO 5459, ISO 5458, ISO 1660, ISO 8015). However, according to ISO 14 253-1, if not otherwise specified, the measurement uncertainty is to be subtracted from the tolerance when the supplier measures and added to the tolerance when the purchaser measures. In the latter case, the tolerance may be actually exceeded by the amount of the measurement uncertainty. If that, for functional reasons, is not acceptable, special procedures are to be agreed upon between the parties. See 2.6.

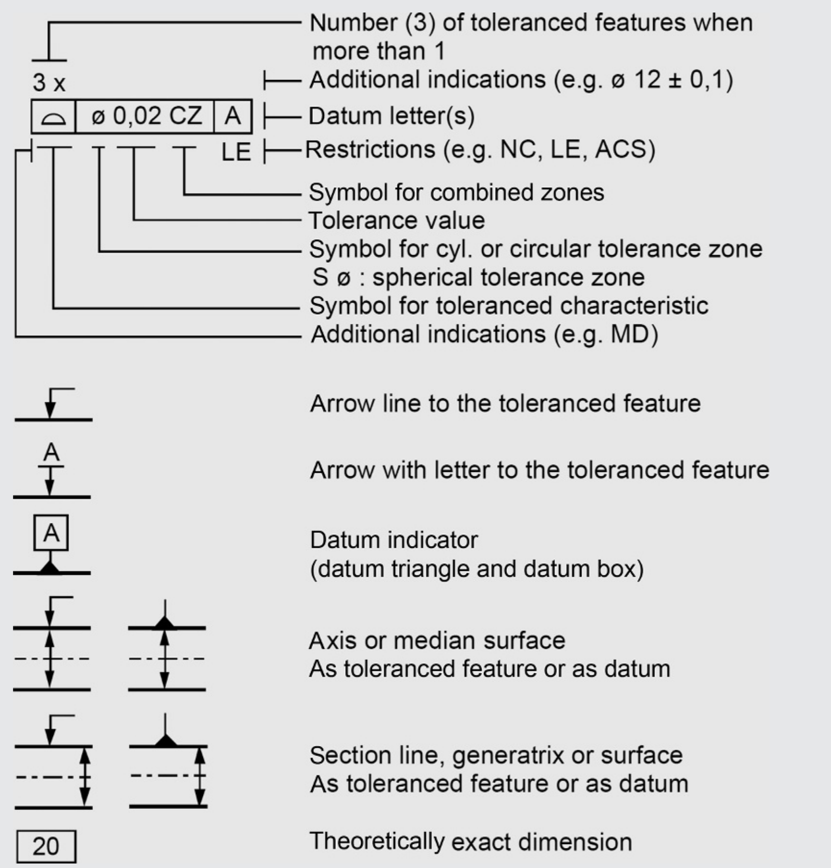
Chapter 3

Basics of Geometrical Tolerancing

3.1 Symbols

The symbols for the drawing indicators of geometrical tolerances according to ISO 1101, ISO 5459, ISO/TS 8062-2 and ISO 10 579 are shown in Tables 3.1 to 3.10.

TABLE 3.1 Symbols for geometrical tolerancing, general, ISO 1101, ISO 5459



Geometrical Dimensioning and Tolerancing for Design, Manufacturing and Inspection
https://doi.org/10.1016/B978-0-12-824061-8.00007-4

TABLE 3.2 Symbols for geometrical characteristics, ISO 1101

		1)	2)	
Form (without datum)	**Line**	⌒	⌖ with SZ[3)]	
	Straightness			—
	Roundness			○
	Surface	⌓	⌖ with SZ[3)]	
	Flatness			▱
	Cylindricity			⌭
Orientation (with datum(s))	**Inclination**	⌒ ⌓	⌖	∠
	Parallelism	with ><		//
	Perpendicularity			⊥
Location (with datum(s))	**Location/position**	⌒ ⌓	⌖	
	Coaxiality			◎
	Symmetry			⌯
Run-out (with datum(s))	**Circular run-out**	↗		
	Total run-out			⌰

1) For **integral** features (intersection lines, surfaces)
2) Für **derived** features (axes, median surfaces)
3) With SZ only when more than 1 feature

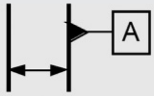
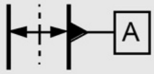
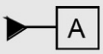
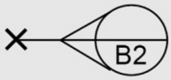
The symbols for toleranced characteristics shown in Table 3.2, columns 1) and 2), are sufficient for all kinds of geometrical tolerancing. The symbols of the last column are superfluous but still frequently used, so they are explained. It is recommended, and in this book applied, to use profile and circular run-out tolerancing for integral features (surfaces) and to use position tolerancing for derived features (median lines, median surfaces).

Some tolerancings can only be expressed by using profile or position tolerancing, e.g. tolerancing of the form of a torus or orientation tolerancing with a datum system where some datums are still valid for location; see Fig. 3.34.

It is further recommended to use a surface profile tolerance as a general tolerance. Then the entire workpiece is completely toleranced. Further necessary additional or alternative tolerances can be chosen as profile or circular run-out tolerances for integral features or as position tolerances for derived features; see 6 and Fig. 6.14.

Table 3.11 shows symbols for **filtering methods**. There is no standardized default for filtering of toleranced features. The usual filtering is the "closed ball" filtering (filtering by a tracing sphere) with the nesting index (tip radius) of 1,5 mm (drawing indication: CB 1,5). This corresponds to a measurement with a dial gauge with 1,5 mm tip radius.

TABLE 3.3 Symbols for datums, ISO 5459

	Datum integral feature (surface, section line)	
	Datum derived feature (axis, median surface)	
A - B	Common datum of A and B	
A - A	Common datum of multiple features A	
[PT]	Situation feature: point	
[SL]	Situation feature: straight line	
[PL]	Situation feature: plane	
><	No translation (orientation only (rotation))	
[-]	Datum only for definition of sec., tert. datum	1)
$[R_x]$, $[R_y]$, $[R_z]$	Locked rotation around x-, y-, z-axis	1)
$[T_x]$, $[T_y]$, $[T_z]$	Locked translation along x-, y-, z-axis	1)
 A1, 2 ,3	Datum defined by three datum targets	
	Fixed datum target	
	Movable datum target	
[CF]	Contacting feature	
[DV]	Dimension (between datums) variable	
[DF]	Dimension (between datums) fixed	

1) ISO/DIS 5459:2017

TABLE 3.4 Further symbols for geometrical tolerancing

Symbol	Description	ISO
LE	Line elements toleranced	1101
ACS	Any cross section (tolerance and datum)	1101
SCS	Specific cross section	1101
ALS	Any longitudinal section	1101
◄─►	Between	1101
─►	Increasing zone (substituted by ◄─►)	
CZ	Combined zones simultaneous gauging	1101
SZ	Separate zone, separate gauging	1101
CT	Common tolerance	14 405-1
UF	United feature	1101
UZ	Unequally disposed tolerance zone	1101
OZ	Offset zone	1660
NC	Not convex	1101
Ⓐ	Derived feature (axis, median feature)	1101
Ⓟ	Projected tolerance zone	1101
Ⓕ	Flexible feature in free state	10 579
LD	(Least) minor cylinder of thread, spline or gear	1101
PD	Pitch cylinder of thread, spline or gear	1101
MD	Major cylinder of thread, spline or gear	1101
Ⓔ	Envelope requirement	14 405-1
Ⓜ	Maximum material requirement	2692
Ⓛ	Least material requirement	2692
Ⓡ	Reciprocity requirement	2692
SIM	Simultaneous requirement	1101
(AD)	Altered (non-ISO) default	8015

TABLE 3.5 Symbols for datum targets, ISO 5459

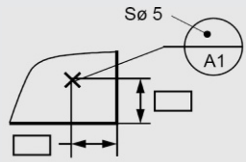
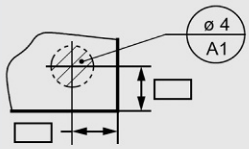
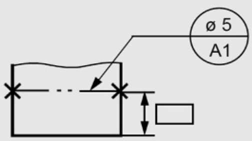
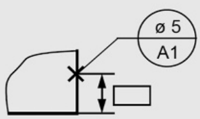
	Datum target point (contacting feature: sphere Sø 5)
	Datum target area (contacting feature: flat area ø 4)
	Datum target line (front view) (contacting feature: cylinder ø 5)
	Datum target line (side view) (contacting feature: Cyinder ø 5)

TABLE 3.6 Symbols for additional indicators (additional to the tolerance indicators), ISO 1101

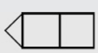
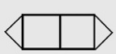
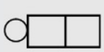
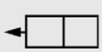
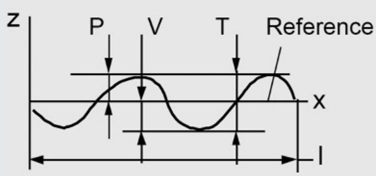
	Intersection plane indicator
	Orientation plane indicator
	All around collection plane indicator
	Measurement direction indicator

TABLE 3.7 Additional symbols for tolerancing of castings, ISO/TR 8062-2

Meaning	
Parting surface	
Parting surface	Movable / fixed
Taper, variable nominal shape	
Taper, variable tolerance	
Moulded condition	
Intermediate (pre)machined	
Final machined	
Provided by supplier	

TABLE 3.8 Symbols for toleranced parameters (characteristics), ISO 1101

Symbol	Parameter (characteristic)
T	Peak to valley (total) = default
V	Reference to valley
P	Reference to peak
Q	Root mean square (RMS)



$$Q = \sqrt{\frac{\int z^2(x)\,dx}{l}}$$

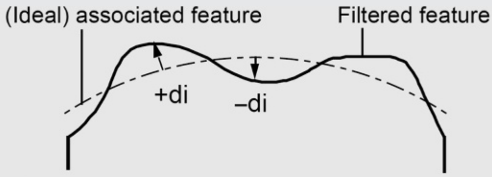
TABLE 3.9 Symbols for associated toleranced features, ISO 1101

	Toleranced features:
Ⓒ	Minimax (Chebyshev) feature[1)]
Ⓖ	Gauss feature
Ⓣ	Tangent feature[2)]
Ⓧ	Maximum inscribed feature
Ⓝ	Minimum circumscribed feature
Ⓟ	Projected tolerance zone
Ⓐ	Derived feature (axis, median surface)

[1)] Indication of Ⓒ may be omitted, because default

[2)] Gauss with constraint outside material

TABLE 3.10 **Symbols for associated reference features (ISO 1101) and for datums (ISO/DIS 5459:2017)**



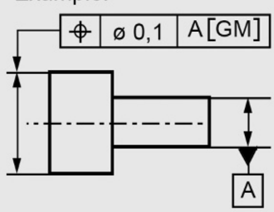
Objective function

Symbol	Name	Description
C	Chebyshev (= $L\infty$ -norm)	(+di max + (|-di| max)) minimized
G	Gauss (= L2-norm)	$\sum\|di\|^2$ minimized
K	min. volume (= L1-norm)	
X	max. inscribed	
N	min. circumscribed	
Lp-norm	Lp-norm p different 1, 2,.∞	$\sum\|di\|^p$ minimized

Material constraint

Symbol	Description
E	External (outside) material
I	Internal (inside) material
M	Without mat, constraint
+	Shifted tangent outside mat.
–	Shifted tangent inside mat.
xx %	Material ratio percentage

Example:



ISO default: for toleranced features: C, for datums : CE

TABLE 3.11 Symbols for filterings, ISO 1101

Designt.	Symbol	Name	Standard	Nesting index
FALG	**G**	Gaussian	ISO 16 610-61	Cutoff length 1)
FALS	**S**	Spline	ISO 16 610-62	Cutoff length 1)
FALW	**W**	Spline Wavelet	ISO 16 610-69	Cutoff length 1)
FAMCB	**CB**	Closing ball	ISO 16 610-81	Ball radius
FAMCH	**CH**	Closing horizontal Segment	ISO 16 610-81	Segment length
FAMOB	**OB**	Opening ball	ISO 16 610-81	Ball radius
FAMOH	**OH**	Opening horizontal Segment	ISO 16 610-81	Segment length
FAMAB	**AB**	Alternating ball	ISO 16 610-89	Ball radius
FAMAH	**AH**	Alternating horizontal Segment	ISO 16 610-89	Segment length
FARG	**RG**	Robust gaussian	ISO 16 610-71	Cutoff length 1)
FARS	**RS**	Robust spline	ISO 16 610-72	Cutoff length 1)
FPLG	**G**	Gaussian	ISO 16 610-21	Cutoff length 1)
FPLS	**S**	Spline	ISO 16 610-22	Cutoff length 1)
FPLW	**W**	Spline wavelet	ISO 16 610-29	Cutoff length 1)
FPMCD	**CD**	Closing disk	ISO 16 610-41	Disk radius
FPMCH	**CH**	Closing horizontal segment	ISO 16 610-41	Segment length
FPMOD	**OD**	Opening disk	ISO 16 610-41	Disk radius
FPMOH	**OH**	Opening horizontal segment	ISO 16 610-41	Segment length
FPMAD	**AD**	Alternating disk	ISO 16 610-49	Ball radius
FPMAH	**AH**	Alternating horizontal segment	ISO 16 610-49	Segment length
FPRG	**RG**	Robust gaussian	ISO 16 610-31	Cutoff length 1)
FPRS	**RS**	Robust spline	ISO 16 610-32	Cutoff length 1)
F2RC	**2RC**	2RC	ISO 3274	Cutoff length 1)

1) for circumferences: cutoff UPR

F, filter; **A,** arial, **P,** profile; **P,** linear; **M,** morphological; **R** robust

Figure 3.1 shows the filtering effect of the closing ball filter CB1,5 (dial gauge).

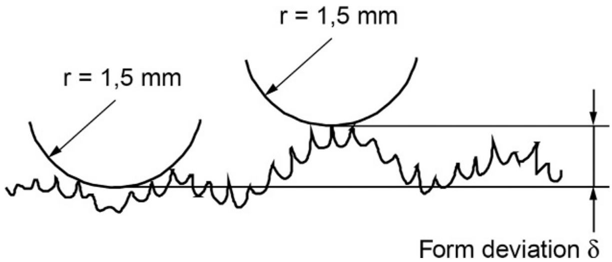


FIG. 3.1 Filtering effect of the closing ball filter (dial gauge)

Figure 3.2 shows the **arrangement** of tolerance indications, ISO 1101.

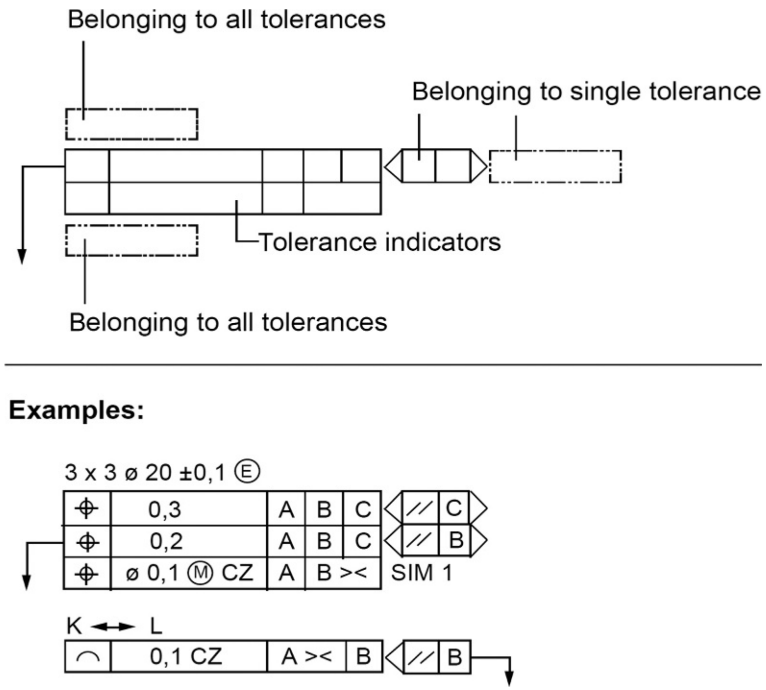


FIG. 3.2 Arrangement of tolerance indications

Figure 3.3 shows **further possibilities** according to ISO 1101 and ISO 5459 for the **indication of the derived feature**. Decisive for the datum is that the datum indicator and the diameter indication are located at the same leader line. This indicator for datum features is only possible for cylinders. It is recommended to **prefer** the indication, as shown in Table 3.1.

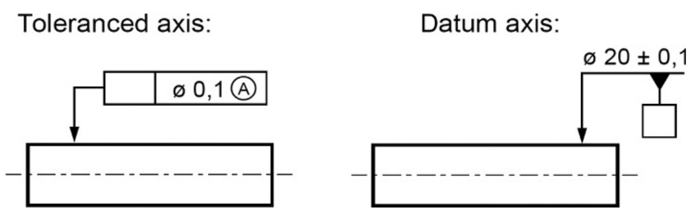


FIG. 3.3 Tolerance for the derived feature, application for datums **not** recommended

Figure 3.4 shows **further indications** according to ISO 1101, which are **not** recommended to use, because they deviate from the general rule to use profile tolerances for **integral features** and position tolerances for **derived features**.

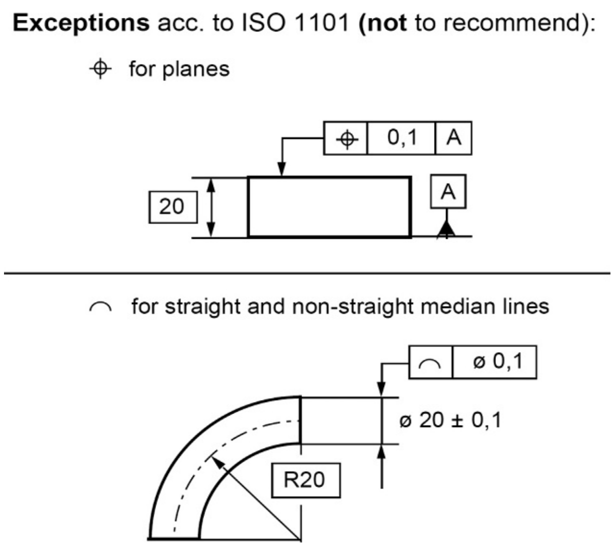


FIG. 3.4 Possibilities for tolerancing planes with position tolerances and derived linear features (lines) with profile tolerances, **not** recommended

Figure 3.5 shows the rule for **screw threads** according to ISO 1101. As default, the pitch diameter applies.

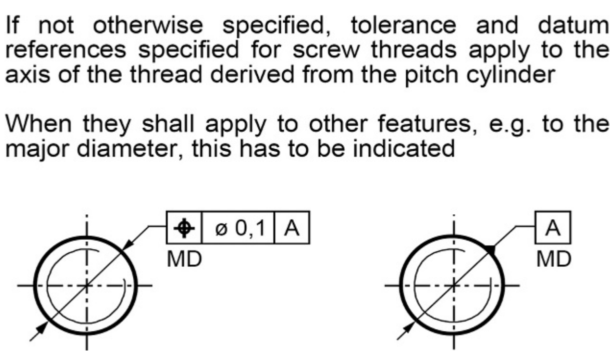


FIG. 3.5 Screw threads, MD (major diameter) applies

The possible **modifiers for threads, gears and splines** are shown in Table 3.12.

TABLE 3.12 Modifiers for threads, gears and splines, ISO 1101

MD	Major diameter
PD	Pitch diameter
LD	Least diameter

For **gears and splines**, the specific feature to which the tolerance or the datum applies must always be indicated. (There is no default.)

Figure 3.6 shows the indication of the **length or area of a specified location**.

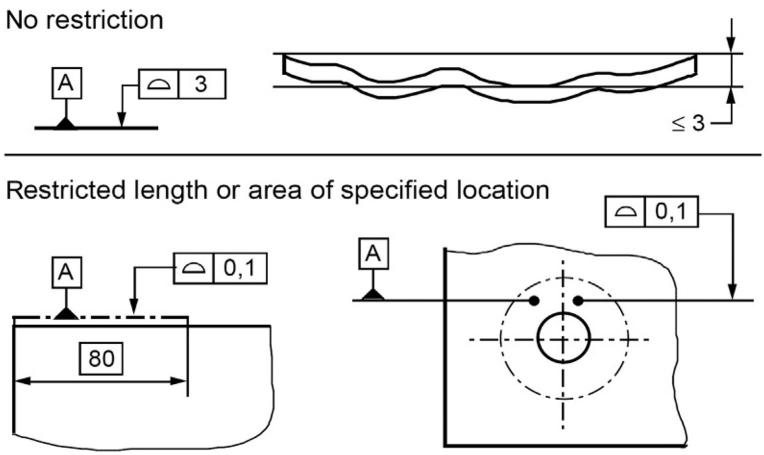


FIG. 3.6 Indication of the length or area of a specified location, ISO 1101, ISO 5459

Figure 3.7 shows the indication of a (straightness) tolerance of a **restricted length located anywhere**.

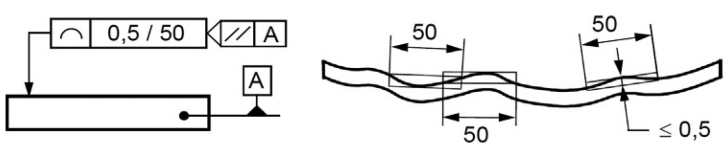


FIG. 3.7 Tolerance for a restricted length located anywhere, ISO 1101

Figure 3.8 shows the application of the **between symbol**. It applies to the surfaces which include the arrow.

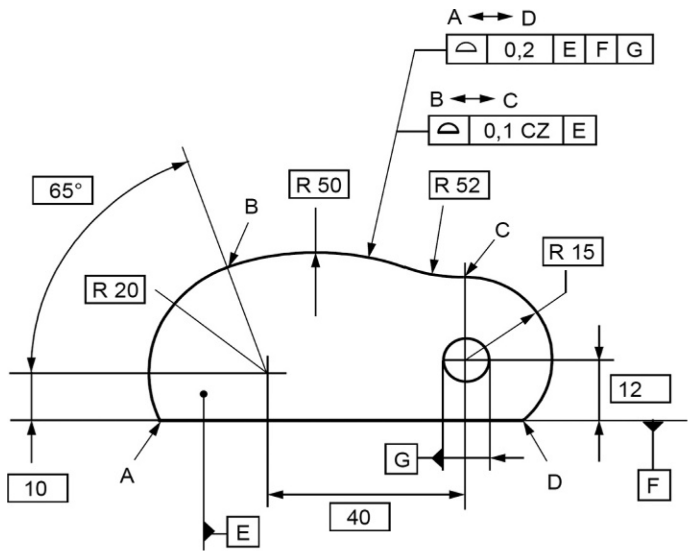


FIG. 3.8 Between symbol, ISO 1101

Figure 3.9 shows the application of the symbols **all around and all over**. With all over, the tolerance applies to all surfaces (the entire workpiece is toleranced).

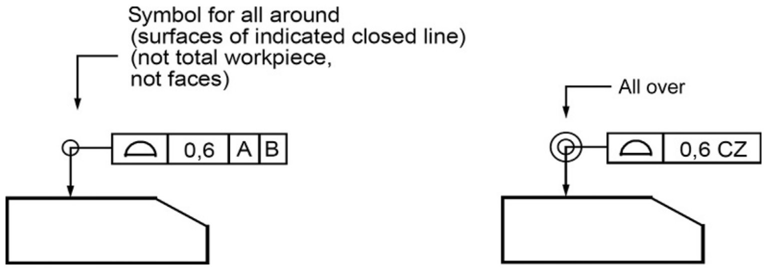


FIG. 3.9 Symbols all around and all over, ISO 1101

Figure 3.10 shows the application of the symbol **CZ combined zones**.

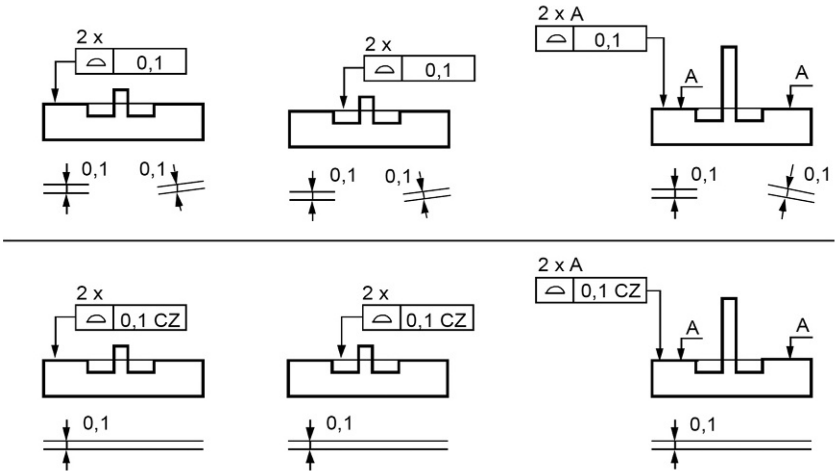


FIG. 3.10 Combined zones CZ, ISO 1101

Figure 3.11 shows no longer allowed **indications on centre lines**. It is unclear to which feature the indication belongs (when the centre line belongs to more than one feature).

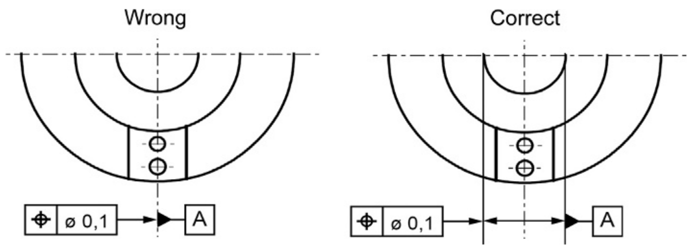


FIG. 3.11 No longer allowed indication on centre line, ISO 1101, ISO 5459

Figure 3.12 shows the application of **letters for the identification of toleranced features**. A distinction must be made between toleranced features and datum features. They have different definitions and have different indication symbols (arrow and triangle).

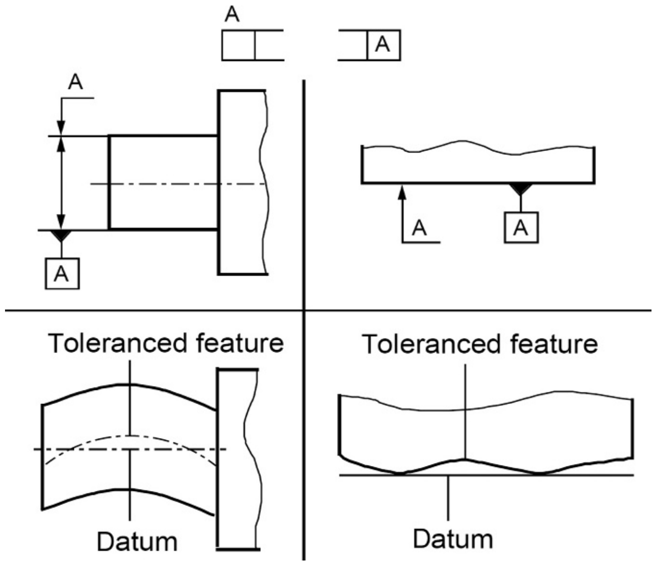


FIG. 3.12 Letters for identification of features, ISO 1101, ISO 5459

Figure 3.13 shows an example for a **simplified drawing indication**. The complete symbol is to be explained near the drawing title block.

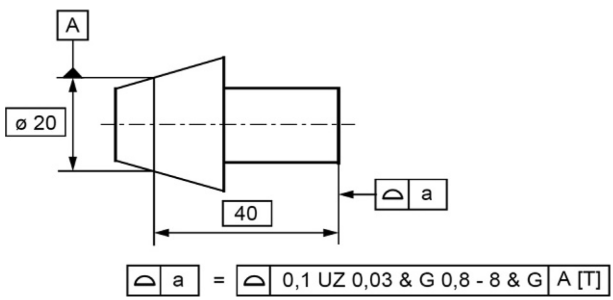


FIG. 3.13 Simplified drawing indication, ISO 1101

Figure 3.14 shows examples of **indications on visible and hidden surfaces**.

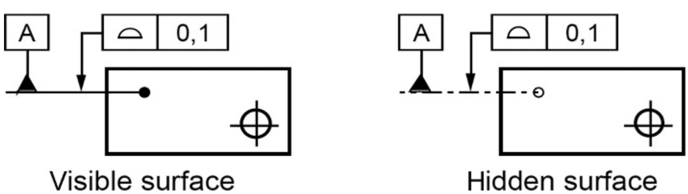


FIG. 3.14 Indications on visible and hidden surfaces, ISO 1101, ISO 5459

Equally spaced on a circle needs no angle indicated. The number of features is given by the tolerance indication. See Fig. 5.26.

The **form of the tolerance zone** depends on the reference (ideal) feature or is established by two parallel planes oriented relative to the datum; see Fig. 3.15 above.

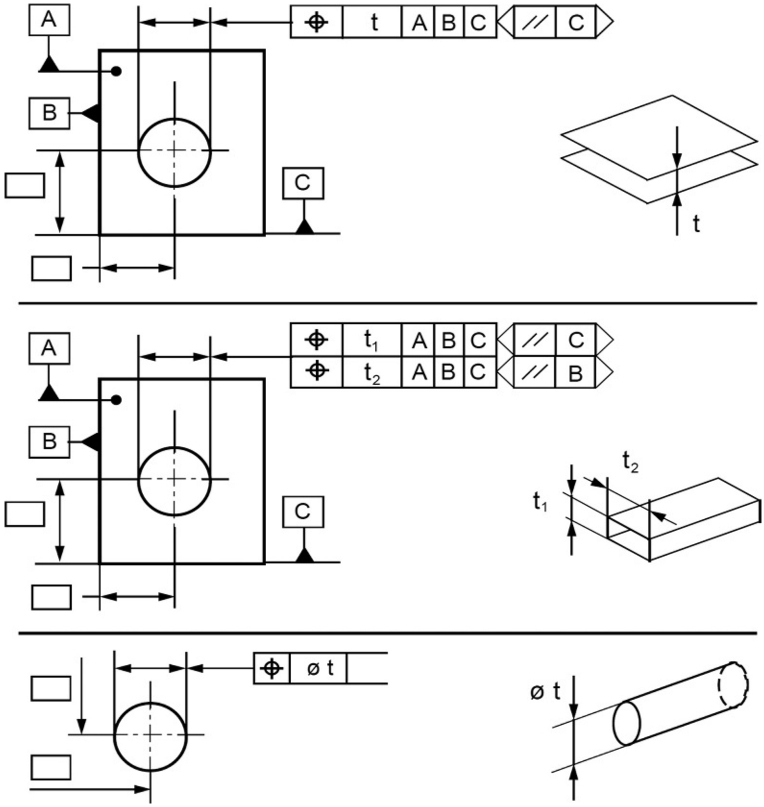


FIG. 3.15 Forms of tolerance zones, ISO 1101

The reference feature is defined by TEDs or, in the case of a straight line or plane, by the drawing outline.

The tolerance zone can be interpreted as the envelope of spheres of diameter equal to the tolerance value, the centres of which are located at the reference feature. See Figs 3.25 and 3.34.

In the case of a straight reference feature and the tolerance without the symbol Ø preceding the tolerance value, the tolerance zone is established by two parallel planes oriented relative to the datum; see Fig. 3.15 centre.

For other forms of tolerance zones, e.g. OZ, UZ for free forms, see 3.2 and 6.1.

Figure 3.16 shows the **tolerance zone plane orientation indicator**. In the past the arrow line indicated the orientation; see Fig. 3.17. According to the new ISO 1101, the indicator shall always be used.

Figures 3.18 and 3.19 show the **section plane indicator**. In Fig. 3.18 only the toleranced features are section lines. In Fig. 3.19, with any cross section (ACS), the datum features are also section lines.

Figure 3.20 shows the **width (measurement direction) of the tolerance zone**. With the exception of roundness tolerances, the width is in the orientation perpendicular to the tangent on the ideal surface in the measurement point. In Fig. 3.20 top, the orientation changes along the feature, which requires changes during the measurement. In Fig. 3.20 bottom the measurement orientation is fixed.

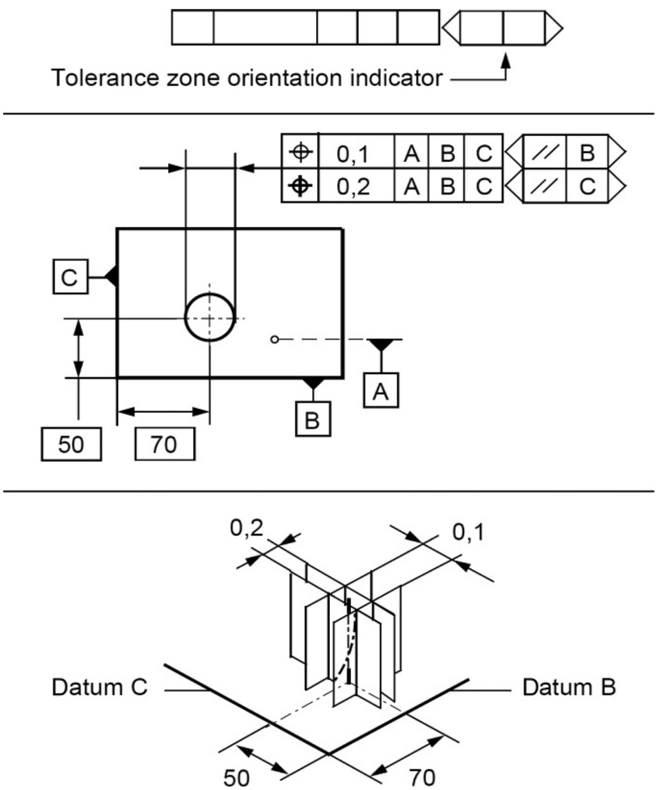


FIG. 3.16 Tolerance zone plane orientation indicator, ISO 1101

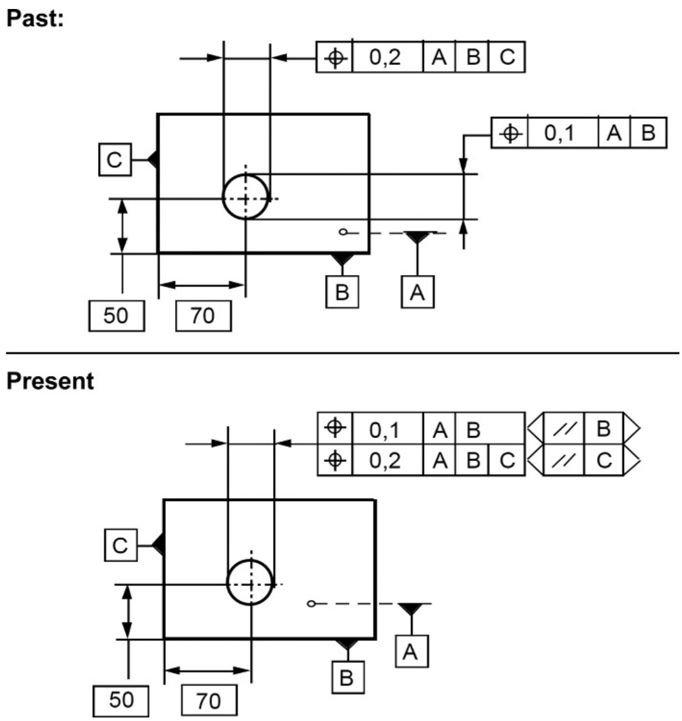


FIG. 3.17 Tolerance zone different in two orientations, past and present, ISO 1101

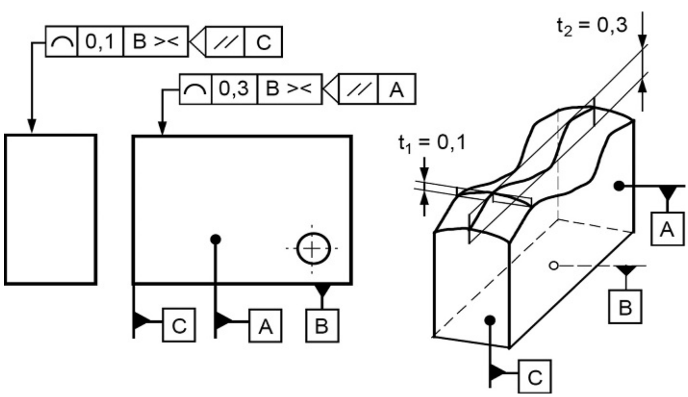


FIG. 3.18 Section plane indicator, ISO 1101

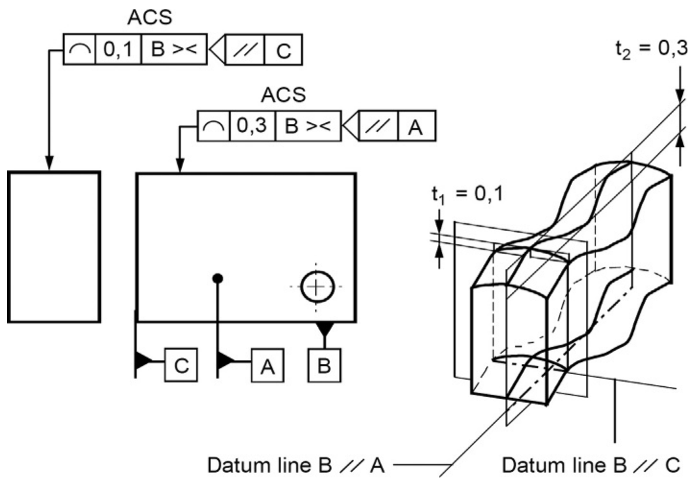


FIG. 3.19 Section plane indicator together with ACS, datum features are also section lines, ISO 1101

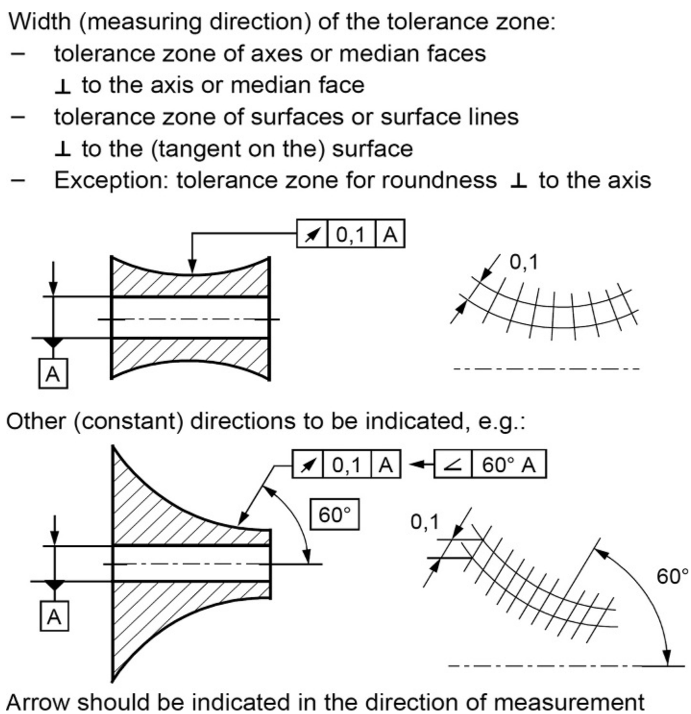


FIG. 3.20 Orientation of the width of the tolerance zone, ISO 1101

With roundness tolerances (symbol ○) the tolerance zone width is perpendicular to the feature axis. Fig. 3.21 shows a possibility for the width perpendicular to the surface.

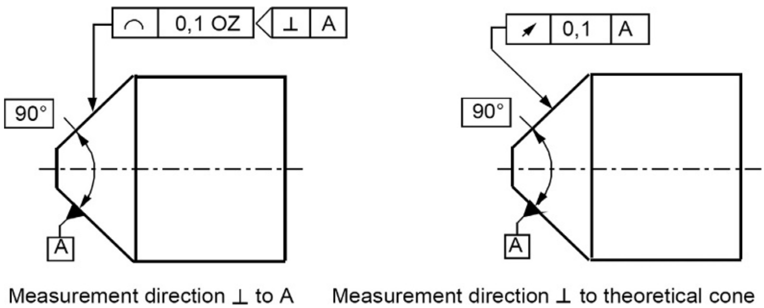


FIG. 3.21 Tolerancing of roundness with the tolerance width perpendicular to the feature axis and perpendicular to the ideal surface, ISO 1101

Figure 3.22 shows a linearly **varying tolerance zone**.

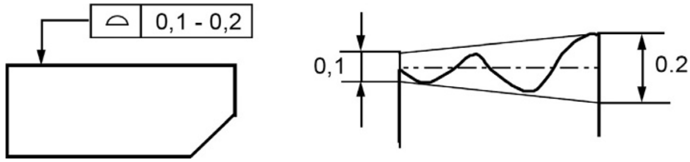


FIG. 3.22 Linearly varying tolerance zone, ISO 1101

Figure 3.23 shows the **possibilities and the sequence of indications of toleranced features**, ISO 1101.

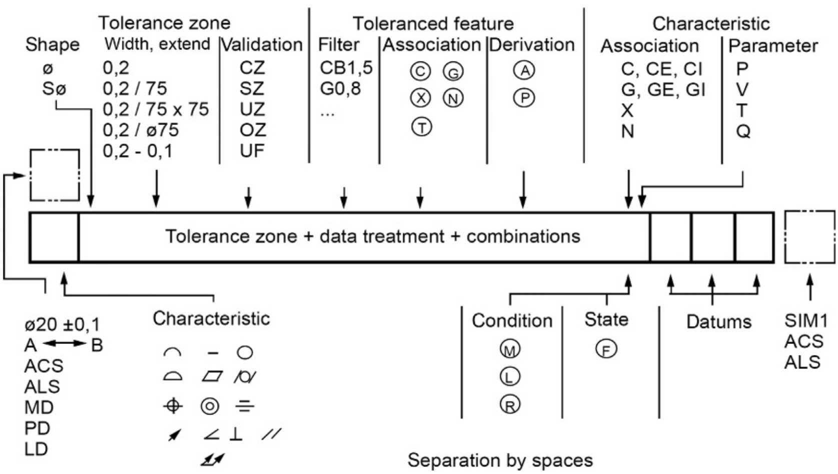


FIG. 3.23 Indication and sequence of toleranced features

The condition symbols Ⓜ, Ⓛ, Ⓡ are to be indicated after the tolerance zone width.

Figure 3.24 shows the **possibilities and the sequence of indication of datums**, ISO 5459.

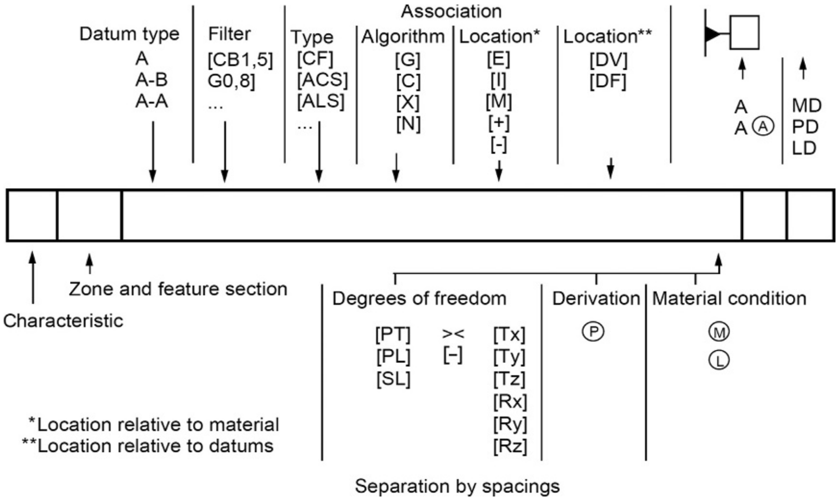


FIG. 3.24 Indication and sequence of datums

For separating, spaces are to be used. Symbols (letters) in circles, do not need spacing.

In this book Ⓜ, Ⓛ, Ⓡ and Ⓟ are located after the tolerance value, because they modify the tolerance value.

3.2 Definitions of geometrical tolerances

3.2.1 General

See ISO 1101 for geometrical tolerances
ISO 5459 for datums
ISO 5458 for group tolerancing

In order to specify geometrical tolerances, the workpiece is considered to be composed of features (geometrical elements), such as planes, cylinders, cones, spheres, tori, etc. (Fig. 1.2).

As a general rule: What is indicated, applies (is to be regarded). When something shall be disregarded, a disclaimer is to be indicated, e.g. when a TED for size shall not be regarded, the disclaimer OZ, VA for size variable is to be indicated. However, there is one exception. When between datums of different order there is a linear TED, this TED is to be disregarded. See Fig. 3.91. (When it is to be regarded, the claimer [DF] for distance fix shall be indicated).

According to ASME Y14.5, this TED is to be regarded (ASME default). When this TED shall be disregarded, according to ASME Y14.5 the symbol ▷ is to be indicated after the datum letter in the tolerance indicator. The ISO symbol is [DV]. In order to keep the rule "what is indicated, applies" it is recommended to always indicate [DV] for distance variable, or [DF].

The ISO defaults, as described in 3.2.3.1.1, apply when there is no modifier indicated, such as Ⓖ for Gauss toleranced feature.

However, the ISO defaults use filtered toleranced features. But the type of filtering of the toleranced feature is not defined (still open). It can be defined by a drawing default, indicated in or near the drawing title block. These drawing defaults can be identified, e.g. by:

"ISO TF: CB 1,5 mm", when the filtering of the toleranced feature for all specifications is performed by a ball of radius 1.5 mm;
"ISO TFF: CB 1,5 mm", when the filtering of the toleranced feature for form specifications is performed by a ball of radius 1.5 mm;
"ISO TFO: CB 1,5 mm", when the filtering of the toleranced feature for orientation specifications is performed by a ball of radius 1.5 mm;
"ISO TFL: CB 1,5 mm", when the filtering of the toleranced feature for location specifications is performed by a ball of radius 1.5 mm.

Figure 3.1 shows ISO TF: CB 1,5 mm. What the tip of the measuring instrument touches establishes the filtered feature (roughness is filtered out). This filter corresponds to a dial gauge. It is often used when no other filtering is specified.

The types of filters are shown in Table 3.11.

The ISO default of filtering datum features is described in 2.4.

The geometrical tolerances are, in the cases of ISO defaults, tolerance zones in which all points of the toleranced filtered feature shall be contained. The tolerance zones are the envelope of spheres of diameters equal to the tolerance value, the centres of which are located on the reference feature. In the case of an integral line, the tolerance zones are the envelope of circles of diameters equal to the tolerance value, the centres of which are located on the reference feature. See Figs 3.25 and 3.34.

The reference feature (nominal surface, theoretically exact feature (TEF), ideal feature) is defined by TEDs or, in cases of a plane or a cylinder, by the drawing outlines. The orientation and the location of the reference feature are, in cases of the ISO defaults, defined by the Chebyshev association with form tolerances (without datums) and by the datums with tolerances of orientation and location.

3.2.2 General types of geometrical tolerances, ISO defaults

Location tolerance is twice (doubled) the permitted maximum value of the location deviation (see 3.2.3.1 and 3.2.3.6). According to ISO 1101, there are defined location tolerance zones within which all points of the filtered toleranced feature must be contained. The location tolerance zone is in the geometrically ideal

orientation and location with respect to the datum(s). The tolerance value defines the width or diameter of this zone, see Figs. 3.25 and 3.79.

Location tolerances (⌒ for integral lines, ⌓ for integral surfaces, ⌖ for derived features, in combination with datum letter(s) in the tolerance indicator) limit the deviations of a feature from its geometrically ideal location with respect to the datum(s). Special symbols for coaxiality (when toleranced feature and datum feature are cylindrical) and symmetry (when at least one of the features concerned is prismatic), where the nominal distance between the axis or median plane of the toleranced feature and the axis or median plane of the datum feature is zero (Table 3.2), are no longer needed.

The location tolerance also limits the orientation deviation and the form deviation of the toleranced feature (integral feature or derived feature), but not the form deviation of the datum feature(s). Form, orientation and location of the datum feature, as appropriate, must be specified separately.

Orientation tolerance is the permitted maximum value of the orientation deviation (see 3.2.3.2 and 3.2.3.7). According to ISO 1101, there are defined orientation tolerance zones within which all points of the filtered toleranced feature must be contained. The orientation tolerance zone is in the geometrically ideal orientation with respect to the datum(s). The tolerance value defines the width of this zone; see Fig. 3.20.

Orientation tolerances (⌒ for integral lines, ⌓ for integral surfaces, ⌖ for derived features, in combination with >< after the datum letter in the tolerance indicator) limit the deviations of a filtered feature from its geometrically ideal orientation with respect to the datum(s). The special symbols for parallelism, perpendicularity and angularity (Table 3.2), are no longer needed.

The orientation tolerance also limits the form deviation of the toleranced feature, but not of the datum feature(s). Form, orientation and location of the datum feature, as appropriate, must be specified separately.

Form tolerance is the permitted maximum value of the form deviation (see 3.2.3.3, 3.2.3.4 and 3.2.3.8). According to ISO 1101, there are defined form tolerance zones within which all points of the filtered toleranced feature must be contained. Within this zone, the feature may have any form, if not otherwise specified. The tolerance value defines the width or diameter of this zone, see Figs. 3.45ff, 3.57ff and 3.79.

Form tolerances (symbol ⌒ for integral lines, ⌓ for integral surfaces, ⌖ for derived features) limit the deviations of a feature from its geometrically ideal line or surface form. The special symbols for straightness and roundness (circularity) and for flatness (planarity) and cylindricity (Table 3.2) are no longer needed.

Run-out tolerances (symbols ↗ and ⌰) are partly tolerances of form, orientation and location.

Radial circular tolerances can be substituted by line profile tolerances; see Fig. 3.65.

Total run-out tolerances can be substituted by surface profile tolerances (⌓) and are no longer needed; see Figs 3.65, 3.67 and 3.69.

Circular run-out tolerances as shown in Figs 3.67 and 3.69 are the only run-out tolerances that may be used.

Location and orientation of the **tolerance zones** are defined

- with form tolerances by the Chebyshev criterion
- with location, orientation and run-out tolerances by the datums.

ISO defaults, as described in following paragraphs, are sufficient for the most functional cases. In very few functional cases are ISO non-defaults necessary. They are also described, but are separated in order to facilitate reading.

3.2.3 Geometric tolerances

3.2.3.1 Location tolerances of integral surfaces

3.2.3.1.1 ISO defaults

Figures 3.25 and 3.26 show tolerance zones for the location of integral surfaces.

Figure 3.27 also shows the indications for the locations of planar integral features. The use of the position symbol is allowed according to ISO 1101, but only for planar features. This is not recommended. The position symbol should be used only for derived features.

Figure 3.28 shows location tolerances for cylindrical features of variable size (OZ). The tolerance zone is coaxial to Datum A. The possible diameters are shown in Fig. 3.29.

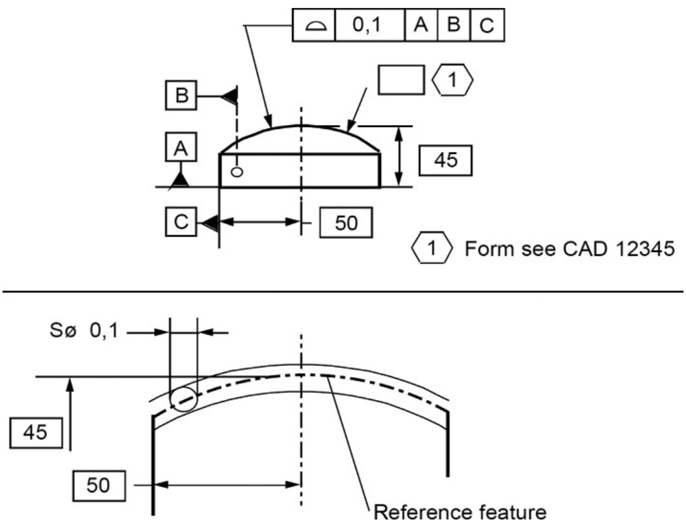


FIG. 3.25 Profile surface tolerance for location of an integral surface

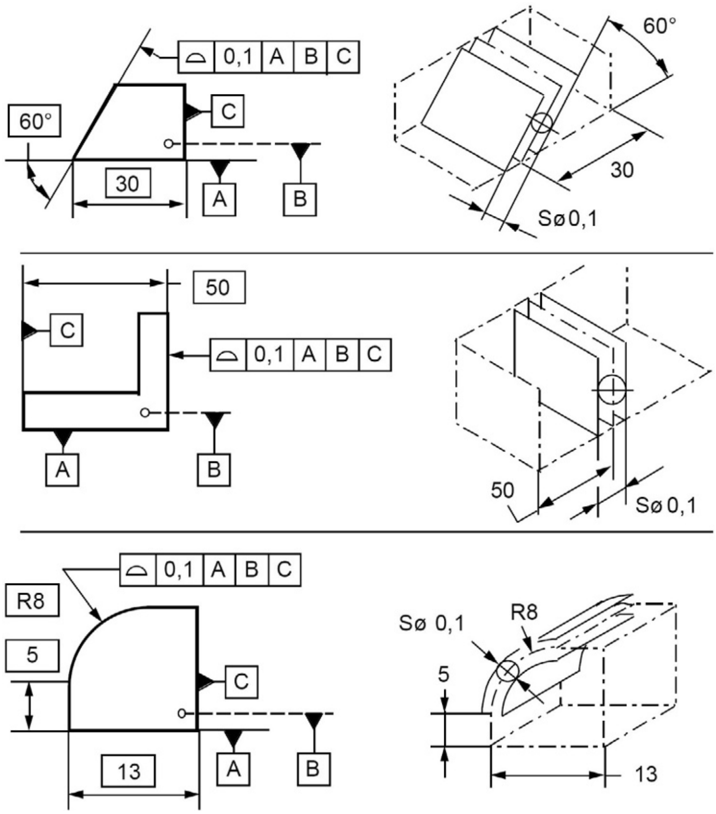


FIG. 3.26 Profile surface tolerances for the location of integral surfaces

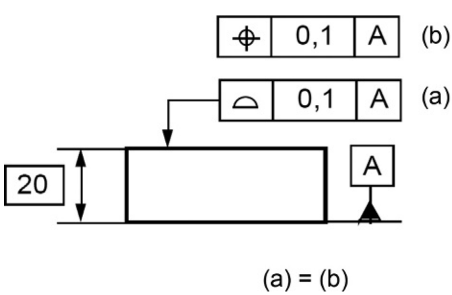


FIG. 3.27 Location tolerancing of a plane, identical meanings

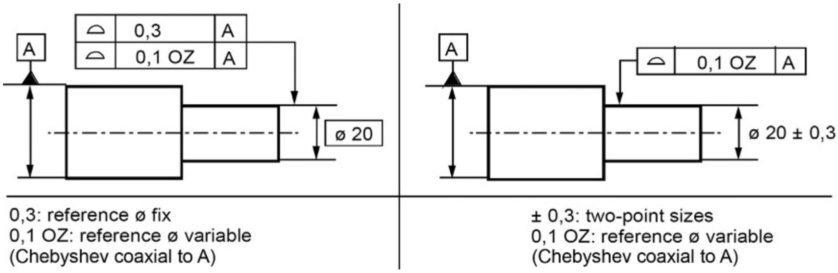


FIG. 3.28 Profile surface tolerances for location for cylindrical surfaces

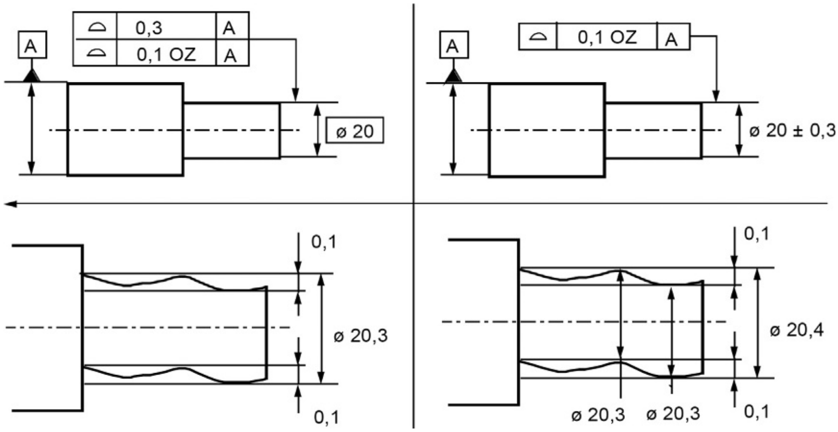


FIG. 3.29 Difference between TED Ø and variable Ø

3.2.3.1.2 ISO non-defaults (specials)

In the following, further tolerancing possibilities (with modifiers) are explained, for special functional cases of location tolerancing and deviating from the ISO defaults.

According to Table 3.10 **tolerancing of associated features**.

Ⓒ tolerance of the associated Chebyshev (minimax) feature
Ⓖ tolerance of the associated Gauss (least squares) feature
Ⓧ tolerance of the associated maximum inscribed feature
Ⓝ tolerance of the associated minimum circumscribed feature
Ⓣ tolerance of the associated tangent feature

Figure 3.30 shows the Chebyshev and Gauss associated toleranced feature.

Figure 3.31 shows the location deviation for the associated tangent feature. This feature is contacting (from outside material) the filtered feature and is oriented according to Gauss outside material (GE).

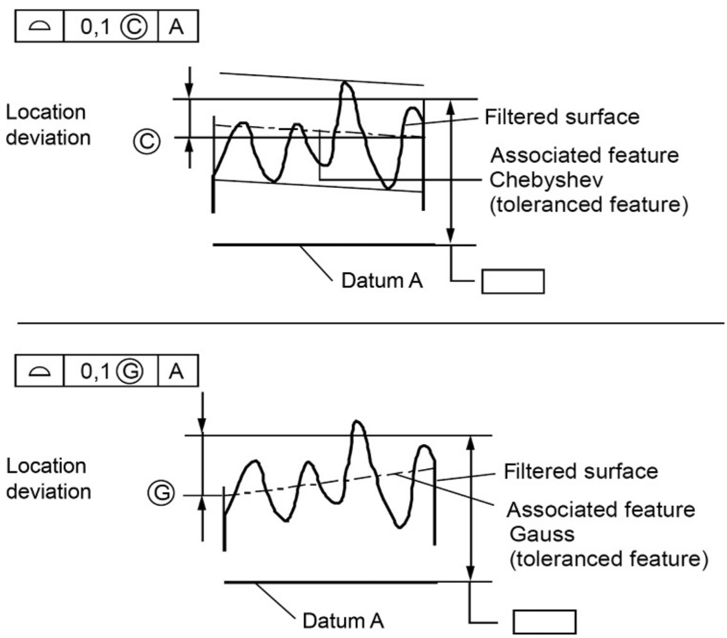


FIG. 3.30 Orientation tolerances of the Chebyshev and Gauss association features

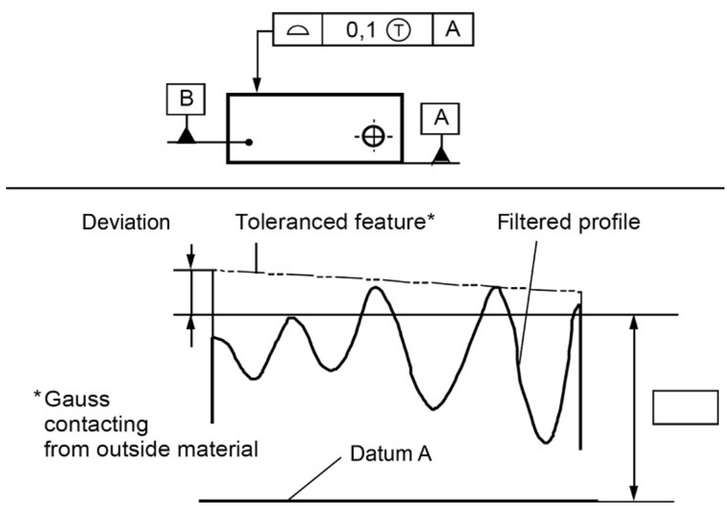


FIG. 3.31 Orientation tolerance with Ⓣ

Figure 3.32 shows the location deviation for the associated tangent feature. This feature is contacting (from outside material) the H0 filtered feature (convex hull) and is oriented according to Gauss external (GE) to the convex hull.

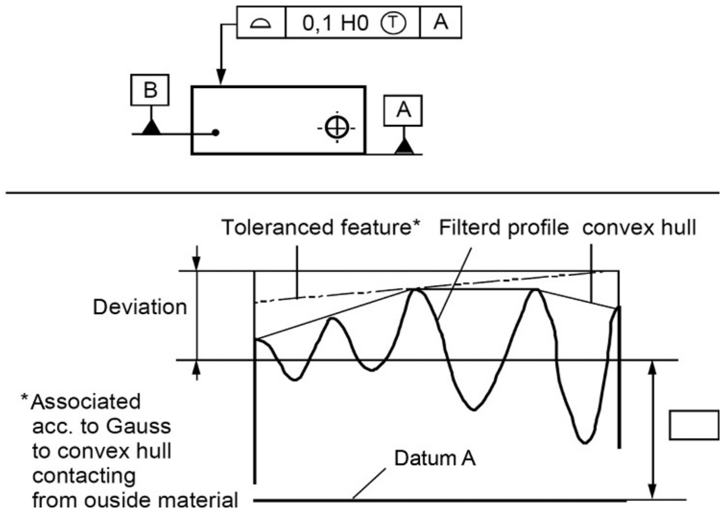


FIG. 3.32 Orientation tolerance with H0 Ⓣ

The convex hull filter can be described as a rubber sheet spanned over the feature.

Figure 3.33 shows the location deviation for the associated tangent feature. This feature is contacting (from outside material) the filtered feature and oriented according to Chebyshev association. A special symbol for this case does not exist.

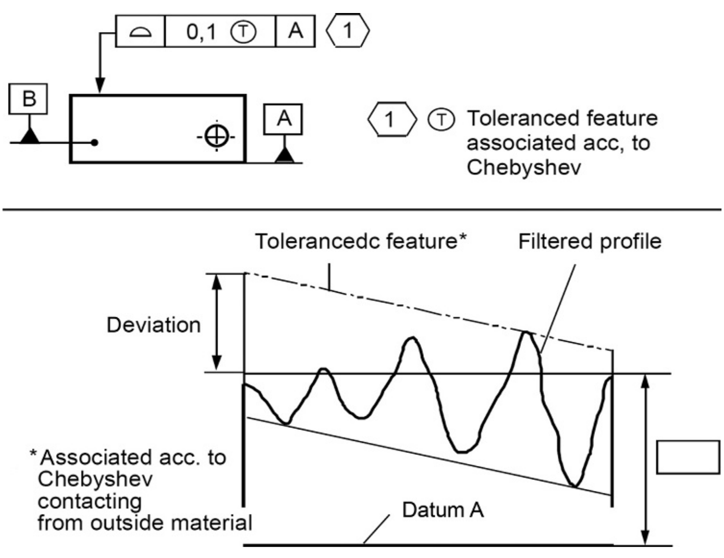


FIG. 3.33 Orientation tolerance with Ⓣ and minimax association

The possible tolerancings of associated features are shown in Table 3.13.

TABLE 3.13 Possible tolerancings of associated features

Type of feature	Ⓒ	Ⓖ	Ⓣ	Ⓧ	Ⓝ
Straight line	•	•	•		
Plane	•	•	•		
Circle	•	•		•	•
Sphere	•	•		•	•
Cylinder	•	•		•	•
Torus	•	•		•	•
Cone	•	•			
Wedge	•	•			
Plane pair size	•	•		•	•

For **filtering**, see Table 3.11.

3.2.3.2 Orientation tolerances of integral features

3.2.3.2.1 ISO defaults

In addition to the location tolerances, smaller orientation tolerances can be added.

The tolerance zones are shown in Fig. 3.34.

The reference feature is oriented (not located) relative to the datum(s). When there is a linear distance between datum and toleranced feature, the datum has the symbol >< following the datum letter in the tolerance indicator; see Fig. 3.34.

Figure 3.34 shows examples in which the location relative to the datums B and C is fixed. The location (not the orientation) relative to datum A is free. The tolerance zone is parallel to A.

With the symbol ⌓ it is possible to combine requirements of orientation and location in the tolerance; see Fig. 3.34 tolerance 0,1.

Figure 3.35 shows orientation tolerances for planar features. In the case above, the symbol >< is needed to disregard the distance (45). In the cases centre and below, the symbol >< is not needed, because only orientation (not location) relative to the datum is possible.

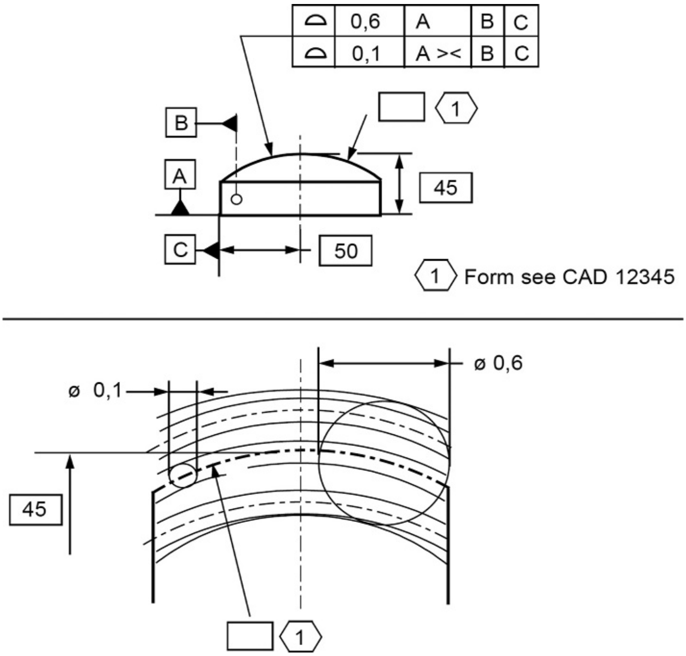


FIG. 3.34 Profile surface tolerance for location and additional for orientation; the orientation tolerance zone is ±0,05 apart from the reference feature, but may vary in height within the 0,6 zone parallel to A; the distances to B and C are fixed

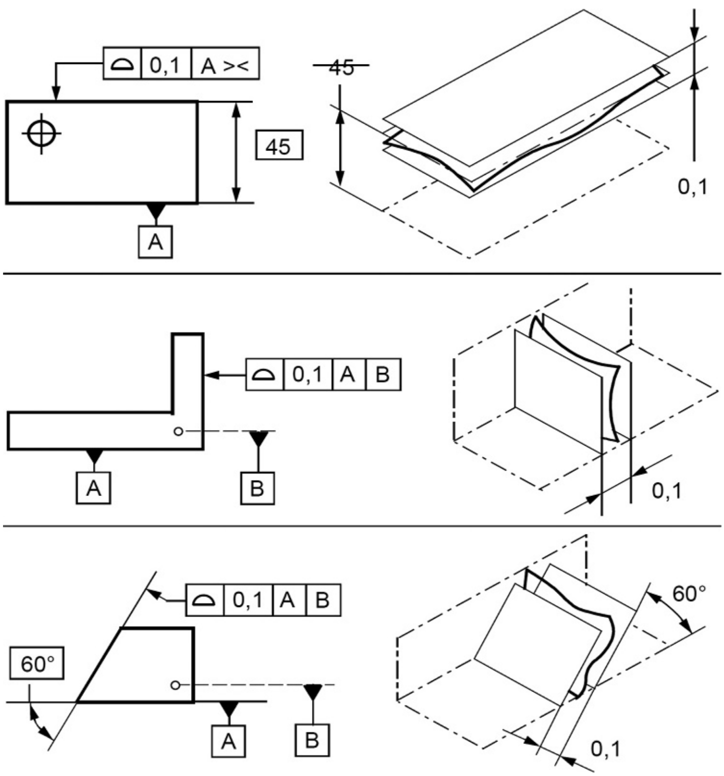


FIG. 3.35 Orientation tolerances for planar surfaces

Figure 3.36 shows orientation tolerances for cylindrical features of variable size (OZ). The tolerance zone is parallel to A (not necessarily coaxial).

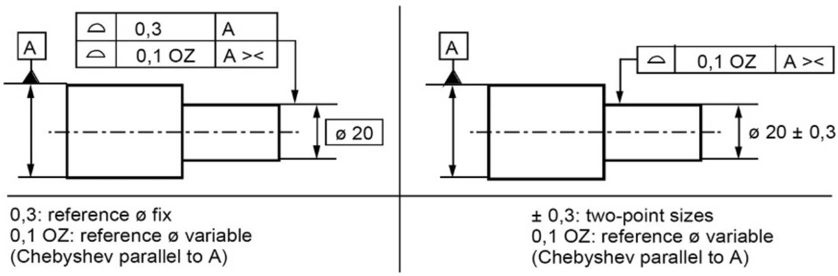


FIG. 3.36 Orientation tolerances for cylindrical features

Figure 3.37 shows the difference between profile tolerance and size tolerance with additional cylindricity tolerance (0.1 OZ).

Figure 3.38 shows drawing indications with identical meanings.

It is recommended to use the symbol ⌓.

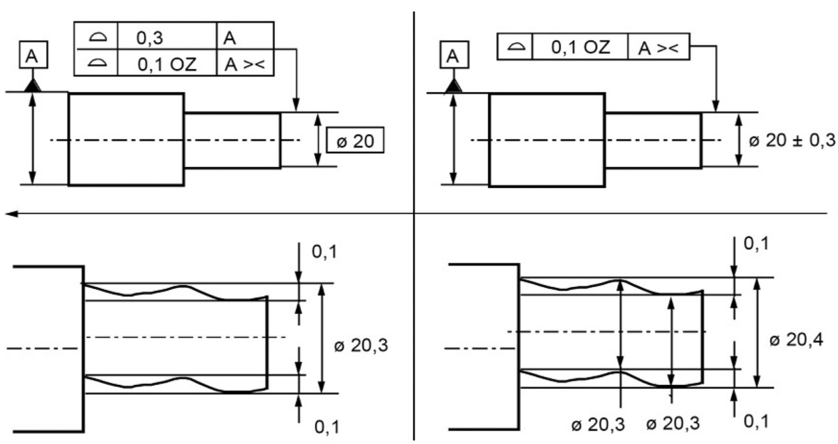


FIG. 3.37 Difference between TED Ø and variable Ø

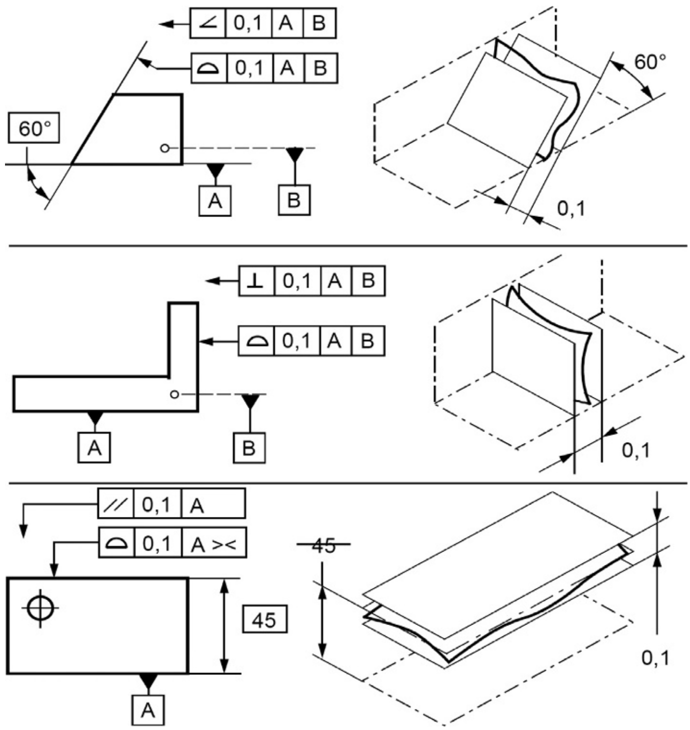


FIG. 3.38 Drawing indications with identical meanings

3.2.3.2.2 ISO non-defaults (specials)

In the following, further tolerancing possibilities (with modifiers) are explained, for special functional cases of orientation tolerancing and deviating from the ISO defaults.

According to Table 3.10, **tolerancing of associated features**

Ⓒ tolerance of the associated Chebyshev (minimax) feature
Ⓖ tolerance of the associated Gauss (least squares) feature
Ⓧ tolerance of the associated maximum inscribed feature
Ⓝ tolerance of the associated minimum circumscribed feature
Ⓣ tolerance of the associated tangent feature

Figure 3.39 shows the Chebyshev and Gauss associated toleranced features.

Figure 3.40 shows the tolerance zone for the associated tangent feature. This feature is contacting (from outside material) the filtered feature and is oriented according to Gauss outside material (GE).

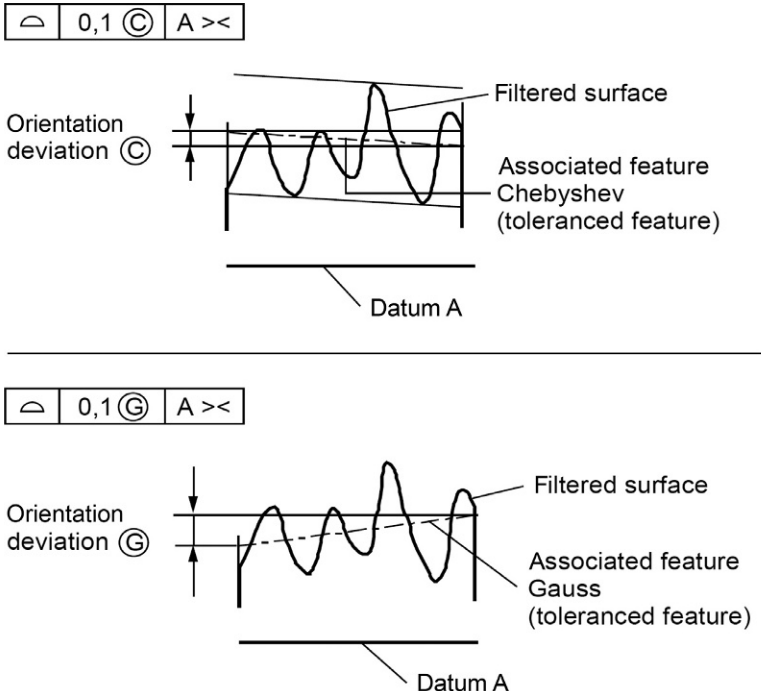


FIG. 3.39 Orientation tolerances of the Chebyshev and Gauss association features

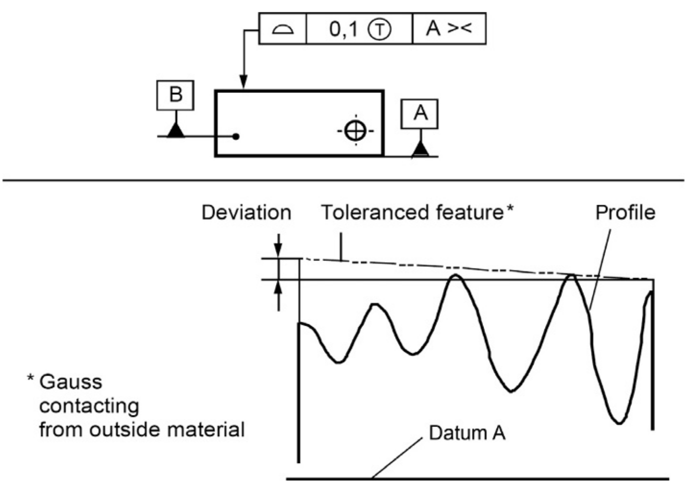


FIG. 3.40 Orientation tolerance with Ⓣ

Figure 3.41 shows the tolerance zone for the associated tangent feature. This feature is contacting (from outside material) the H0 filtered feature (convex hull) and is oriented according to Gauss external (GE) to the convex hull. The convex hull filter can be described as a rubber sheet spanned over the feature.

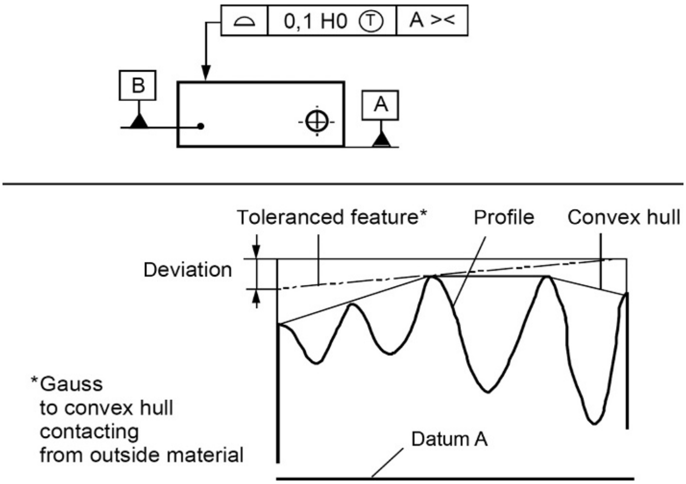


FIG. 3.41 Orientation tolerance with H0 Ⓣ

Figure 3.42 shows the tolerance zone for the associated tangent feature. This feature is contacting (from outside material) the filtered feature and oriented

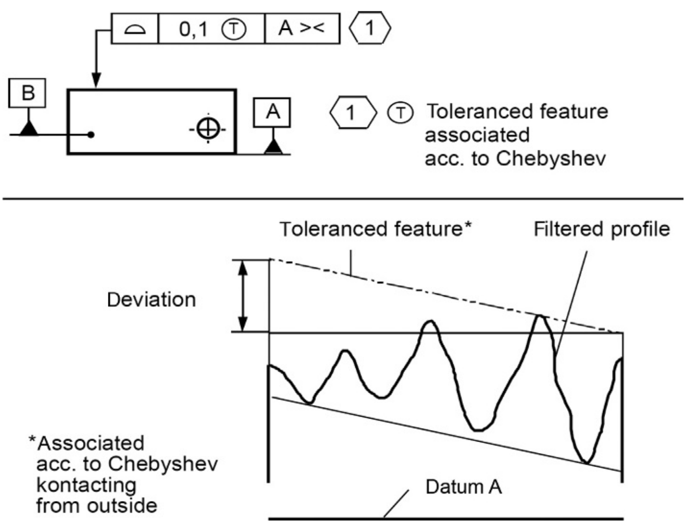


FIG. 3.42 Orientation tolerance with Ⓣ and minimax association

according to minimax (Chebyshev) criterion. A special symbol for this case does not exist.

The possible tolerancings of associated features are shown in Tables 3.9 and 3.13.

For **filtering**, see Table 3.11.

3.2.3.2.3 Angular size tolerances

Angular size tolerances according to ISO 14 405-3 are specified in angular grades or minutes following the nominal angular size (e.g. 45°±30′). Angular size tolerances define ranges of angles (e.g. 44°30′ to 45°30′) for the direction of contacting straight lines in section planes.

The direction of the contacting straight lines are derived from the actual surface. The maximum distance of the actual line from the contacting line is minimized, i.e. the contacting line (e.g. leg of the angle measuring instrument, protractor) contacts the actual surface on the highest point(s) in an average direction (Fig. 3.43). Therefore the angular size tolerance does not limit the form deviations of the lines or surfaces constituting the angle. The theoretical definitions of the angles are given in 4.8.

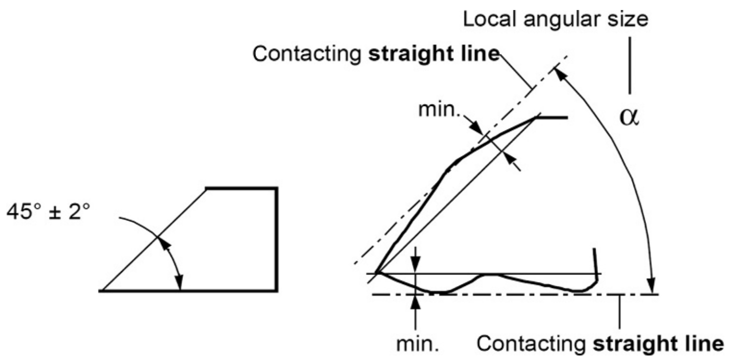


FIG. 3.43 Angular dimension tolerance, actual angular size

Profile (inclination) tolerances, specified as shown in Fig. 3.44, define tolerance zones within which all points of the toleranced surface or line must be contained. Therefore the geometrical tolerance also limits the form deviations of the toleranced feature. This in contrast to angular size tolerances. See also 4.8, Angular sizes. (As the drawing is to be completely toleranced, the datum feature is toleranced and therefore also these form deviations are limited.)

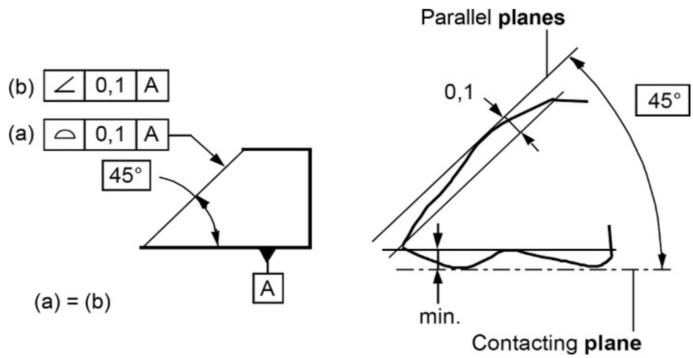


FIG. 3.44 Angularity (inclination) tolerance, tolerance zone

3.2.3.3 Form tolerances of integral surfaces

3.2.3.3.1 ISO defaults

In addition to the location and orientation tolerances, smaller form tolerances can be added. The tolerance zones are shown in Fig. 3.45.

In the case of a cylinder, there are two different possibilities:

– Figure 3.46, upper diagram, the TED of the cylinder diameter defines the total tolerance zone limiting the diameter deviations; the unrelated offset zone from the TED cylinder (zone of variable Ø defined by the Chebyshev association) limits the form deviations (cylindricity).

– Figure 3.46, lower diagram, the ± tolerance limits the two-point sizes of the diameter; the unrelated (offset) zone (zone of variable Ø defined by the Chebyshev association) limits the form deviations (cylindricity).

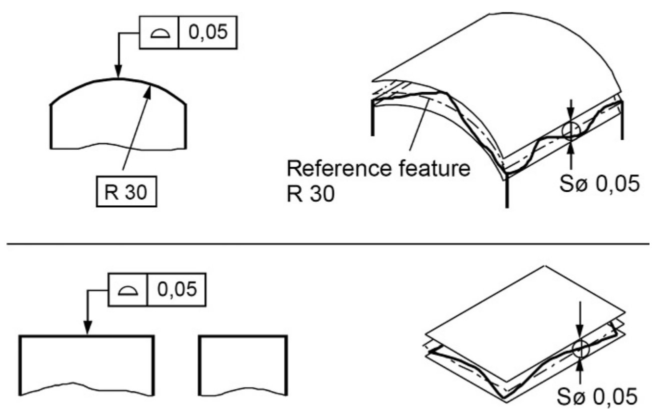


FIG. 3.45 Form tolerances of surfaces: drawing indications, tolerance zones

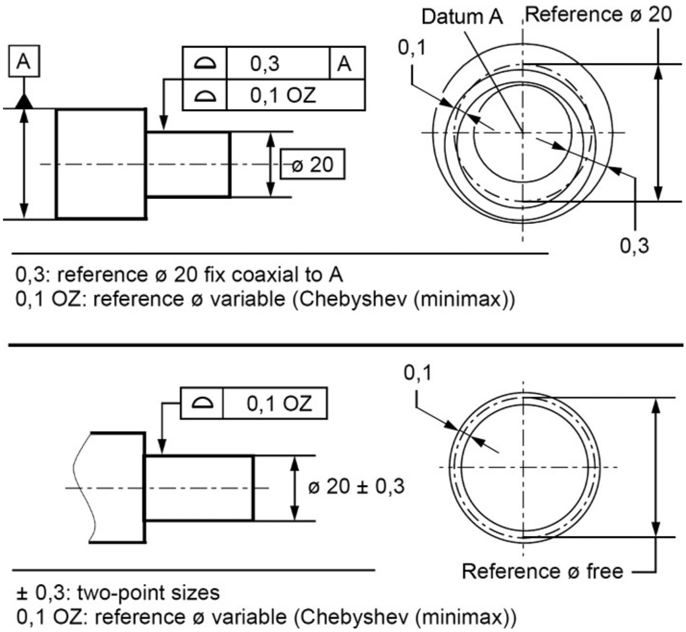


FIG. 3.46 Form tolerance of a cylinder, upper diagram with the diameter fix, lower with the diameter variable, location 0.1 zone defined by Chebyshev

The difference between these two possibilities is shown in Fig. 3.47. The diameters of the maximum material enveloping cylinders are different.

Figure 3.48 shows tolerance indications with identical meanings.

It is recommended to use only the symbol ⌓; see Fig. 3.48.

Figure 3.49 shows the differences between OZ (offset zone) and VA (variable angle) in the case of cones and wedges. With OZ, the TED angle remains

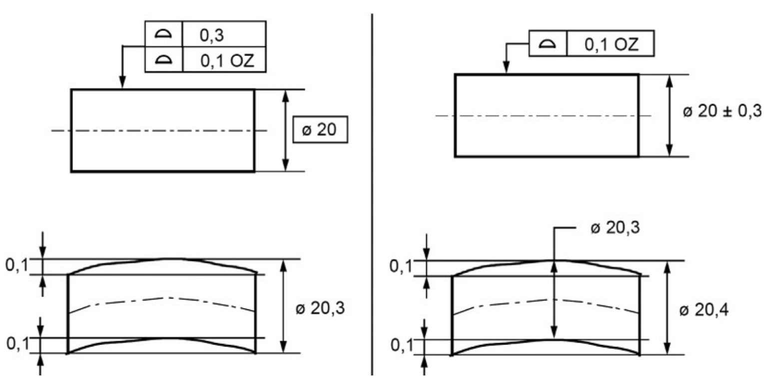


FIG. 3.47 Difference between form tolerances with fixed and with variable diameter

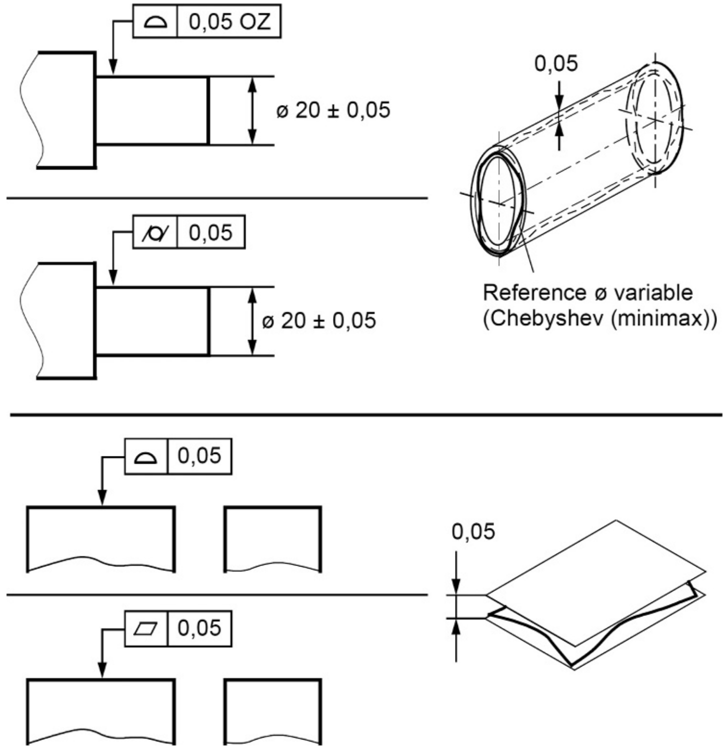


FIG. 3.48 Drawing indications with identical meanings

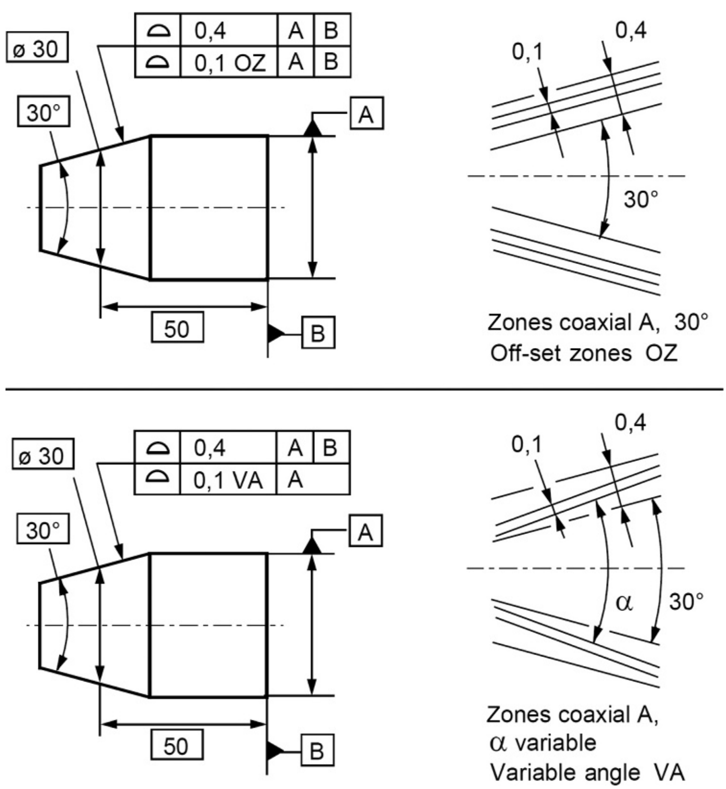


FIG. 3.49 Difference between OZ and VA in the case of a cone (or wedge)

for the offset zone (0,1). With VA, the angle is variable within the total tolerance zone (0,4) and defined by the minimax (Chebyshev) criterion.

Figure 3.50 shows the development regarding OZ for a cylinder (or sphere or torus) and VA for a cone or wedge.

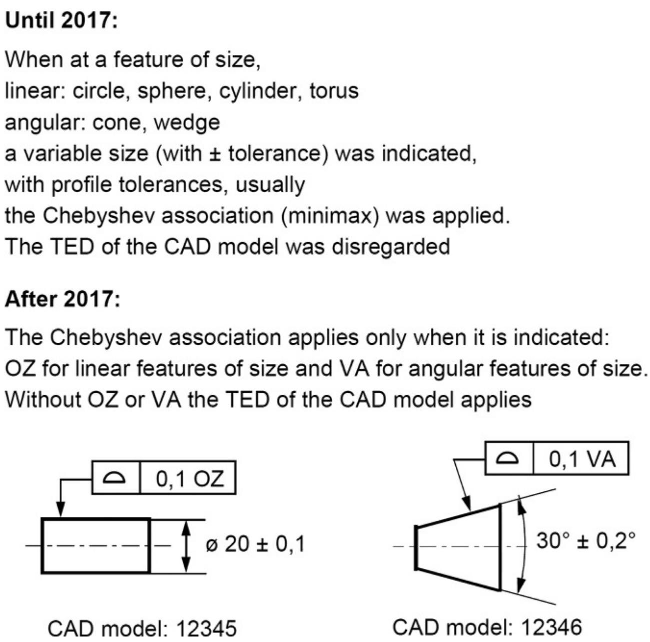


FIG. 3.50 Development regarding OZ (offset zone) and VA (variable angle)

3.2.3.3.2 ISO non-defaults (specials)

Regarding the **association methods** C, CI, CE, G, GI, GE, X, N, see Table 3.10; regarding filtering, see Table 3.11; and regarding **toleranced parameters**, see Table 3.8.

Figure 3.51 shows a form tolerance with a Gauss reference feature (instead of Chebyshev reference feature). This corresponds to the usual form deviation measured with CMMs in the past.

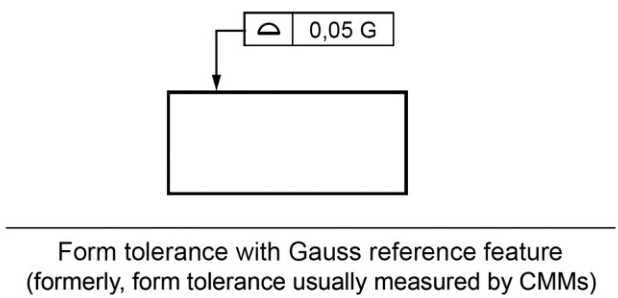


FIG. 3.51 Form tolerance with Gauss reference feature

Figure 3.52 shows a form tolerance by a toleranced parameter (characteristic) P (reference to peak) from a filtered cylindrical feature. The reference feature is Gauss. See Tables 3.8 and 3.11.

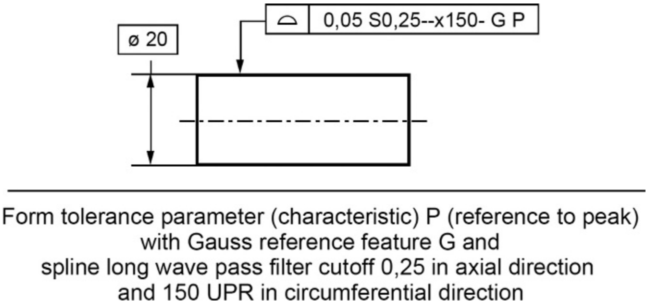


FIG. 3.52 Form tolerance P

Figure 3.53 shows a form tolerance using a toleranced parameter (characteristic) V (reference to valley) from a filtered cylindrical feature. The reference feature is Chebyshev (ISO default). See Tables 3.8.

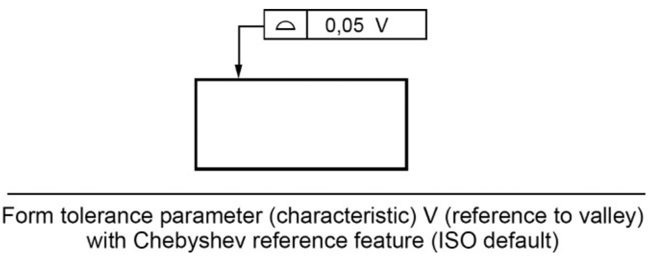


FIG. 3.53 Form tolerance V, reference feature C

Figure 3.54 shows the difference of the tolerance zones between Chebyshev and Gaussian reference feature.

Figure 3.55 shows the reference feature of the smallest envelope.

Figure 3.56 shows the association methods according to Table 3.8. (The deviations are exaggerated in order to show the differences.)

C, CI, CE designate the Chebyshev association and all three have the same result.

G, GI, GE designate the Gaussian association and are not shown in Fig. 3.56.

X and N can have the same zones as C (see Fig. 3.56 right upper and centre lower) or different zones (see Fig. 3.56 centre upper and right lower).

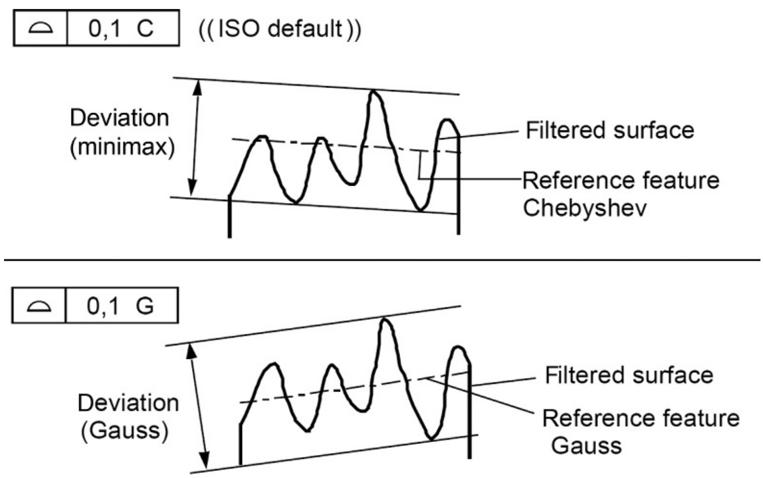


FIG. 3.54 Difference between Chebyshev and Gaussian reference feature

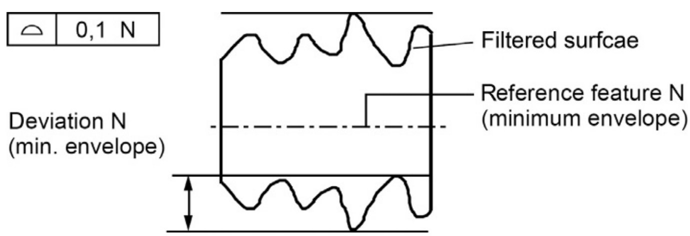


FIG. 3.55 Reference feature minimum envelope

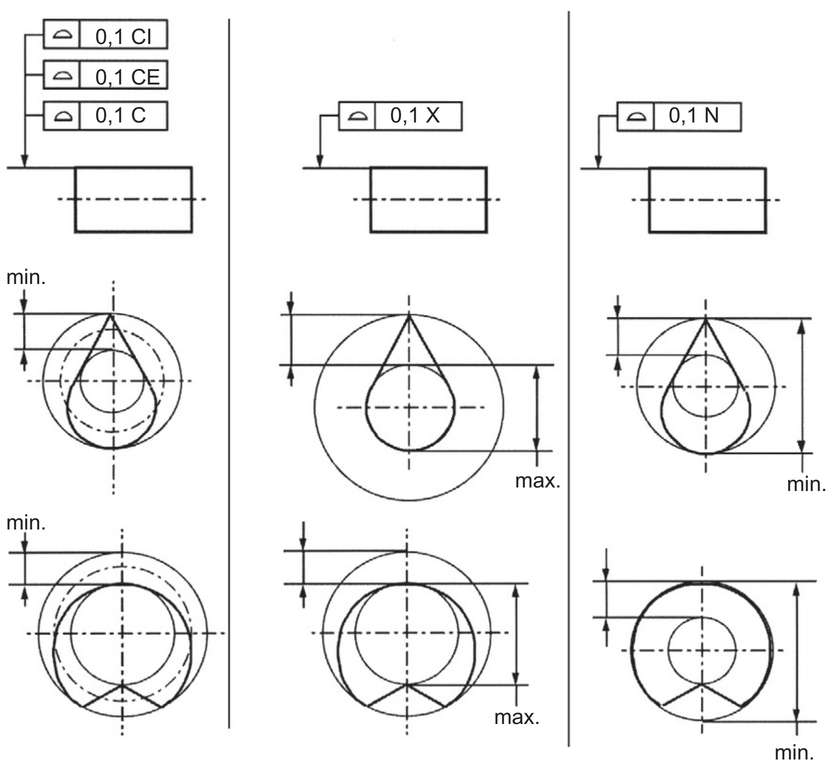


FIG. 3.56 Form tolerances, associations C, CI, CE, X, N

3.2.3.4 Tolerances of integral lines

3.2.3.4.1 Form tolerances of integral lines, ISO defaults

In addition to the location tolerances and the orientation tolerances and the form tolerances of integral surfaces, form tolerances of integral lines can be added.

The tolerance zones are shown in Fig. 3.57.

It was former practice not to use the section plane indicator, but to define the section plane as parallel to the plane of projection in which the indication is shown.

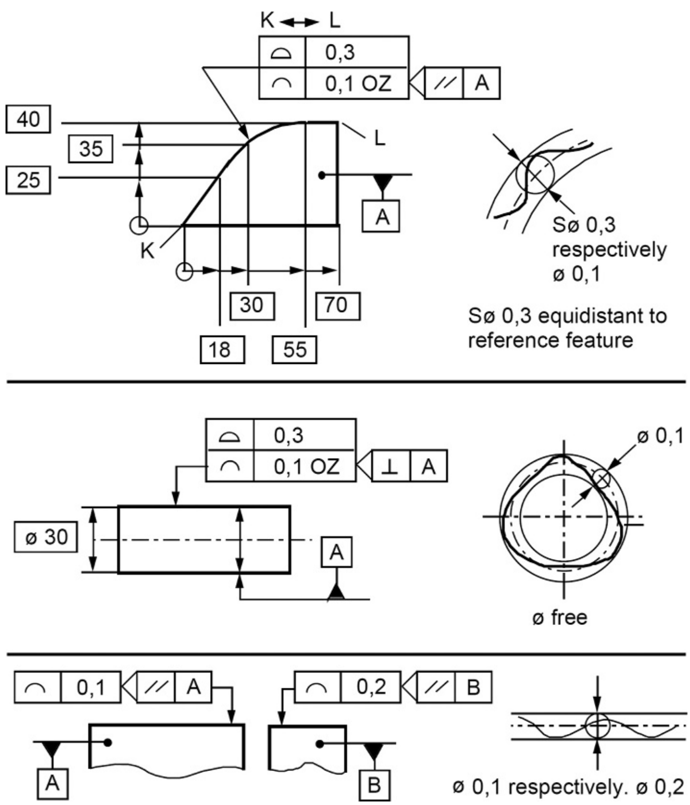


FIG. 3.57 Form tolerances of integral lines (in section planes)

In the case of a cylinder, there are two different possibilities:

– Figure 3.58 upper, the TED of the cylinder diameter defines the total tolerance zone limiting the diameter deviations; the unrelated offset zone from

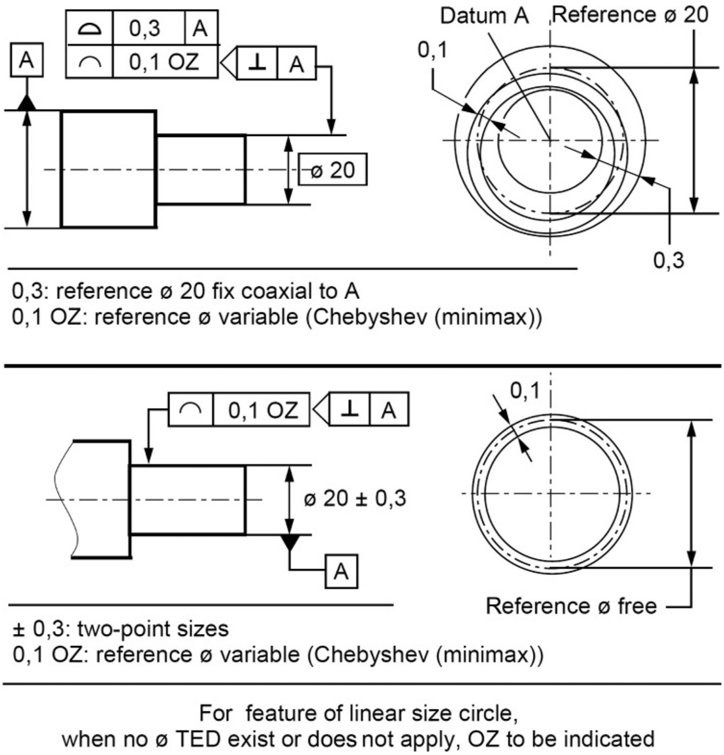


FIG. 3.58 Form tolerances of integral lines of a cylinder, upper, with the diameter fixed, lower, with the diameter variable, location zone 0,1 defined by Chebyshev

the TED circle (zone of variable Ø defined by the Chebyshev association within the section plane) limits the form deviations (roundness).

– Figure 3.58 lower, the ± tolerance limits the two-point sizes of the diameter; the unrelated (offset) zone (zone of variable Ø defined by the Chebyshev association within the section plane) limits the form deviations (roundness).

The difference between the two possibilities is shown in Fig. 3.59. The diameters of the maximum inscribed circles are different.

Figure 3.60 shows the development regarding OZ for profile line tolerances. Figure 3.61 shows tolerance indications with identical meanings.

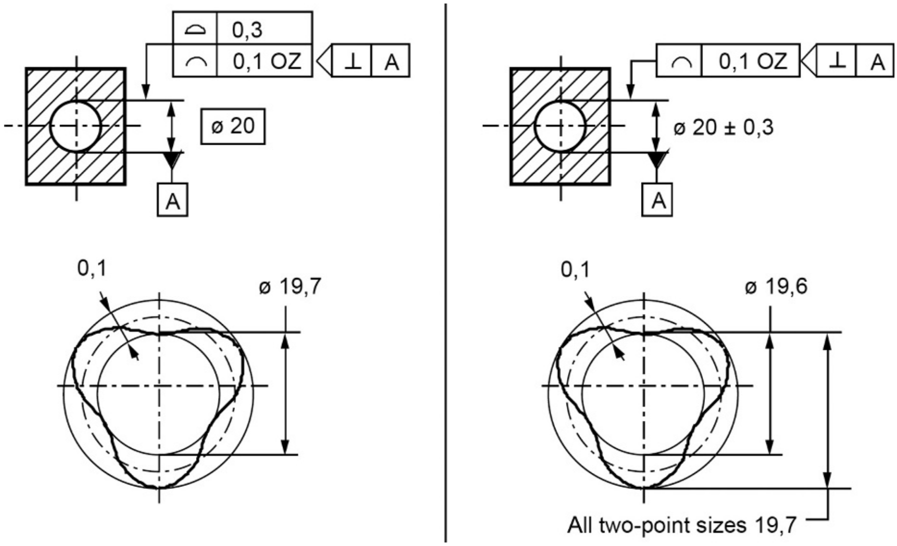


FIG. 3.59 Difference between form tolerances with fixed and variable diameter

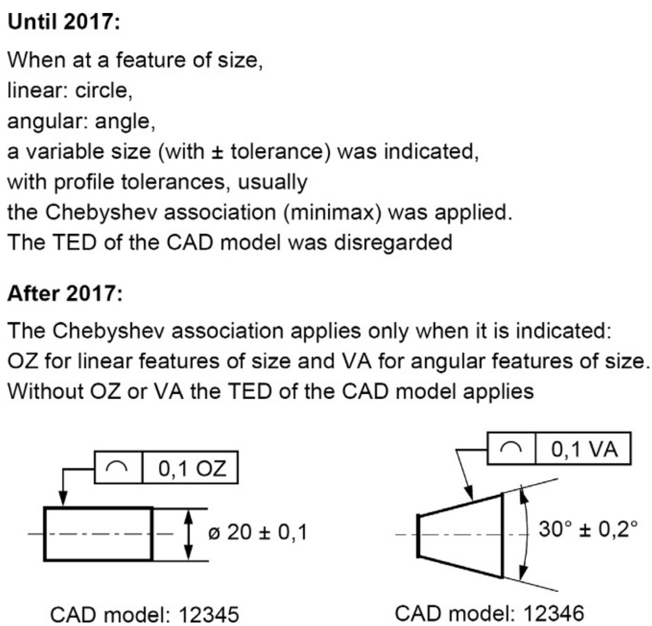


FIG. 3.60 Development regarding OZ (offset zone) and VA (variable angle)

It is recommended to use the symbol ⌒ only; see Fig. 3.38.

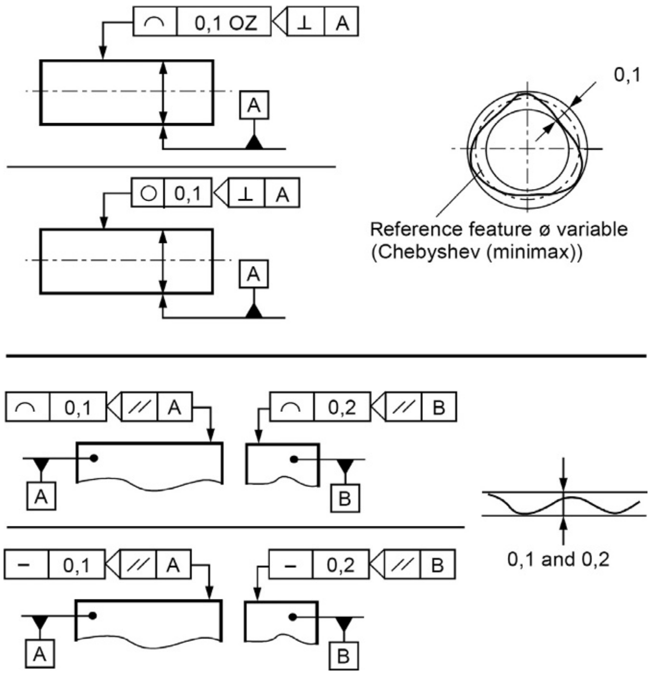


FIG. 3.61 Drawing indications with identical meanings

3.2.3.4.2 ISO non-defaults (specials)

In the following text, **further tolerancing possibilities** (with modifiers) are explained, for special functional cases of tolerancing of the form of lines, which are not ISO defaults.

Figure 3.62 shows the **association methods of the reference features**, defining the orientation and location of the tolerance zone for form tolerances. The Chebyshev (minimax) association minimizes the maximum distance and is the default. The modifier C is usually omitted (because it is the ISO default).

The Gaussian (least squares, LS) association minimizes the sum of the squares of the distances. Figure 3.62 shows the differences between Chebyshev and Gauss.

The possible association methods are listed in Table 3.8.

Figure 3.63 shows the principles of the association methods according to Table 3.8. (The deviations are exaggerated, in order to show the differences.)

C, CI, CE refer to the minimum zone (minimax, Chebyshev) reference features. Regardless of which is the reference feature (e.g. centre circle, material external circle, material internal circle), the three zones are identical.

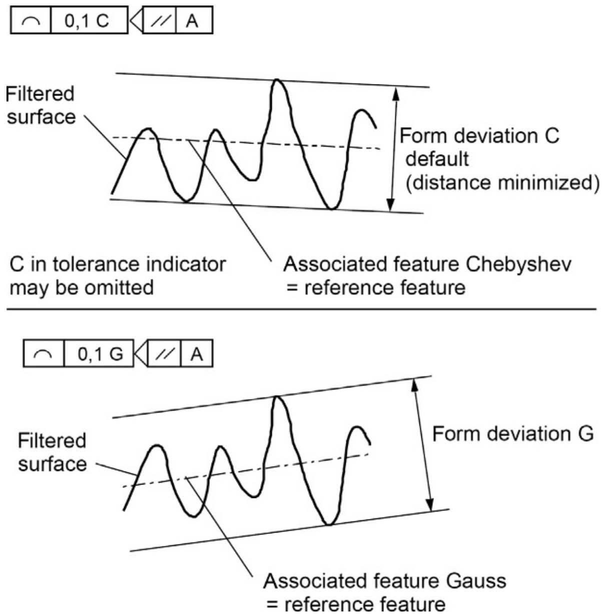


FIG. 3.62 Association methods C and G for form tolerances

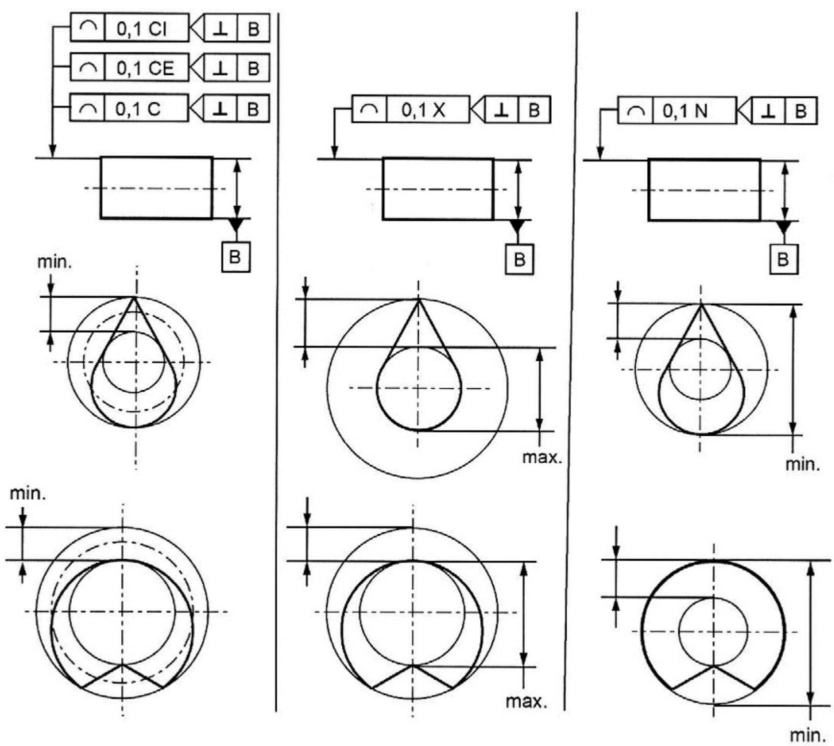


FIG. 3.63 Principles of association methods C, CE, CI, X, N

G, GI, GE (see Table 3.8) refer to the least-squares (Gaussian) reference features (not shown in Fig. 3.63). As the minimized sums of the squares of the deviations are different, these three possibilities lead to different tolerance zones.

X (reference feature = maximum inscribed) and N (reference feature = minimum circumscribed) can have the same zone as C (see Fig. 3.63 upper right and lower centre) or different zones (see Fig. 3.63 upper centre and lower right), depending on the shape of the feature.

Figure 3.64 shows a roundness tolerance, filtered with a long pass Gaussian filter 50 UPR (shorter waves than 50 UPR are partially or totally filtered out).

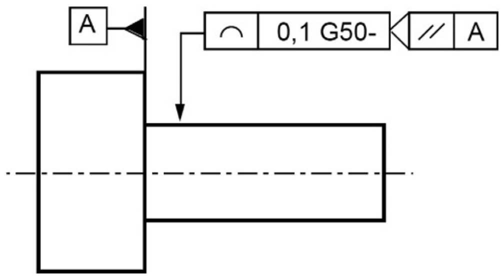


FIG. 3.64 Roundness tolerance with long pass filter G50 UPR

3.2.3.4.3 Orientation and location tolerances of integral lines

For these tolerances, symbol ⌒, the same rules apply as for integral surfaces, with the addition that the section plane indicator is to be indicated.

3.2.3.5 Run-out tolerances

3.2.3.5.1 Radial run-out tolerances

Circular radial run-out (Fig. 3.65 top). In each plane perpendicular to the common datum axis A–B, the line profile (circumference) shall be contained between two circles concentric with the datum axis A–B and with a radial distance of 0.1.

Total radial run-out tolerance (Fig. 3.65 bottom). The surface shall be contained between two cylinders, coaxial with the datum axis A–B and with a radial distance of 0.1.

During checking of the circular radial run-out deviation, the positions of the dial indicator are independent of each other. However, during checking of the total radial run-out deviation, the positions of the dial indicator are along a guiding (straight) line parallel to the datum axis A–B (see clause 13.7.2).

Therefore the straightness deviations and the parallelism deviations of the generator lines of the toleranced cylindrical surface are limited by the total radial run-out tolerance, but not by the circular radial run-out tolerance (Figs 3.65 upper and 3.66).

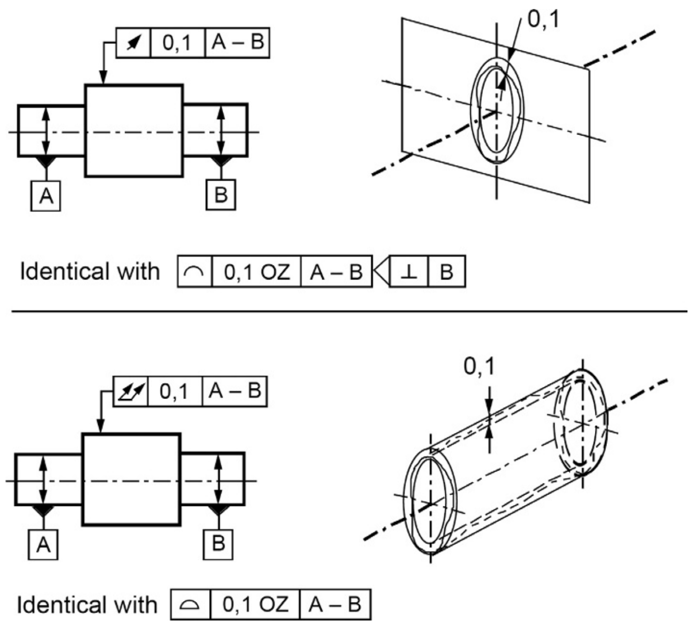


FIG. 3.65 Circular radial run-out tolerance, total radial run-out tolerance

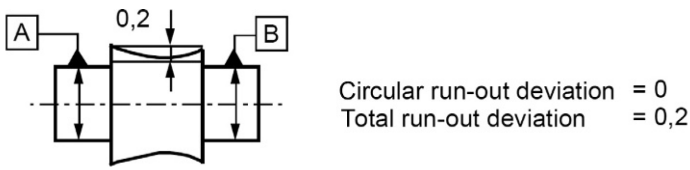


FIG. 3.66 Workpiece with total radial run-out deviation, but without circular radial run-out deviation

Radial run-out tolerances can be substituted by profile tolerances; see Fig. 3.65.

3.2.3.5.2 Axial run-out tolerances

Circular axial run-out (Fig. 3.67 upper). In each cylindrical section coaxial to the datum D the section line shall be contained between two circles 0,1 apart and perpendicular to the datum D.

Total axial run-out tolerance (Fig. 3.67 lower). The surface shall be contained between two parallel planes 0.1 apart and perpendicular to the datum axis D.

During checking of the circular axial run-out deviation, the positions of the dial indicator are independent of each other. However, during checking of the

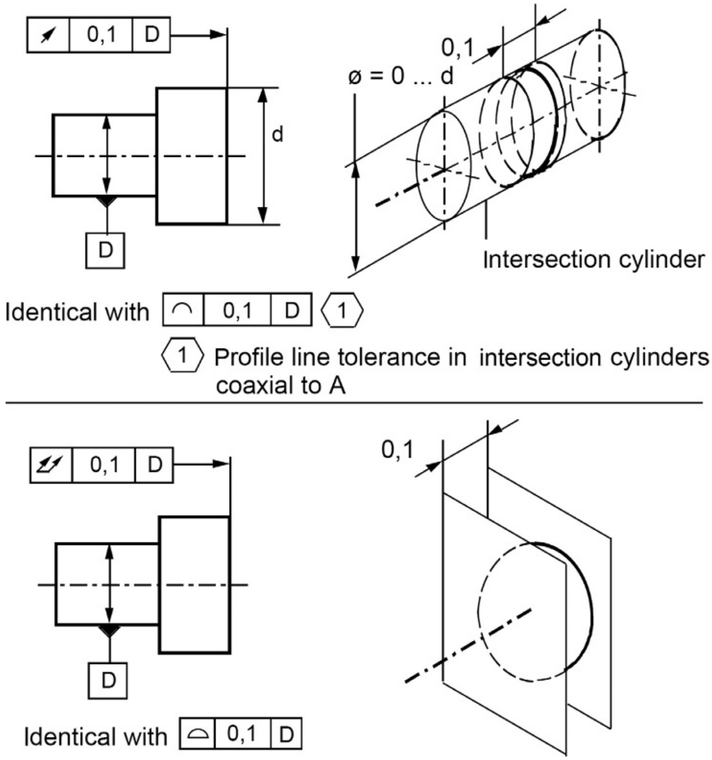


FIG. 3.67 Circular axial run-out tolerance, total axial run-out tolerance: drawing indications, tolerance zones

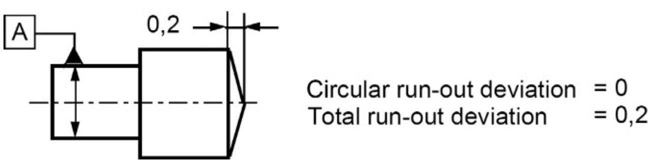


FIG. 3.68 Workpiece with total axial run-out deviation, but without circular axial run-out deviation

total axial run-out deviation, the positions of the dial indicator are along a guiding (straight) line perpendicular to the datum axis D (see clause 13.7.10.2.2).

Therefore the flatness deviations of the toleranced surface are limited by the total axial run-out tolerance, but not by the circular axial run-out tolerance (Figs 3.67 upper and 3.68).

Axial total run-out tolerances can be substituted by profile tolerances; see Fig. 3.67.

3.2.3.5.3 Run-out tolerances in any direction

Circular run-out tolerance in any direction (Fig. 3.69 upper). In each conical section (measuring cone) coaxial with the datum axis A and perpendicular to the nominal toleranced surface, the section line shall be contained between two circles 0,1 apart.

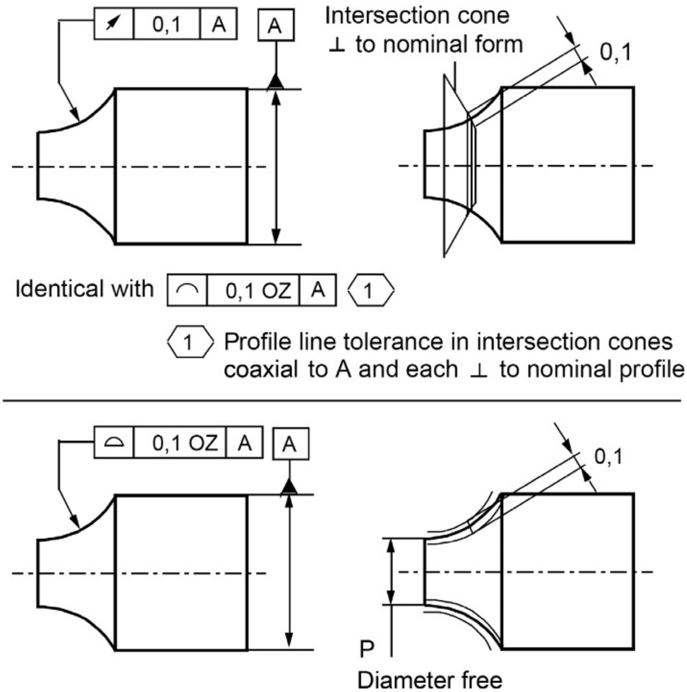


FIG. 3.69 Circular total run-out tolerance in any direction, profile tolerance

Total run-out tolerance in any direction. There is no definition in ISO 1101. It is recommended to use surface profile tolerancing instead; see Fig. 3.69 lower.

During checking of the circular run-out deviation in any direction, the positions of the dial indicator are independent from each other.

Therefore the deviations of the generator line of the toleranced feature are limited by the profile tolerance, but not by the circular run-out tolerance in any direction (Figs 3.69 upper and 3.70).

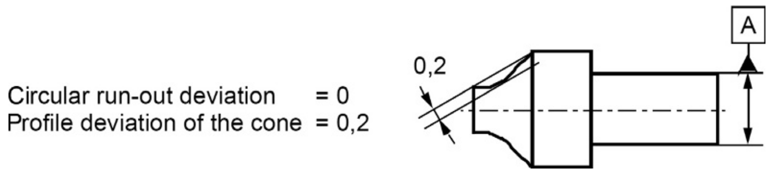


FIG. 3.70 Part with profile deviations, but without circular run-out deviations

Run-out tolerances in any direction can be substituted by profile tolerances; see Fig. 3.69.

3.2.3.6 Location tolerances of derived features

3.2.3.6.1 ISO defaults

The tolerance zones are shown in Figs 3.71 and 3.72.

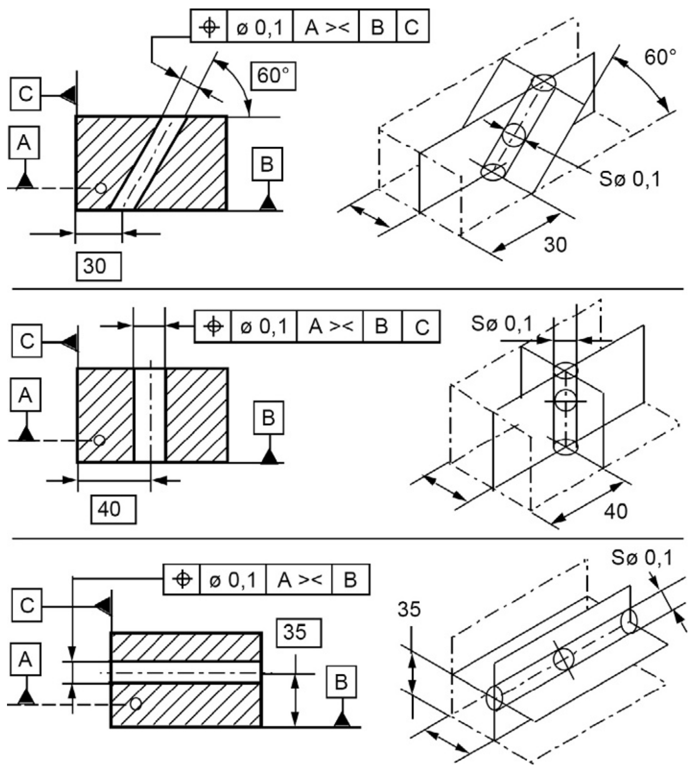


FIG. 3.71 Cylindrical position tolerances for location of derived features; TEDs between A and toleranced feature to be disregarded

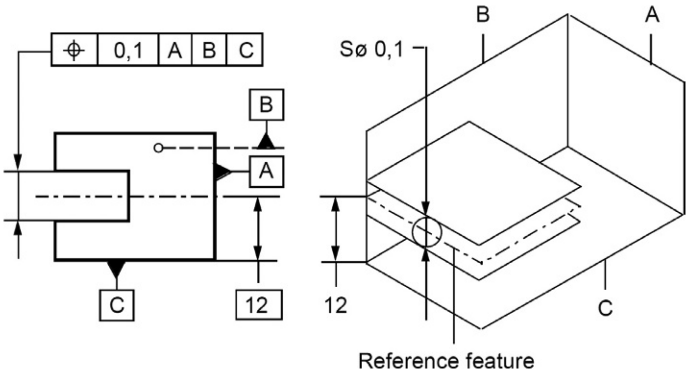


FIG. 3.72 Position tolerances for location, established by parallel planes, for derived features

The location tolerances of derived features are to be used together with sizes and size tolerances. They allow smaller size tolerances than profile tolerances; see 5.4. They can be used with the maximum or least material requirement; see 4.4 and 4.6.

In Figs. 3.71 and 3.72 are the sizes and the size tolerances to be added.

Figure 3.73 shows drawing indications with identical meanings.

It is recommended to use the symbol ⌖.

Concentricity (symbol ⌖ or ◎) applies to section planes and needs the indication ACS above the tolerance indicator. The datum is the centre point in the section plane.

Position tolerances are used together with size tolerances. See, e.g. Figs 2.15 and 3.73.

3.2.3.6.2 ISO non-defaults (specials)

Regarding the **association methods** G, GI, GE, X, N, see Table 3.10. Regarding **filtering**, see Table 3.11. Regarding the derived features of **associated features** Ⓖ, ⊗, Ⓝ, see Table 3.9.

Figure 3.74 shows tolerancing a cylindrical shaft by a position tolerance Ø 0.1 for the minimum circumscribed cylinder Ⓝ at the defined location. Size and form of the shaft are toleranced by the profile tolerance 0.2.

Figure 3.75 shows tolerancing of a cylindrical hole by a position tolerance Ø 0.1 for the maximum inscribed cylinder ⊗ at the defined location. Size and form of the hole are toleranced by the profile tolerance 0.2.

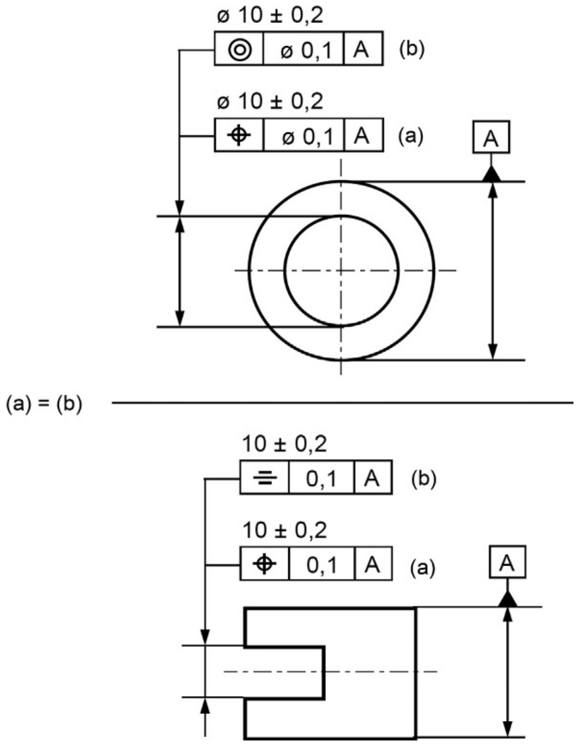


FIG. 3.73 Drawing indications with identical meanings

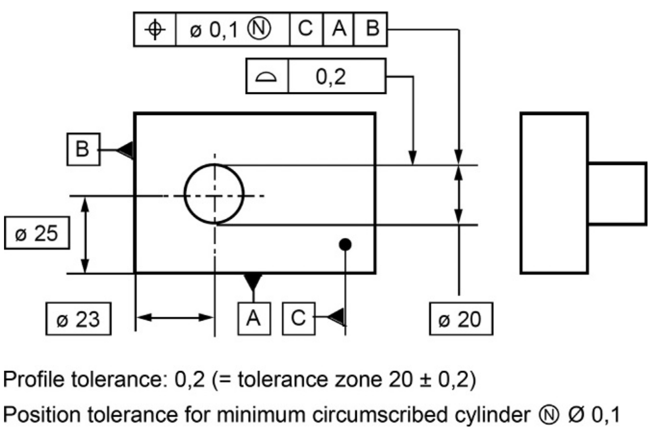


FIG. 3.74 Form and location tolerance, minimum circumscribed cylinder Ⓝ

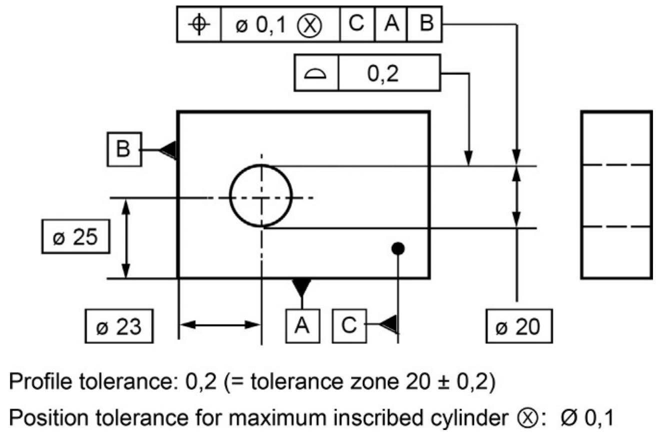


FIG. 3.75 Form and location tolerance, maximum inscribed cylinder Ⓧ

3.2.3.7 Orientation tolerances for derived features, ISO defaults

In addition to the location tolerances of derived features, smaller orientation tolerances of derived features can be added. The tolerance zones are shown in Figs 3.76 and 3.77.

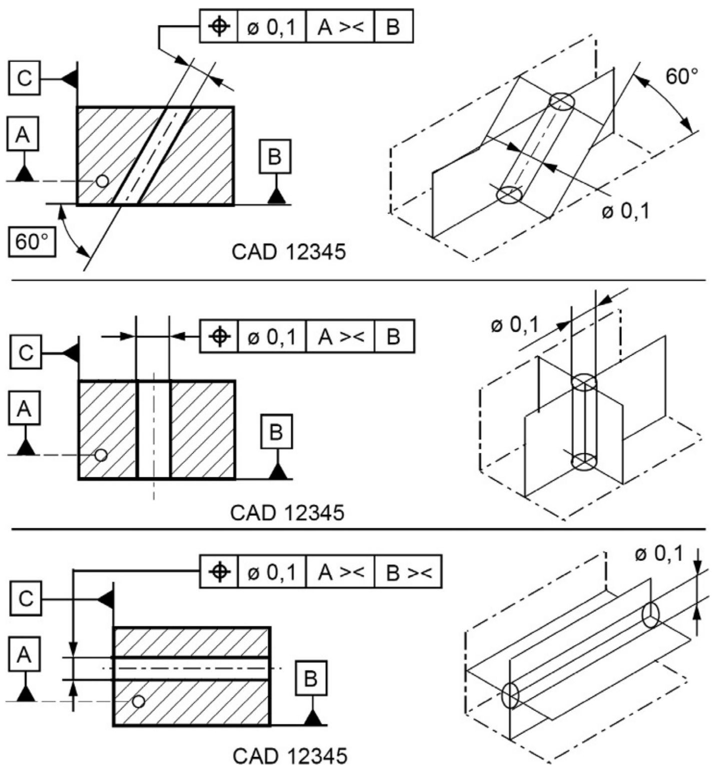


FIG. 3.76 Cylindrical position tolerances for orientation

The reference feature is oriented (not located) relative to the datum(s). When there is a linear distance between datum and toleranced feature, the datum has the symbol >< following the datum letter in the tolerance indicator; see Fig. 3.76.

Figure 3.77 shows the position tolerance for orientation of a median surface.

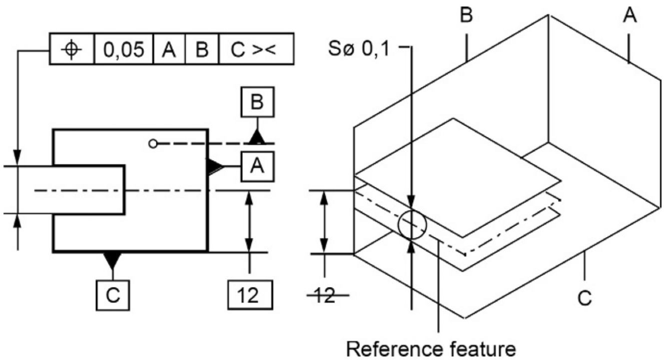


FIG. 3.77 Position tolerance for orientation established by two parallel planes

Figure 3.78 shows drawing indications with identical meanings.

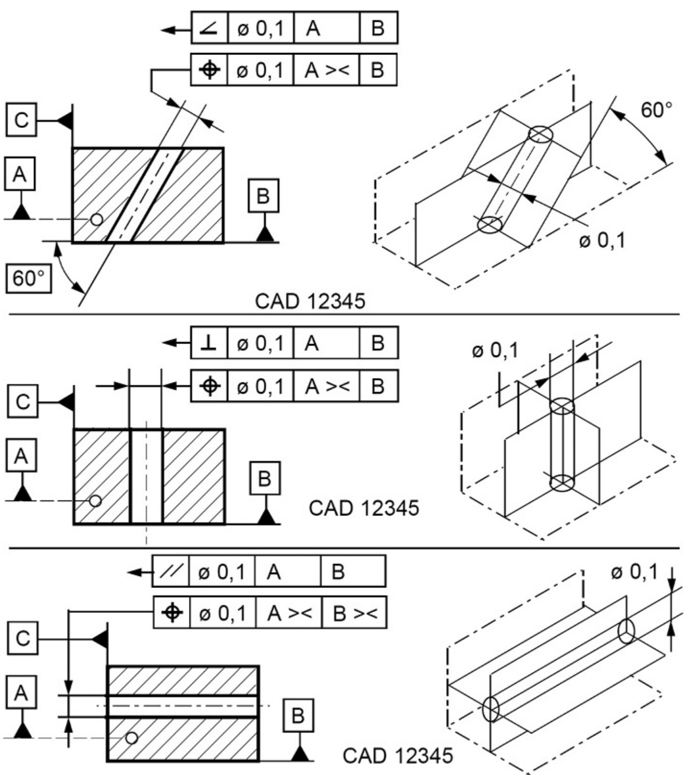


FIG. 3.78 Drawing indications with identical meanings; the TEDs between A and the toleranced features are to be disregarded

3.2.3.8 Form tolerances of derived features, ISO defaults

In addition to the location tolerances and the orientation tolerances of derived features, smaller form tolerances of derived features can be added. The tolerance zones are shown in Fig. 3.79.

Figure 3.80 shows indications of identical meanings.

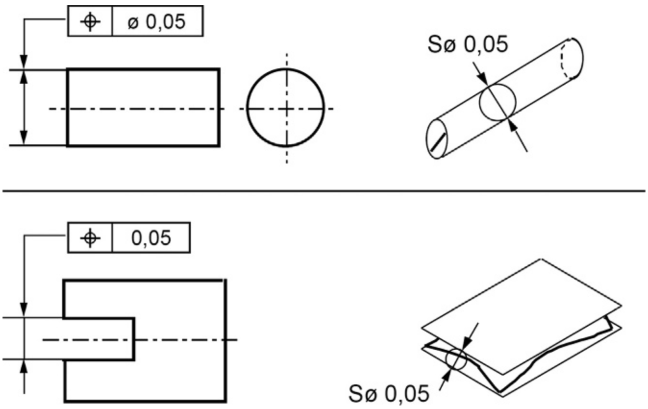


FIG. 3.79 Form tolerances of derived features

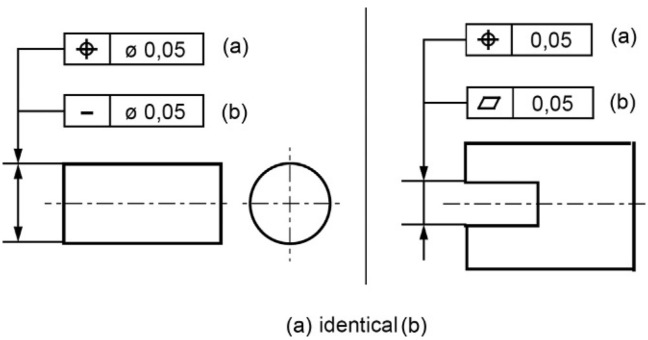


FIG. 3.80 Drawing indications with identical meanings

3.3 Datums

3.3.1 General

A datum feature is a real (non-ideal) integral feature (surface) used for establishing a single datum. A datum (situation feature) is an associated (ideal) feature (point, straight line, plane) derived from one or more datum features. The association operation fits the ideal feature (situation feature) to the non-ideal filtered datum feature.

The history of the definition of the datum was described in 2; see Fig. 2.12.

When the orientation of the situation feature is not stable, the minimum rock requirement according to ISO 5459:1981 and ISO TR 5460 may be used as a near approximation. It defines a median orientation. See Fig. 3.81.

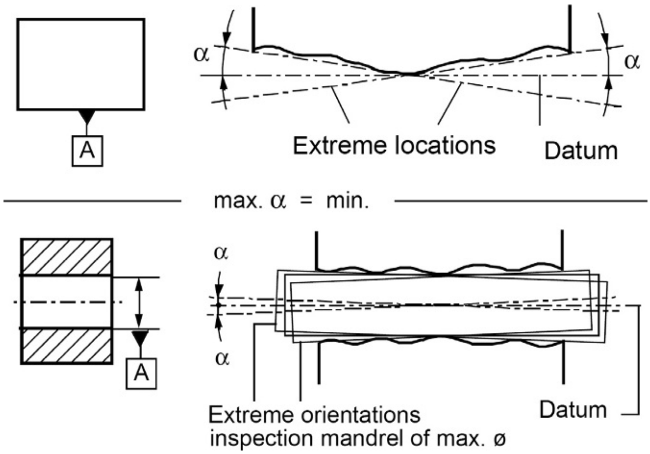


FIG. 3.81 Minimum rock requirement for the datum

The minimum rock requirement minimizes the maximum angle between the extreme locations of the possible associated features. See Fig. 3.81. The associated features are planes for planar datum features. Plane pairs for plain pair datum features, cylinders for cylindrical datum features.

The problems with the minimum rock requirement, and how to solve them, are described in 2.4.

For datum features of variable linear size (± size tolerance, cylinder, sphere, parallel opposite planes (plane pairs)), the contacting ideal feature (establishing the situation feature point, straight line or plane) is the maximum inscribed for holes and the minimum circumscribed for shafts with or without constraints. There are no constraints for primary datums; there are orientation and sometimes location constraints for secondary and tertiary datums; see, e.g. Figs 3.87 and 3.91.

For datum features of fixed linear size (TED for size), the associated ideal feature is the TED feature with the maximum distance to the filtered feature minimized (Chebyshev).

For primary datum features of angular size (cone, wedge), the contacting ideal feature (situation feature) is the TED feature inscribed for inner features, circumscribed for outer features and the maximum distance to the filtered feature minimized (Chebyshev).

There are **single datums, common datums and datum systems**. See Fig. 3.82.

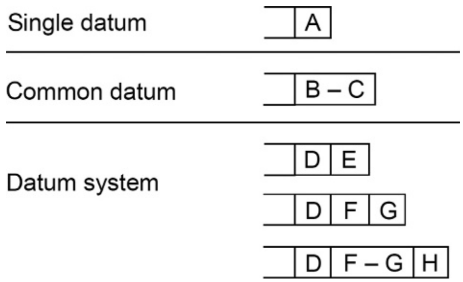


FIG. 3.82 Datums

Common datums are established by more than one datum feature (mostly two) and are to be handled as single datums.

Figure 3.83 shows a new (simplified) option for indicating common datums.

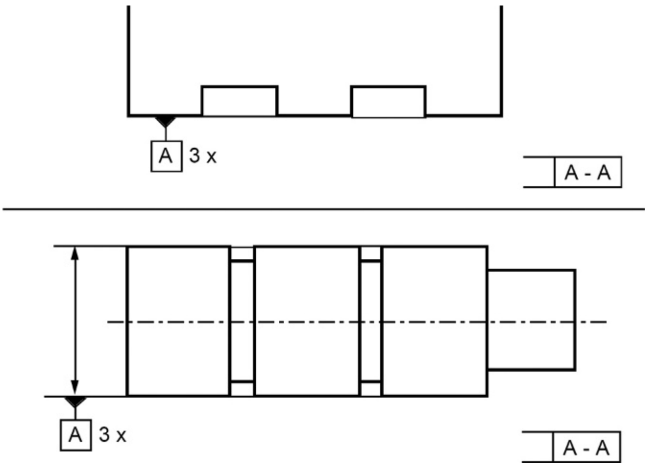


FIG. 3.83 Simplified indication of common datums

A datum system is an ordered set of datums. The primary datum is not influenced by other datums. When not stable in orientation, the minimum rock requirement may be applied; see Fig. 3.81. The secondary datum is perpendicular to the primary datum. When not stable in orientation, the minimum rock requirement may be applied. The tertiary datum is perpendicular to the primary and to the secondary datum. See Fig. 3.84.

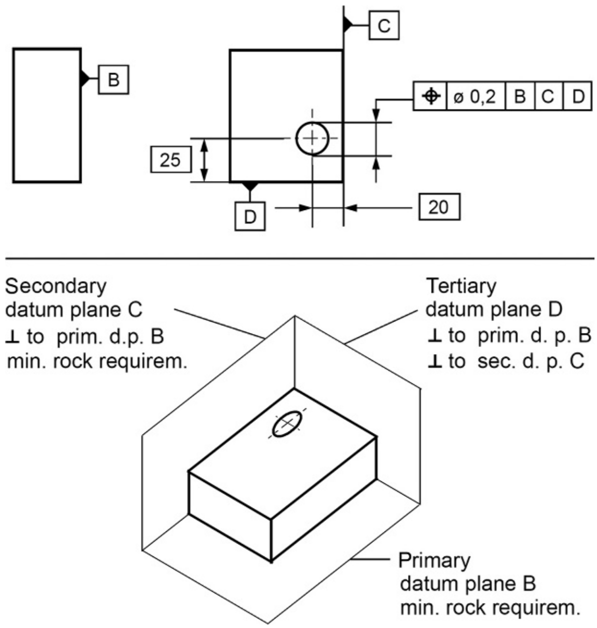


FIG. 3.84 Datum system

In the past there was a possibility of specifying **datum systems with undefined order** by indicating the datum letters in the tolerance indicator without separating them, e.g., AB. A can be primary or secondary. This is now explained in ISO 1101 Annex, Former practice, in order to read old drawings. It shall no longer be used, because of the differences in the meaning; see Fig. 3.85.

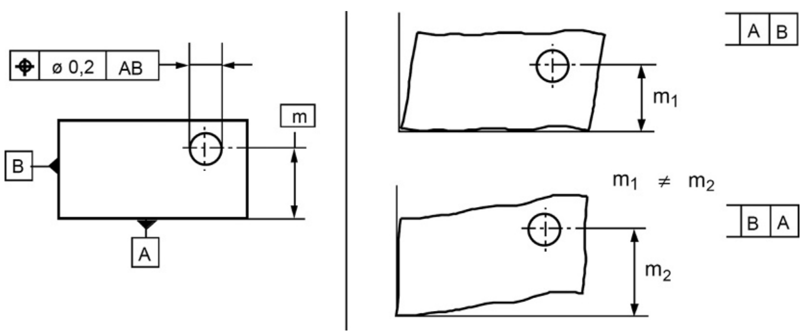


FIG. 3.85 Former practice, undefined order of datums, permissible possibilities

When the datum applies to the orientation only and there is a linear distance involved, the symbol >< is to be indicated after the datum letter in the tolerance indicator.

Within a Cartesian coordinate system, a part has six degrees of freedom (DOFs), three translations along the axes T_x, T_y, T_z; and three rotations around these axes R_x, R_y, R_z. When these six DOFs are locked, the part keeps the location and orientation; see Fig. 3.86.

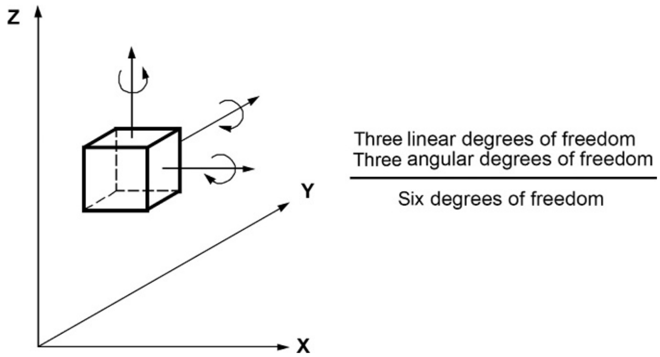


FIG. 3.86 Degrees of freedom

Each complete datum system establishes three datum planes perpendicular to each other; see Figs 3.84 and 3.87. It depends on the order of the datum as to which datum defines the datum planes. Datums can only lock degrees of freedom which are not yet locked by preceding datums; see Figs 3.87 to 3.91.

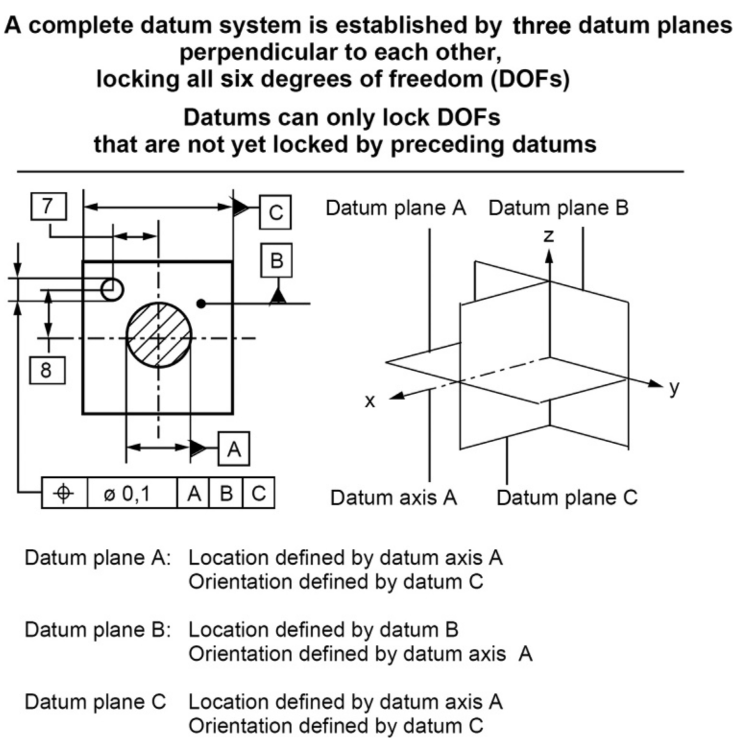


FIG. 3.87 Datum system established by three mutually perpendicular planes

In Fig. 3.87, the datum A is the straight median line of the minimum circumscribed cylinder. Datum plane A contains this median line. Datum plane B is perpendicular to datum A and to datum C. Datum C is parallel to the median plane of the smallest plane pair where the median plane contains the axis A.

In Figs 3.88 to 3.90, the datum plane B contains the straight median line of the minimum circumscribed cylinder perpendicular to the datum plane A. In Fig. 3.88 the datum plane C is defined by the median plane of the largest plane pair containing datum B. Datum planes B and C are perpendicular to each other.

In Fig. 3.89 the datum plane C is defined by the median plane of the smallest median plane pair containing datum B. Datum planes B and C are perpendicular to each other.

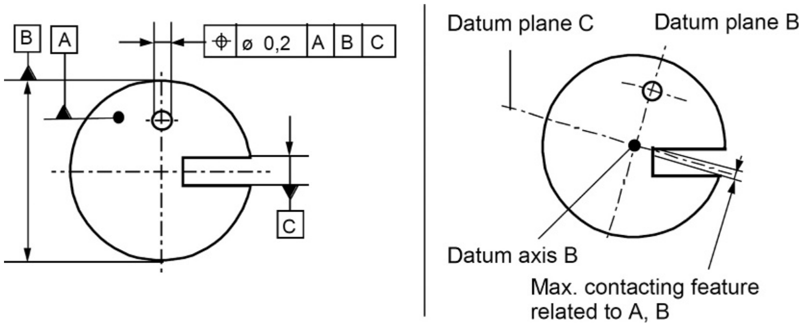


FIG. 3.88 Datum system: tertiary datum defines orientation

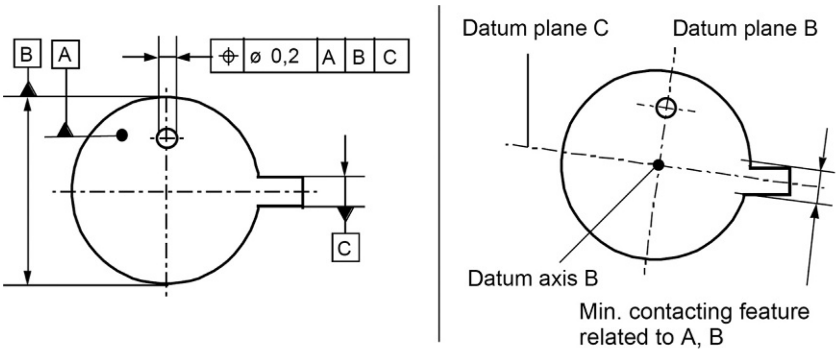


FIG. 3.89 Datum system: tertiary datum defines orientation

In Fig. 3.90, datum plane C is defined by the median plane of the largest plane pair five apart from datum B. Datum planes B and C are perpendicular to each other.

When within the datum system there is a linear TED between datums of different order, this TED is to be disregarded, unless there is a claimer [DF] indicated. That the TED is to be disregarded can also be indicated by the

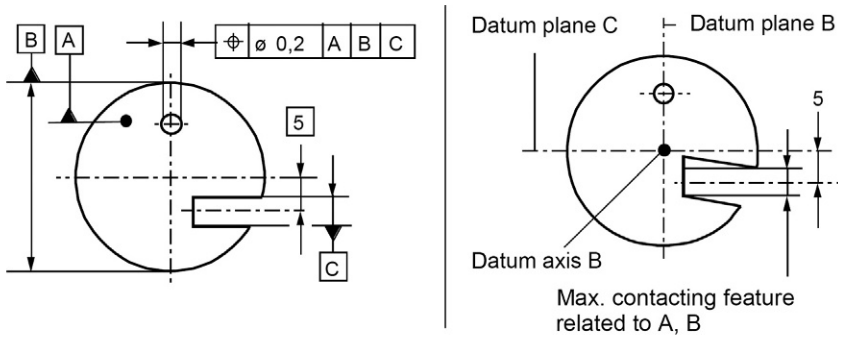


FIG. 3.90 Datum system: secondary datum defines location, and tertiary datum defines orientation

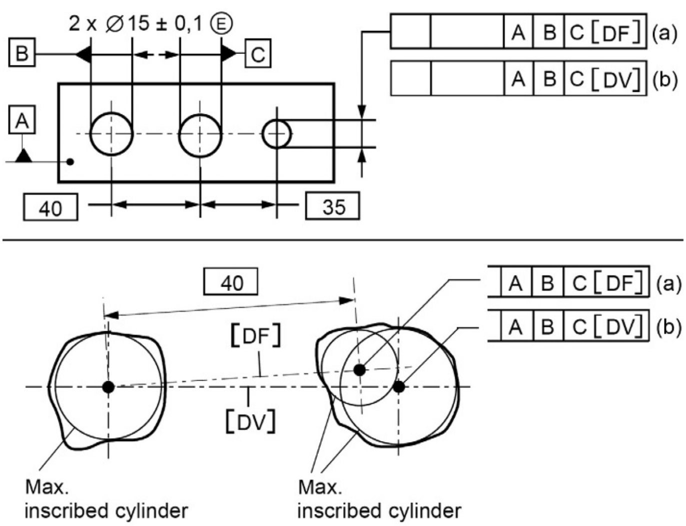


FIG. 3.91 Datum system with a linear TED between datums of different order, variable size

disclaimer [DV] (dimension variable). See Fig. 3.91. In order to follow the general rule "What is indicated, applies" (without exceptions), it is recommended to always indicate whether [DF] or [DV] applies.

Figure 3.92 shows an example with a TED for the size of the tertiary datum feature. In this case, it comes to the same result in orientation for both [DF] and [DV], so these modifiers may be omitted. Figure 3.93 shows an example of a planar datum feature, where a similar situation applies.

Situation features (point, straight line, plane) derived from the datum features are explained in 2.2 and 2.4.

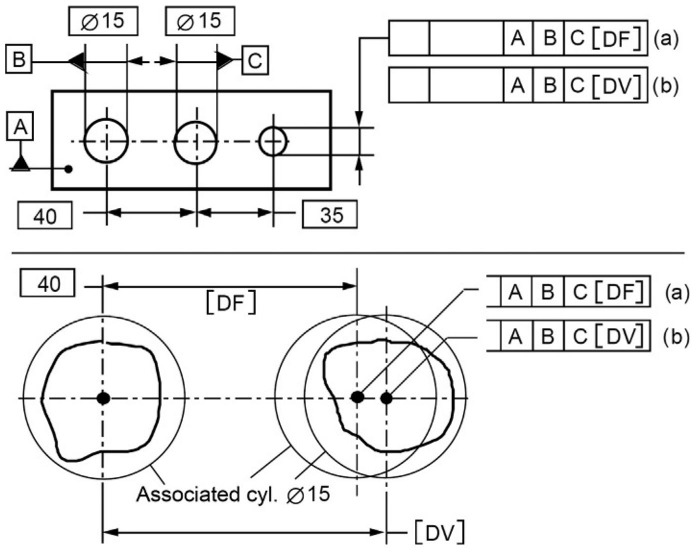


FIG. 3.92 Datum system with a linear TED between datums of different order, TED for size (diameter fix)

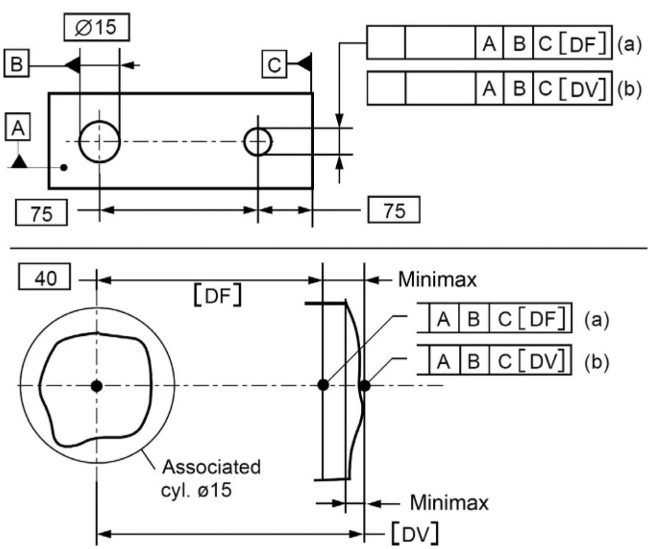


FIG. 3.93 Datum system with a linear TED between datums of different order, planar datum feature

When the drawing indication for the datum in the tolerance indicator contains the datum letter only, all situation features are applied. When only one situation feature is to be applied, e.g. the apex of the cone and not the axis, then the applied situation feature is to be indicated in square brackets, e.g. [PT] for the apex of the cone; see Fig. 3.94.

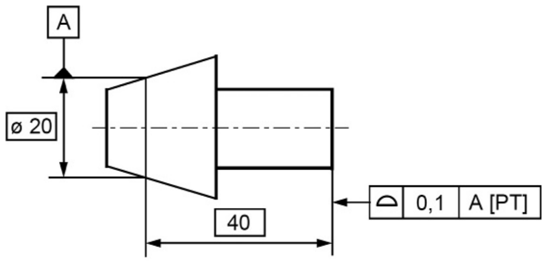


FIG. 3.94 Selected situation feature point

Drawing indications for selected situation features are:

Point	Drawing indication:	[PT]
Straight line	Drawing indication:	[SL]
Plane	Drawing indication:	[PL]

When a right-handed Cartesian **coordinate system X, Y, Z** is indicated, selected degrees of freedom, DOFs, (for translation T_x, T_y, T_z and for rotation R_x, R_y, R_z) can be locked by a datum feature. Notice that the indicated DOFs are locked, while the others are free. See, e.g. Fig. 6.3 tolerances p, q, r.

Figure 3.95 shows a datum established by **restricted areas**.

Form deviations of datum features are not limited by the indication of a datum. A form or orientation or location tolerance for the datum feature is to be indicated.

With **features of size**, there are three types of datums; see Fig. 3.96. Without modifier, the ideal feature with mating size and with Chebyshev orientation applies. With Ⓜ the datum feature is movable within the gauge. With Ⓛ the datum feature may move around the cut-out contour.

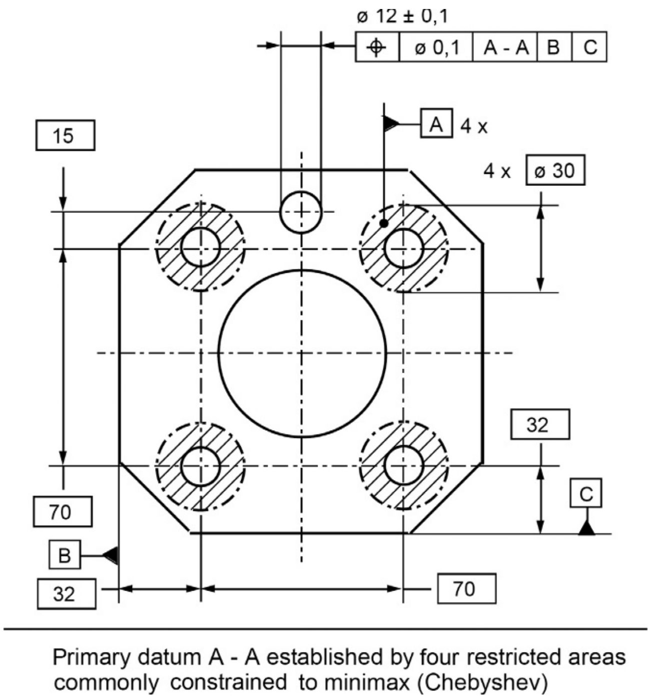


FIG. 3.95 Datum from restricted areas (instead of datum targets)

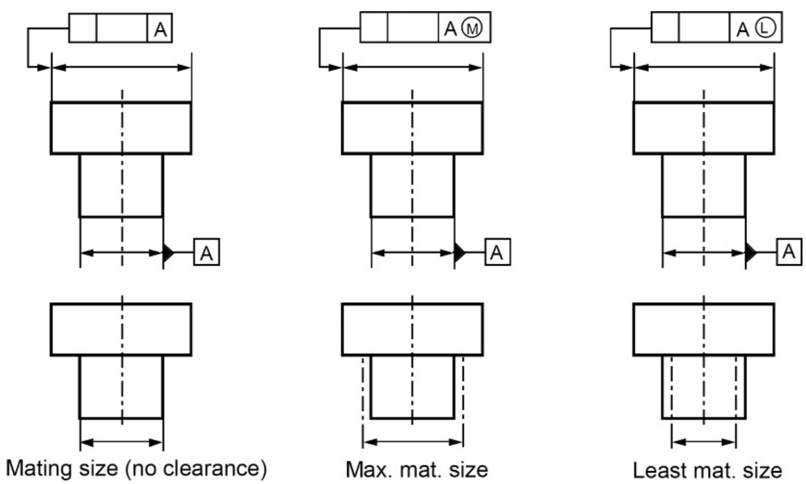


FIG. 3.96 Types of datums

3.3.2 Datum targets

Datum systems can be established from integral features and from restricted areas of integral features; see previous, or from datum targets, see, e.g., Fig. 3.97.

Datum targets are points (contacting spheres), small areas (contacting planes), or straight lines (contacting cylinders) contacting the part in fixed locations relative to each other or movable in a specified direction. They define an ideal support device for the part. The symbols for datum targets are shown in Table 3.5.

For **rigid parts** there are two types of datum target systems:

a) The 3-2-1 method. The primary datum has three datum targets, the secondary two and the tertiary one. See Fig. 3.97. The part does not rock when it rests in the supporting device. The supporting device is shown in Fig. 3.98.

b) Method of stable primary datum. The primary datum has the minimum number of datum targets in order to lock all its degrees of freedom: in the case of a cylinder, four datum targets. See Fig. 3.102. Six datum targets are used in total. The part does not rock when it rests in the supporting device.

When the contacting features define a datum that is different from the datum feature (e.g. a plane instead a cylinder) the symbol [CF] (meaning contacting feature) shall be indicated; see Figs. 3.102 and 3.106.

The total number of datum targets for rigid parts is always six, in accordance with the six degrees of freedom in a coordinate system (three translations along x, y, z and three rotations around x, y, z). Each datum target removes one of the degrees of freedom. See Fig. 3.100.

Figure 3.98 shows the supporting device according to Fig. 3.97.

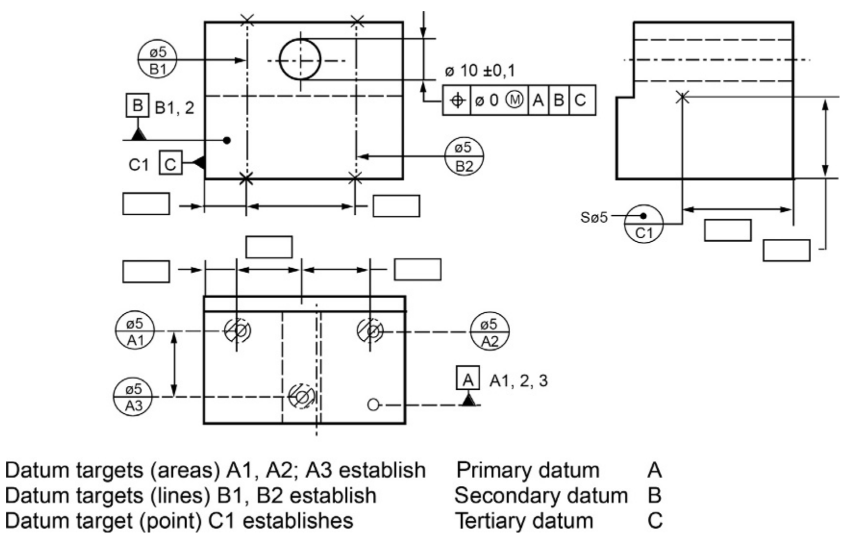


FIG. 3.97 Example of datum targets according to the 3-2-1 method

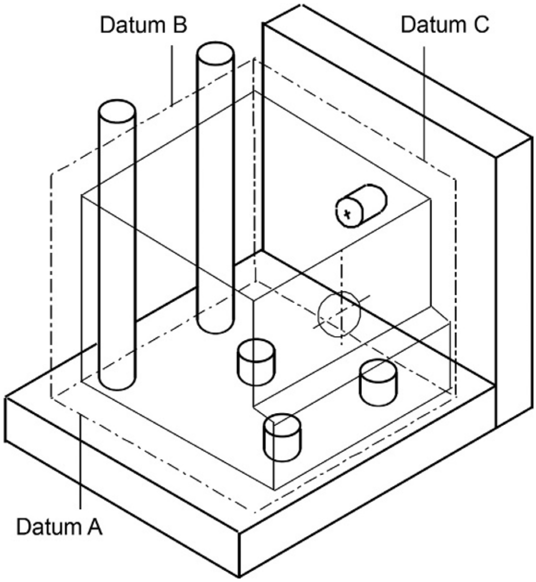


FIG. 3.98 Supporting device according to datum targets of Fig. 3.97

Figure 3.99 shows how the datum planes are located relative to the contacting features.

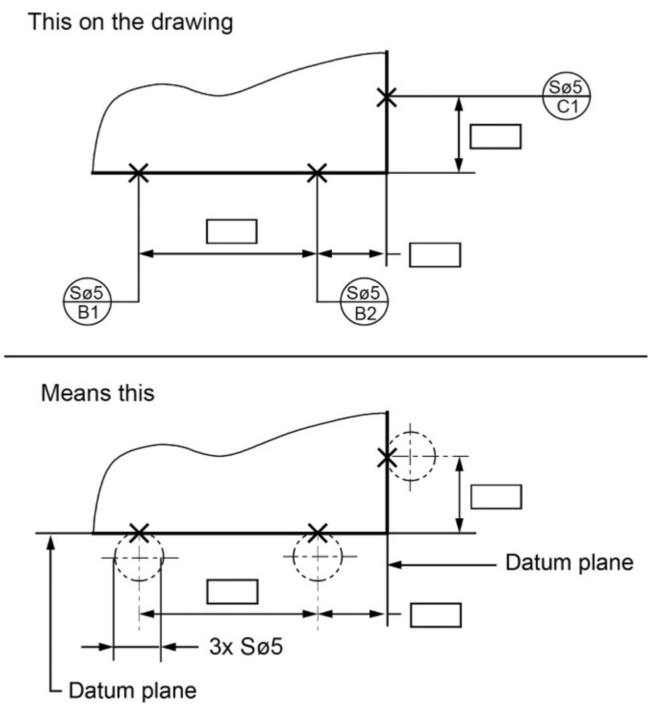


FIG. 3.99 Datum planes located relative to the contacting features

Fig. 3.100 shows the 6 degrees of freedom for a workpiece.

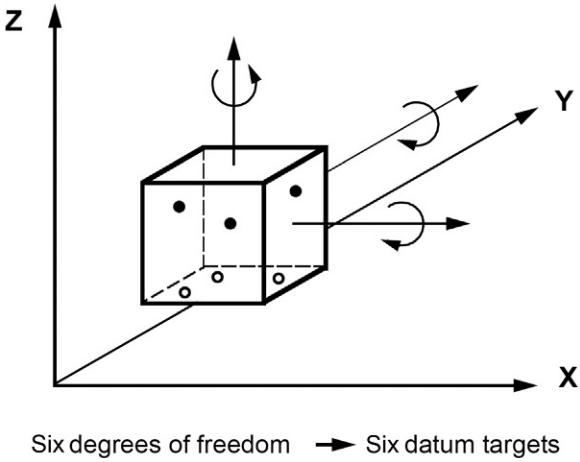
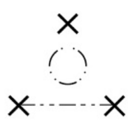


FIG. 3.100 Datum targets, 3-2-1 method; each datum target locks one DOF

Figure 3.101 shows the rules for locating datum targets, in order for each to lock a DOF. E.g. three datum target points for a primary datum must not lie on a straight line. The two secondary datum targets must not lie on a line normal to the primary datum, etc.

Datum targets:



PT point (support on a sphere)

PL small flat surface in a (common) plane

SL straight line (support on a cylinder)

Datum targets are supports located in such a way that the workpiece does not rock

E.g.:

Prim. datum: 3 PT forming a triangle or
3 PL in a plane forming a triangle or
2 PT + 1 SL apart from each other or
2 PL in a plane + 1 SL apart from each other

Sec. datum: 2 PT // to prim. datum or
2 SL ⊥ to prim. datum or
1 PT + 1 SL ⊥ to prim. datum and apart from each other

Tert. datum: 1 PT or
1 SL or
1 PL

FIG. 3.101 Rules for locating datum targets

When the 3-2-1 method is used for a cylindrical datum feature, the primary datum with three datum targets is not yet stable (it becomes stable together with the two datum targets of the secondary datum feature).

Figure 3.102 shows the datum targets to be used (four) for the cylindrical primary datum feature, in order to make it stable.

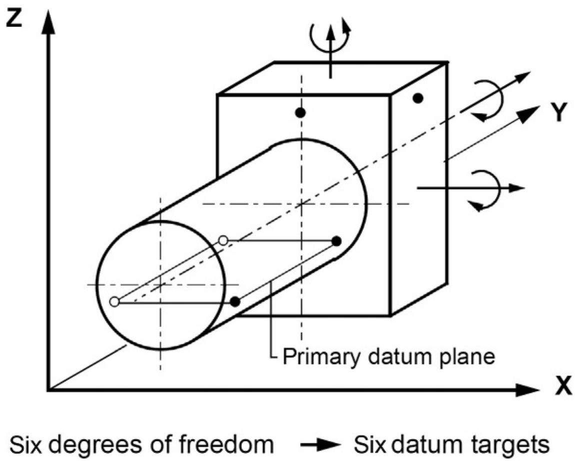
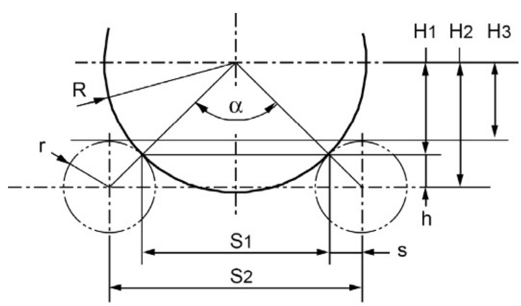


FIG. 3.102 Primary datum targets on a cylindrical feature to make the datum stable

Figure 3.103 shows the nominal dependency of datum target points or straight lines and the cylinder centre line.



$$R^2 = (S1/2)^2 + H1^2$$

$$H1 = \sqrt{(R^2 - (S1/2)^2)}$$

$$\alpha/2 = \arctan \frac{S1}{2\,H1}$$

$$H1 = R \cos(\alpha/2)$$

$$S1 = 2\,R \sin(\alpha/2)$$

$$H2 = (R + r)\cos(\alpha/2) \qquad h = r\cos(\alpha/2)$$

$$S2 = 2\,(R + r)\sin(\alpha/2) \qquad s = r\sin(\alpha/2)$$

$$H3 = (R + r)\cos(\alpha/2) - r$$

FIG. 3.103 Nominal dependency of datum target points or straight lines and the cylinder centre line.

When on a cylindrical datum feature three or four spherical datum targets are used, the datum plane (start of dimensions) may be defined by the centres or highest or lowest points of the spheres or by the points of nominal contact.

In order to centre the part in the fixture, **movable datum targets** can be used. Figure 3.104 shows the symbol for the drawing indication.

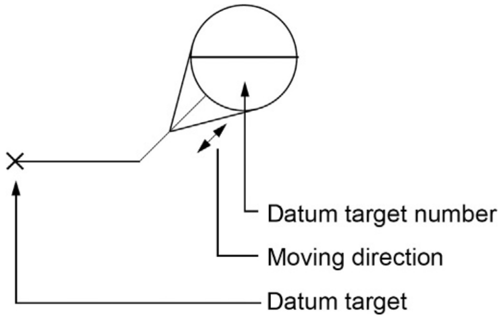


FIG. 3.104 Symbol for a movable datum target

When three-jaw chucks (movable datum targets) are used to establish a point or part of the axis, the total number of datum targets may be different from 6 datum targets (Fig. 3.105).

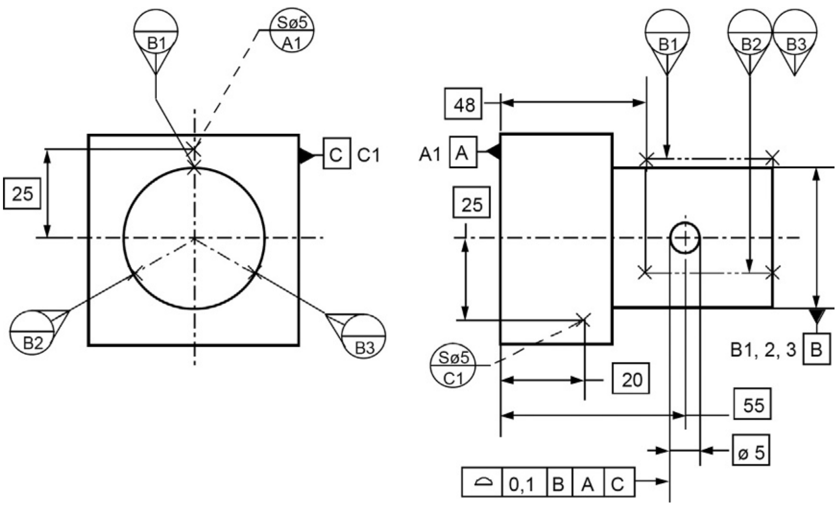
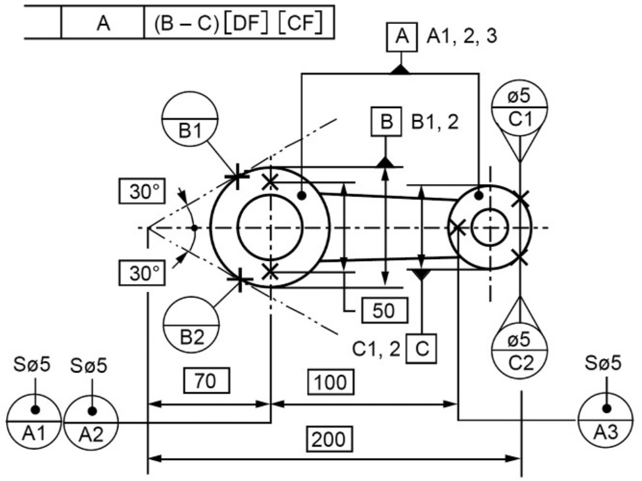


FIG. 3.105 Datum target system using a three-jaw chuck

Figures 3.106 to 3.108 show the use of movable datum targets with and without [DV].

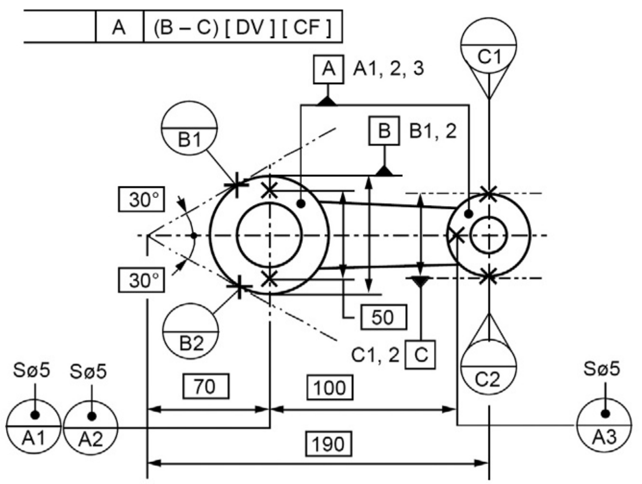


Constraint:

set of movable datum targets in theor. exact orientation and location relative to other datums of the datum system, i.e.

centre C1, C2 on median plane B1, B2
moving direction perpendicular to median plane B1, B2
distance 200 fixed

FIG. 3.106 Movable datum targets, linear TED with [DF]



Constraint:

set of movable datum target planes in theor. exact orientation and location relative to other datums of the datum system, i.e.

centre C1, C2 on median plane B1, B2
moving direction perpendicular to median plane B1, B2
(distance 190 variable)

FIG. 3.107 Movable datum target, linear TED with [DV]

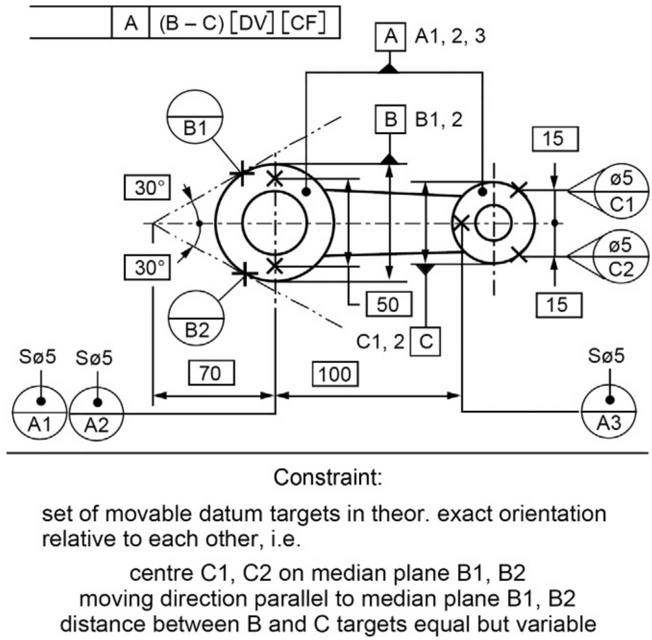


FIG. 3.108 Movable datum target, linear TED with [DV]

3.3.3 Change of the datum system

Sometimes, e.g. for manufacturing reasons, the datum system is changed. In this case, in order to meet the original tolerances, the tolerances related to the new datum system are to be decreased.

The inclination or the height offset of the new primary datum relative to the original primary datum requires a reduction of the new tolerances; see Fig. 3.109.

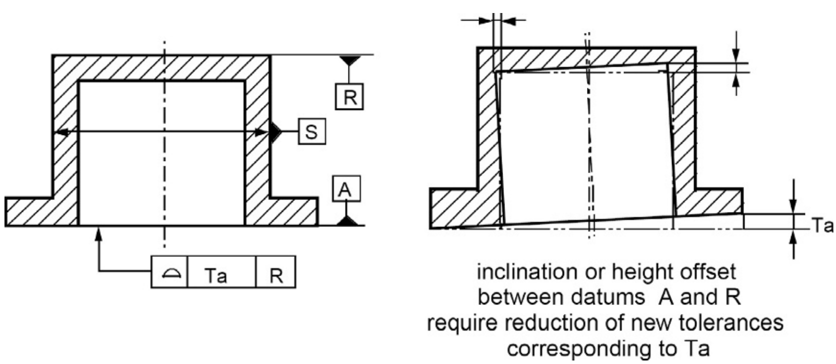


FIG. 3.109 Change of the primary datum, resulting decrease of the tolerances of the new datum system

The offset of the secondary datum relative to the original secondary datum requires an additional reduction of the new tolerance; see Figs 3.110 and 3.111.

Because of the tolerance decrease, changing of the datum system should be avoided. If this is not possible, the tolerances of the new datum features relative to the original datums should be as small as possible.

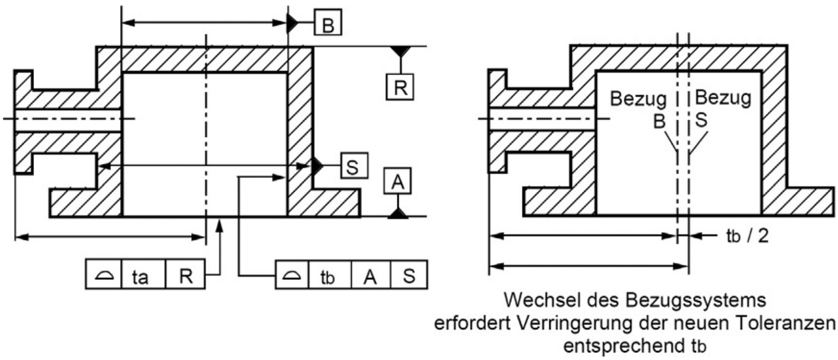


FIG. 3.110 Change of the secondary datum, resulting decrease of the tolerances of the new datum system

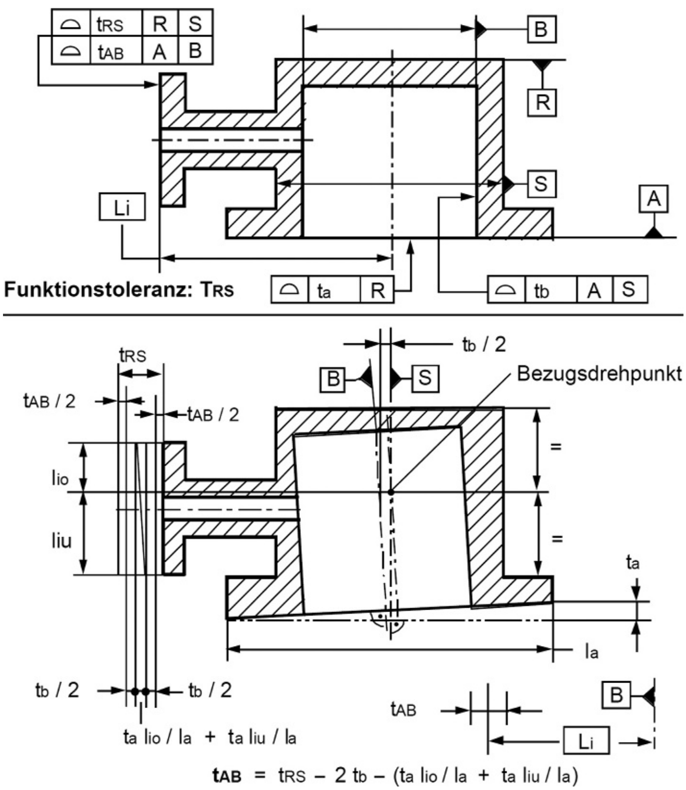


FIG. 3.111 Change of the datum system, resulting decrease of the tolerances of the new datum system

3.3.4 Interchanging of toleranced feature and datum

Figure 3.112 shows the same workpiece with different datums. The effect is quite different. Therefore toleranced feature and datum must not be interchanged. A change of the drawing is needed.

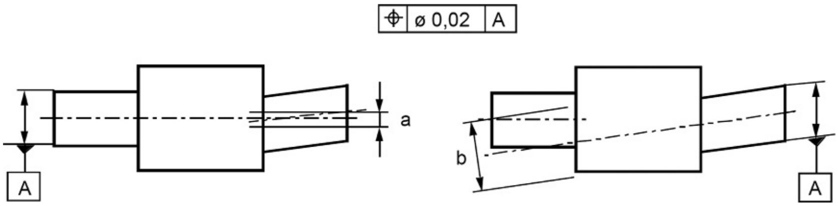


FIG. 3.112 Effect of interchanging toleranced feature and datum

3.3.5 Inspection appropriate tolerancing

What cannot be measured cannot be manufactured intentionally. Such drawings are useless and wrong. When the inspection is not possible with sufficient accuracy (i.e. when the measuring uncertainty becomes too large), it is not inspection appropriate. For example, tolerancing in Fig. 3.113 left is not inspection appropriate, because the datum feature is too short.

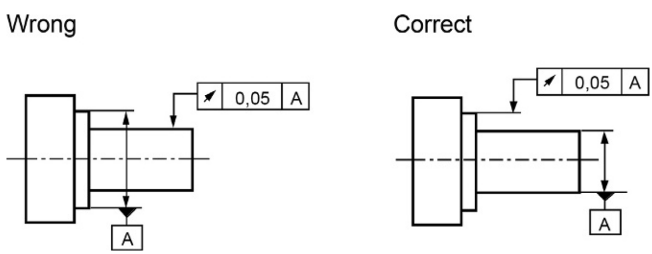


FIG. 3.113 Tolerancing: inspection appropriate (right) and not inspection appropriate (left)

If necessary, a further datum is to be indicated (e.g. the flange surface) in order to stabilize the orientation of the workpiece (Fig. 3.114).

Figure 3.115 shows tolerancing that is not inspection appropriate. The measurement uncertainty of the location of the axis derived from a part of a cylinder smaller than half of the cylinder is very large (often larger than the tolerance).

Figure 3.116 shows tolerancing that is inspection appropriate. The deviation of the surface from the theoretically exact form at the specified location can be measured with a smaller measurement uncertainty.

Figure 3.117 left shows a wrong datum indication. The datum indicator is allocated to the centre line that stands for several median lines (three median lines of three cylinders, one median surface of the slot, one median surface

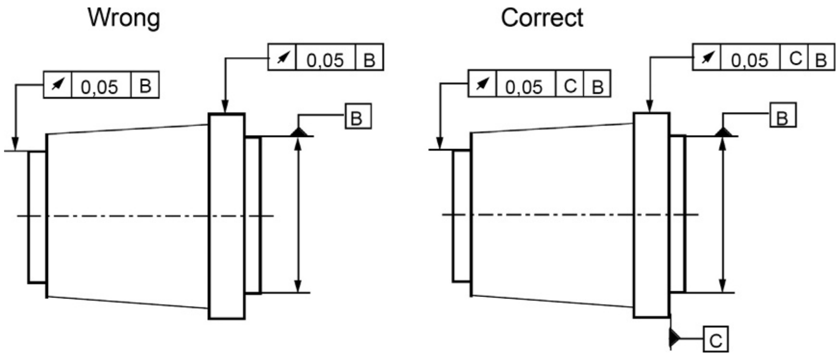


FIG. 3.114 Short datum and further datum in order to stabilize the orientation of the workpiece

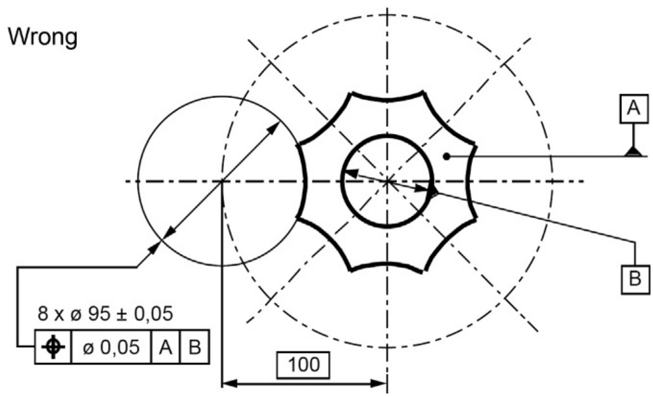


FIG. 3.115 Not inspection appropriate tolerancing

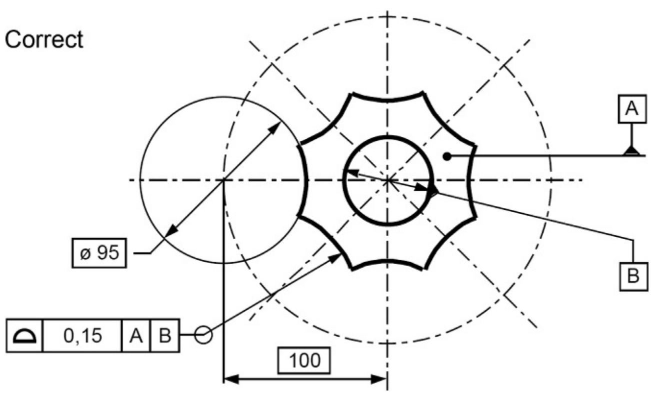


FIG. 3.116 Inspection appropriate tolerancing

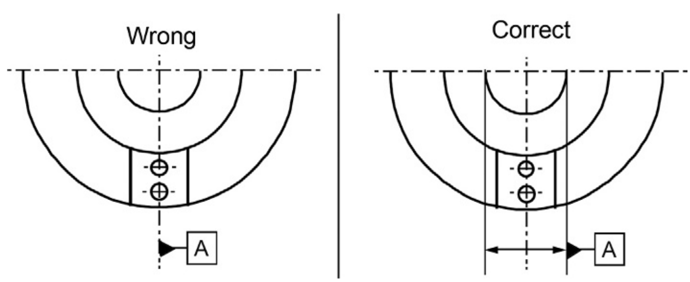


FIG. 3.117 Indication of datums

of the two holes). Each of them yields different measuring results. Therefore, the datum indicator shall always be indicated to the particular feature; see Fig. 3.117 right.

3.4 Flexible parts

According to the “rigid workpiece principle” in ISO 8015, the ISO default is to consider the workpiece as rigid. When the workpiece is flexible (possibly takes a different form when assembled), the reference “ISO 10 579-NR” (NR = non-rigid) shall be placed in the drawing in or near the title block.

The tolerances without the symbol Ⓕ apply to the restrained condition of the workpiece. When the restrained condition is not defined by the datum system, it is to be defined near the drawing title box. The restrained condition should correspond to the assembly.

For **flexible parts** (e.g. for a part of thin sheet metal), more than six datum targets are possible. All datum targets must contact the (flexible) part, so the flexible part must be able to deform accordingly. This can be assured by specifying the maximum force necessary to bring the part to contact or by tolerancing the shape of the part in the free state using the symbol Ⓕ; see Fig. 3.118.

Figure 3.118 shows an example of tolerancing a flexible part.

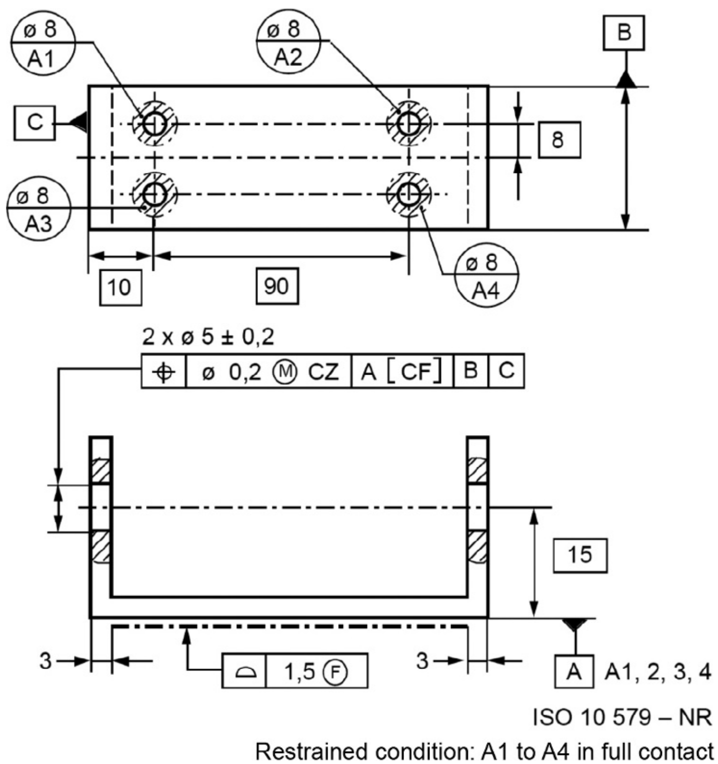


FIG. 3.118 Tolerancing of a flexible part

3.5 Decomposition of measurement results

When a location tolerance (profile surface or position) is exceeded, it is not evident whether the exceeding is caused by deviations of form or of orientation or of location; see Fig. 3.119. Furthermore, it is not evident in which direction the deviation requires adjustment of the machine tool, i.e. in which direction the manufacturing process is to be adjusted. Therefore **decomposition of the inspection results** is necessary (ISO 20 170).

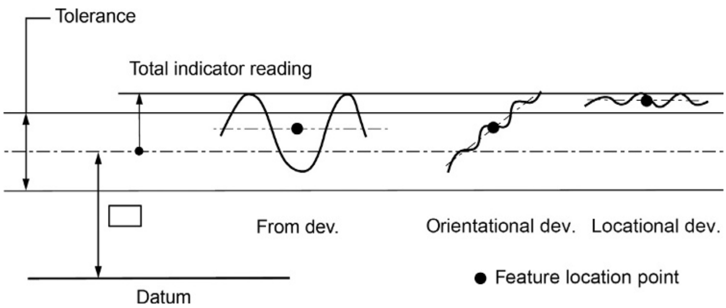


FIG. 3.119 Exceeding a location tolerance

Decomposition according to ISO 20 170 yields the required information. It uses an imaginary geometrically ideal feature (e.g. a plane or a cylinder) associated to the real feature (real surface). This associated feature is called a substitute feature; see Fig. 3.120. Decomposition requires a Cartesian coordinate system.

The association follows an objective function. The objective function can be:

Ⓒ Minimum zone (Chebyshev, minimax)
Ⓛ Least squares (Gauss)
Ⓧ Maximum inscribed
Ⓝ Minimum circumscribed

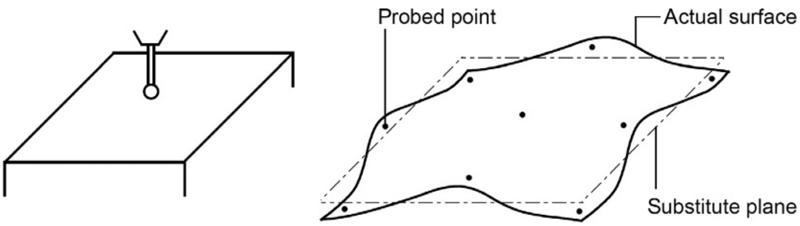


FIG. 3.120 Substitute feature

These association methods may yield different results, although they use the same extracted feature (same sensed points). Only the Gaussian association Ⓖ always leads to a unique result.

The orientation of the substitute feature is defined by the orientation of vector V_O. This is the unit vector (with the value 1) perpendicular to the substitute plane (situation plane) and parallel to the axis of the substitute cylinder (situation straight line). The positive direction of the orientation vector is out of material in the case of a substitute plane and away from the coordinate system origin in the case of a substitute cylinder. See Figs 3.121 and 3.122.

The location of the substitute feature is defined by the location vector V_L. This is the vector to the point (location point) of the substitute plane (situation plane) or of the axis of the substitute cylinder (situation straight line) which is nearest to the coordinate system origin. See Figs 3.121 and 3.122. The location point may be outside the part.

The orientation vector V_O and the location vector V_L are substituted by their vector triples (components) in the directions of the coordinate axes.

Vector triples are designated according to ISO 20 170 as:

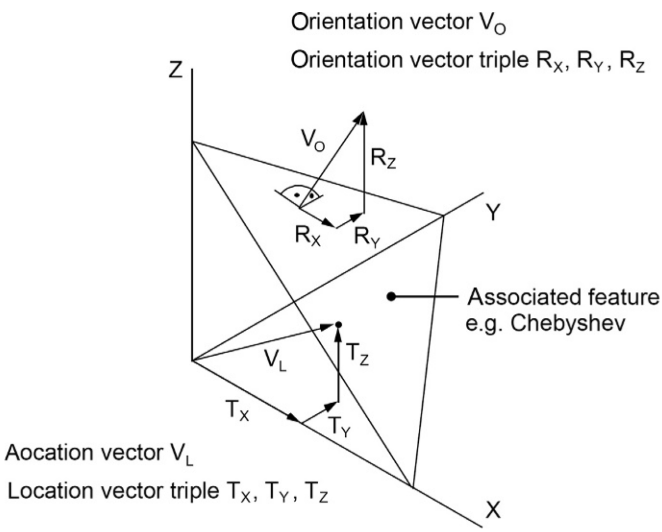


FIG. 3.121 Orientation and location vectors of a plane

The following further indices designate:

T	Theoretically exact (nominal)
A	Actual (real)
a	Angular (in angular units)
Δ	Deviation (real minus nominal)

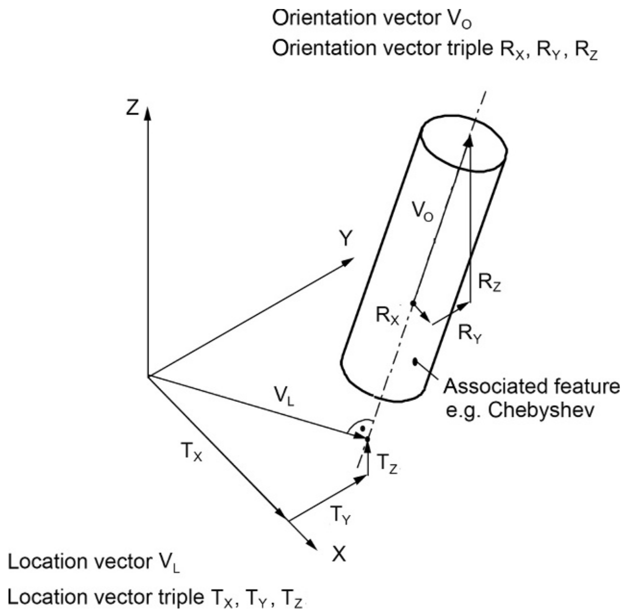


FIG. 3.122 Orientation and location vectors of a cylinder

According to ISO 20 170, the following symbols are standardized for global characteristics (global means: expressed by one single value) in length units:

A	Actual (real) value of a specified GPS characteristic (global result of deviation of size, form, orientation or location)
GF	(Global) form characteristic (result of form measurement)
GS	Global size characteristic (result of size measurement)
GO	(Global) orientation characteristic corresponding to R_X, R_Y, R_Z of the substitute feature (result of orientation measurement)
GL	(Global) location characteristic corresponding to T_X, T_Y, T_Z of the substitute feature (result of location measurement)

Figure 3.123 shows the orientation vector projected onto the datum planes. The orientation vector may be the nominal V_{OT} or the real V_{OA}. The difference $(V_{AO} - V_{TO})$ results in the deviation $V\Delta_O$. The same applies to the components R_X, R_Y, R_Z. This allows the expression in angular grades.

Figure 3.124 shows an example of a set of GPS characteristics. Figure 3.125 shows the corresponding measurement report.

As for specification No 2 the nominal orientation vector for R_Z is 0; $R_{X\Delta}$ and $R_{Y\Delta}$ provide all necessary information for manufacturing control and $R_{Z\Delta}$ is not needed.

When a datum system is applied for manufacturing, different from the drawing specification, the tolerances have to be decreased by the effect of the

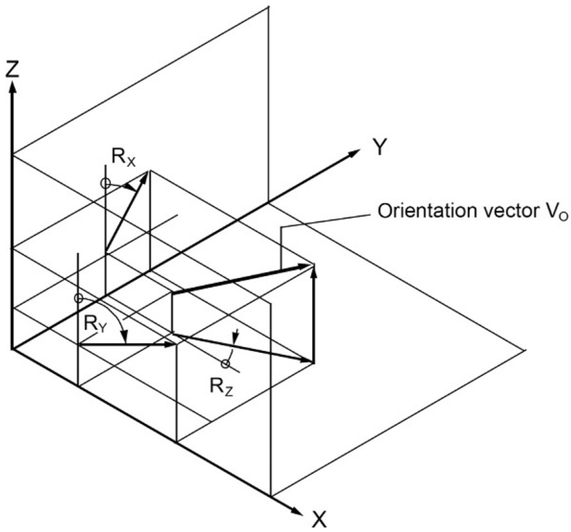


FIG. 3.123 Orientation vector components projected into the coordinate planes

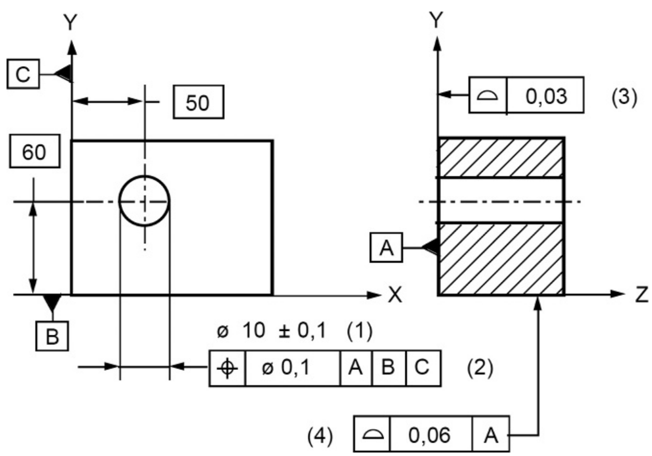


FIG. 3.124 Example of a set of GPS specifications

deviations of the manufacturing datum system from the (original) drawing datum system, This is in order to meet the original tolerances. See 3.3.3.

Figure 3.126 shows a position tolerance for a hole and the procedure for an automated manufacturing process control.

The nominal matrix shows the nominal values for the location and orientation vector triples. D_{GX} means the diameter of the maximum inscribed cylinder.

Characteristic	GPS specification			
	No. 1	No. 2	No. 3	No. 4
A (global result)	+0,02; –0,03	0,08	0,022	0,068
U (uncertainty of A)	0,003			
$G_{S\Delta}$ (size deviation result)	0,05			
$G_{F\Delta}$ (form result)		0,03	0,022	0,04
$G_{O\Delta}$ (orientation result)		0,022		0,028
$R_{X\Delta}$, $R_{Y\Delta}$, $R_{Z\Delta}$		+0,03°; –0,06°; 0°	0°; 0°; 0°	+0,02°; 0°; +0,01°
$G_{L\Delta}$ (location result)		0,014		
$T_{X\Delta}$, $T_{Y\Delta}$, $T_{Z\Delta}$		+0,01; +0,01; 0°		

FIG. 3.125 Measurement report for Fig. 3.124

Identification of the surface: 12345

ø 10 ±0,1

Drawing tolerance for hole: | ⌖ | ø 0,2 Ⓜ | A | B | C |

1 Normal surface matrix

T_{XT}	T_{YT}	T_{ZT}	R_{XT}	R_{YT}	R_{ZT}	D_{GXT}
50	60	0	0	0	0	ø 10

2 Real surface matrix

T_{XA}	T_{YA}	T_{ZA}	R_{XA}	R_{YA}	R_{ZA}	D_{GXA}
50,01	59,99	0	+0,06°	–0,04°	0°	ø 10,05

3 Deviation matrix (2 - 1)

$T_{X\Delta}$	$T_{Y\Delta}$	$T_{Z\Delta}$	$R_{X\Delta}$	$R_{Y\Delta}$	$R_{Z\Delta}$	$D_{GX\Delta}$
+0,01	–0,01	0	+0,06°	–0,04°	0°	+ ø 0,05

4 Correction matrix (3 with reversed signs)

T_{XC}	T_{YC}	T_{ZC}	R_{XC}	R_{YC}	R_{ZC}	D_{GXC}
–0,1	+0,01	0	–0,06°	+0,04°	0°	– ø 0,05

FIG. 3.126 Geometrical tolerancing, automated manufacturing process control

The real surface matrix shows the real (measured) values of the inscribed maximum material cylinders related to datum A. The deviation matrix shows the difference between the real surface matrix and the nominal surface matrix. Reversing the signs in the deviation matrix results in the correction matrix for the manufacturing process control.

It is possible to transform the components (tolerances and deviations) into the axes of the machine tool, in order to achieve an automated manufacturing process control.

3.6 Vectorial dimensioning and tolerancing VD&T

It is possible, but not yet introduced into industrial practice, to define the vectors and their tolerances for each surface, as shown in Fig. 3.126. The list is to be extended, as appropriate, by form, additional profile or position, Ⓔ, Ⓜ, Ⓛ, Ⓟ, roughness, waviness. Then an automated manufacturing process control is possible, as described in Fig. 3.126.

In 3D CAD systems and in coordinate measuring systems (CMSs), these vectors often already exist.

This page intentionally left blank

Chapter 4

Size Tolerancing

4.1 Linear sizes

Features of linear size are circle, two opposite parallel straight lines, sphere, cylinder, two opposite parallel planes (plane pair) and torus.
Linear size is the characteristic dimension of a feature of linear size (diameter, width).
Size is the characteristic dimension of a feature of size (diameter, width).

ISO 14 405-1 defines the following sizes:

Local sizes (varying within the feature of the workpiece, depending on the location of the size):

(LP)	Two-point measurements
(LS)	Ball (cram sphere) size

Global sizes (constant over the feature length):

(GX)	Max. inscribed size
(GN)	Min. circumscribed size
(GG)	Least squares size (Gauss)
(GC)	Minimax size (Chebyshev)

Calculated size (calculated from):

(CC)	Circumference ø size
(CA)	Area ø size
(CV)	Volume ø size

Geometrical Dimensioning and Tolerancing for Design, Manufacturing and Inspection
https://doi.org/10.1016/B978-0-12-824061-8.00018-9

For local sizes, the following statistical characteristics (**statistical sizes**, rank order sizes) may be toleranced:

- (SX) Max. size
- (SN) Min. size
- (SA) Average size
- (SM) Median size
- (SD) Mid-range size
- (SR) Range of sizes
- (SQ) Standard deviation of sizes

Global sizes may be applied to cross sections lying anywhere within the feature; see Fig. 4.1 for an example. They then become local sizes.

10 ±0,1 (GX) / 0

FIG. 4.1 Global size applied to cross sections

Global sizes may be applied to restricted lengths lying anywhere within the feature: for example, when a washer shall fit to a longer cylinder (see Fig. 4.2).

10 ±0,1 (GX) / 5

FIG. 4.2 Global size applied to a restricted length lying anywhere

Figures 4.3 to 4.5 show examples of linear size.

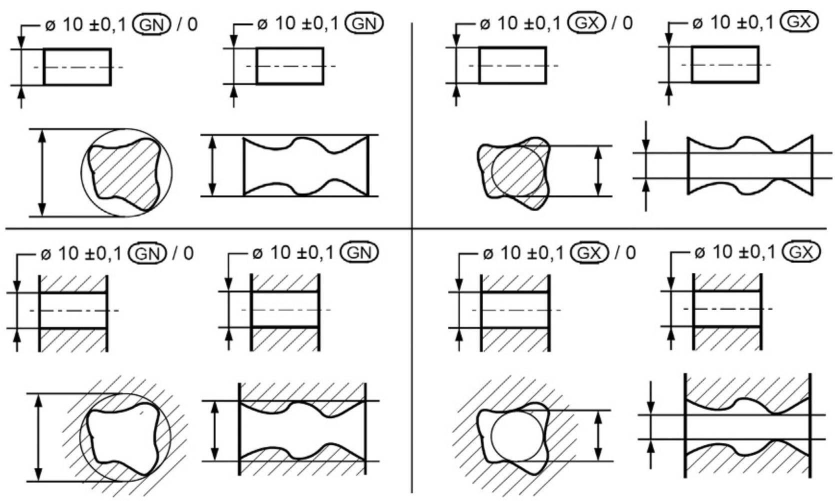


FIG. 4.3 Linear sizes, global minimum circumscribed, global maximum inscribed, in cross sections and over the entire length

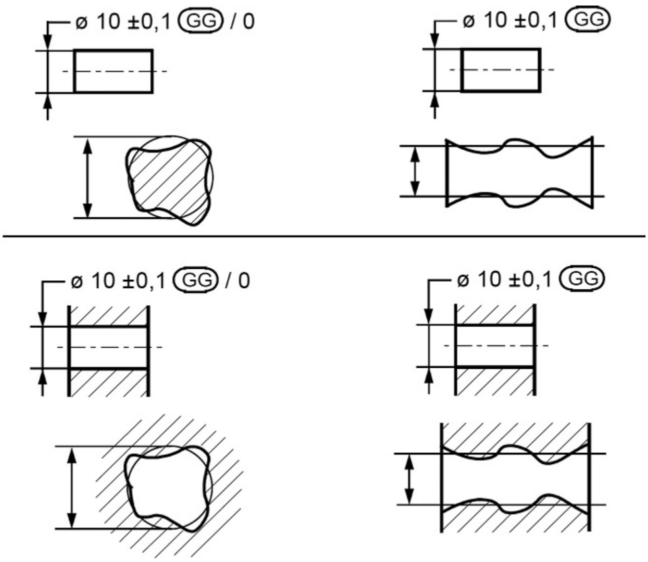


FIG. 4.4 Linear sizes, global Gauss in cross sections and over the entire length

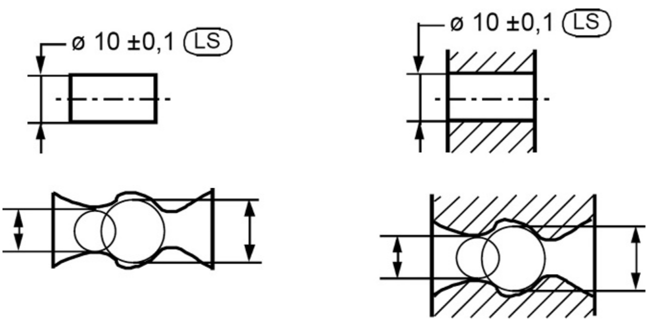


FIG. 4.5 Linear sizes, sphere sizes

Mating size: size of the smallest circumscribed (with outer features) or largest inscribed (with inner features) cylinder or plane pair of the real feature of size.

Maximum material size: maximum size (with outer features) or minimum size (with inner features).

When features are regarded as a **united feature**, e.g. as one cylinder in Fig. 4.6, **UF** is to be indicated.

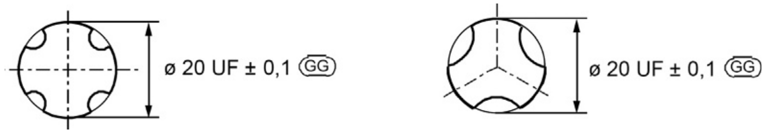


FIG. 4.6 Unified feature (UF)

When a tolerance applies simultaneously to more than one feature, **CT (common tolerance)** is to be indicated; see Fig. 4.7.

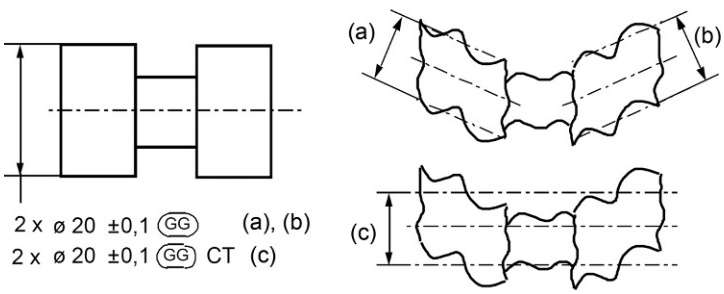


FIG. 4.7 Common tolerance (CT)

Feature of size: Cylinder or two opposite parallel planes (plane pair).

Figure 4.8 shows the definition of a **two-point size** for a cylindrical feature. First a Gaussian element is associated. With cylinders, intersections perpendicular to the Gaussian element are then determined. In each intersection, the Gaussian circle is associated. The two-point sizes are the distances in these Gaussian circles which meet the centre points of the Gaussian circles. With plane pairs, the two-point sizes are the distances perpendicular to the median plane of the two independent Gaussian associated planes; see Fig. 4.9.

Figure 4.9 shows the definition of a two-point size for a plane pair.

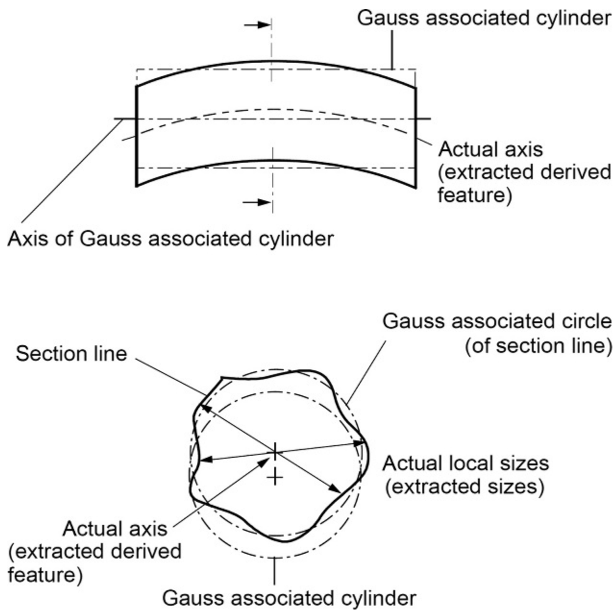


FIG. 4.8 Two-point size of a cylindrical feature

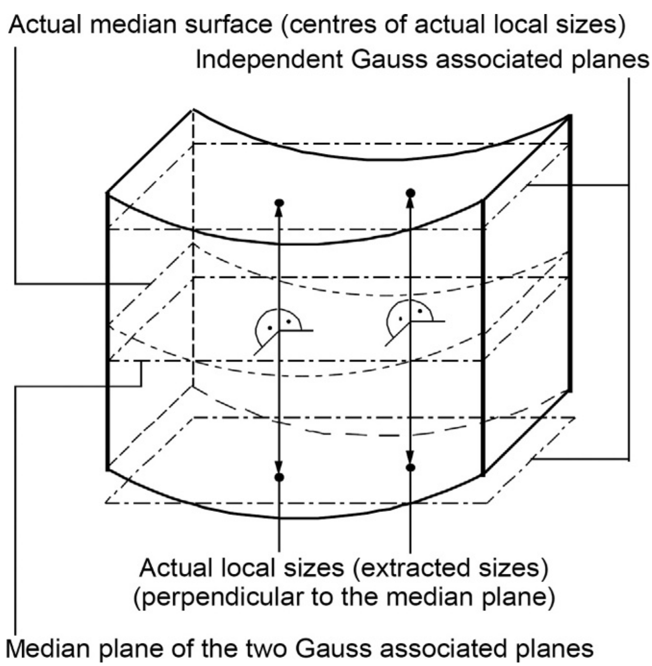


FIG. 4.9 Two-point sizes of a plane pair

4.2 Equal thickness

Some functions (e.g. of thin discs of multiple-disc clutches) require small limits on the variation of the actual local sizes (e.g. of thickness) within the same feature of a single workpiece. The size deviations of different workpieces, however, may vary over a larger range. The parallelism tolerance is not appropriate for this purpose, since a workpiece may have a considerable parallelism deviation but almost no variation in its actual local sizes (Fig. 4.10).

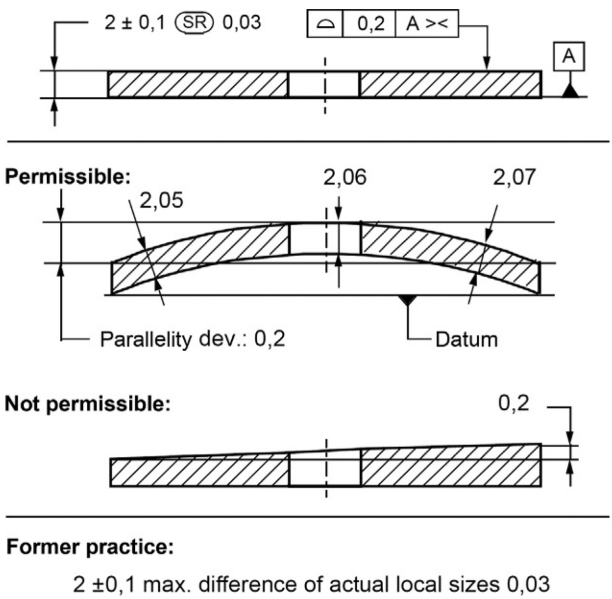


FIG. 4.10 Workpiece with considerable parallelism deviation but almost no variation in its actual local size

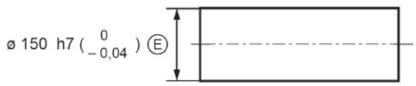
4.3 Envelope requirement

4.3.1 Definition, history

The history of the envelope requirement is described in Fig. 4.11.

ISO 8015:1985 specified:
“Envelope requirement: … The requirement means that the envelope of perfect form at maximum material size of the feature shall not be violated.”

Further was specified:



“This means:
- each actual local diameter of the shaft shall remain within the size tolerance of 0,04 and, therefore, may vary between Ø150 and Ø149,96
- the entire shaft shall remain within the boundary of the envelope cylinder of perfect form and of Ø150.”

ISO 14405-1:2010 specified:
“3.12 envelope requirement
simultaneous use of the combination of the two-point size (…) as the specification operator applied for the least material limit of size (…) and either the minimum circumscribed size (…) or the maximum inscribed size (…) as the specification operator applied to the maximum material limit of size.”

FIG. 4.11 History of the envelope requirement

Consequently both standards require that
- the limits of size must be respected by two-point sizes
- the feature of size has to fit into an ideal gauge of maximum material size.

The wording is different, but the result is the same.

When more than one feature is to fit simultaneously into the gauge, in the past CZ was indicated after Ⓔ. According to ISO 14 405-1, now CT (common tolerance) is to be indicated after Ⓔ; see Fig. 4.12.

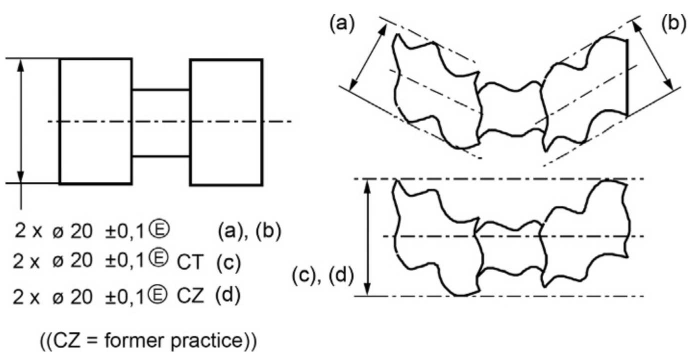


FIG. 4.12 Envelope requirement, simultaneous requirement

The envelope requirement cannot be applied to features for which a form tolerance is specified that is larger than the size tolerance. According to ASME Y14.5, the envelope requirement also does not apply to features for which a straightness tolerance of the axis (even if smaller than the size tolerance) is specified.

4.3.2 Application of the envelope requirement

The envelope requirement is applicable to features of size (cylindrical surfaces or features established by two parallel opposite planar surfaces, plane pairs).

The envelope requirement may be applied to features of size that are to be mated with a clearance fit and the location of the feature is of no interest. (When the location is involved, the maximum material requirement Ⓜ is to be applied.)

Figure 4.13 shows the application of the envelope requirement for a cylindrical feature. The size tolerance requires that the actual local sizes be within the limits of size. In addition, the envelope requirement is specified.

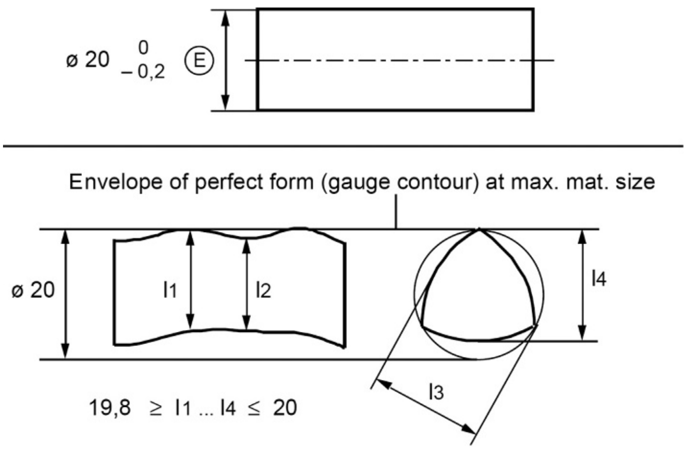


FIG. 4.13 Envelope requirement

When simple measuring devices, like calliper gauges or micrometer screws, are used to measure actual local sizes (two-point sizes), the cross sections may have lobed forms and go beyond the circles of maximum material size; see Fig. 4.14. Then the envelope requirement is violated. Even when size tolerances and line profile tolerances are indicated, this may happen; see Fig. 4.14. Even when size tolerances and surface profile tolerances are indicated, the envelope requirement may be violated; see Fig. 4.15.

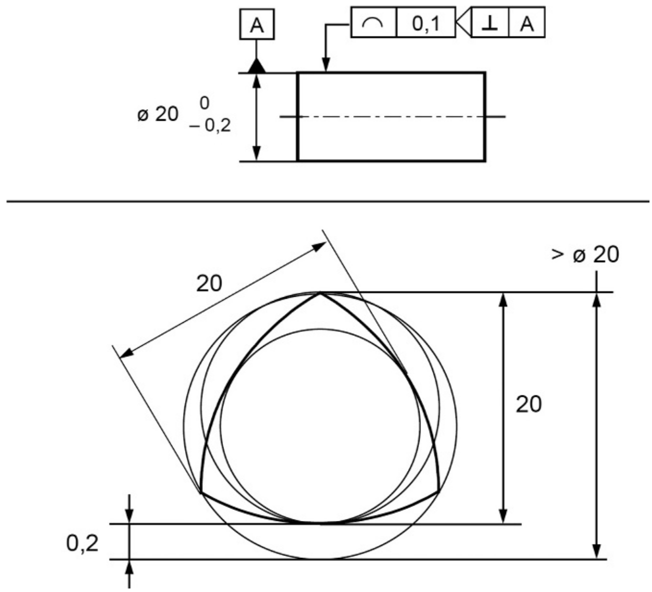


FIG. 4.14 Violation of the envelope requirement even though size and profile line tolerance are respected

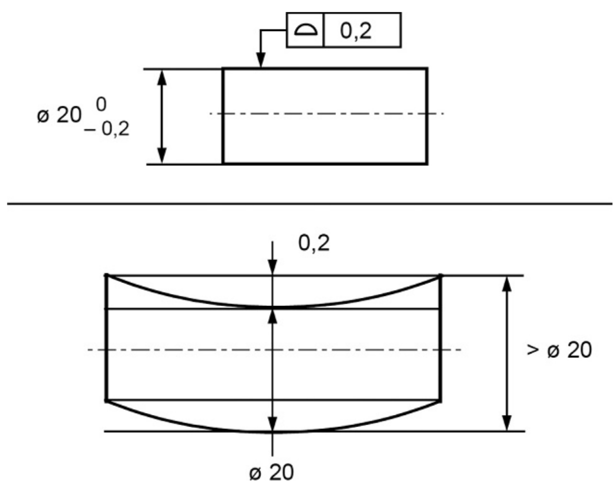


FIG. 4.15 Violation of the envelope requirement even though size and profile surface tolerance are respected

When a gap gauge is used, the gap gauge cannot detect lobed forms. The width of the gage may be shorter than the feature length. Gap gauges cannot verify the envelope requirement. Therefore in order to verify the envelope requirement, full form gauging or its simulation, such as by a coordinate measuring machine, must be executed (see 13.7.11).

Figure 4.16 shows inspection of the envelope requirement using a gap gauge. The gauge cannot detect whether the feature has a lobed form and violates the envelope requirement. Furthermore, in many cases the gauge is smaller than the feature is long. Thus the envelope requirement may be violated, and the gap gauge cannot verify the envelope requirement.

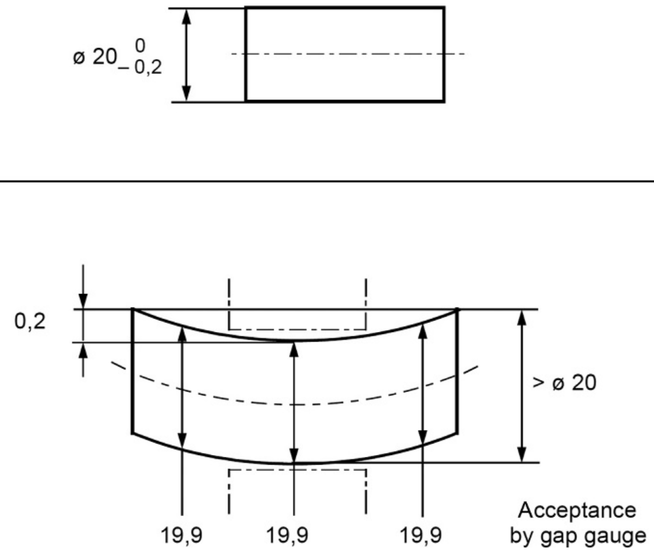


FIG. 4.16 Inspection by gap gauge

Figure 4.17 shows an example where the envelope requirement applies over a length of 10 but throughout the whole cylinder: that is, when a ring of 10 width is moved along the cylinder. The width of the gauge is 10.

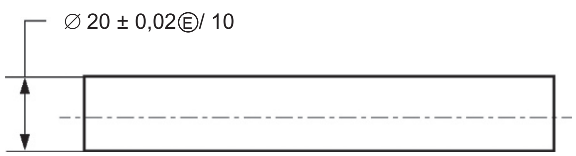


FIG. 4.17 Envelope requirement over a restricted length lying anywhere

The envelope requirement may also be indicated by a position tolerance with 0 Ⓜ applied. Then it can be related to a datum with or without Ⓜ. See Fig. 4.18. This is not possible with Ⓔ.

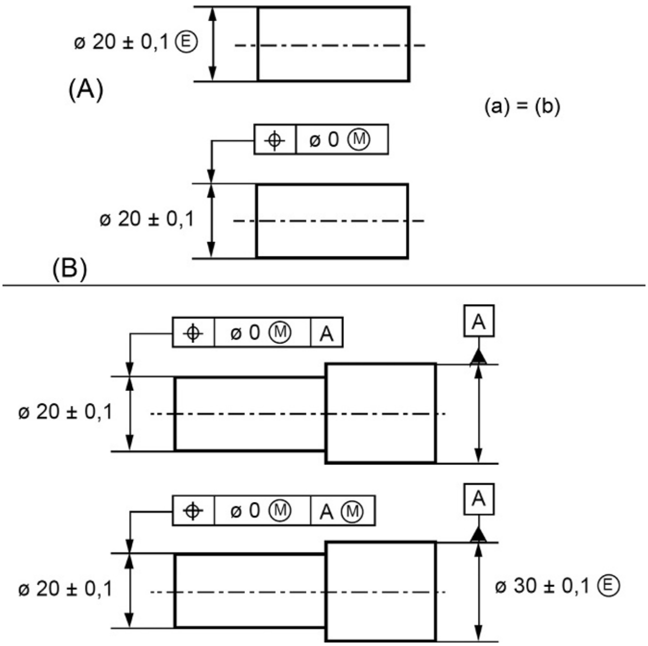


FIG. 4.18 Drawing indications for the envelope requirement

Ⓔ cannot be applied to a related feature; instead, 0 Ⓜ is to be used.

The envelope requirement should not be used for threads. Instead, gauging according to ISO 1502 applies. In addition, position tolerancing with Ⓜ for the axis may be applied; see Table 5.1 and ISO 4759-1:2000 par. 3.2.2.2.

In manufacturing, to respect the envelope requirement, the size tolerance should be reduced at the maximum material side corresponding to the profile deviation δ that is expected in the manufacturing process, i.e. by 2δ.

4.3.3 Tolerancing principle

There are two tolerancing principles. In the first one, the independency principle applies and the symbol Ⓔ is to be used, when the envelope requirement applies. This is the ISO default. The second tolerancing principle applies when the envelope requirement applies to all features of size without any individual indication. Then it is "ISO 14 405 Ⓔ", to be indicated in or near the drawing title box. However, this principle is not recommended, because special effort is required in manufacturing and inspection, although this is not necessary in most functional cases.

4.4 Maximum material requirement

4.4.1 Definitions

The maximum material requirement can only be applied to linear features of size, such as cylinders and plane pairs.

Actual local size (two-point size): Any individual distance at any cross section of a feature, i.e. any size measured between any two opposite points (two-point measurement) (ISO 286, ISO 2692), ISO 17 450-3. See Figs 4.8 and 4.9. Each feature of an individual workpiece theoretically has an infinite number of actual local sizes.

When sizes are measured with simple measuring instruments, see 4.3.2.

In the past, the definition of actual local size was not unambiguously standardized. The problems were the definitions of "opposite" and of the directions of the cross sections. The new standard ISO 17 450-3 provides a unique definition of two-point size (see Fig. 4.8). ISO 14 405-1 defines various other sizes. The two-point size is the default (i.e. if not otherwise specified).

Maximum material condition (MMC): The state of the considered feature in which the feature is everywhere at that limit of size where the material of the feature is at its maximum, e.g. minimum hole diameter and maximum shaft diameter (ISO 2692). MMC is of perfect form.

Mating size for an external feature: The dimension of the smallest perfect feature (e.g. cylinder or plane pair) that can be circumscribed about the feature so that it just contacts the surface at the highest points (Fig. 4.19).

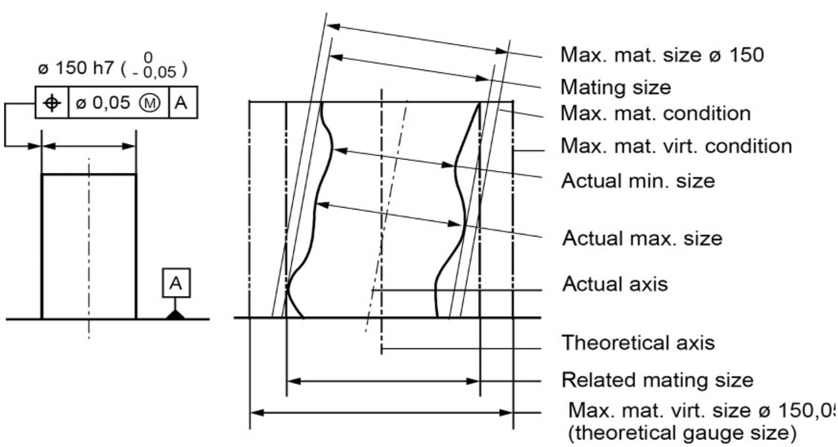


FIG. 4.19 Maximum material requirement, sizes and conditions

Mating size for an internal feature: The dimension of the largest perfect feature (e.g. cylinder or plane pair) that can be inscribed within the feature so that it just contacts the surface at the highest points (ISO 2692).

Maximum material size (MMS): The dimension defining the maximum material condition of a feature (Fig. 4.19) (ISO 2692), i.e. the limit of size where the material is at the maximum, such as maximum limit of size of a shaft or minimum limit of size of a hole.

Maximum material virtual size (MMVS): Size generated by the collective effect (concerning mating) of the maximum material size (MMS) and the geometrical tolerance followed by the symbol Ⓜ:

for shafts, MMVS = MMS + geometrical tolerance
for holes, MMVS = MMS − geometrical tolerance

The MMVS represents the design dimension of the functional gauge (ISO 2692).

Maximum material virtual condition (MMVC): This comprises the features limiting boundary of perfect (geometrical ideal) form and of maximum material virtual size (MMVS) (Fig. 4.19). When more than one feature or one or more datum features are applied to the geometrical tolerance, the MMVCs (of the toleranced features and the datum features) are in the theoretically exact locations and orientations relative to each other (ISO 2692). They represent the ideal gauge.

4.4.2 Description of the maximum material requirement

Maximum material requirement (MMR): Requirement, indicated on drawings by the symbol Ⓜ, placed after the geometrical tolerance of the toleranced feature or after the datum letter in the tolerance indicator, that specifies the following:

– when **applied to the toleranced feature**, the maximum material virtual condition (MMVC) of the toleranced feature shall not be violated (see Fig. 4.19). That is, the toleranced feature must fit into an ideal gauge (gauge contour), the size of the gauge is equal to the maximum material size of the toleranced feature, + with shafts, − with holes or slots, the geometrical tolerance indicated in the tolerance indicator and followed by Ⓜ.
– when **applied to the datum**, the related maximum material virtual condition (MMVC) of the datum feature shall not be violated. That is, the datum feature must fit into an ideal gauge, where the size of the gauge is:

- equal to the maximum material size of the datum feature, when there is no geometrical tolerance with Ⓜ at the datum feature; see Fig. 4.20. In this case, the datum triangle is to be indicated as an extension of the dimension; see Fig. 4.20.

- equal to the maximum material size of the datum feature, + with shafts, − with holes or slots, the geometrical tolerance indicated in the tolerance indicator and followed by Ⓜ, at the datum feature; see Fig. 4.21. In this case, the datum triangle is to be indicated at the tolerance indicator of the datum feature; see Fig. 4.21.

However, only those geometrical tolerances with Ⓜ come into consideration that are unrelated or related to datums of the considered datum indicator. Geometrical tolerances of the datum feature followed by Ⓜ that are related to other datums do not come into consideration; see Fig. 4.22.

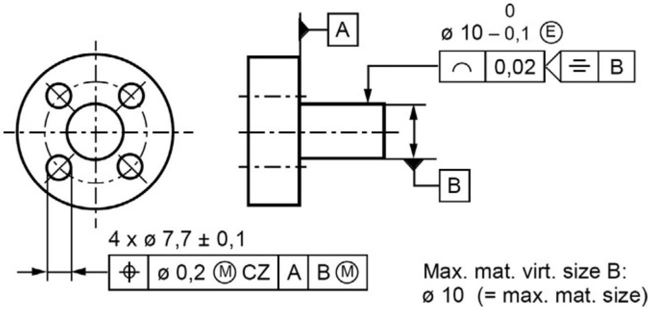


FIG. 4.20 Maximum material virtual size of datum B; form tolerance (straightness) to be disregarded for B Ⓜ

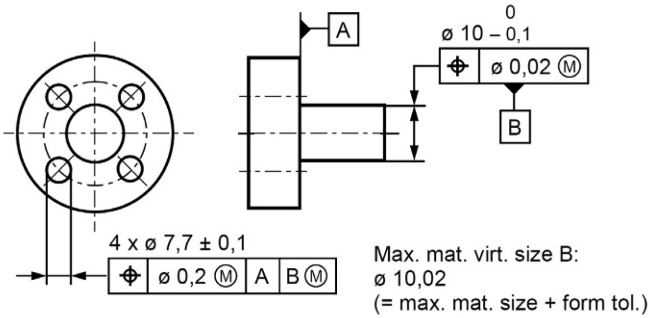


FIG. 4.21 Maximum material virtual size of datum B; form tolerance (straightness) to be regarded for B Ⓜ

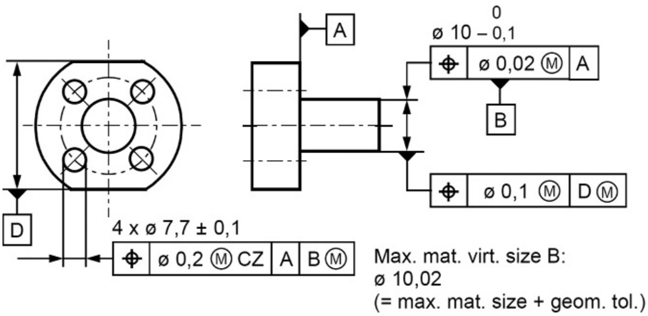


FIG. 4.22 Maximum material virtual size of datum B; perpendicularity tolerance to be regarded, symmetry tolerance 0,1 to be disregarded for B Ⓜ

The gauge contours of the toleranced feature(s) and of the datum feature(s) are in the theoretically exact position relative to each other.

The maximum material requirement can be explained as a (functional) gauging requirement. The maximum material virtual conditions at the toleranced feature and at the datum(s) describe together the (functional) gauge contour. The workpiece must fit into this gauge. The gauge also represents the most unfavourable counterpart. When the workpiece fits into the gauge, it also fits into all counterparts.

With Ⓜ, the meaning of the tolerance indicator changes. Without Ⓜ, the derived feature(s) are toleranced and the relation(s) to the datum(s) are fixed. With Ⓜ, the toleranced feature(s) and the datum feature(s) must fit into a gauge.

4.4.3 Application of the maximum material requirement

4.4.3.1 General

The maximum material requirement can be applied only to those features with an axis or a median plane (cylindrical features or plane pair features). It cannot be applied to a planar surface.

The maximum material requirement can be applied when there is a functional relationship between size and form, orientation and/or location, i.e. when the geometrical deviation may be larger if the size deviation is smaller. This applies normally to clearance fits. For transition fits and interference fits and for kinematic linkages (e.g. distances of axes of gears) the maximum material requirement is normally not appropriate. This is because enlarging the geometrical tolerance (when the size tolerance is not fully used) is detrimental to the function of the part.

4.4.3.2 Maximum material requirement for the toleranced feature

The maximum material requirement for the toleranced feature allows an increase in the geometrical tolerance when the feature deviates from its

maximum material condition (in the direction of the least material condition), provided that the maximum material virtual condition (gauge contour) is not violated (Fig. 4.23 ff). That is, the maximum material requirement specifies that the indicated geometrical tolerance applies when the feature is in its maximum material condition (largest shaft, smallest hole). When the feature deviates from the maximum material condition (thinner shaft, larger hole), the geometrical deviation may be larger without endangering the mating capability.

The maximum material virtual condition represents the theoretically functional gauge at the toleranced feature. The maximum material virtual size represents the theoretical size of the functional gauge.

The maximum material requirement has two effects. It assures the function of a clearance fit, and it is an enlargement of the tolerance. See Fig. 4.23, which shows the application of Ⓜ. The workpiece must fit into a gauge ø 40/ø 20. When the diameters of the workpiece are at the least material sizes (39,7/19,9), the actual position tolerance increases from 0,1 to 0,4.

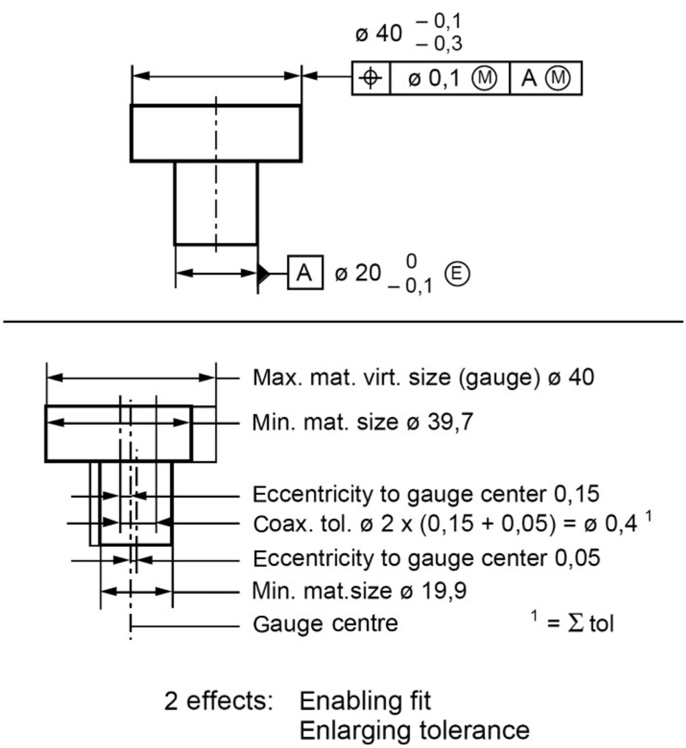


FIG. 4.23 Maximum material requirement, maximum possible coaxiality deviation

Figure 4.24 shows another example of using Ⓜ. The gauge of ø 9,9 − ø 0,28 = ø 9,62 at the theoretically exact location must fit into the hole.

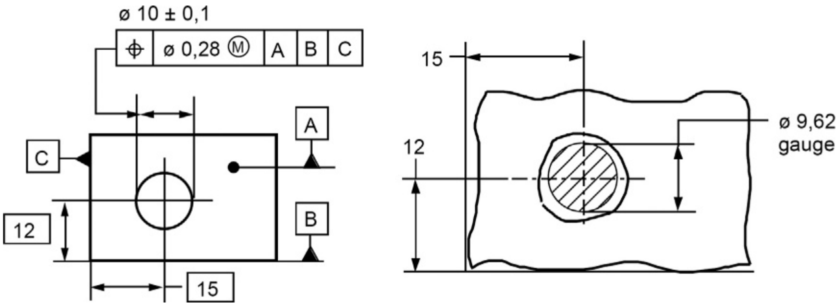


FIG. 4.24 Maximum material requirement at a fixed location

4.4.3.3 Maximum material requirement 0 Ⓜ

When the functional tolerance is **not** distributed on size and position, but is provided for both (size and position) for random distribution, this is to be indicated on the drawing by 0 Ⓜ. See Fig. 4.25; the gauge and the area ø 10,4 are at the fixed location. The hole must be in the area. That determines the diameter and the location of the hole.

Figure 4.25 shows the differences between ø 0 Ⓜ and ø 0,3 Ⓜ. In both cases the MMVS (gauge size) is the same (ø 9,6). However, with ø 0 Ⓜ the maximum size of the hole is larger. This may be detrimental for holes for fasteners, because of the larger pressure under the nut or screw head.

Sometimes on drawings the indication on the right of Fig. 4.26 appears. This has the same meaning as using the symbol Ⓔ according to ISO 14 405-1 and shown on the left of Fig. 4.26. In both cases the envelope requirement applies, i.e. the boundary of geometrically ideal form and maximum material size must not be violated. The symbol Ⓔ has been standardized because the drawing indication is simpler than with 0 Ⓜ.

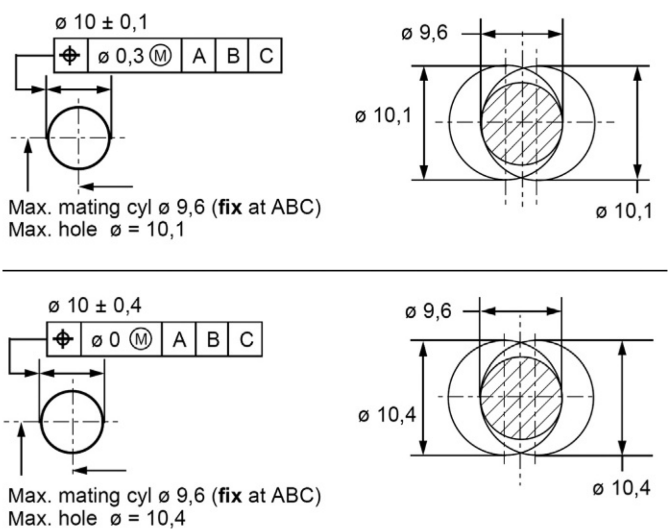


FIG. 4.25 Maximum material requirement 0 Ⓜ

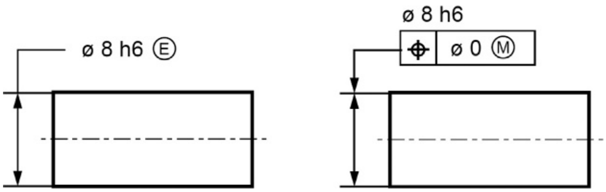


FIG. 4.26 Envelope requirement

4.4.3.4 Comparison of 0.1 Ⓜ and 0 Ⓜ

Figure 4.27 shows a plug and socket that fit. The same maximum material virtual conditions (gauge contours) apply to both, in cases a) as well as in cases b). The difference between cases a) and b) is the size tolerance and therefore the possible clearance between bolt and hole. In case a) the clearance may be 0 (hole Ø 4 and bolt Ø 4). In case b) the clearance is at least 0,2 but may be utilized by the straightness deviations of the axes (bent hole and bent bolt) (Fig. 4.27). In case b), the manufacturer obtains a recommendation for the distribution of the in-total provided tolerance (0,2) on the size (0,1) and on the distance (0,1) (see also 4.4.3.6). However, workpieces with diameters <Ø 4.1 left and >Ø 3.9 right, which fit into the gauges and also into the counterparts, are not allowed. In the case of 0 Ⓜ, these workpieces are allowed. Therefore 0 Ⓜ reflects the functional requirement and should be used by the designer.

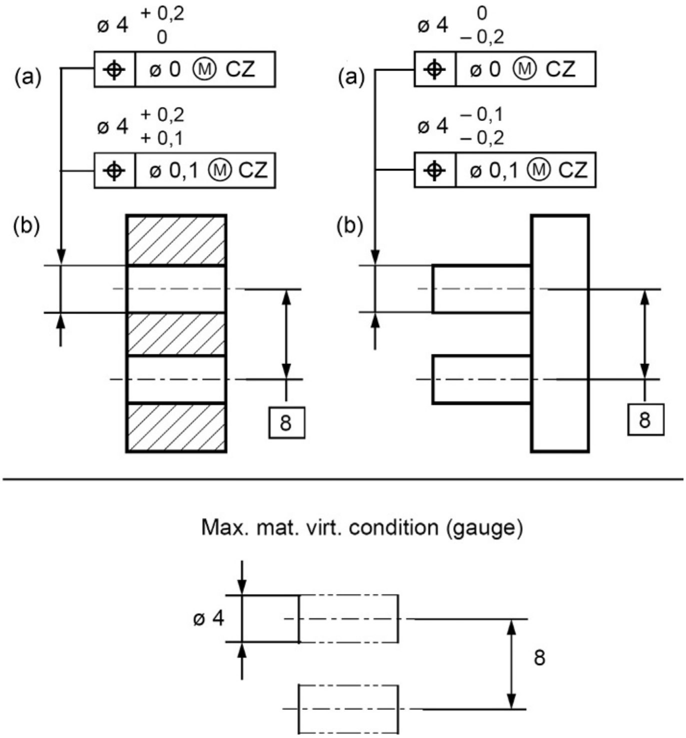


FIG. 4.27 Comparison of 0,1 Ⓜ and 0 Ⓜ with the same maximum material virtual conditions (gauge contours)

4.4.3.5 Maximum material requirement for the datum

The maximum material requirement for the datum permits floating of the datum axis or datum median plane relative to the toleranced features pattern when the datum feature deviates from its maximum material condition (in the direction of the least material condition). A prerequisite is that the datum feature does not violate its maximum material virtual condition, which is geometrically ideally positioned in relation to the geometrical ideal position of the toleranced features. Within this boundary, the datum feature may take the position where the requirements at the toleranced features are fulfilled.

The requirement is that the datum feature must fit into the gauge contour which is in the theoretically exact location. See Figs 5.9 and 5.10.

The deviation of the datum feature from its maximum material virtual size does not increase the tolerance of the toleranced features relative to each other. It only permits a displacement of the pattern of tolerance zones (maximum material virtual conditions of the toleranced features) relative to the actual axis or actual median surface of the datum feature (Fig. 4.24). However, when only

two features are related, as in Fig. 4.23, the floating (displacement) has the effect of enlarging the tolerance of the toleranced feature.

The functional gauge embodies the maximum material virtual condition of the datum. The maximum material virtual size represents the theoretical size of the gauge. In other words, the maximum material requirement describes a gauge contour (at the toleranced feature(s) and at the datum feature(s)), located and oriented theoretically exactly in relation to each other, in which the actual features must be contained (the workpiece must fit into the gauge). See also 4.4.2.

It should be noted that the indication of the maximum material requirement Ⓜ at the datum has a meaning different from ISO 5459. With Ⓜ at the datum, the datum cylinder may float within the maximum material virtual condition (gauge). Without Ⓜ, the datum cylinder is fixed. Figure 4.28 shows an example with fixed datum. Compare with Fig. 4.21, where the datum may float within the gauge contour.

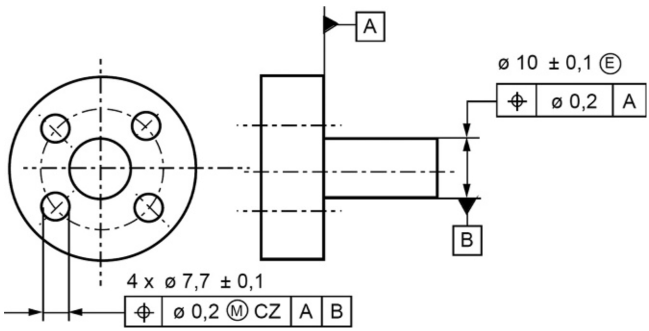


FIG. 4.28 Datum fix (without Ⓜ)

4.4.3.6 Reciprocity requirement together with the maximum material requirement

Figure 4.29 shows three fits (a), (b) and (c). In all cases the same maximum material virtual conditions (gauge contours) apply for part and counterpart. The differences between cases a) and b) are as described in 4.4.3.4. In case b) the coaxiality tolerance will be enlarged by the size tolerance not utilized, but not vice versa. The size tolerance cannot be enlarged by the non-utilized coaxiality tolerance, although the function (clearance fit) would allow this. To allow this on the drawing, the reciprocity requirement with the symbol Ⓡ after the symbol Ⓜ after the geometrical tolerance can be applied.

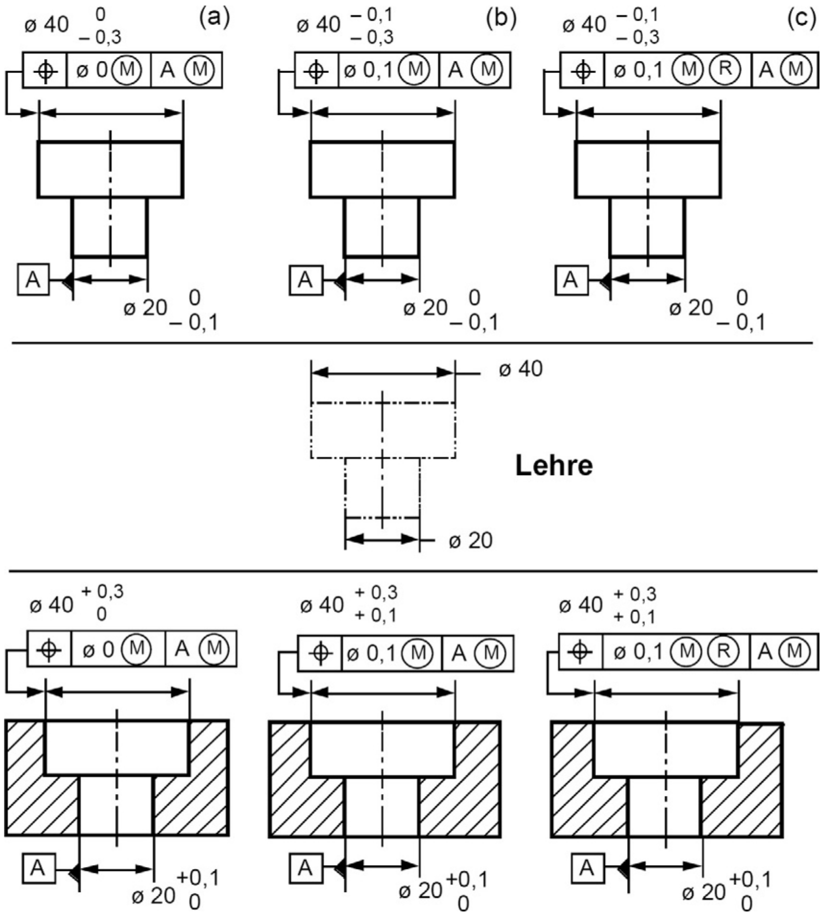


FIG. 4.29 Maximum material requirement (Ⓜ after the geometrical tolerance and after the datum letter) and reciprocity requirement Ⓡ after Ⓜ (after the geometrical tolerance)

Reciprocity requirement associated with the maximum material requirement RR: This requirement, indicated on drawings by the symbol Ⓡ placed after the symbol Ⓜ after the geometrical tolerance in the tolerance indicator, specifies that the maximum material virtual condition (MMVC) of the toleranced feature shall not be violated. Deviations of size, form, orientation and location may take full advantage of the total tolerance (sum of tolerances).

The reciprocity requirement associated with the maximum material requirement has the same effect as 0 Ⓜ, i.e. the total tolerance may be utilized for deviations of size, form, orientation or location in an arbitrary way. However, in contrast to 0 Ⓜ, the drawing indication with the reciprocity requirement Ⓡ gives a recommendation to the manufacturer for the distribution of the total tolerance on size and geometrical characteristics. Thus the reciprocity requirement provides communication between production planning and the workshop. The design requirement (function related tolerance) is “0 Ⓜ”.

ⓂⓇ is a means for communication between the manufacturing planner and the manufacturer. The designer should use the function related tolerance 0Ⓜ (because he does not know who the manufacturer is and how to split the total tolerance).

However, the 0Ⓜdrawing may also be used for manufacturing. In this case, the manufacturer must reduce the size tolerance at the maximum material side corresponding to the maximum geometrical deviation to be expected on his machine tool.

4.4.4 Education

The fact that the application of the maximum material requirement appears rather complicated sometimes leads to the false opinion that such application is not practicable and therefore to be ignored.

However, in many cases the precise functional requirements can only be indicated with the aid of the maximum material requirement. Only then do the largest possible tolerances appear. Therefore this is often unavoidable when economical production is to be achieved.

A prerequisite for application of the maximum material requirement is appropriate education and appropriate planning of the manufacturing and inspection. For education, in most cases it should be sufficient to explain the maximum material requirement by the following gauging rule:

Where Ⓜ occurs, gauging is required or to be simulated. At the toleranced feature, the gauge size is to be calculated in the following way from the maximum material size and the geometrical tolerance, which is followed by the symbol Ⓜ:

for shafts: maximum material size + geometrical tolerance
for holes: maximum material size – geometrical tolerance

At the datum feature the gauge size is to be calculated in the same way when at the datum feature a geometrical tolerance followed by the symbol Ⓜ is indicated (and the datum triangle is connected directly to this tolerance indicator (Fig. 4.21)).

When there is no geometrical tolerance followed by the symbol Ⓜ indicated at the datum, the gauge size is equal to the maximum material size (Fig. 4.20).

Designers should know that at the datum (of the concerned geometrical tolerance) only geometrical tolerances (tolerance indicators) should be connected with the datum triangle that have no relationship to other features (see Fig. 4.21) or that are related to datums occurring in the datum system of the concerned tolerance (Fig. 4.22).

According to ANSI Y 14.5 M – 1982 and to ASME Y14.5M – 1995 there are different rules. Connection of the datum triangle to the considered tolerance indicator is not mandatory. The standard requires an analysis of tolerance controls applied to a datum feature in determining the size of the gauge. Further, the standard specifies rules for cases when at the datum feature geometrical tolerances are indicated that are not followed by the symbol Ⓜ.

4.5 Tolerance chains (accumulation of tolerances)

When workpieces are assembled, size deviations accumulate. In order to assess the resulting clearance or interference, tolerance line-up calculations are performed. Arithmetical tolerance calculations are based on extreme cases (worst cases) when all sizes are at their favourable or unfavourable limit of size. Statistical tolerance calculations take into account the form of distribution of the actual sizes and give the clearance or interference that will not be exceeded with a certain statistical probability (see 4.7).

The procedure for an arithmetical tolerance line-up calculation is as follows:

1. Define the dimension scheme, showing all dimensions and their tolerances that form the chain, i.e. all dimensions that contribute to the clearance or interference.
2. Dimensions whose upper limits lead to an increase in the closing dimension are drawn in the positive direction, and others in the negative direction.

The arithmetical sum of the maximum limits of size of the positive chain links and subtracting the sum of the minimum limits of size of the negative chain links gives the **maximum value of the closing dimension** (maximum clearance, minimum interference).

The arithmetical sum of the minimum limits of size of the positive chain links and subtracting the sum of the maximum limits of size of the negative chain links gives the **minimum value of the closing dimension** (minimum clearance, maximum interference).

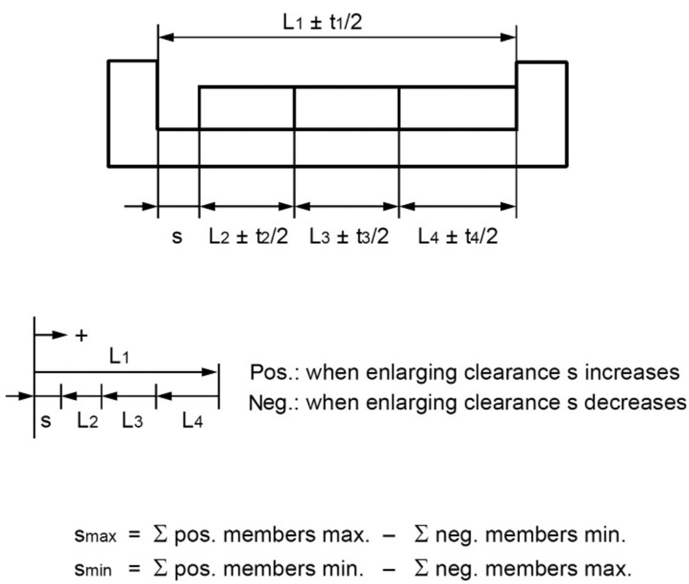


FIG. 4.30 Arithmetical tolerance line-up calculation

The tolerance line-up calculation according to Fig. 4.30 considers size tolerances only. A prerequisite is that the workpiece surfaces respect the geometrical ideal boundaries of maximum material size.

Figure 4.31 shows the effect of geometrical deviations at the gear. The resulting width must not exceed the maximum limit of size used in the calculation. Figure 4.32 shows the appropriate tolerancings. Both tolerancings lead to the same gauge.

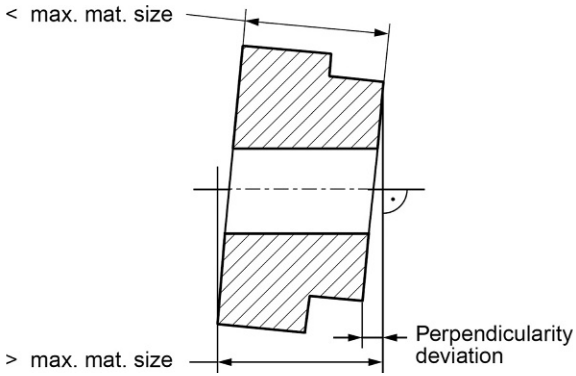


FIG. 4.31 Effective width of the gear in the assembly

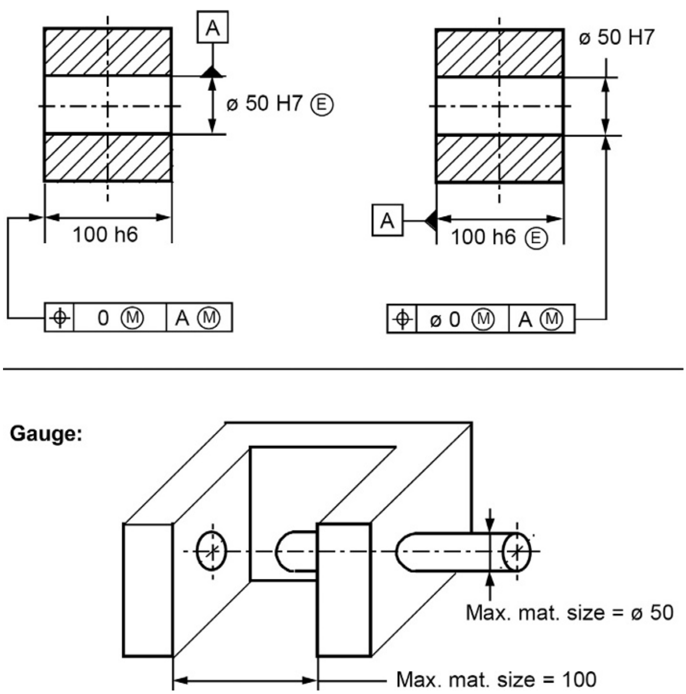


FIG. 4.32 Appropriate tolerancings for the gear

A similar situation applies to the casing. Figure 4.33 shows the effect of geometrical deviations at the casing. The resulting width of the assembly space must not become smaller than the minimum limit of size used in the calculation. Figure 4.34 shows the appropriate tolerancings. Both tolerances lead to the same gauge.

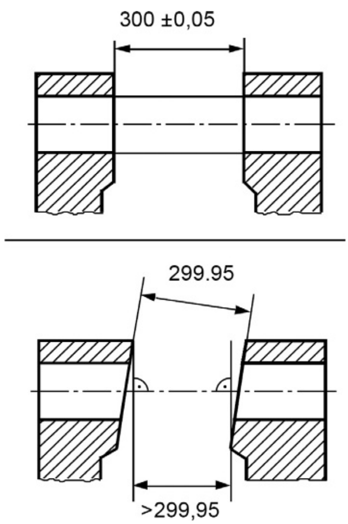


FIG. 4.33 Effective width of the assembly space of the casing

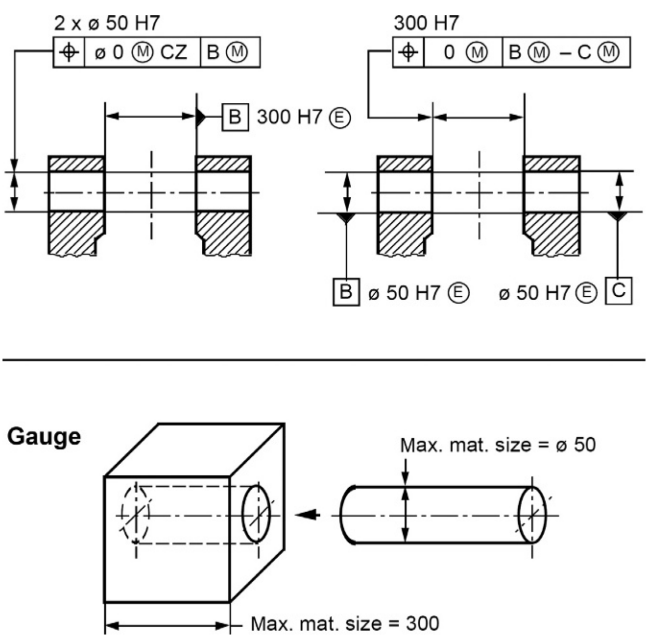


FIG. 4.34 Appropriate tolerancings for the assembly space of the casing

Figure 4.35 shows an assembly where adjacent parts have different diameters. In order to avoid jamming, as shown in Fig. 4.36, the tolerance zone for the part with the smaller diameter must be enlarged; see Fig. 4.37.

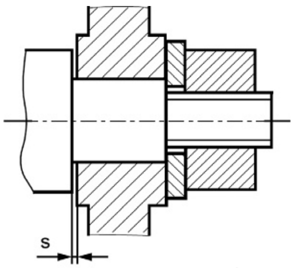


FIG. 4.35 Assembly of parts with different diameters

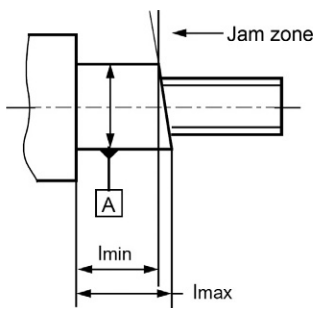


FIG. 4.36 Possible jam in the assembly

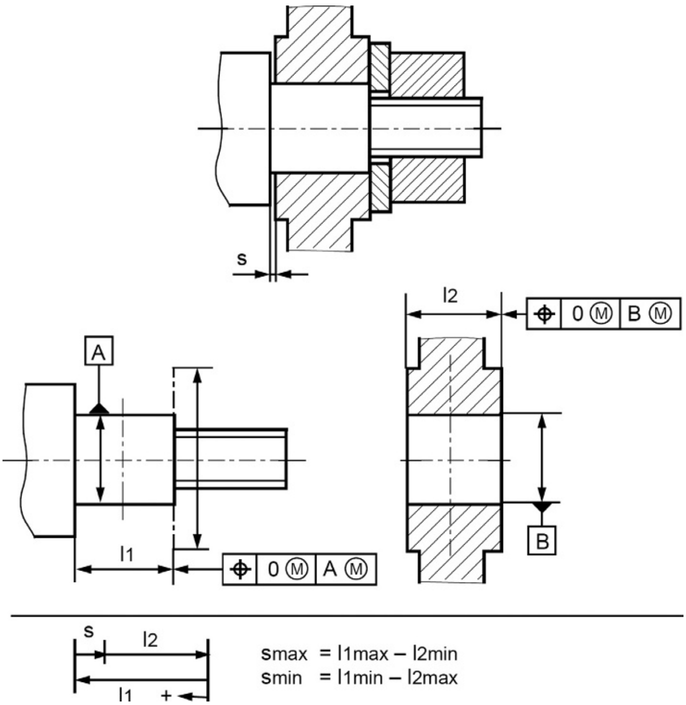


FIG. 4.37 Appropriate tolerancing with enlarged zone

4.6 Least material requirement

4.6.1 Definitions

The least material requirement can only be applied to linear features of size such as cylinders and plane pairs.

Least material condition (LMC): The state of the considered feature in which the feature is everywhere at that limit of size where the material of the feature is at its minimum, e.g. maximum limit of size of a hole or slot (maximum hole diameter or slot width) and minimum limit of size of a shaft or tab (minimum shaft diameter or minimum tab width) (ISO 2692).

The actual axis of the feature need not to be straight.

Least material size (LMS): The dimension defining the least material condition of a feature of size (ISO 2692), i.e. the limit of size where the material is at the minimum (e.g. maximum limit of size of a hole or slot and minimum limit of size of a shaft or tab).

Least material virtual size (LMVS): Size generated by the collective effect (with regard to what can be cut out of the material) of the least material size and the geometrical tolerance followed by the symbol Ⓛ, i.e.:

for shafts: LMVS = LMS − geometrical tolerance
for holes: LMVS = LMS + geometrical tolerance

Least material virtual condition (LMVC): The features limiting boundary of perfect (geometrical ideal) form and of least material virtual size. When more than one feature, or one or more datum features with Ⓛ, are applied to the geometrical tolerance, the LMVCs of the toleranced features and the LMVCs of the datum features are in theoretically exact locations and orientations relative to each other (ISO 2692). They represent the ideal cut-out contour.

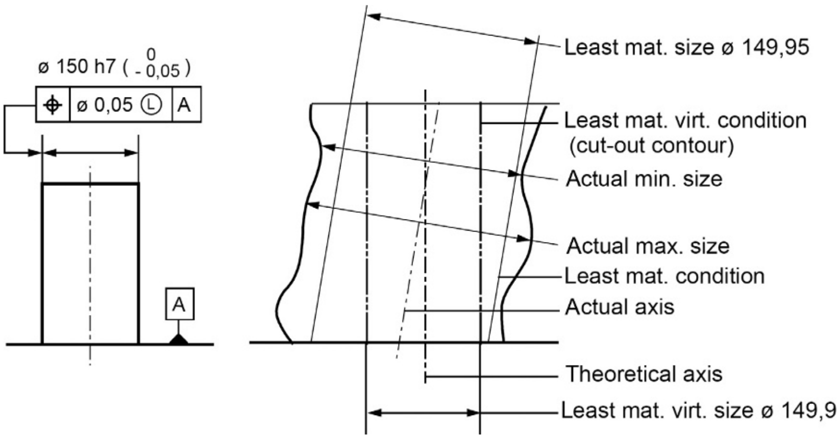


FIG. 4.38 Least material requirement, sizes and conditions

4.6.2 Description of least material requirement

Least material requirement (LMR): Requirement, indicated on drawings by the symbol Ⓛ placed after the geometrical tolerance of the toleranced feature or after the datum letter in the tolerance indicator, that specifies the following:

– when applied to the toleranced feature, the least material virtual condition (LMVC) of the toleranced feature shall not be violated (see Fig. 4.38). That is, the toleranced feature must be fully contained within the cut-out contour (LMVC): the size of the cut-out contour is equal to the least material size of the toleranced feature, − with shafts or tabs, + with holes or slots, the geometrical tolerance, indicated in the tolerance indicator and followed by Ⓛ.
– when applied to the datum, the related least material virtual condition (LMVC) of the datum feature shall not be violated. That is, the datum feature material must fully contain the cut-out contour. The size of the cut-contour is:

- equal to the least material size of the datum feature when there is no geometrical tolerance with Ⓛ at the datum feature; see Fig. 4.39. In this case, the datum triangle is to be indicated as an extension of the dimension; see Fig. 4.39.

- equal to the least material size of the datum feature, − with shafts or tabs, + with holes or slots, the geometrical tolerance indicated in the tolerance indicator and followed by Ⓛ, at the datum feature; see Fig. 4.40. In this case, the datum triangle is to be indicated at the tolerance indicator of the datum feature; see Fig. 4.40.

– However, only those geometrical tolerances with Ⓛ that are unrelated or related to datums of the considered datum indicator come into consideration. Geometrical tolerances of the datum feature followed by Ⓛ that are related to other datums do not come into consideration; see Fig. 4.41.

The least material requirement (LMR) Ⓛ has the effect that the least material virtual condition is entirely contained in the material of the feature and, for example, can be cut out. The mutual dependence of size and form and location and orientation is thereby taken into consideration.

With Ⓛ, the meaning of the tolerance indicator changes. Without Ⓛ, the derived feature(s) are toleranced and the relation(s) to the datum(s) are fixed. With Ⓛ, the toleranced feature(s) and the datum feature(s) must have the cut-out contour fully within the material. A typical application is a casting when the final machined part shall be achievable.

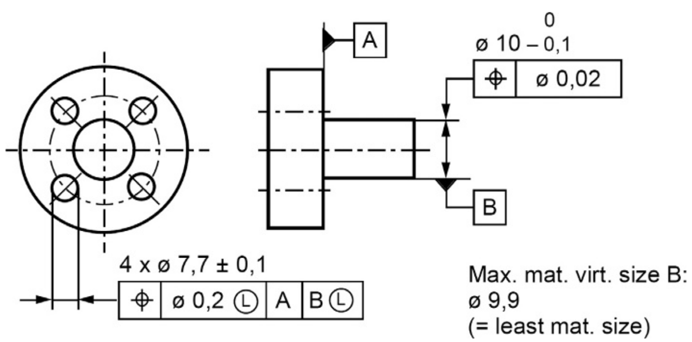


FIG. 4.39 Least material requirement, least material virtual condition (cut-out contour) at datum B, form tolerance to be disregarded for B

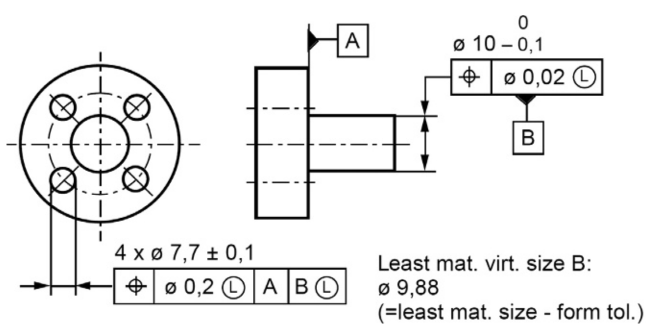


FIG. 4.40 Least material requirement, least material virtual condition (cut-out contour) at datum B, form tolerance to be regarded for B

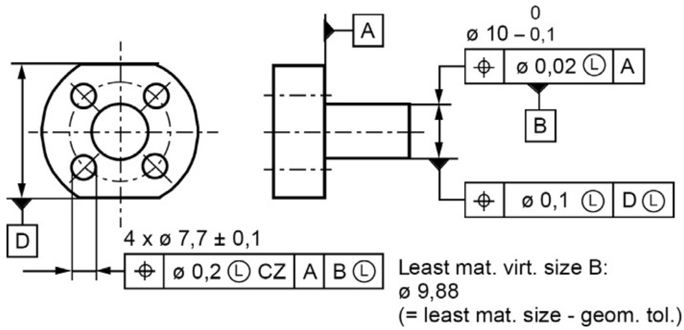


FIG. 4.41 Least material requirement, least material virtual condition (cut-out contour) at datum B, B with Ⓜ, datum D to be disregarded, because not involved in tolerance of holes

4.6.3 Application of least material requirement

4.6.3.1 General

The least material requirement can be applied only to features of size such as cylindrical features or plane pairs. It cannot be applied to a planar surface.

The least material requirement may be applied where a minimum material thickness must be respected and this minimum thickness depends on the mutual effect of deviations of size and form, location or orientation (cut-out contour).

4.6.3.2 Least material requirement for the toleranced feature

The least material requirement for the toleranced feature allows an increase in the geometrical tolerance when the feature deviates from its least material condition (in the direction of the maximum material condition), provided that the least material virtual condition (cut-out contour) is entirely within the material; see Fig. 4.42. That is, the least material requirement specifies that the indicated geometrical tolerance applies when the feature is in its least material condition (smallest shaft, largest hole) and of geometrical ideal form. When the feature deviates from the least material condition (larger shaft, smaller hole) the geometrical deviation may be larger without violating the least material virtual condition (LMVC) (cut-out contour).

Figure 4.42 shows an example in which a minimum ridge thickness is to be contained in the material, e.g. of a casting that is to be machined, and Fig. 4.43 shows a permissible workpiece. The least material virtual condition has the size of 50 (least material size minus position tolerance), is perpendicular to the datum A and 60 apart from the datum B. The more the ridge thickness deviates from the least material size, the more the actual median surface may deviate from the theoretical exact location. In the example, the smallest actual local size is 53. In this case, the position deviation of the actual median surface can be 1,5, which corresponds to a position tolerance of 3.

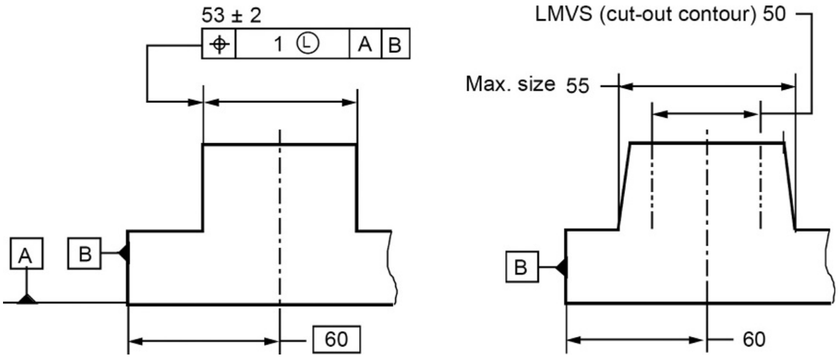


FIG. 4.42 Least material requirement

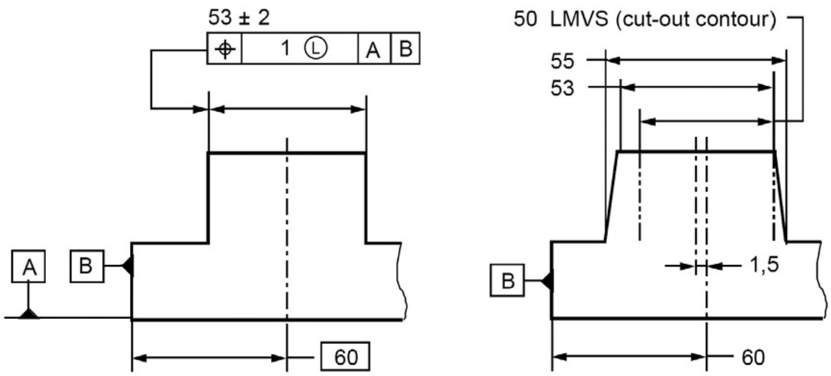


FIG. 4.43 Least material requirement, permissible workpiece

Figure 4.44 shows an example where the geometrical ideal form of least material size shall be contained in the material (must be possible to cut out). Here the least material virtual condition has least material size.

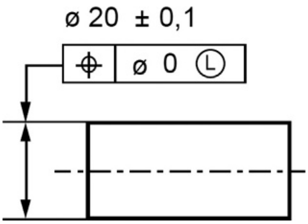


FIG. 4.44 Straightness tolerance of the axis with least material requirement

Without this indication (without position tolerance with Ⓛ), the feature may be bent or may be of a lobed form in the cross sections. The maximum (geometrical ideal) cylinder contained in the material (possible to cut out) may have a diameter smaller than the least material size.

4.6.3.3 Least material requirement for the datum

The least material requirement for the datum permits floating of the datum axis or datum median plane relative to the toleranced feature(s) when the datum feature deviates from its least material condition (in the direction of the maximum material condition). A prerequisite is that the surface of the datum feature does not violate the least material virtual condition (which is geometrically ideally positioned in relation to the geometrical ideal position of the toleranced features). Around this boundary, the datum feature may take, if possible, the position where the requirements of the toleranced features are fulfilled.

It should be noted that the indication of the least material requirement Ⓛ at the datum has a meaning different from ISO 5459. Furthermore, with A Ⓛ, the datum cylinder A of least material (virtual) size must be contained in the material (and is not allowed to lie outside of the material, as applicable to the maximum material virtual condition of A Ⓜ or A without a modifier).

Figure 4.45 shows on the left an example where a geometrical ideal pipe with a minimum wall thickness of 5 shall be contained in the material (must be possible to be cut out). On the right are shown the coaxial least material virtual cylinders (ø 40 and ø 50) that must not be violated by the surfaces. In the centre, both surfaces (features) have least material size and therefore are coaxial. On the right are shown the extreme permissible coaxiality deviations with maximum material sizes.

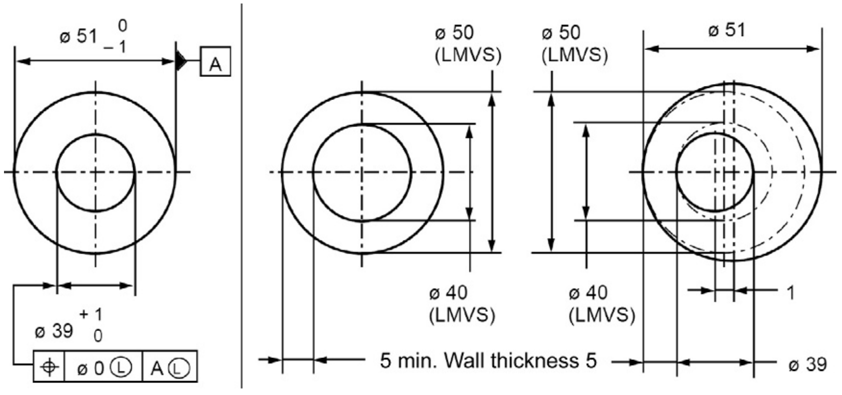


FIG. 4.45 Coaxiality tolerance with least material requirement in order to secure the minimum wall thickness

4.6.4 Reciprocity requirement together with the least material requirement

Figure 4.46 shows three rings. In all cases, the same least material virtual conditions (boundaries to be entirely contained within the material) apply. In the case on the left, the whole tolerance is indicated at the size. The tolerance may be utilized by size deviations and by coaxiality deviations in an arbitrary way. In the case in the centre, the tolerance is distributed on size and coaxiality. The coaxiality tolerance will be enlarged by the size tolerance not utilized, but not vice versa. The size tolerance cannot be enlarged by the non-utilized coaxiality tolerance. In the case on the right, the size tolerance can also be enlarged by the non-utilized coaxiality tolerance.

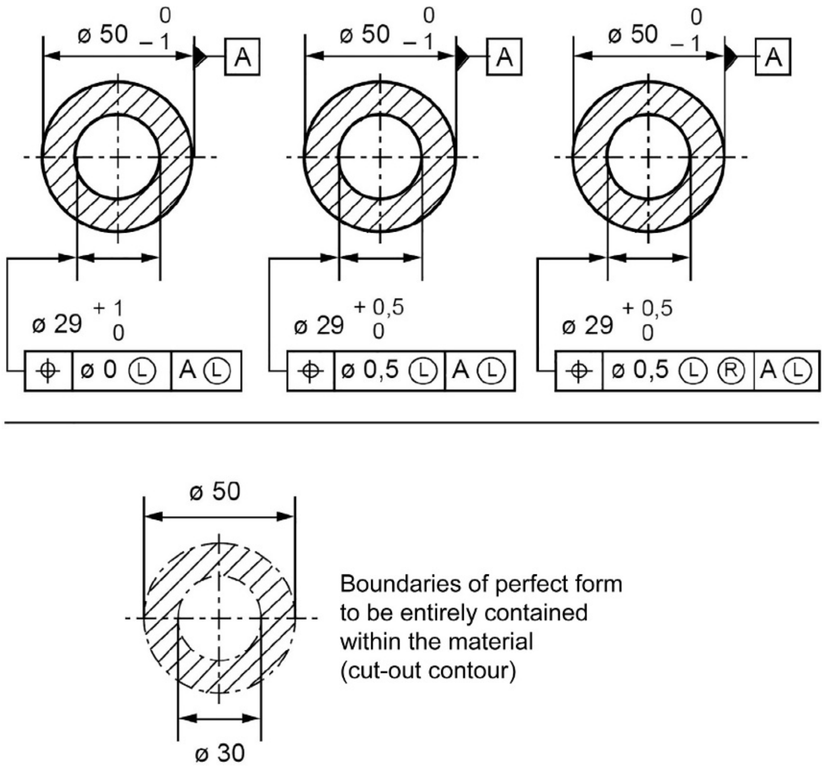


FIG. 4.46 Least material requirement (Ⓛ after geometrical tolerance and after datum letter) and reciprocity requirement (Ⓡ after Ⓛ after geometrical tolerance)

Reciprocity requirement (RR) associated with the least material requirement: This requirement, indicated on drawings by the symbol Ⓡ placed after the symbol Ⓛ after the geometrical tolerance in the tolerance indicator,

specifies that the least material virtual condition (LMVC) of the toleranced feature shall not be violated. Deviations of size, form, orientation and location may take full advantage of the total tolerance (sum of tolerances).

The reciprocity requirement has the same effect as 0 Ⓛ, i.e. the total tolerance may be utilized for deviations of size, form, orientation or location in an arbitrary way. However, in contrast to 0 Ⓛ, the drawing indication with the reciprocity requirement Ⓡ gives a recommendation to the manufacturer for the distribution of the total tolerance on size and geometrical characteristics. Thus the reciprocity requirement provides a communication between production planning and the workshop (manufacturing-related tolerance). The design requirement (functional-related tolerance) is 0 Ⓛ. There may be several manufacturing-related tolerances (with Ⓡ) derived from the same functional-related tolerance, according to the needs of different workshops.

However, the 0 Ⓛ drawing may also be used for manufacturing. In this case, the manufacturer must reduce the size tolerance at the minimum material side by the maximum geometrical deviation to be expected on his machine tool: in the case of a cylinder, two times the cylindrical deviation.

4.7 Statistical tolerancing

With the arithmetical tolerance line-up calculation (worst-case tolerancing), the tolerances of the links of the dimension chain (e.g. chain of single parts of an assembly) are defined in such a way that the assembly still functions when all links of the chain have actual sizes equal to their limits of size; see Fig. 4.47 and 4.5.

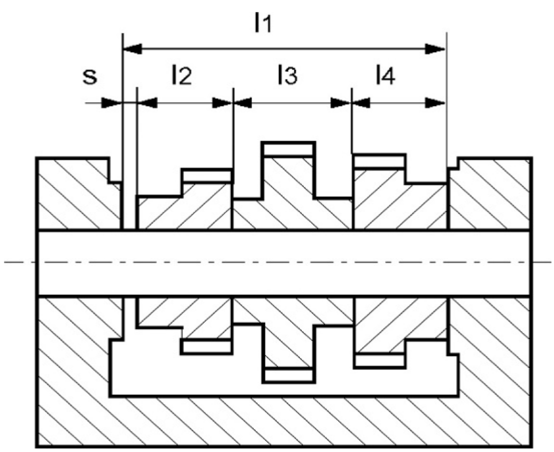


FIG. 4.47 Assembly with required clearance s (functional closing tolerance T)

However, in practice the actual sizes are subject to variations, and are therefore statistically distributed (e.g. according to a normal distribution) (Figs 4.48 and 4.49). With this supposition, it is most unlikely that an assembly contains only parts with actual sizes equal to the limits of size (Fig. 4.50).

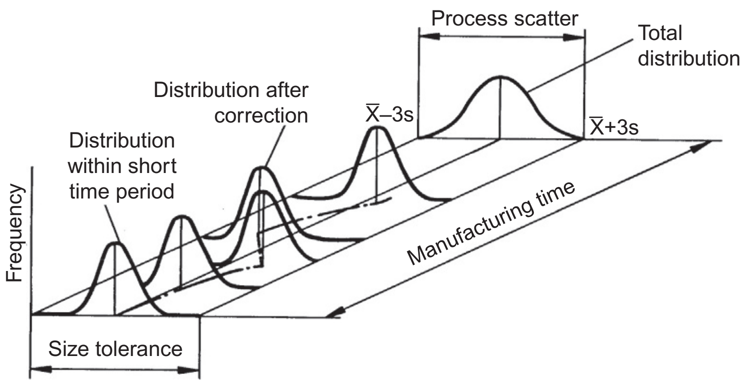


FIG. 4.48 Distribution of actual sizes during a manufacturing process

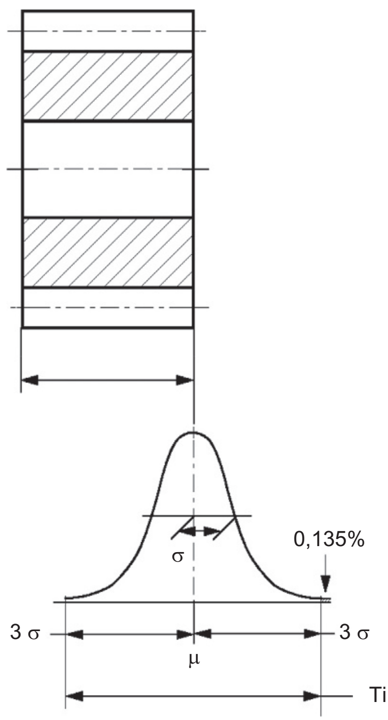


FIG. 4.49 Distribution of actual sizes (size deviations) of a part

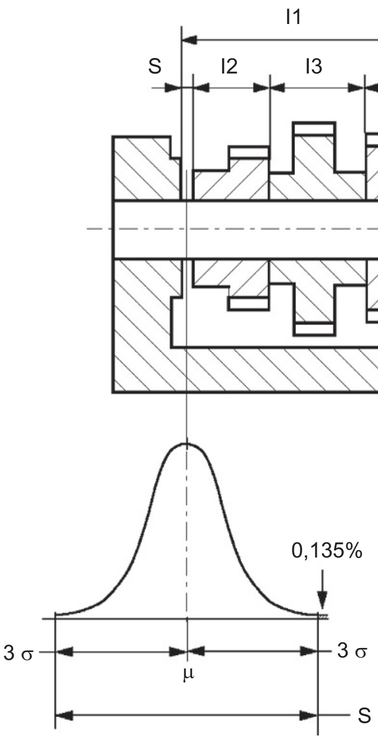


FIG. 4.50 Distribution of the clearance

In contrast to the arithmetical tolerance line-up calculation, in the statistical tolerance line-up calculation it is assumed that this extreme case will not occur. The single tolerances are chosen larger than in arithmetical tolerancing.

The variability Δ_s of the clearance or interference, i.e. functional closing tolerance T of the assembly, derives from n single tolerances with arithmetical tolerancing according to

$$\Delta_s = T = T_a = T_1 + T_2 + \cdots + T_n$$

with statistical tolerancing according to

$$\Delta_s = T = T_s = \sqrt{\left(T_{s1}{}^2 + T_{s2}{}^2 + \cdots + T_{sn}{}^2\right)}$$

With the assembly of Fig. 4.47 (with a chain of four parts), the single tolerance is

$T_{ai} = \Delta_s/4 = 0{,}25\ \Delta_s$ with arithmetical tolerancing
$T_{si} = \Delta_s/\sqrt{4} = 0{,}5\ \Delta_s$ with statistical tolerancing.

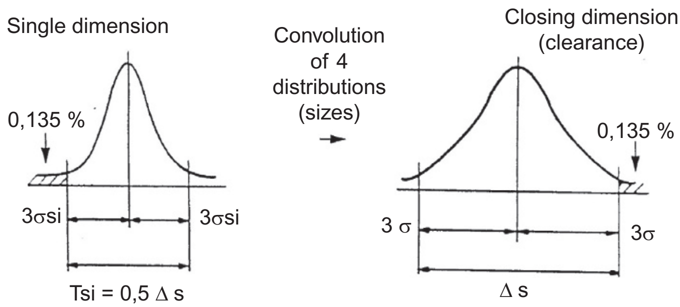


FIG. 4.51 Probability of exceeding the statistical closing tolerance of a chain of four parts. Probability of exceeding $T_s = \Delta_s$: 0,135%, i.e. 1,35 of 1000 assemblies

Figure 4.51 shows the distribution of the closing dimension derived from the distributions of the single dimensions. The probability of exceeding the closing tolerance is 0,135%, i.e. 1,35 of 1000 assemblies may jam and at least one part must be changed for another one.

Here it is assumed that the actual sizes are normally distributed, their standard deviation is one-sixth of the statistical single tolerance, the distribution is centred in the tolerance and the (dimensions of the) parts are independent of each other.

Deviations from these assumptions can be covered by a coverage factor c, so that

$$T_s = \sqrt{\Sigma c^2 T_{si}{}^2}$$

Depending on the manufacturing method (form of distribution) and according to the experiences of the manufacturer, the coverage factor c ranges practically between 1,3 and 1,7.

In Ref. [1] coverage factors c are indicated, depending on the form of distribution of the single sizes (equal, trapeze, triangular, normal). Figure 4.52 shows coverage factors c often used in industrial practice (the practically worst case c = 1,73 occurs very seldom).

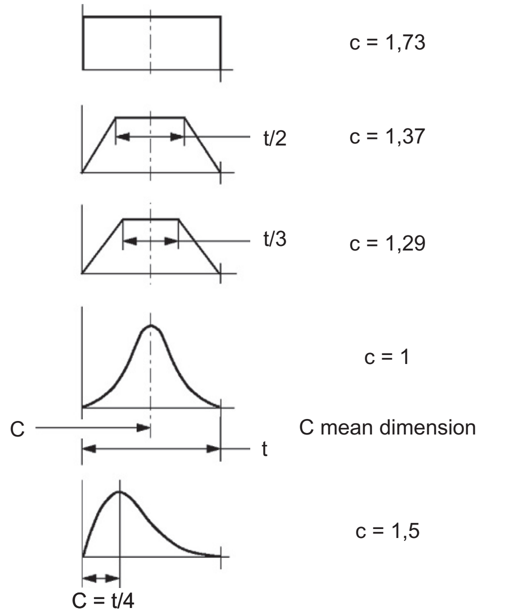


FIG. 4.52 Coverage factors *c* according to the distribution form

According to the central limit theorem of mathematical statistics, in any case, independent of the kind of distribution of the single sizes, the closing dimensions are approximately normally distributed if more than four members contribute to the closing dimension.

In order to stipulate and to verify the assumptions of statistical tolerancing, properly defined terms are needed. An international standard on this subject does not yet exist. The former planned German Standard DIN 7186 defines terms as follows:

Mean size C: arithmetical mean of the limits of size.

Dimension chain: geometrical representation of several cooperating dimensions that are independent of each other (Figs 4.47 and 4.53).

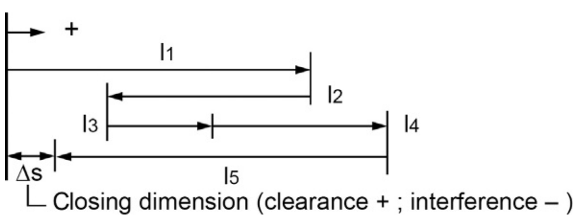


FIG. 4.53 Dimension chains

Single dimension: dimension **acting** as a link in a dimension chain.
Closing dimension: result of arithmetic **addition** of an independent single dimension within a dimension chain.
Single tolerance T_i: tolerance of a single dimension.
Statistical tolerance T_{si}: single tolerance with specifications regarding the distribution of the actual sizes within the tolerance zone.
Arithmetical closing tolerance T_a: sum of the single tolerances within a dimension chain.
Statistical closing tolerance T_s: closing tolerance, smaller than T_a, specified according to the actual size distribution.
Side zones and central zone: for the specification of statistical tolerances, the tolerance zone is divided into zones. In general, three zones predominantly symmetrical with respect to the mean size C are sufficient (Fig. 4.54).

The lower side zone B_l is the zone adjacent to the minimum limit of size.
The upper side zone B_u is the zone adjacent to the maximum limit of size.
The central zone B_m is the zone between the upper and the lower side zone.

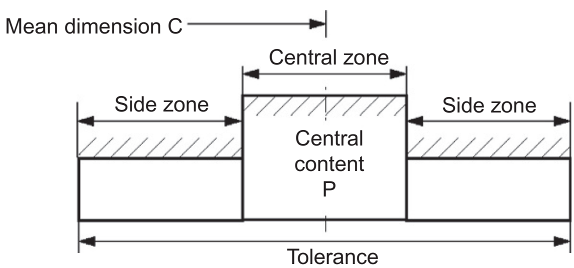


FIG. 4.54 Statistical tolerancing, tolerance zones

Central content **P:** percentage of actual sizes (more precise: mating sizes or related mating sizes) of a manufacturing lot within the central zone; see Fig. 4.54.

Side content P_l and P_u: percentage of actual sizes (more precise: mating sizes or related mating sizes) of a manufacturing lot within the side zone (Figs. 4.54 and 4.56).

If not otherwise specified, according to DIN 7186 the contents P_l and P_u must not be more than (100% – P%)/2 each. (From this it follows that, with an asymmetrical distribution of the actual sizes where there is no lower side zone, the content that would go into this zone actually goes into the central zone (Fig. 4.56).)

For **machining processes (chip removal)**, the following specifications are predominant:

– central zone (width) and side zones (widths) each 1/3 of the tolerance T_{si}
– central content at least 50%, side contents not more than 25% each.

Figure 4.55 shows the drawing indication according to DIN 7186.

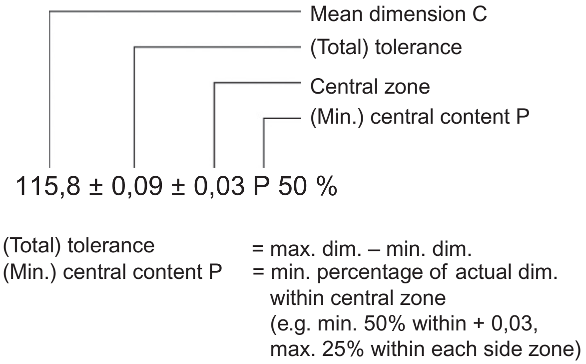


FIG. 4.55 Statistical tolerancing, drawing indication and interpretation according to DIN 7186

Deviations of form, orientation and run-out are not symmetrically distributed. Then the following specifications are predominant: coverage factor $c = 1{,}5$; central zone equal to half the tolerance wide and adjacent to zero located; central content at least 75%. (The left side zone is contained within the central zone). Fig. 4.56 shows the two predominant possibilities of central zones.

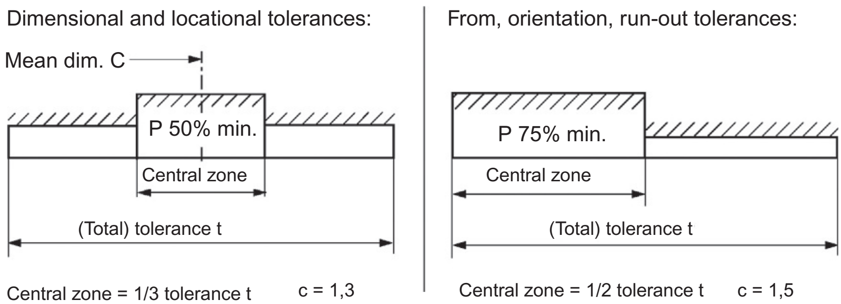


FIG. 4.56 Central zones for statistical tolerancing

An international standard on drawing indications for statistical tolerances has not, thus far, been published. The planned standard will probably adopt the symbol "ST" of the American standard ASME Y14.5 followed by boxed specifications regarding the distribution of the specified property (e.g. size). Figure 4.57 shows some possibilities. The upper three examples specify central

zones and central contents. The other examples specify process capability parameters Cp, Cpk, Cc, Cpu, Cpm which have to be defined in referenced specifications and which determine limits of the distribution of the specified property.

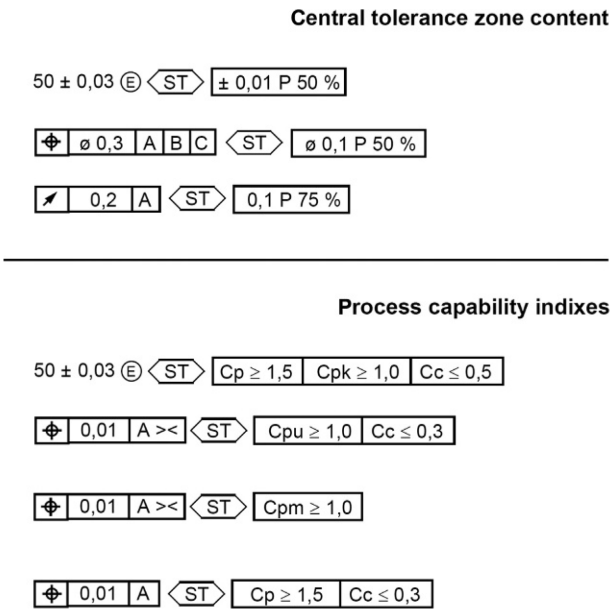


FIG. 4.57 Drawing indications for statistical tolerances

Figure 4.58 shows an example of an assembly. With arithmetical tolerancing, the tolerance for the dimensions l1 … l7 is 0,05 (± 0,025). With statistical tolerancing, the tolerance is 0,1 (± 0,05). Figure 4.59 shows tolerancing of the parts of the assembly. The symbol Ⓨ indicates related mating sizes for the dimensions l1 … l7, which must be explained on the drawing because this symbol is not yet standardized.

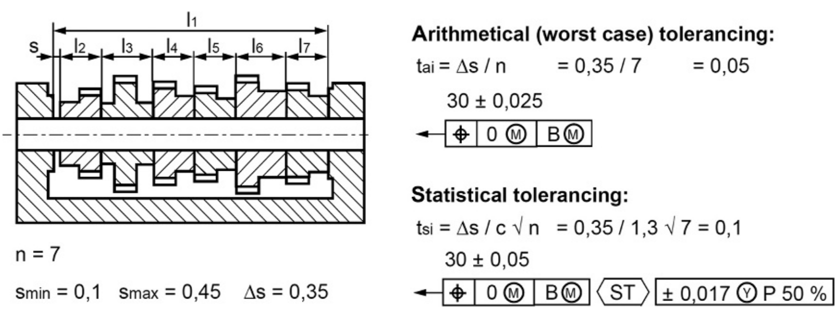


FIG. 4.58 Example of an assembly arithmetically toleranced and statistically toleranced under the condition of the same limits for the clearance (comparison)

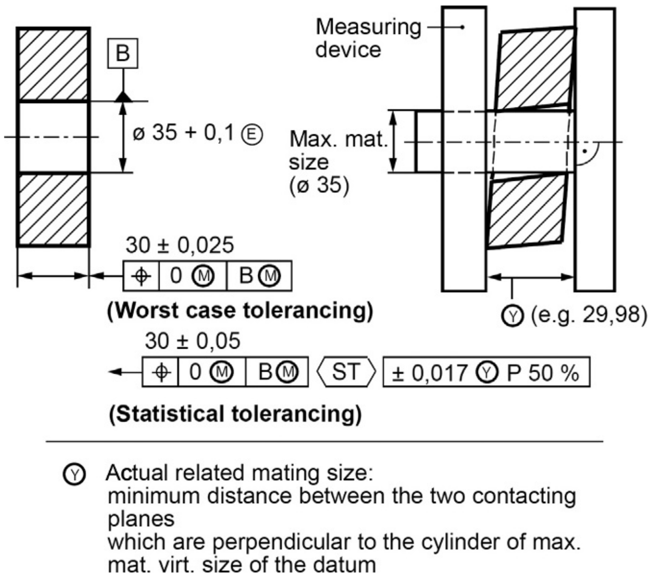


FIG. 4.59 Arithmetical and statistical tolerancing of a component of the assembly

Statistical tolerancing allows specifications of larger single tolerances than arithmetical tolerancing without detriment to the function of the workpiece. In many cases, this results in a gain in manufacturing economy. Because of the current trend to greater miniaturization and to more precise products, statistical tolerancing has become more important. Sometimes the smaller arithmetical tolerances are not even achievable. However, for manufacturing, the following prerequisites should exist:

- the actual sizes of a single dimension are approximately normally distributed or
- deviations from the normal distribution are taken care of by the coverage factor c;
- the mean values of the symmetrical distributions coincide approximately with the mean size;
- the ratios of single tolerance Tsi and standard deviation σ are of the same approximate order of magnitude, e.g. $T_{si}/\sigma = 6$;
- there is no mutual dependence between the dimensions within the dimension chain.

Statistical tolerancing is more advantageous:

- the larger the number of members in the dimension chain,
- the larger the manufacturing lot,
- the better manufacturing and inspection can satisfy the prerequisites for statistical tolerancing (form and location of the distribution; see previous).

Sizes produced by punching usually have very small variability within a delivery lot, i.e. the distribution is very small. For example, regarding the size of punched holes, when the tool is new the distribution starts near the least material size of the hole. Due to the wear of the tool over the years, the distribution drifts towards the maximum material size of the hole. As for a delivery lot, normally it is not known where within the tolerance the distribution is located, so the whole tolerance must be taken into account for the statistical line-up calculation (Fig. 4.60).

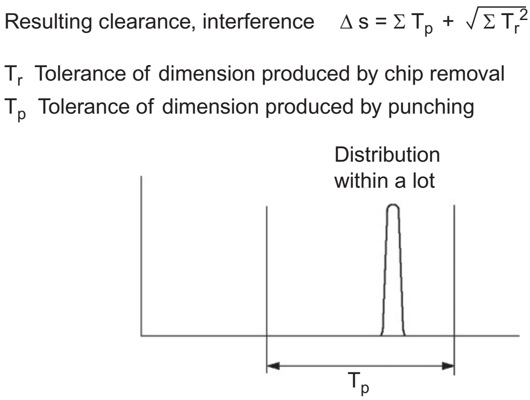


FIG. 4.60 Statistical tolerancing with chain links produced by punching

There are computer programs available for two- or three-dimensional statistical tolerance line-up calculations. These programs use methods of assembly simulation (Monte Carlo method) or convolution of distributions. These programs facilitate statistical tolerancing considerably.

In mass production, statistical tolerancing allows more realistic tolerance calculations, leading to larger tolerances with lower production costs without loss of product quality.

4.8 Population specifications

In mass production there is the possibility to tolerance population (delivery lots) characteristics according to ISO 18391, e.g.: mean μ, standard deviation σ. The tolerances are preceded by the symbol ⟨ST⟩, e.g.:

Ø 10 ± 0,1 (GG) ⟨ST⟩ μ 10/ −0,002; 0,005

or

Ø 10 ± 0,1 (GG) ⟨ST 1⟩

⟨ST 1⟩ = μ 10/ −0,002; 0,005

Population tolerances may be used for size tolerances as well as for geometrical tolerances.

For more information see ISO 18391.

4.9 Angular sizes

ISO 14 405-3 defines the following sizes:

Local angular sizes (varying within the feature of the workpiece, depending on the location along or around the feature of angular size):

(LC) Two-line angular size, minimax (Chebyshev)
(LG) Two-line angular size, Gauss

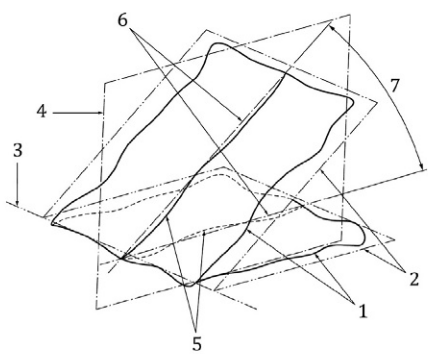
Global angular sizes (constant over the feature):

(GG) Global least squares size (Gauss)
(GC) Global minimax size (Chebyshev)

Statistical (rank order) **sizes**:

(SX) Max. angular size
(SN) Min. angular size
(SA) Average angular size
(SM) Median angular size
(SD) Mid-range size
(SR) Range of angular sizes
(SQ) Standard deviation of angular sizes

Figure 4.61 shows the definition of the two-line angular size for a wedge.



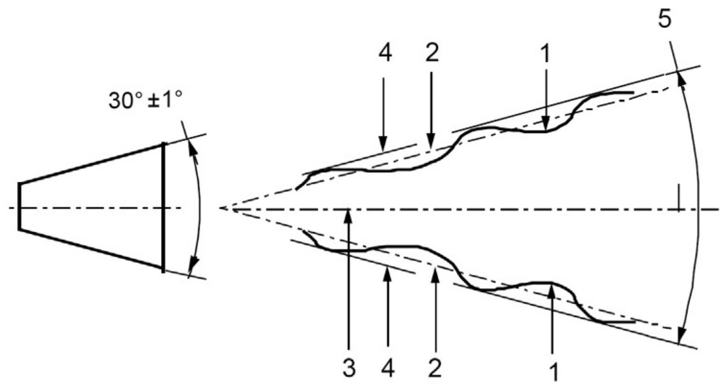
(1) Real angular feature of size
(2) Associated planes with 1 (least squares)
(3) Intersection straight line of 2
(4) Perpendicular cross section to 3
(5) Two extracted lines
(6) Two associated straight lines (Chebyshev)
(7) Two line angular size acc. to Chebyshev

FIG. 4.61 Two-line angular size of a wedge

Angular size is the angular dimension of a cone or wedge, between two coplanar non-parallel straight lines or between two opposite non-parallel planes.

In Fig. 4.61, for the two-line angular size of a wedge, the operations (definition) are explained. (1) The first operation is to associate two planes according to Gauss. (2) The next operation is to add a section plane perpendicular to the intersection line of the two planes. (3) The last operation is to associate straight lines according to minimax (Chebyshev) to the two (extracted) section lines.

The two-line angular size of a cone is defined within a section plane containing the axis of the associated cone according to Gauss. The last operation is to associate straight lines according to minimax (Chebyshev) to the two (extracted) section lines. See Fig. 4.62.



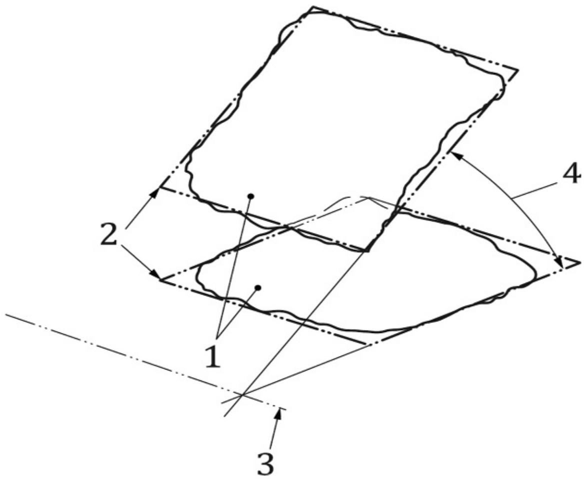
(1) Actual cone
(2) Associated Gaussian cone
(3) Median line of Gaussian cone
(4) Associated chebyshev lines in section containing the gauss median line containing
(5) (Chebyshev) two-line size

FIG. 4.62 Two-line size of a cone

ISO 3040 contains examples of tolerancing of selected properties of a cone, which do not tolerance the cone completely. In order to tolerance the cone completely, it is recommended to use profile tolerancing. See 6.2.8.

Figure 4.63 shows the definition of a global Gaussian size.

See also 3.2.3.2.3.



(1) Actual surfaces
(2) Associated Gaussian planes
(3) Intersection line of the Gaussian planes
(4) Global Gaussian size

FIG. 4.63 Global Gaussian size of a wedge

Chapter 5

Position Tolerancing

5.1 Definitions

The position tolerances for derived features are defined in ISO 1101. ISO 5458 deals with tolerancing of groups (patterns). See 3.2.3.6 to 3.2.3.8.

In the method of position tolerancing (for the location of derived features), theoretically exact dimensions (TEDs) and position tolerances determine the location of features (points, axes, median surfaces) relative to each other or in relation to one or more datum(s). The tolerance zone is defined by the envelope in spheres of diameter of the tolerance and located at the reference feature at the theoretically exact location.

Due to this definition, position tolerances do not accumulate where theoretically exact dimensions are arranged in a chain (Fig. 5.1). This contrasts with dimensional tolerances (± tolerances) arranged in a chain.

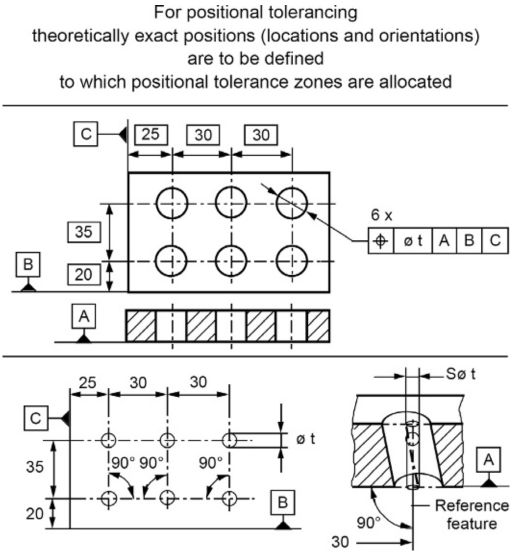


FIG. 5.1 Position tolerancing (The figure is to show the meaning of position tolerances. In practice Ⓜ will be applied to the tolerance related to the datum system (ABC))

Geometrical Dimensioning and Tolerancing for Design, Manufacturing and Inspection.
https://doi.org/10.1016/B978-0-12-824061-8.00005-0

When, in Fig. 5.1, the datums are omitted, the tolerance zones are parallel to each other, but not necessarily perpendicular to the upper or lower surface.

5.2 Theoretically exact dimensions, CZ, SZ, SIM, ± tolerances

According to ISO 1101, theoretically exact dimensions (TEDs) are to be indicated in rectangular frames. However, the following theoretically exact dimensions, that determine the theoretically exact locations of position tolerance zones, are not to be indicated:

a) theoretically exact angles between features (e.g. holes) equally spaced on a complete pitch circle (Fig. 5.26) and
b) theoretically exact dimensions 90° and 0°, 180° or distance 0 (features located theoretically on a straight line) between
- position-toleranced features not related to a datum (Fig. 5.13),
- position-toleranced features related to the same datum(s) (Fig. 5.1),
- position-toleranced features and their related datum(s) (Fig. 5.1).

Formerly, when the position-toleranced features were drawn on the same centre line, they were regarded as related features having the same theoretically exact location (Fig. 5.2 without the indications SIM), unless otherwise specified, e.g. by relation to different datums, or by an appropriate note on the drawing (Fig. 5.3). In Fig. 5.3, the locations of the two patterns of features (holes) are independent of each other.

When different position tolerances related to the same datum system (e.g. with Ⓜ in the datum system) are to be considered simultaneously, SIM shall be indicated after the tolerance indicators; see Fig. 5.2. When more simultaneous groups are concerned, SIM shall be enumerated, e.g. SIM1, SIM2.

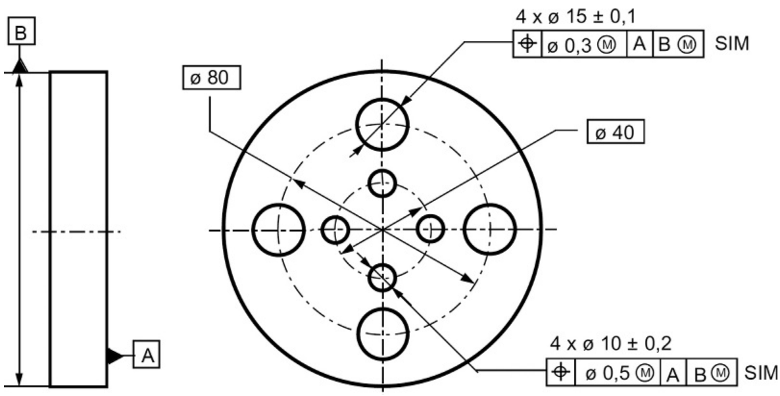


FIG. 5.2 Position-toleranced features (holes) equally spaced on a complete pitch circle, related to the same datum system, simultaneous requirements (former practice: without SIM, simultaneous requirements because drawn on the same centre line)

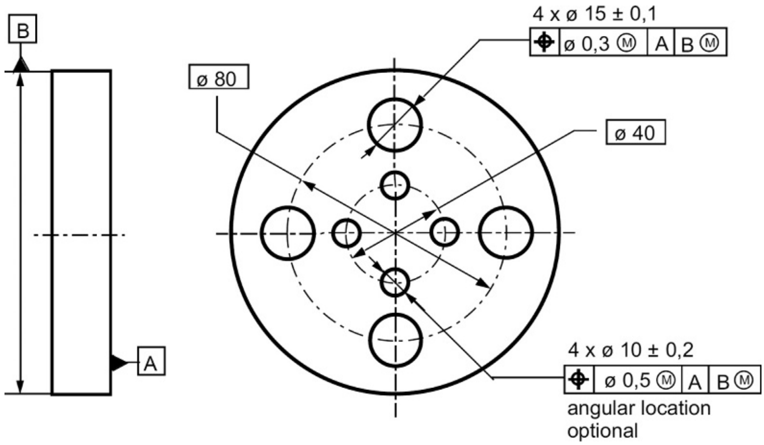


FIG. 5.3 Position-toleranced features (holes) drawn on the same centre line, but with angular location of the two patterns relative to each other being optional, former practice

Figures 5.4 and 5.5 show position tolerances for two patterns, both related to the same datum system that contains Ⓜ. When the requirements of both patterns

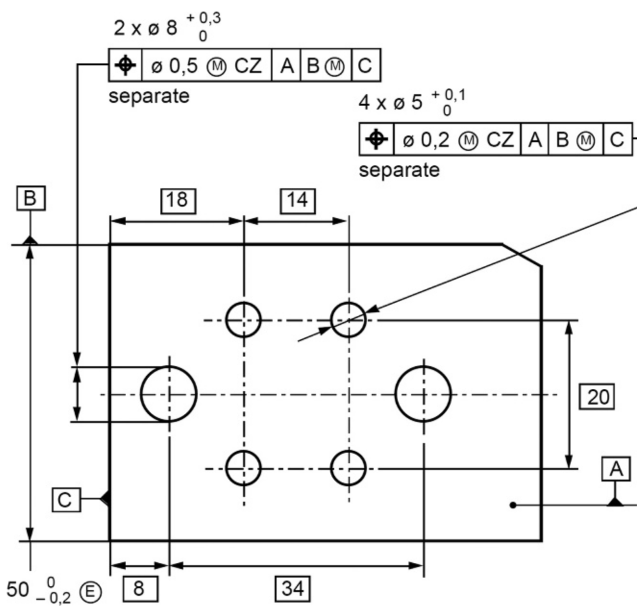


FIG. 5.4 Multiple patterns of holes, with same datum system including Ⓜ, separate requirements

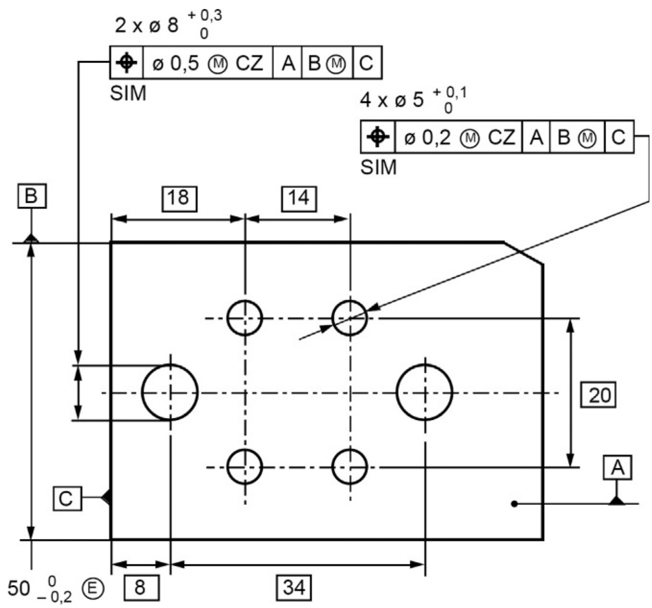


FIG. 5.5 Multiple patterns of holes, with same datum system including Ⓜ, simultaneous requirements

apply simultaneously (inspected by a combined gauge), SIM is to be indicated (Fig. 5.5). According to ISO 8015, without SIM the independency principle applies (inspected by separate gauges).

However, according to ASME Y14.5 the opposite applies: without the indication SIM the simultaneous requirement applies. Therefore, to avoid misinterpretations, it is recommended to indicate "separate", when separate requirements apply.

However, this indication is only necessary when Ⓜ or Ⓛ is in the datum system. Without Ⓜ or Ⓛ in the datum system, it defines the exact location, so there is no difference between SIM and separate.

It was former practice to consider multiple features, position toleranced, as if CZ were indicated (the tolerance zones consider the relation of the features relative to each other). It is planned for the future to also apply the independency principle to position-toleranced features. This then is the opposite. During this intermediate period, according to ISO 1101 and ISO 5458, with position tolerances for patterns, it shall always be indicated whether CZ (combined zones) or SZ (separate zones) applies. When the position tolerances are related to a complete datum system, locking all degrees of freedom of the workpiece, CZ may be omitted, because the positions are fixed; see Fig. 5.6.

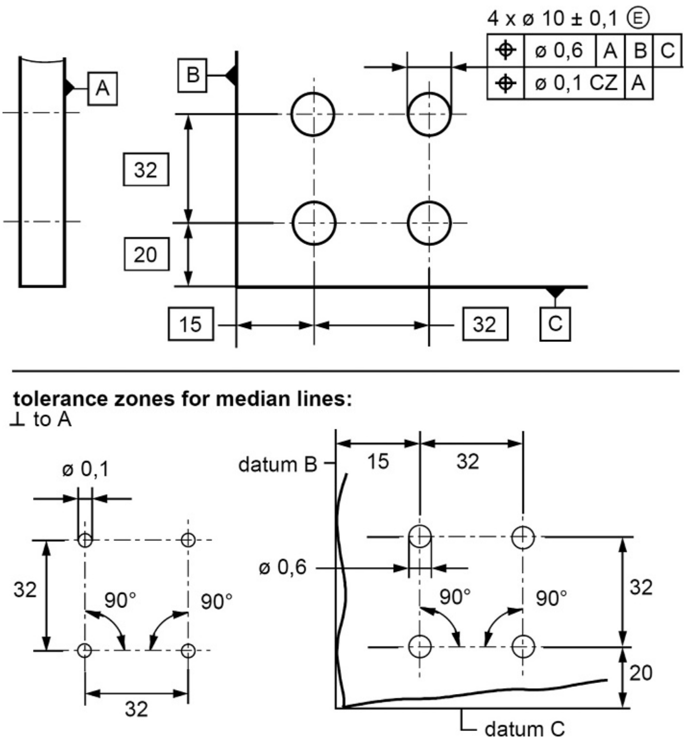


FIG. 5.6 Combination of position tolerancing with different datums (The figure shows the meaning of position tolerances. In practice Ⓜ will be applied to the tolerance related to the datum system (A, B, C))

It was former practice to tolerance patterns of holes by position tolerances relative to each other and to tolerance the location of the pattern by ± tolerances; see Fig. 5.7. This leads to ambiguities in the meaning of the ± tolerances. It is unclear what the centres of the holes are (maximum inscribed cylinder axes, centre of two-point sizes) and from what to measure (real surface of the

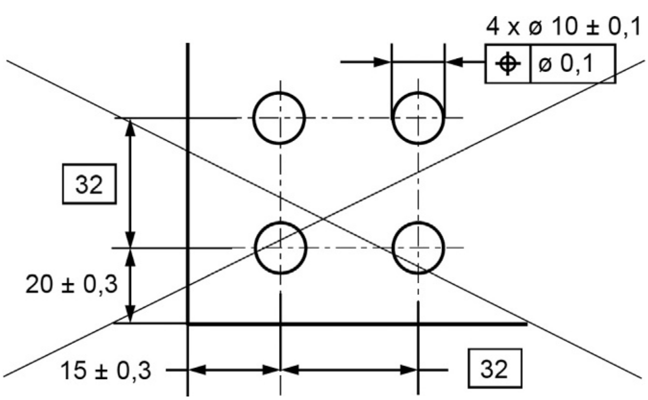


FIG. 5.7 Dimensional coordinate tolerances adjacent to position tolerances, no longer used

workpiece, contacting plane) and what is primary, secondary, or tertiary. Therefore, ± tolerances for distances shall not be used (ISO 14 405-2).

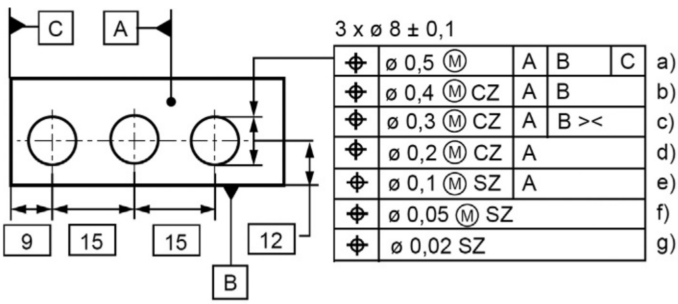
Instead, position tolerancing shall be used for both distances between the holes and location in the datum system; see Fig. 5.6.

Each requirement shall be met independently. For example, the actual axis of each of the four holes in Fig. 5.6 shall be within the cylindrical position tolerance zone of ø 0,1. The position tolerance zones are located in their theoretically exact positions in relation to each other and perpendicular to datum A.

The actual axis of each hole shall be within the cylindrical position tolerance zone of ø 0,6. These position tolerance zones are located in their theoretically exact positions in relation to the datums A, B and C.

It should be noted that datums, drawn perpendicular to each other, are not necessarily perpendicular to each other when they are not in the same datum system. (This applies to all datums of any geometrical tolerancing.)

Figure 5.8 shows the possibilities of tolerancing a pattern (group) of holes. Tolerancing a) is mandatory; without this tolerance, the part is not completely toleranced.



a) gauge, all TEDs regarded **mandatory**

optional:

b) gauge, TEDs 15 & 12 regarded

c) gauge, TEDs 15 regarded, ⊥ A, // B

d) gauge, TEDs 15 regarded, ⊥ A, coplanar

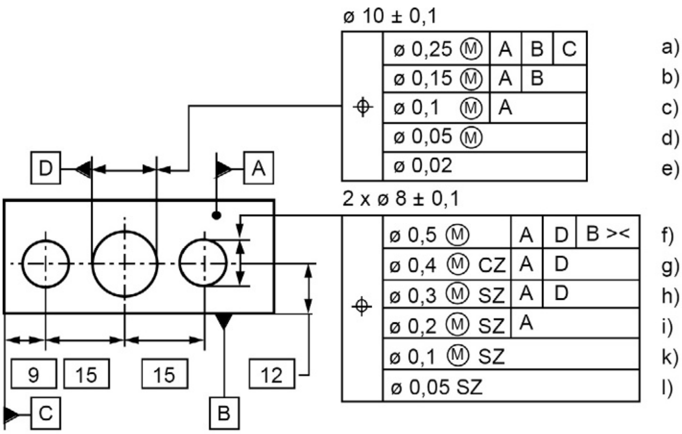
e) separate gauges, ⊥ A, TEDs 15 disregarded

f) separate gauges, TEDs 15 disregarded

g) separate zones, all TEDs disregarded (= straightness of axes)

FIG. 5.8 Pattern of holes, tolerancing possibilities

Figure 5.9 shows the possibilities of tolerancing a pattern (group) of holes related to a datum hole and all toleranced holes are parallel to datum B toleranced. The tolerances a) and f) are mandatory; without these tolerances the part is not completely toleranced.



a) gauge, all TEDs regarded, **mandatory**
b) gauge, related to A & B (TED 12 regd.., TEDs 15+9 =24 disregarded)
c) gauge, ⊥ A
d) gauge, unrelated
e) zone, unrelated (= straightness of axis)
f) gauge, ⊥ A, centred at D (TEDs 15 regarded), // B, **mandatory**
g) gauge, related to A & D, (TEDs 15 regarded, coplanar, orientation free)
h) separate gauges, related to A & D (TEDs 15 regarded, orientation free)
i) separate gauges ⊥ A
k) separate gauges, unrelated
l) separate zones, unrelated (= straightness of axes)

FIG. 5.9 Pattern of holes, related to a datum hole, tolerancing possibilities

Figure 5.10 shows a group of threaded holes related to a datum hole with a large distance to datum C.

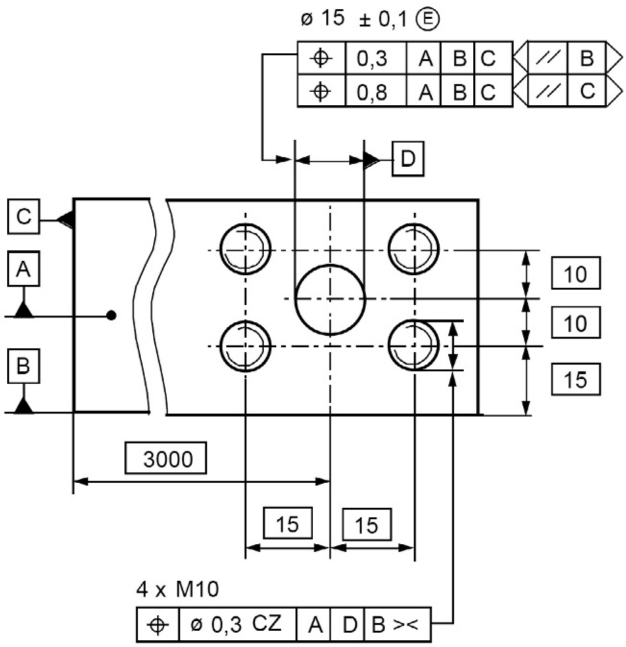


FIG. 5.10 Group of treaded holes related to a datum hole with a large distance to datum C

5.3 Form of the position tolerance zone

It was former practice that the width of the tolerance zone had the direction of the leader line arrow (which connects the toleranced feature with the tolerance indicator) (Fig. 5.11). With this practice, it was unclear which workpiece surface determined the direction. According to the current standard ISO 1101, the direction of the width of the tolerance zone shall be specified by the tolerance zone plane indicator; see Fig. 5.11.

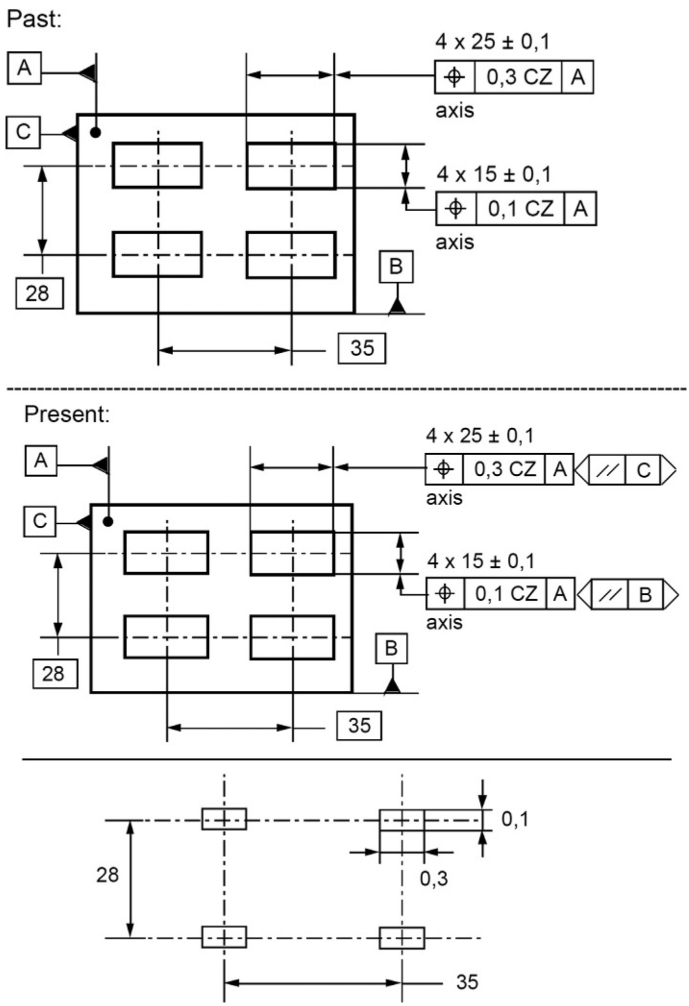


FIG. 5.11 Position tolerance zones of axes with perpendicular cross sections

The indication "axis" in Fig. 5.11 refers to the section line of the two median surfaces. If this indication is missing, it means the median surfaces of the rectangular holes. The tolerance zones are then defined by two parallel planes a distance of 0,3 and 0,1 apart along the entire holes.

When the symbol ø appears before the tolerance value, the tolerance zone is cylindrical.

For cylindrical features the tolerance zone is normally cylindrical, because the function permits the same deviation in all orientations.

Position tolerances can have cylindrical tolerance zones. This is in contrast to size tolerances, which permit only tolerances in two directions perpendicular to each other. The cylindrical tolerance zone is 57% larger than the rectangular tolerance zone; see Fig. 5.12.

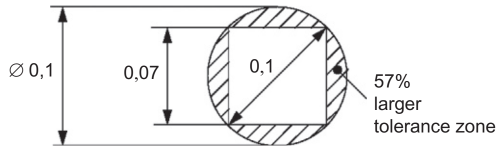


FIG. 5.12 Comparison of tolerance zones, cylindrical and rectangular

5.4 Calculation of position tolerances

A distinction must be made between floating and fixed fasteners (Figs. 5.13 and 5.14).

For **floating** fasteners (Fig. 5.13), the position tolerance t_{cd} of the holes is

$$t_{cd} = D_{min} - d_{max}$$

D_{min}: maximum material size of internal dimension (minimum size of the hole).
d_{max}: maximum material size of external dimension (maximum size of the bolt).

Figure 5.13 shows the extreme locations of the holes of maximum material size that still allow assembly with the bolt of maximum material size. From these locations, the position tolerance zone ø ($D_{min} - d_{max}$) is derived.

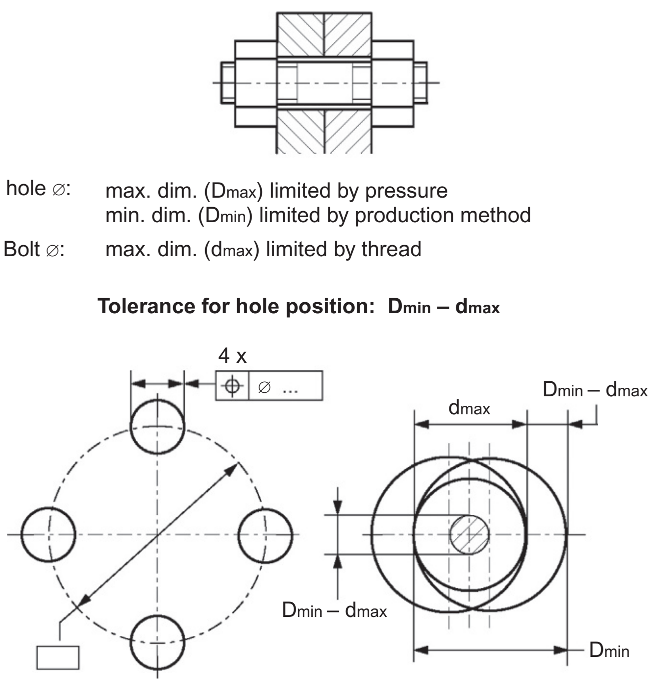


FIG. 5.13 Position tolerance with floating fastener

For **fixed** fasteners (Fig. 5.14), the position tolerance t_{ce} of the holes is

$$t_{ce} = (D_{min} - d_{max})/2$$

Figure 5.14 shows the extreme locations of the holes of maximum material size that still allow assembly with a bolt of maximum material size. From these locations, the position tolerance zone ø $(D_{min} - d_{max})/2$ is derived.

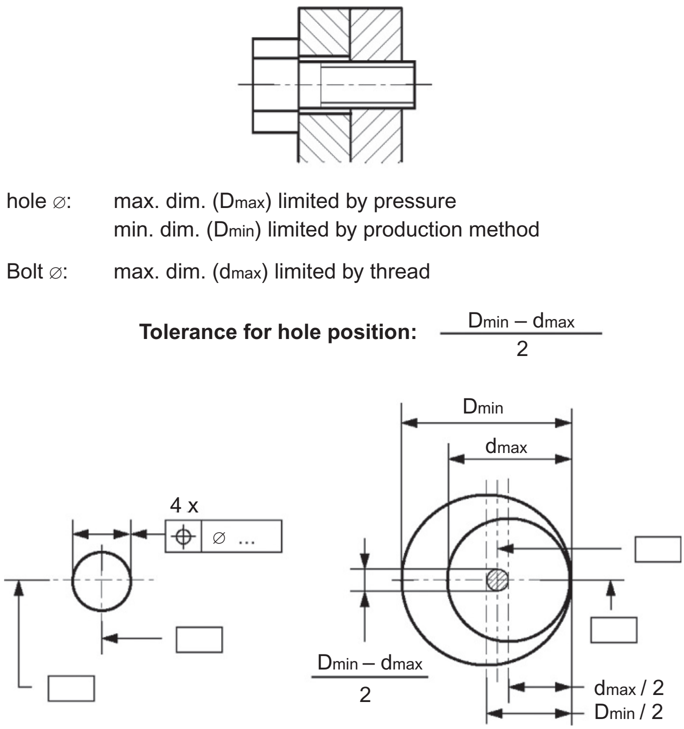


FIG. 5.14 Position tolerance with fixed fastener

The formulae are also applicable to non-cylindrical features (e.g. keys and keyways) (see also Fig. 5.61). See Table 5.1.

The indicated position tolerances t_{ce} and t_{cd} in Table 5.1 do not take account of the straightness tolerances of the screw and hole, the minimum clearance between bolt and hole, or any measurement uncertainty. If necessary, the values are to be decreased accordingly. For head screws the under-head fillet may require chamfered holes or washers. Table 5.1 bottom shows the formulae for the straightness tolerances t of the screws according to ISO 4759-1. Accordingly, for the position tolerances t_c the values t_{cd} and t_{ce} are to be decreased by t.

TABLE 5.1 Position tolerances for holes according to ISO 273 and screws t_{ce} position tolerance for through-hole and threaded hole for stud screw or head screw (Fig. 5.14) t_{cd} position tolerance for through-hole and threaded bolt with nuts at both ends or head screw with nut (Fig. 5.13)

hole ø Dmin: ISO 273 medium   tce = (Dmin – d) / 2   tcd = Dmin – d

thread ø	hole ø	tce	tcd	thread ø	hole ø	tce	tcd	thread ø	hole ø	tce	tcd
1	1,2	0,1	0,2	14	15,5	0,75	1,5	64	70	3	6
1,2	1,4	0,1	0,2	16	17,5	0,75	1,5	68	74	3	6
1,4	1,6	0,1	0,2	18	20	1	2	72	78	3	6
1,6	1,8	0,1	0,2	20	22	1	2	76	82	3	6
1,8	2,1	0,15	0,3	22	24	1	2	80	86	3	6
2	2,4	0,2	0,4	24	26	1	2	85	91	3	6
2,5	2,9	0,2	0,4	27	30	1,5	3	90	96	3	6
3	3,4	0,2	0,4	30	33	1,5	3	95	101	3	6
3,5	3,9	0,2	0,4	33	36	1,5	3	100	107	3,5	7
4	4,5	0,25	0,5	36	39	1,5	3	105	112	3,5	7
4,5	5	0,25	0,5	39	42	1,5	3	110	117	3,5	7
5	5,5	0,25	0,5	42	45	1,5	3	115	122	3,5	7
6	6,6	0,3	0,6	45	48	1,5	3	120	127	3,5	7
7	7,6	0,3	0,6	48	52	2	4	125	132	3,5	7
8	9	0,5	1	52	56	2	4	130	137	3,5	7
10	11	0,5	1	56	62	3	6	140	147	3,5	7
12	13,5	0,75	1,5	60	66	3	6	150	158	4	8

position tolerance tc:

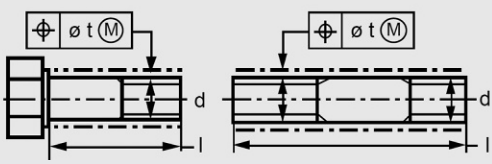
only through holes   tc = tcd – t

through holes and threaded holes   tc = tce – t

ISO 4759-1:

d ≤ 8: t = 0,002 l + 0,05

d > 8: t = 0,0025 l + 0,05



The prerequisite for these position tolerances is that the form of the holes allows them to fit the largest bolt. However, the ± tolerances for the hole diameter apply to two-point sizes. Inspected with plain measuring instruments, e.g. vernier callipers, the form of the hole may be lobed (produced by drilling, reaming) (see Fig. 5.15 bottom) or deformed (see Fig. 5.15 top) (deformed by shrinking).

Then the mating size of the hole may be smaller than the lower limit of size, although the measurement results are within the size tolerance. Therefore, in order to assure the assembling ability, the maximum material requirement Ⓜ is to be applied with the position tolerance for holes for fasteners; see Fig. 5.16.

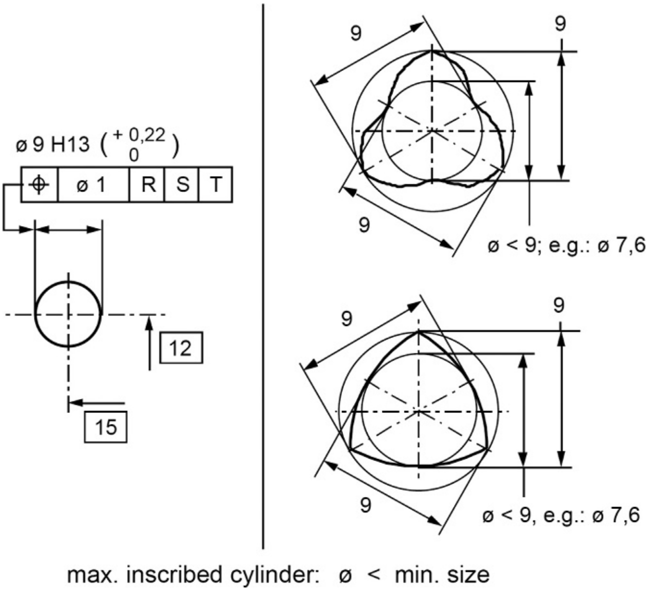


FIG. 5.15 Possible shapes of holes for fasteners

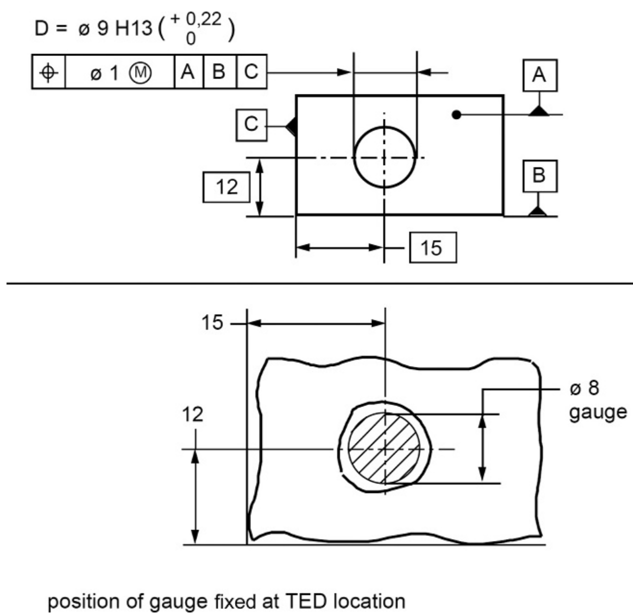


FIG. 5.16 Correct tolerancing of a hole for a fastener

Figure 5.16 shows position tolerancing with Ⓜ of through-holes for a bolt M8 (see Table 5.1).

It is also possible to tolerance the through-holes with position tolerancing with Ⓔ; see Fig. 5.17.

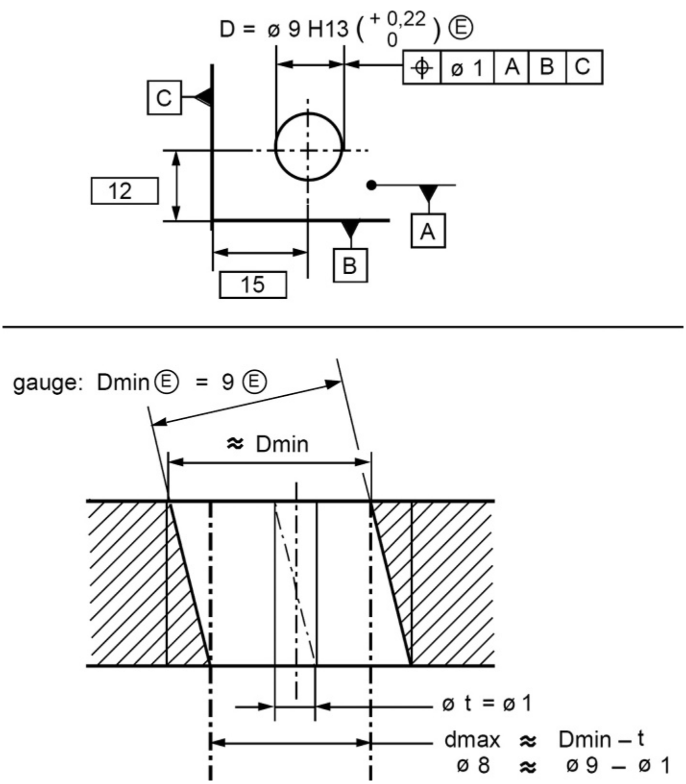


FIG. 5.17 Tolerancing of a hole for a fastener with Ⓔ

Tolerancing with Ⓔ is more stringent than tolerancing with Ⓜ. Since with Ⓜ the gauge is smaller than with Ⓔ, tolerancing with Ⓜ allows larger form deviations of the hole. Therefore tolerancing with Ⓜ is recommended. This also corresponds to the practice according to ASME Y14.5.

5.5 Projected tolerance zones

Position tolerances of threaded holes and through-holes for fixed fasteners (Figs 5.14 and 5.18) are usually calculated from the maximum diameter of the bolt and the minimum diameter of the through-hole (see Fig. 5.14).

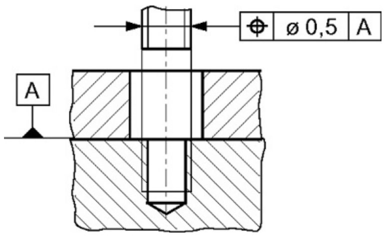


FIG. 5.18 Position tolerance of a threaded hole

A prerequisite for these calculations is that the axis must be perpendicular to the joint surface. In the extreme case (Fig. 5.19), the parts still fit.

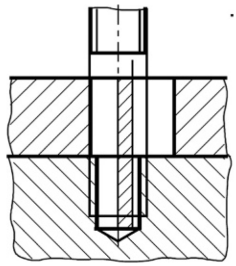


FIG. 5.19 Position tolerance zones of threaded hole and through-hole

When the axis of the threaded hole deviates from the perpendicular orientation (but remains in the position tolerance zone), the parts do not fit (Fig. 5.20).

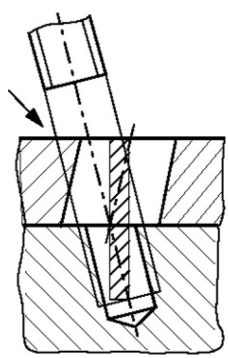


FIG. 5.20 Orientation deviations taking advantage of the position tolerance zones

However, when the position tolerance zone is located outside the part and applies to the external projection of the feature (axis), Fig. 5.21, the part fits in any case (Fig. 5.22). The external projection has the length of the

through-hole in the case of head screws (Fig. 5.23), and the length of the outstanding part of the stud or dowel pin (Fig. 5.24). The projected tolerance zone applies only to these lengths, not to the (length of the) axis within the workpiece.

ISO 1101 defines the projected tolerance zone as follows:

The **projected (position) tolerance zone** applies to the external projection of the feature indicated on the drawing by the symbol Ⓟ placed in the tolerance frame after the position tolerance of the toleranced feature. The minimum extent and the location of the projected tolerance zone are shown in the corresponding drawing view by Ⓟ preceding the projected dimension (Fig. 5.21).

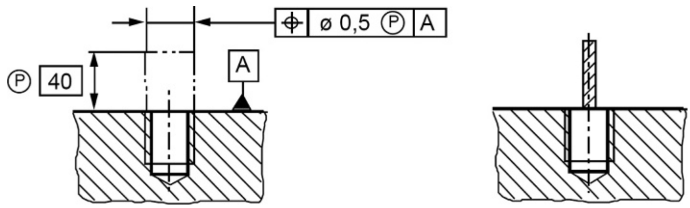


FIG. 5.21 Projected tolerance zone

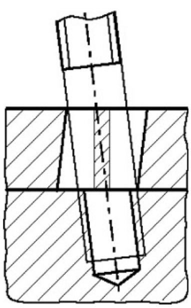


FIG. 5.22 Taking advantage of the position zone by inclined axes

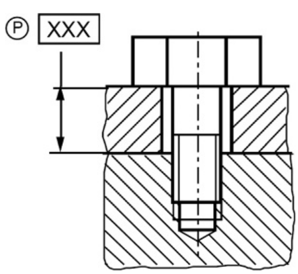


FIG. 5.23 Projected tolerance zone for threaded hole and head screw

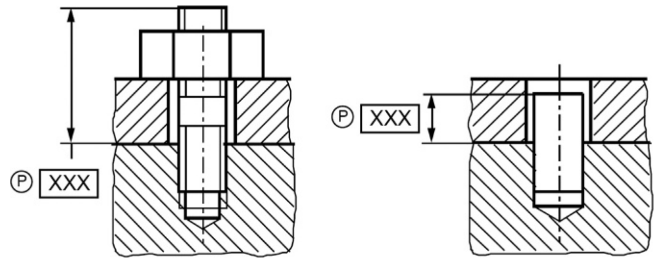


FIG. 5.24 Projected tolerance zone for threaded hole and stud or for hole and pin

The exact definition of the axis to be extended (projected) has not yet been standardized (see 3.5). From a practical point of view, the following definitions of the axis may be applicable:

– for plain holes (e.g. interference fits): the axis of the contacting Chebyshev cylinder;
– for threaded holes: the axis of a (nearly) geometrical ideal screw of maximum material sizes (go gauge) associated according to Chebyshev.

The length of the projected zone may be (simplified) indicated in the tolerance indicator after the symbol Ⓟ; see Fig. 5.25.

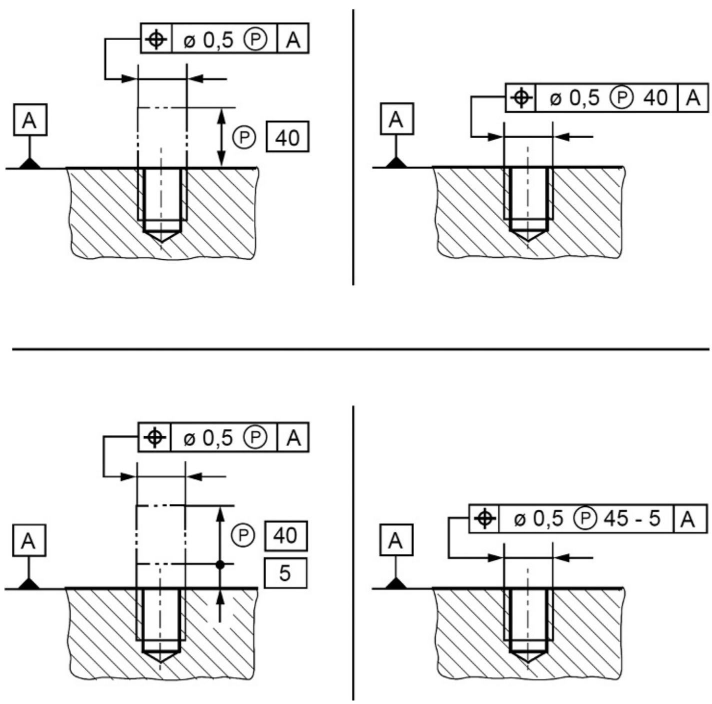


FIG. 5.25 Drawing indications of projected tolerance zones; right, simplified drawing indications

When there is a sealing with a larger hole between the flanges, the projected tolerance zone starts after the sealing; see Fig. 5.25 lower left. The simplified drawing indication is shown in Fig. 5.25 lower right.

5.6 Further examples

5.6.1 Holes for fasteners

Fastener assemblies may be designed with through-holes and (floating) bolts and nuts (Fig. 5.26) or with threaded holes and head screws (Fig. 5.27) or with threaded holes and (fixed) stud screws (Fig. 5.28). In the cases shown in Figs 5.27 and 5.28, the through-holes in the other flange have the same tolerance but without Ⓟ.

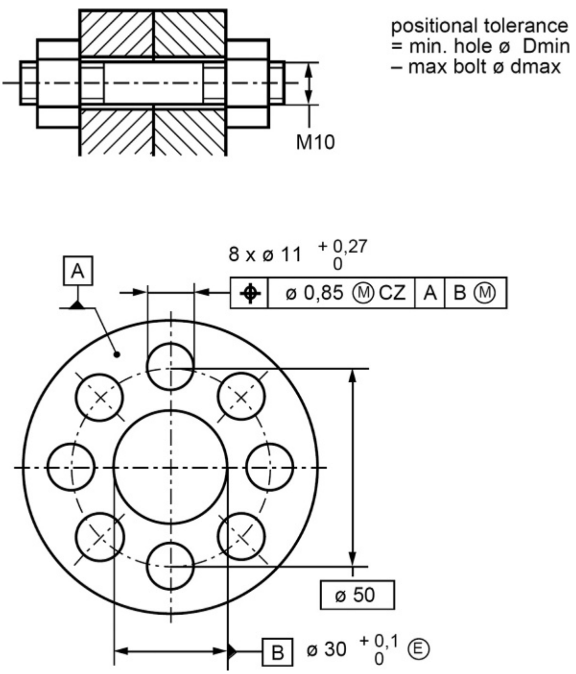


FIG. 5.26 Through-holes for floating fasteners (bolts and nuts)

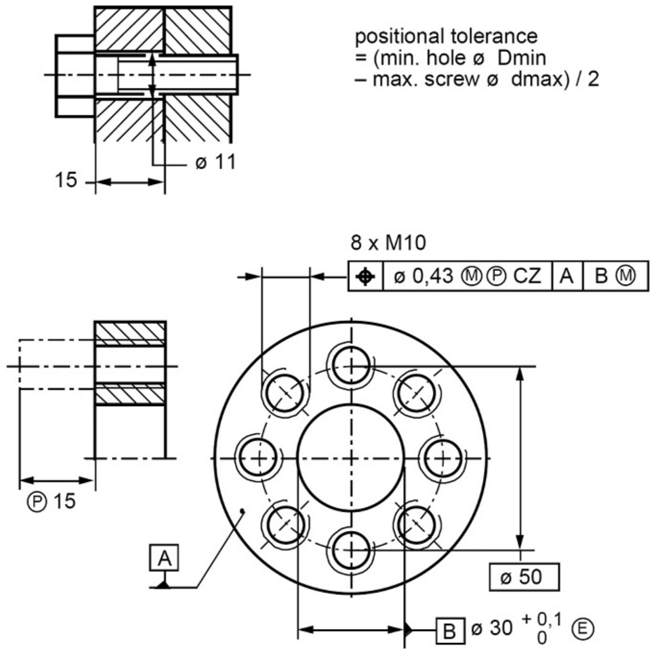


FIG. 5.27 Threaded holes for fixed fasteners (head screws)

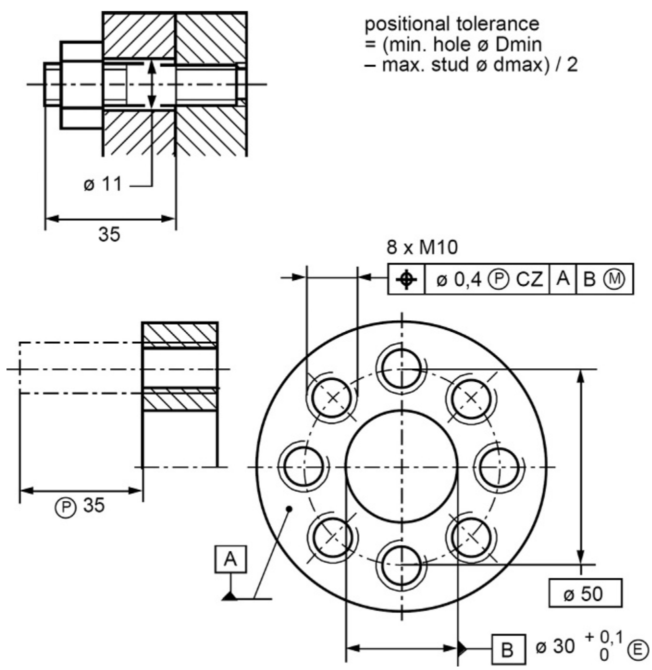


FIG. 5.28 Threaded holes for fixed fasteners (stud screws and nuts)

5.6.2 Counterbores

Figure 5.29 shows tolerancing of separate counterbores, each related to the individual hole (each separately gauged).

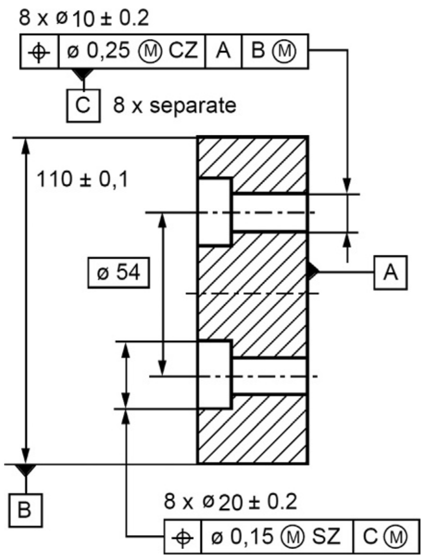


FIG. 5.29 Tolerancing of counterbores

5.6.3 Holes in a line

Figure 5.30 shows an example in which the four gauges are in a plane parallel to datum B. The distances of the axes to the upper plane may vary within Ø 0,5.

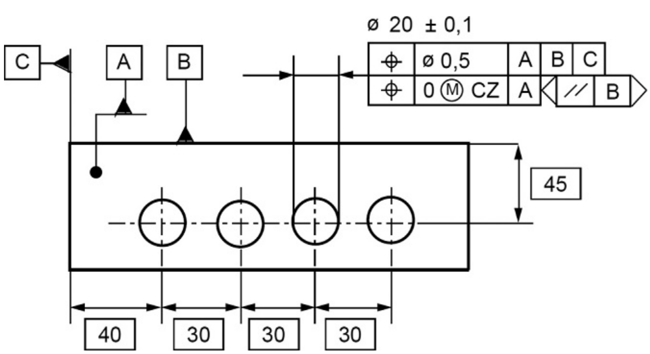


FIG. 5.30 Cosymmetry of holes in line

5.6.4 Group of holes with and without datum hole

Figure 5.31 shows the difference between with and without datum hole for a group of holes.

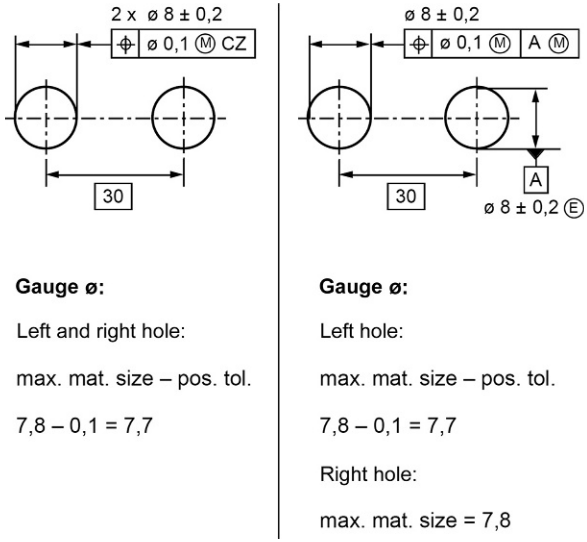


FIG. 5.31 Group of holes with and without datum hole

5.6.5 Crossed axes

Fig. 5.32 shows an example of tolerancing of crossed axes for bevel gears. The projection (extension) of the horizontal axis must be contained between two vertical planes 0,1 apart, between two horizontal planes 0,05 apart, over the length of 75. The projection must be within the zone of ø 0,02 75 apart from B (at the datum A).

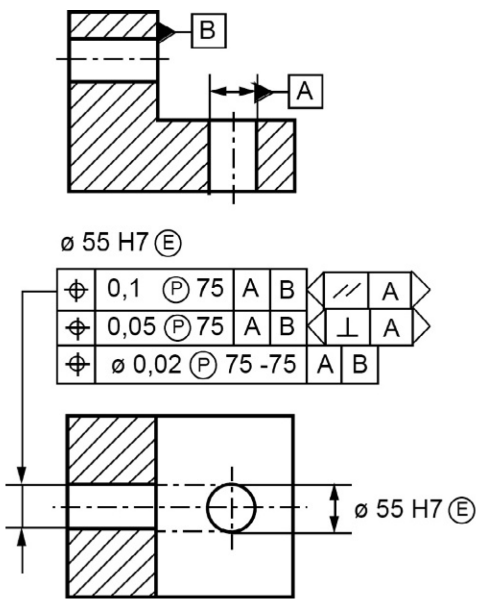


FIG. 5.32 Tolerancing of crossed axes

Figure 5.33 shows a) the position (and orientation) tolerancing of the hole relative to the datum hole A. The case b) shows an additional restriction (constraint) for orientation.

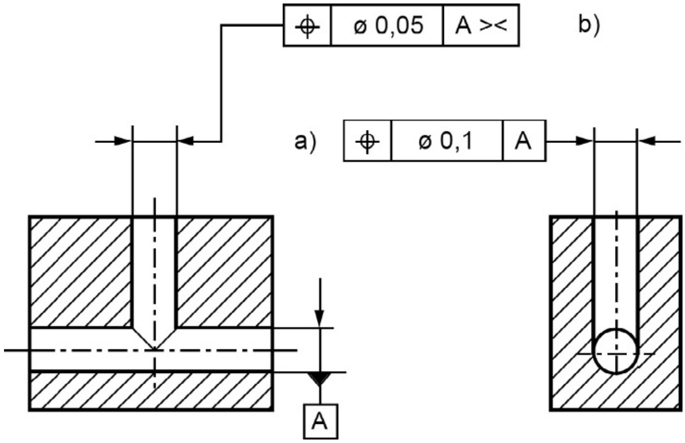


FIG. 5.33 Tolerancing of axes of holes, a) location tolerance, b) additional orientation tolerance

5.6.6 Different location tolerances for different features drawn on the same centre line

When on a drawing features are drawn on the same centre line, it is difficult to allocate different distance tolerances to the different features. For example, in Fig. 5.34 the indication applies for the holes as well as for the slots. Position tolerancing, however, provides the possibility of allocating different tolerances to the different features (Figs 5.5 and 5.35).

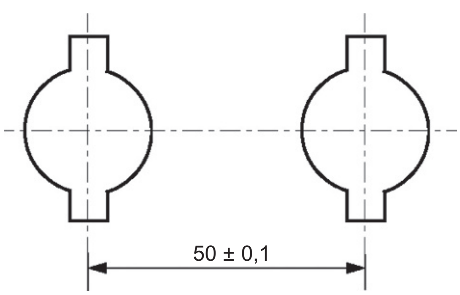


FIG. 5.34 Distance tolerance of features drawn on the same centre line (to be avoided)

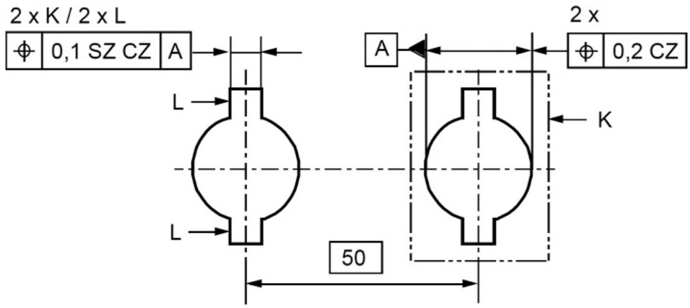


FIG. 5.35 Features drawn on the same centre line, but with different position tolerances

In Fig. 5.35, the holes are toleranced by a position tolerance 0,2 relative to each other. The slots are toleranced by a position tolerance 0,1 relative to their holes, but independent for each hole (two independent groups K).

The position tolerances shown in Fig. 5.36 lower allow position tolerancing of the median surfaces of the elongated holes in relation to the median plane A. With the indication 10 ±0,1 instead of the position tolerances (Fig. 5.36 upper), the location of the elongated holes depends on the actual locations of the two cylindrical holes; this should be avoided.

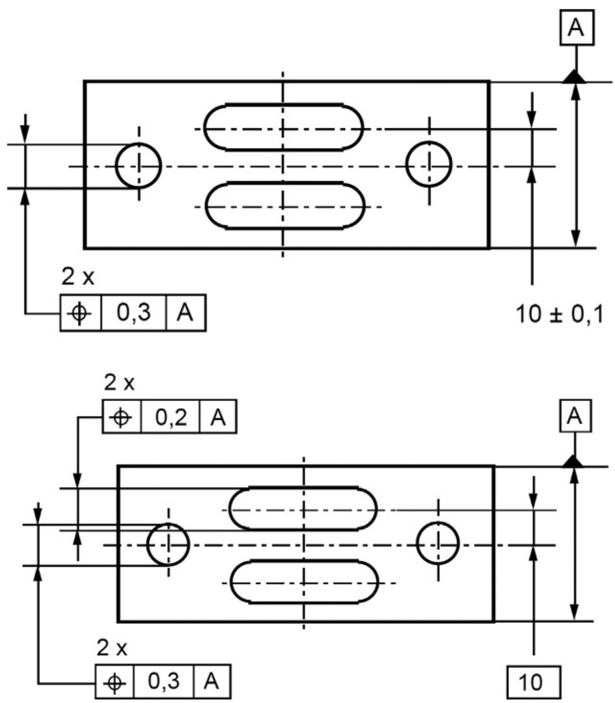


FIG. 5.36 Differentiation of location tolerances in relation to the same median plane

5.6.7 Toleranced coplanar and coaxial feature and datum

Figure 5.37 shows toleranced coplanar and coaxial feature and datum.

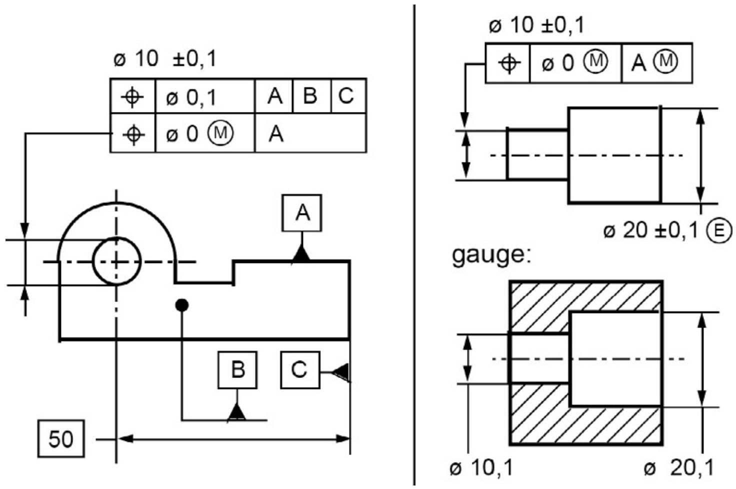


FIG. 5.37 Toleranced coplanar and coaxial feature and datum

5.6.8 Planar surface as datum for a position tolerance, with offset

Normally, the datum for a run-out tolerance is an axis. With machines (e.g. electrical motors) it is possible that the datum axis cannot be detected. In such cases the housing base or housing flange may be indicated as the datum (Fig. 5.38).

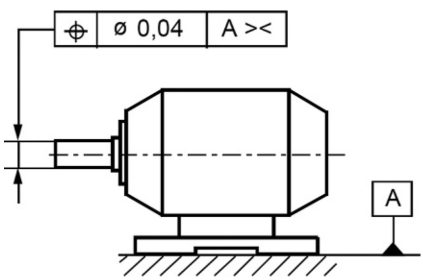


FIG. 5.38 Planar surface as datum for a run-out tolerance

5.6.9 Sequence and maximum material requirement for datums

Figure 5.39 shows different possibilities for datums. In case a), the surface B must contact the gauge according to the minimum rock requirement. The diameter of the gauge is equal to the maximum material size of the datum feature A.

In case b), the gauge must enclose the datum feature A with the least possible diameter (mating size of datum feature A). The datum feature is oriented according to the minimum rock requirement. The surface B contacts the gauge on one point only.

In case c), the surface B must contact the gauge according to the minimum rock requirement. In this orientation, the gauge must enclose the datum feature A with the least possible diameter.

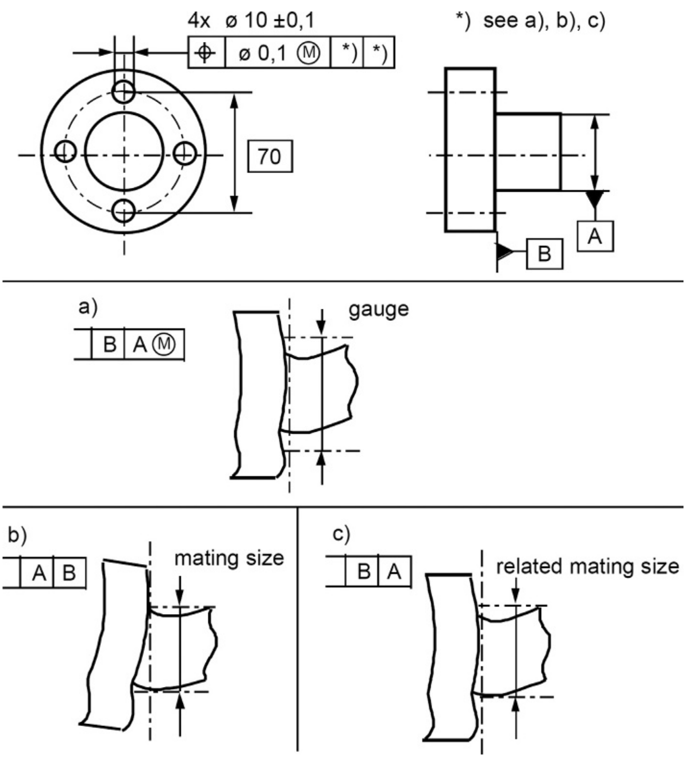


FIG. 5.39 Sequence and maximum material requirement for datums

5.6.10 Datum systems with and without Ⓜ

Figures 5.40 and 5.41 show position-toleranced holes with and without application of the maximum material requirement for the datums.

In Fig. 5.40, the surface A must contact the gauge according to the minimum rock requirement. The maximum possible cylinder perpendicular to the gauge surface A must enclose the datum feature B. The slot C must be enclosed by the maximum possible parallelepiped (plane pair) perpendicular to the gauge surface A. The median plane of the parallelepiped contains the axes of the gauge cylinder B. The maximum material virtual conditions of the four holes (gauge) are ø 7,5.

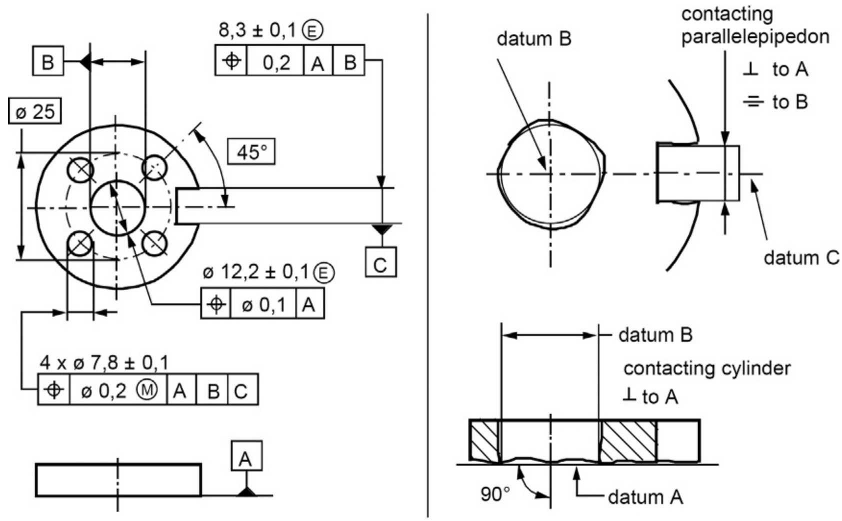


FIG. 5.40 Datums without maximum material requirement

As seen in Fig. 5.41 also, surface A must contact the gauge according to the minimum rock requirement. The gauge cylinder B and the gauge parallelepiped C are perpendicular to the gauge surface A, and the symmetry plane of the gauge parallelepiped contains the axis of the gauge cylinder B. However, the gauge cylinder B and gauge parallelepiped C have maximum material virtual size (ø 12 and 8). The maximum material virtual conditions of the four holes (gauge) are ø 7,5.

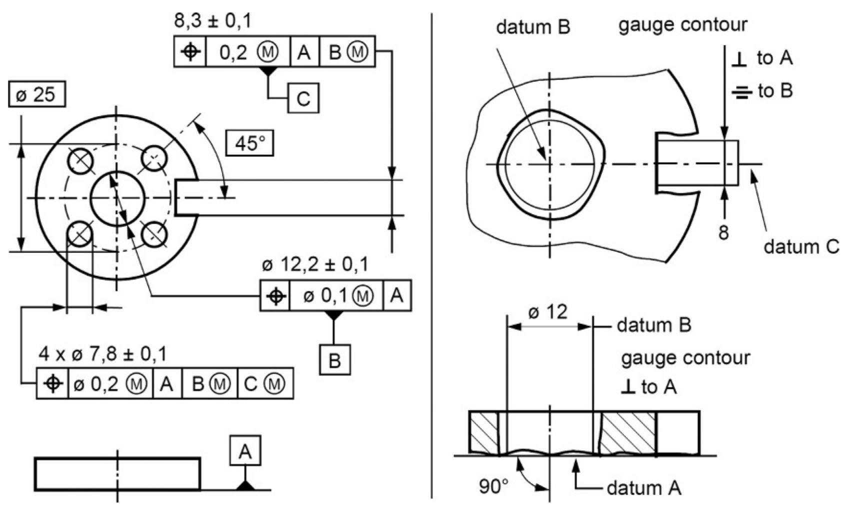


FIG. 5.41 Datums with maximum material requirement

5.6.11 Coaxial shafts (e.g. for bearings)

See Figs 5.42 and 5.43.

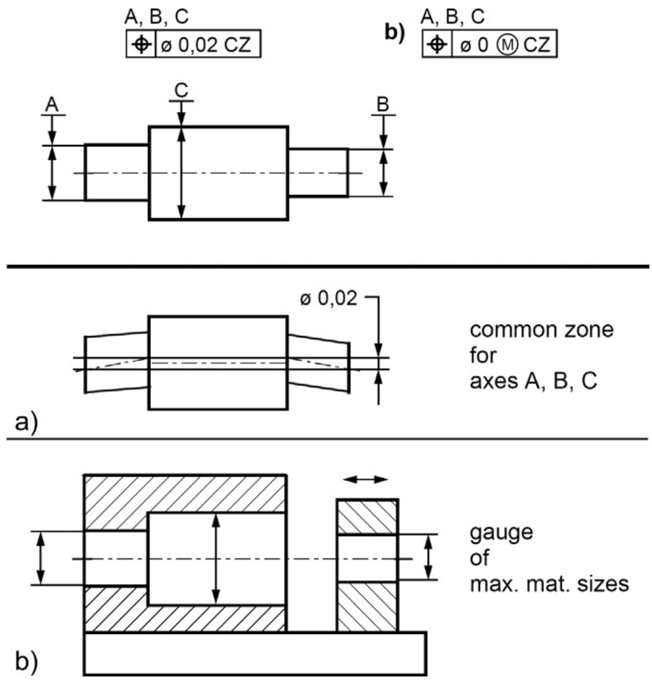


FIG. 5.42 Coaxial shafts

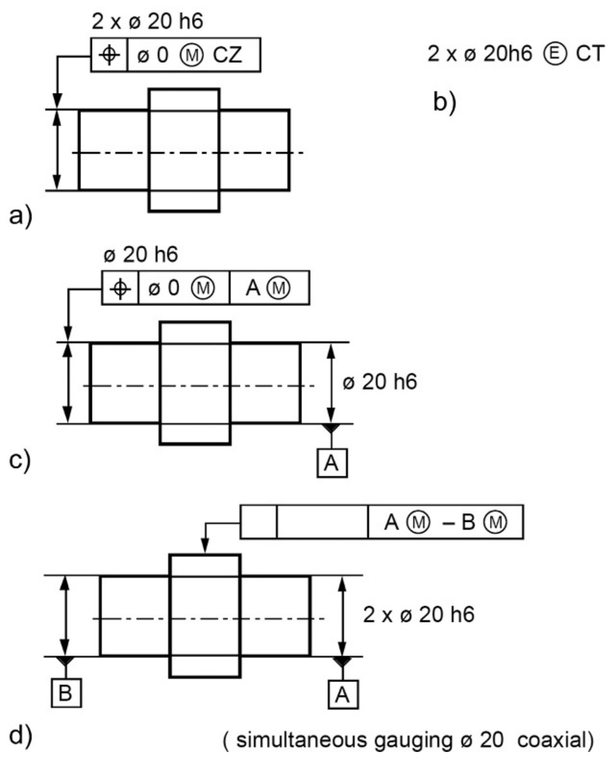


FIG. 5.43 Indications with same meaning

5.6.12 Coaxial holes (e.g. for hinges, bearings)

Figure 5.44 shows two possibilities for tolerancing coaxial holes in relation to a supporting surface (e.g. a hinge). On the right, the relevant inspection or gauge is shown. In the case below, the hinge will be adjusted in the assembly.

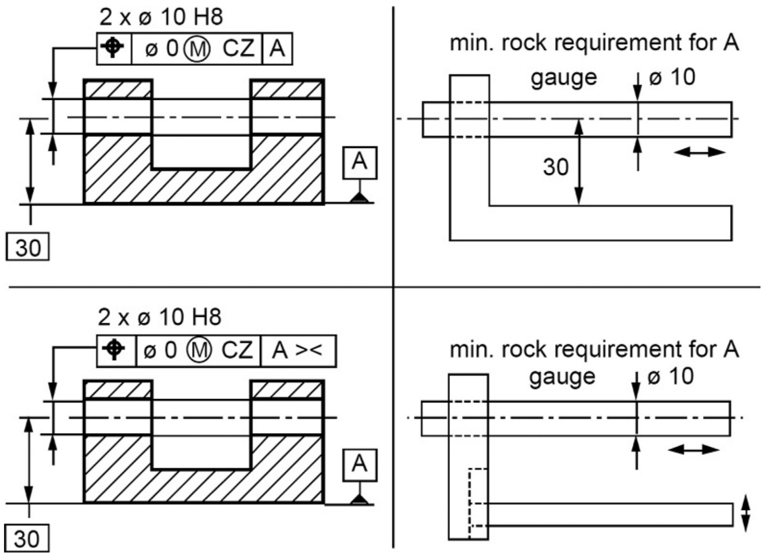


FIG. 5.44 Position tolerancing of the holes of a hinge

Figure 5.45 shows the tolerancing of the gap of the hinge to fit with the tab. The maximum material virtual condition is perpendicular to the datums A and B (minimum rock requirement).

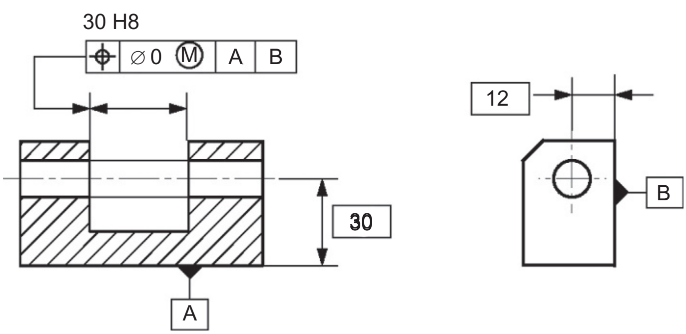


FIG. 5.45 Tolerancing of a hinge gap

Figure 5.46 shows the indication for coaxial holes with the same meanings.

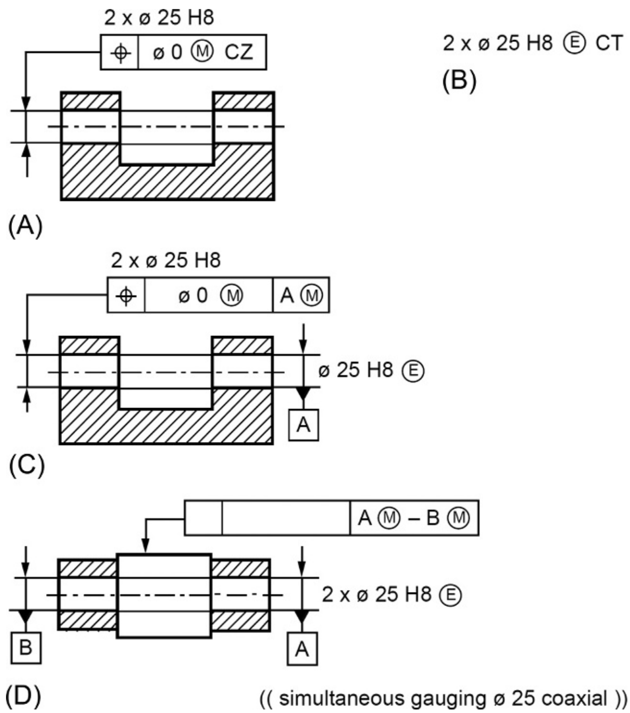


FIG. 5.46 Indications with same meaning

Figure 5.47 shows a hinged frame with holes ø 8 N9 (0/−0,036) Ⓔ that shall fit together with a bolt ø 8 h9 (0/−0036) Ⓔ (transition fit).

Figure 5.47 (lower) shows the extreme case when all actual sizes are at the worst limit. In this case, the coaxiality deviation (eccentricity) of the two holes is 0,01. The bolt is guided without clearance by the right hole, and must deflect when entering the left hole. Here the indication of the maximum material requirement Ⓜ would be detrimental because it would result in a greater deflection of the bolt. The maximum material requirement would allow larger coaxiality deviations with larger holes.

Inspection can be performed with two mandrels that fit into the holes without clearance.

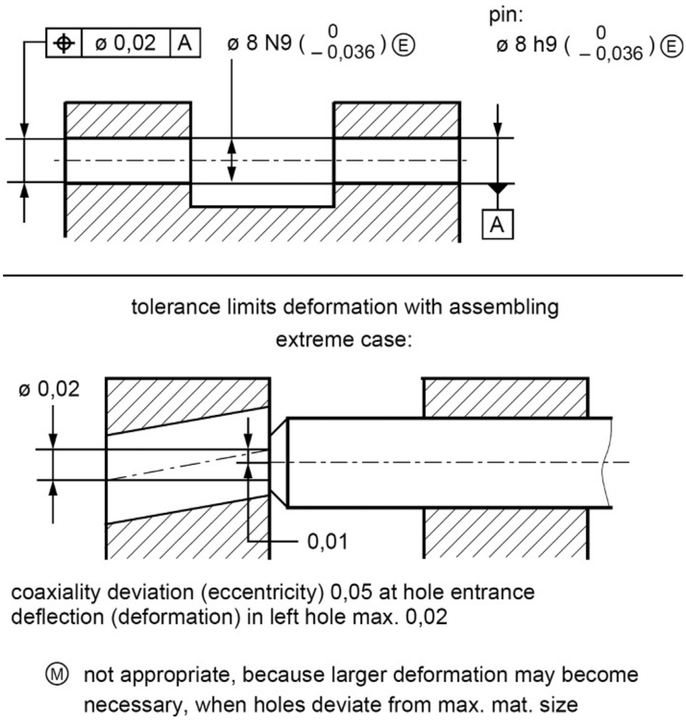


FIG. 5.47 Tolerancing of a hinge: transition fit

5.6.13 Hexagon fit

Figure 5.48 shows a hexagonal fit with a minimum clearance of 0,1. The total tolerance is indicated at the nominal size.

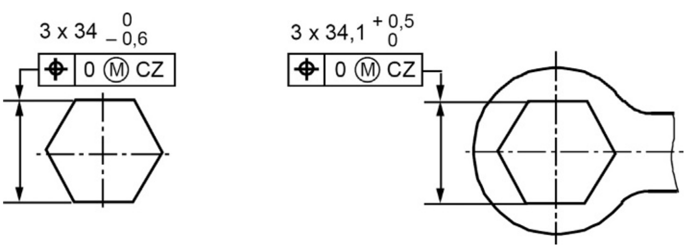


FIG. 5.48 Hexagonal fit with minimum clearance 0.1: total tolerance at the size

Figure 5.49 shows a similar fit using surface profile tolerances for the single plane surfaces.

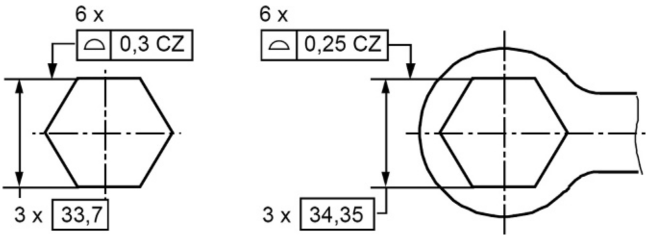


FIG. 5.49 Hexagonal fit with minimum clearance 0.1: surface profile tolerances for the single plane surfaces

Figure 5.50 shows the effect of these tolerances. The limitation at the maximum material limit (go gauge) is the same as in Fig. 5.48. At the minimum material limit, the geometrical ideal form and orientation apply according to Fig. 5.50, but not according to Fig. 5.49. The flatness deviation may occur up to 0,6 or 0,5, respectively (= size tolerance) according to Fig. 5.48. According to Fig. 5.49, when the features are at the least material virtual size (33,4 and 34,6), the surfaces must be perfectly plane. This may be an unnecessary restriction and could be avoided. Tolerancing according to Fig. 5.48 is recommended.

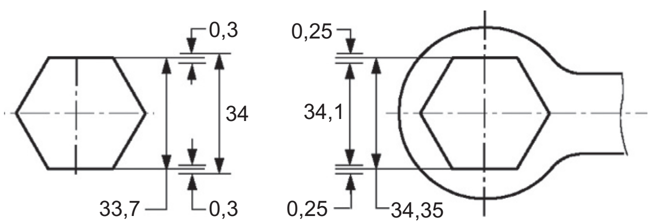


FIG. 5.50 Effect of position tolerancing according to Fig. 5.49

5.6.14 Splines

Figure 5.51 shows a spline fit. The total tolerance is indicated at the nominal size. Figure 5.52 shows the gauges.

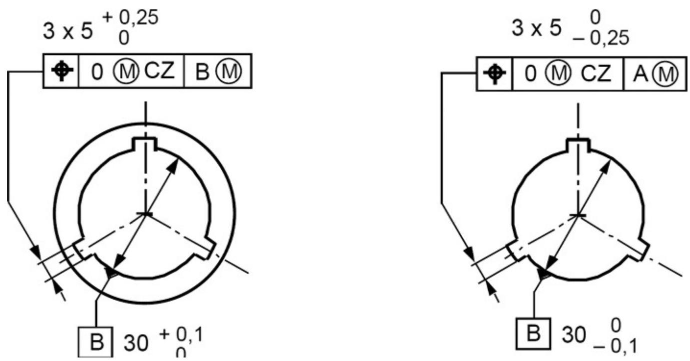


FIG. 5.51 Spline fit: total tolerance at the size

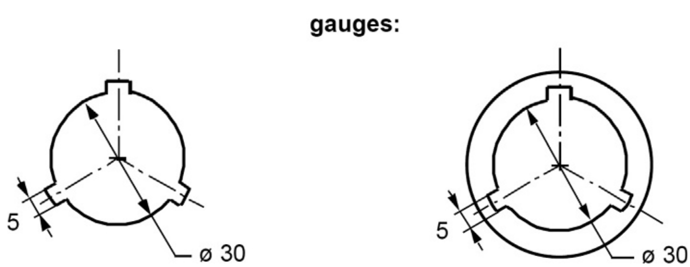


FIG. 5.52 Go gauges for D&T according to Fig. 5.51

5.6.15 Plug and socket

Figures 5.53 and 5.54 show a plug and socket fit where the socket has a frame and forms a further fit with the plug circumference, which is related to the fits of the pins and holes. At the frame is a minimum clearance of 0,5, that is the difference between the maximum material sizes.

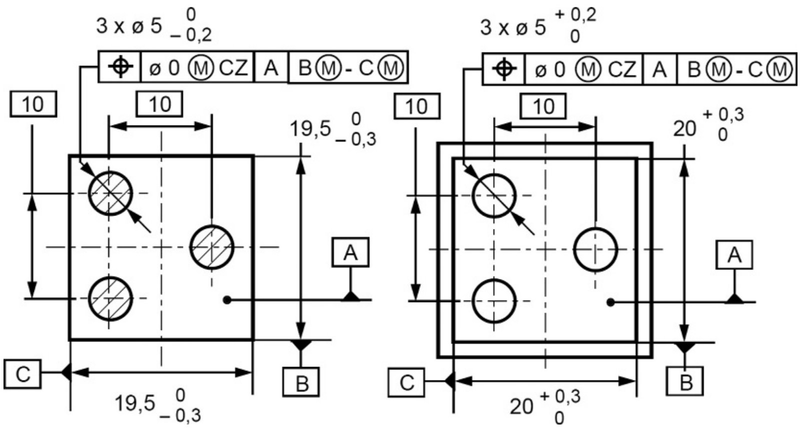


FIG. 5.53 Plug and socket fit with frame fit: total tolerance at the sizes

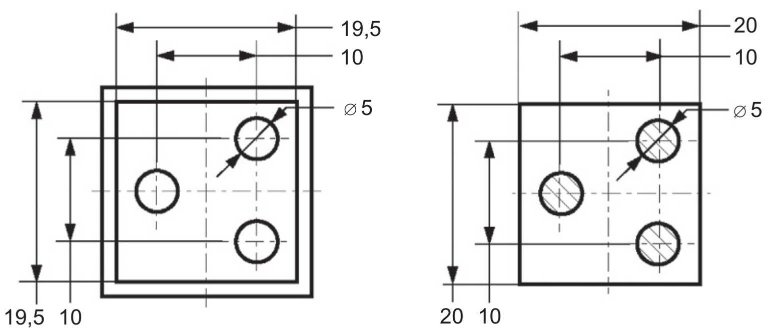


FIG. 5.54 Go gauges for GD&T according to Fig. 5.53

Figure 5.54 shows the go gauges for GD&T according to Fig. 5.53.

5.6.16 Spacings

According to Fig. 5.55, the median surfaces of the slots shall not deviate by more than ±0.025 from the symmetrical position in relation to the axis of the datum hole B and the theoretically exact spacing. The position tolerance does not depend on the actual sizes of the slots and the hole.

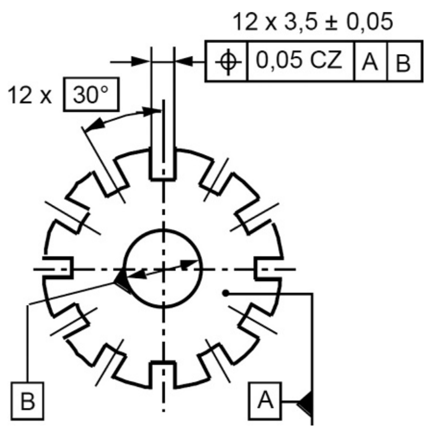


FIG. 5.55 Spacing: position toleranced slot median face

According to Fig. 5.56, slots and hole must fit into a geometrically ideal counterpart (go gauge) of maximum material size at the hole and maximum material size (3,4) at the slots. With this requirement the position tolerance depends on the actual sizes and actual forms of the slots and of the hole.

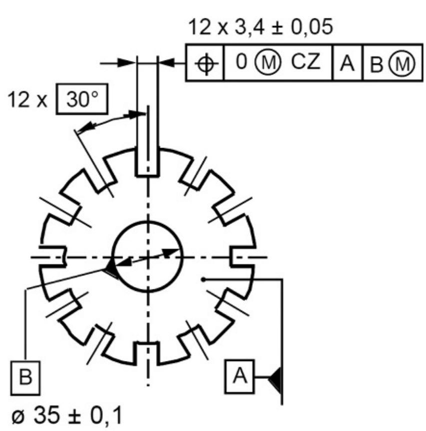


FIG. 5.56 Spacing: position toleranced with Ⓜ

5.6.17 Minimum wall thickness

When the function requires a minimum wall thickness, and the wall thickness and position of the wall depend on each other, then the permissible position deviation of the wall becomes larger the more the wall deviates from its least material size (the more the wall becomes thicker).

Figure 5.57 shows tolerancing to ensure a minimum wall thickness (2,705) between hole and recess. When the hole is at its least material size (ø 4,2), the hole axis must be contained in a tolerance zone ø 0,25. In the worst case, when the hole axis has the largest position deviation, the remaining wall thickness is 2,705 (Fig. 5.58 left). When the hole is at its maximum material size (ø 3,95), the tolerance zone of the hole axis increases by the amount of the size tolerance (ø 0,25) to become ø 0,5. Then, also, in the worst case the remaining wall thickness is 2,705 (Fig. 5.58 right).

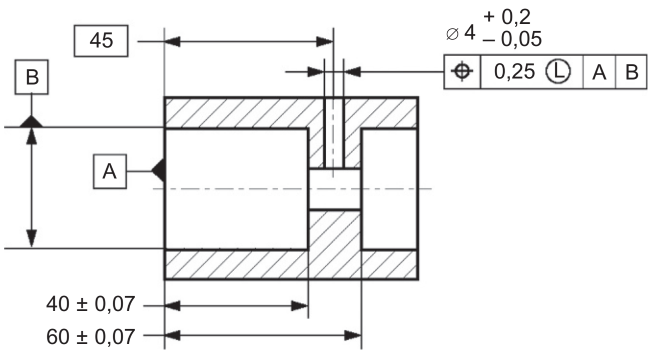


FIG. 5.57 Position tolerance with least material requirement to ensure a minimum wall thickness

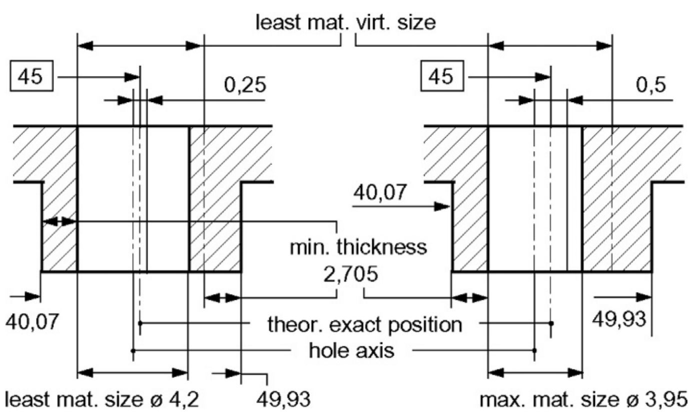


FIG. 5.58 Calculation of the minimum wall thickness 2,705 for tolerancing according to Fig. 5.57: left: hole least material size; right: maximum material size

In other words, the hole must respect (it must be contained within the material) the least material virtual cylinder of diameter given by the least material size plus the position tolerance (4,2+0,25=4,45) that is in the theoretically exact position. Between the least material virtual cylinder and the recesses, there remains a minimum wall thickness of 2,705 (Fig. 5.58).

Figure 5.59 shows tolerancing to ensure a minimum wall thickness of 8,5. The permissible position deviation of the inner cylinder increases when the diameter of the inner cylinder decreases. The permissible position deviation of the outer cylinder increases when the diameter of the outer cylinder increases.

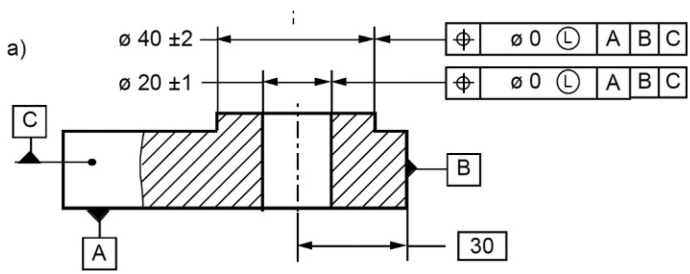


FIG. 5.59 Position tolerances with least material requirements to ensure a minimum wall thickness

Figure 5.60 show the theoretically exact position of the least material virtual cylinders that must be respected by the workpiece surfaces.

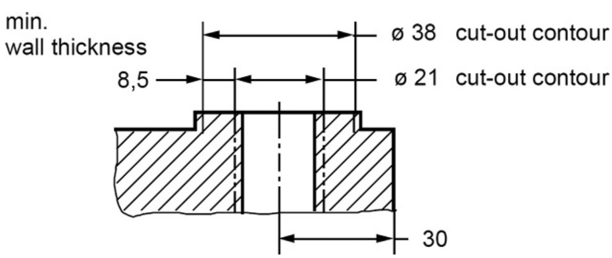


FIG. 5.60 Calculation of the minimum wall thickness 8,5 for tolerancing according to Fig. 5.59

5.6.18 Keyways

With **floating keys** (key slot width e.g. D10) is the position tolerance; see Fig. 5.61,

$$t = \text{min . slot width} - \text{max . key width.}$$

This corresponds with floating fasteners, see Fig. 5.13.

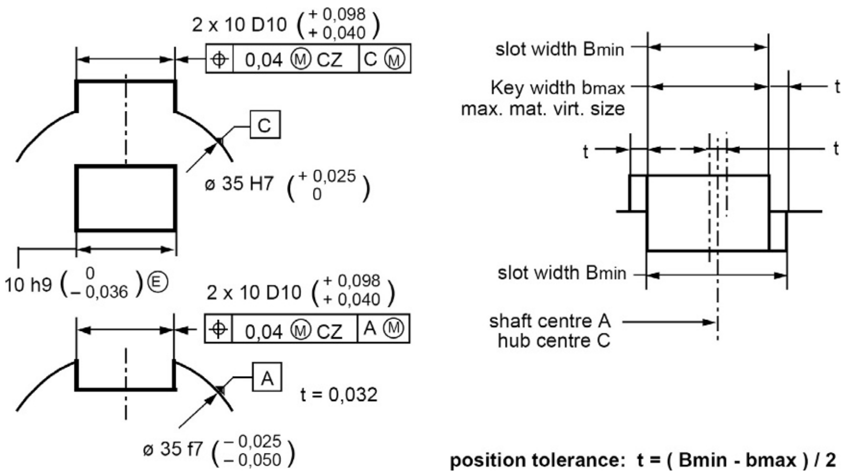


FIG. 5.61 Keyways for two floating keys

With **fixed keys** (key slot width e.g. P9) is the position tolerance, see Fig. 5.62,

$$t = (\text{min . slot width} - \text{max . key width})/2$$

This corresponds with fixed fasteners, see Figs. 5.14 and 5.28. With only one key the position tolerance is

$$t' = t + (\text{minimum clearance shaft \& hub})/2$$

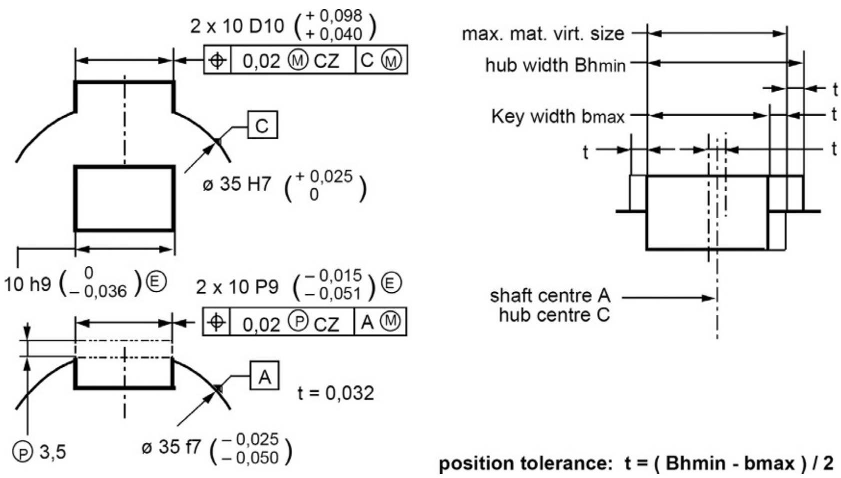


FIG. 5.62 Keyways for two fixed keys

With **screwed keys** (in order to prevent from falling off when the slot is open at the end) is the most detrimental case when the slot has least mat. size. Then the key has the largest distance from the shaft centre. The smaller the slot width the larger is the position tolerance. This corresponds to the least material requirement. See Fig. 5.63.

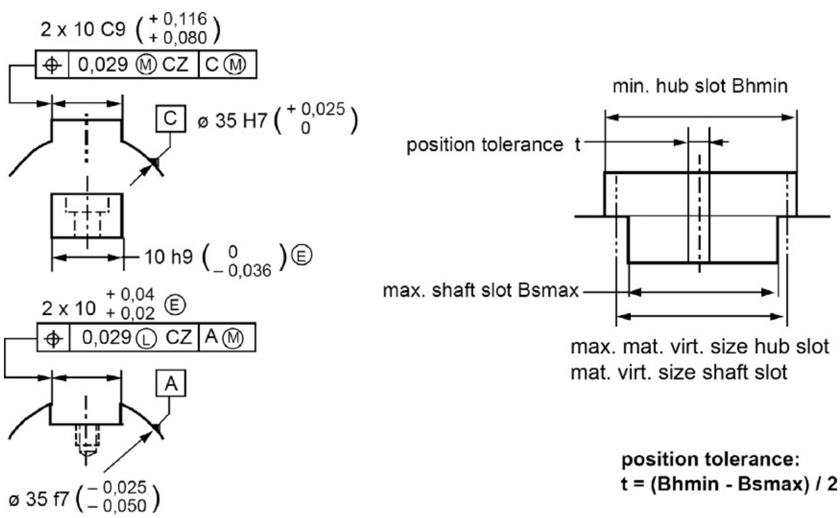


FIG. 5.63 Keyway for two screwed keys

Figures 5.64–5.67 show tolerancings with only one key.

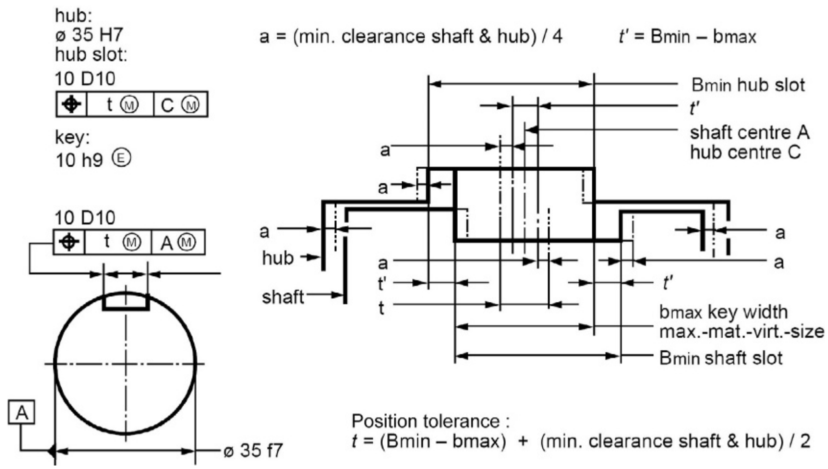


FIG. 5.64 Additional position tolerance 2a = (minimum clearance between shaft and hub diameter) / 2

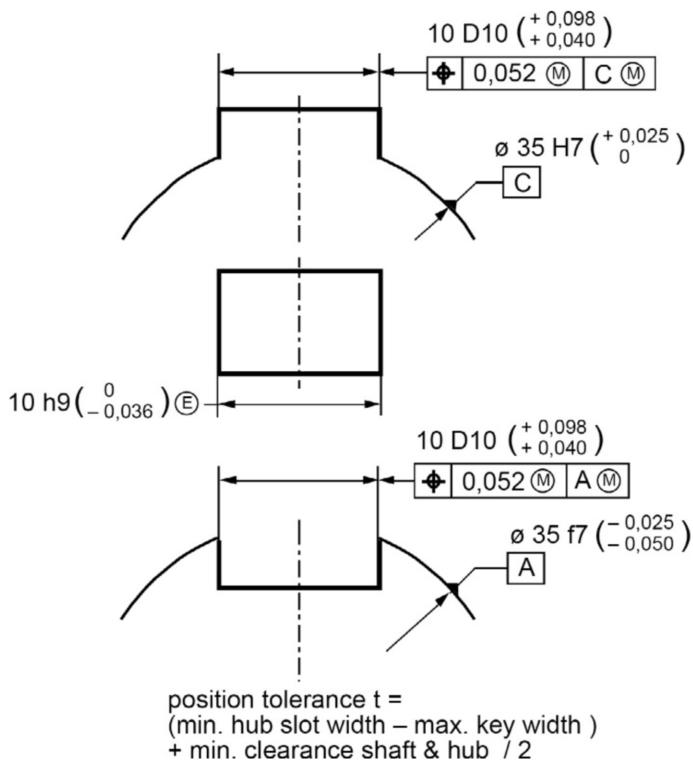


FIG. 5.65 Tolerancing of a key way for a floating key

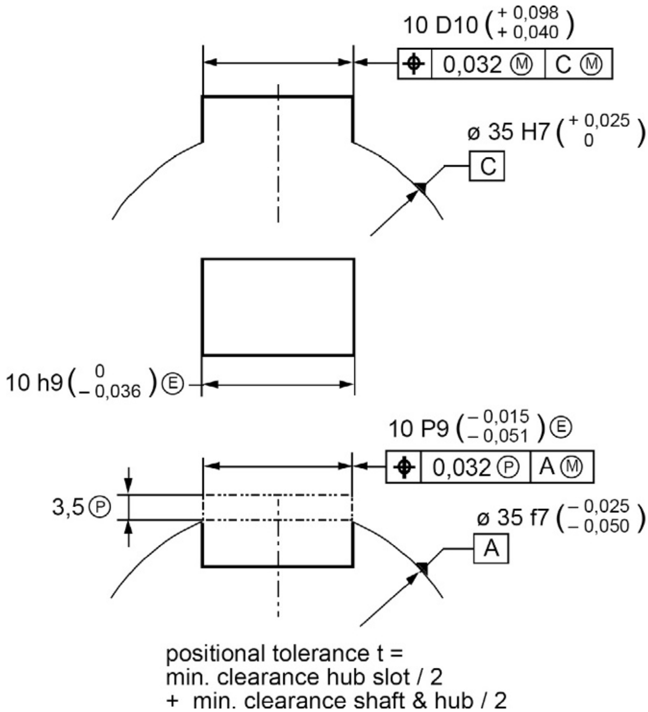


FIG. 5.66 Tolerancing of a key way for a fixed key

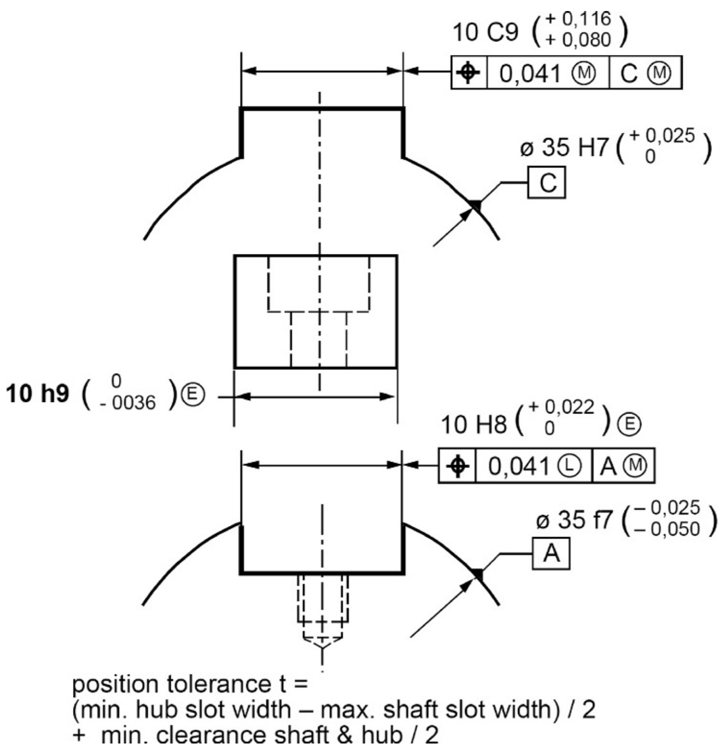


FIG. 5.67 Tolerancing of a key way for a screwed key

When there is only one key and a minimum clearance between shaft and hub (clearance fit), each slot can have an additional eccentricity component of 1/4 of the minimum clearance between shaft and hub or an position tolerance component of 1/2 of the minimum clearance between shaft and hub (Fig. 5.65). Then the clearance between shaft and hub allows additional position deviation. The position tolerance is then

$$t' = t + (\text{minimum clearance shaft \& hub})/2.$$

When the minimum clearance = 0 between key way and key is acceptable, the 0 Ⓜ method, similar as in Fig. 5.45, may be used.

Figure 5.68 shows an example of a key assembly with interference fits at the key and at the shaft & hub. Here all geometrical deviations lead to deformations of the parts. Figure 5.68 shows the extreme possible deviations.

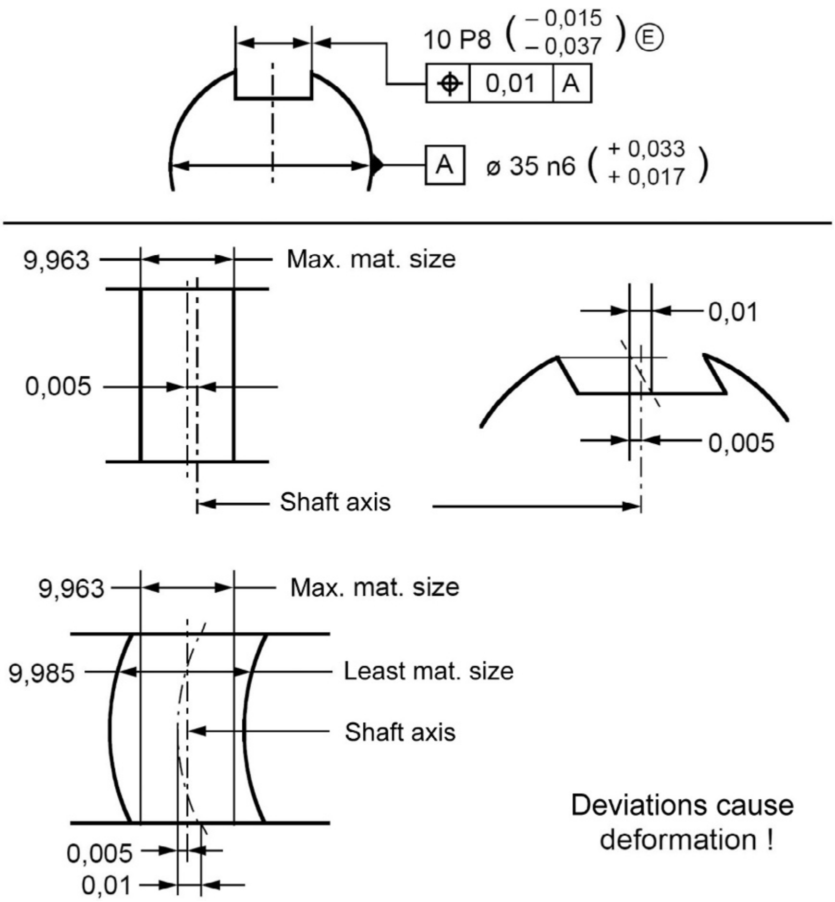


FIG. 5.68 Key assembly with interference fits

5.6.19 Rings and bushes

Figure 5.69 shows tolerancing of a ring or bush for a clearance fit.

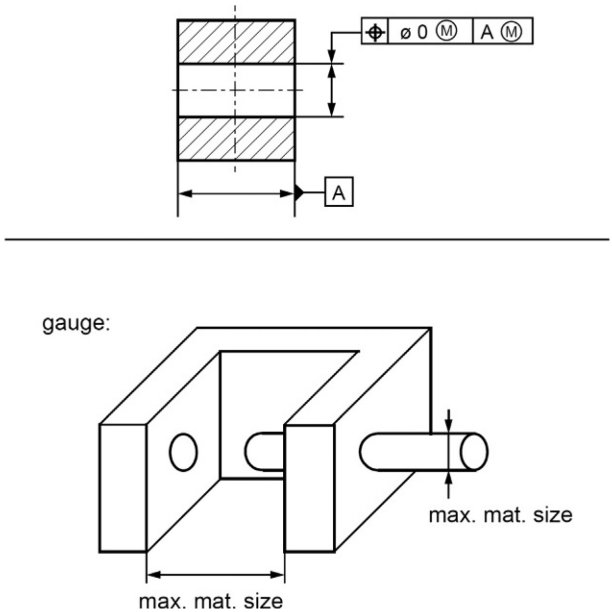


FIG. 5.69 Tolerancing of a ring or bush for a clearance fit

Figure 5.70 shows tolerancing of the perpendicularity.

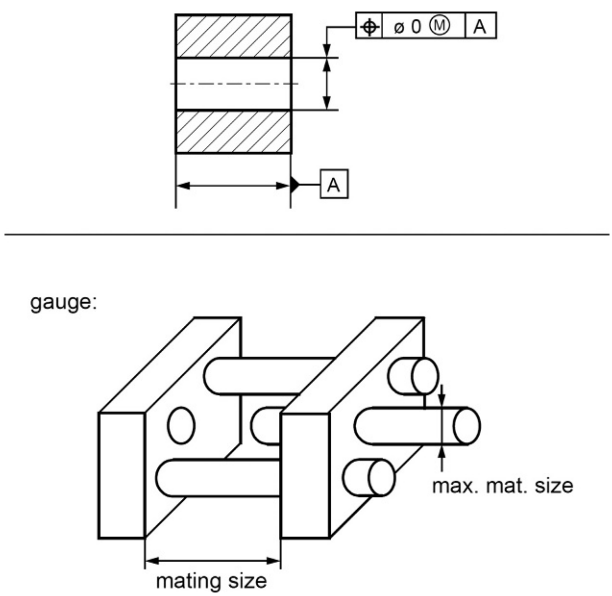


FIG. 5.70 Tolerancing of the perpendicularity

Figures 5.71 and 5.72 show tolerancing of bushes with everywhere clearance fits.

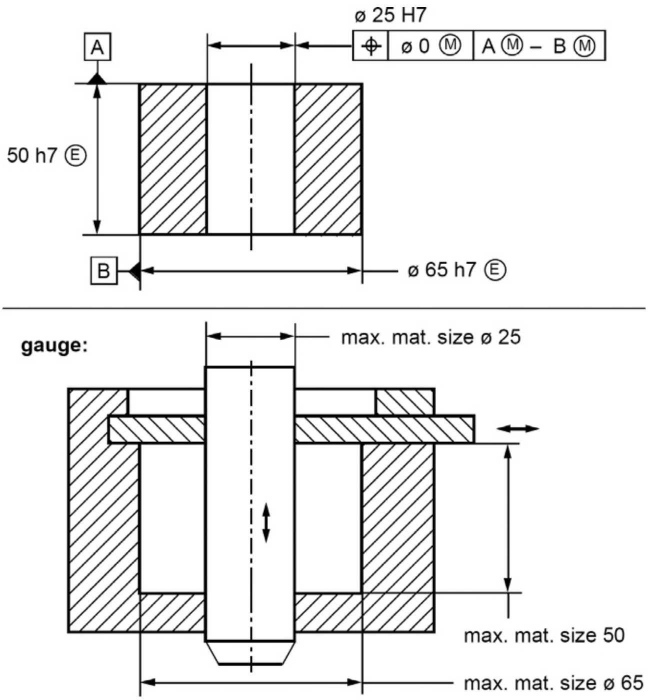


FIG. 5.71 Tolerancing of a bush with everywhere clearance fits

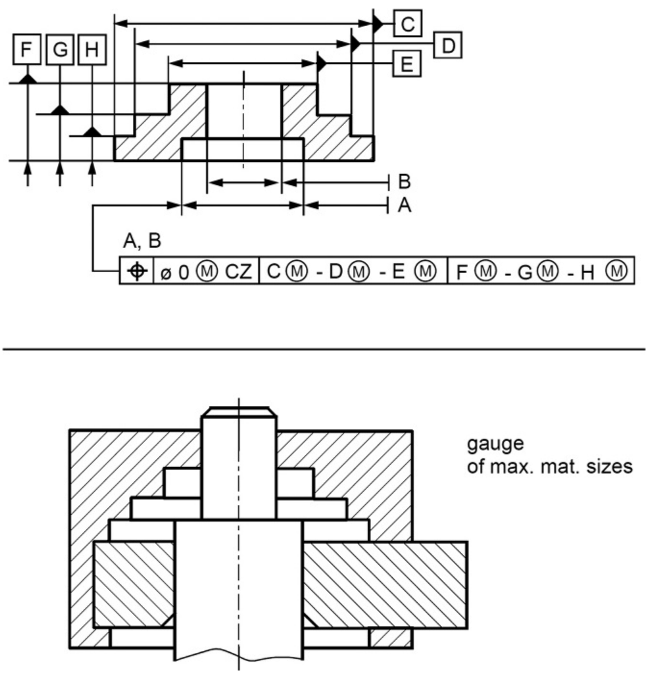


FIG. 5.72 Tolerancing of a bush with everywhere clearance fits

5.6.20 Rectangular fit

Figure 5.73 shows position tolerancing of a rectangular fit, e.g. a cover sheet in a casing. Compare this with Fig. 6.28 profile tolerancing. The maximum possible form deviation is twice as much as with profile tolerancing.

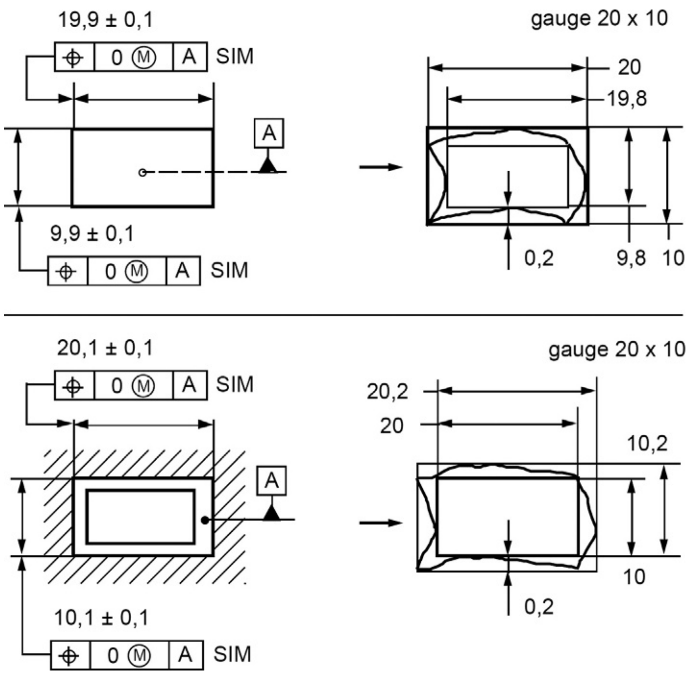


FIG. 5.73 Rectangular fit

5.6.21 Kinematics

Figure 5.74 shows tolerancing according to the kinematics of a gearbox and tolerancing for fitting the gears.

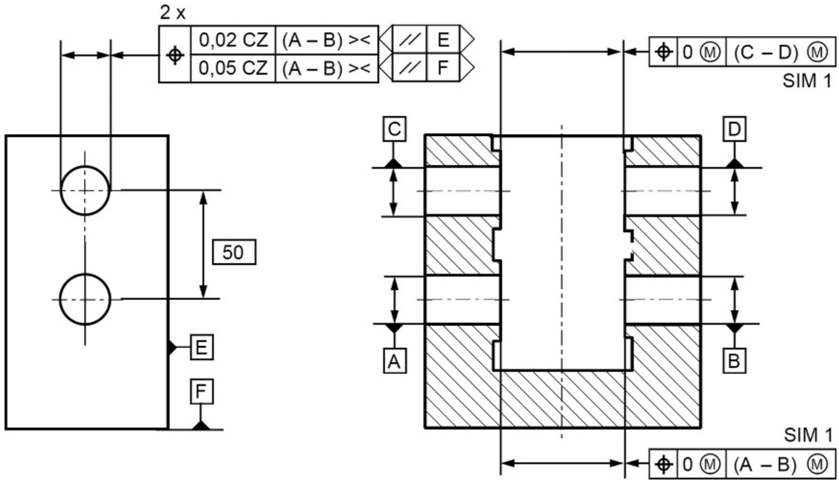


FIG. 5.74 Tolerancing with and without Ⓜ, gearbox

Figure 5.75 shows the more complete tolerancing of the gearbox.

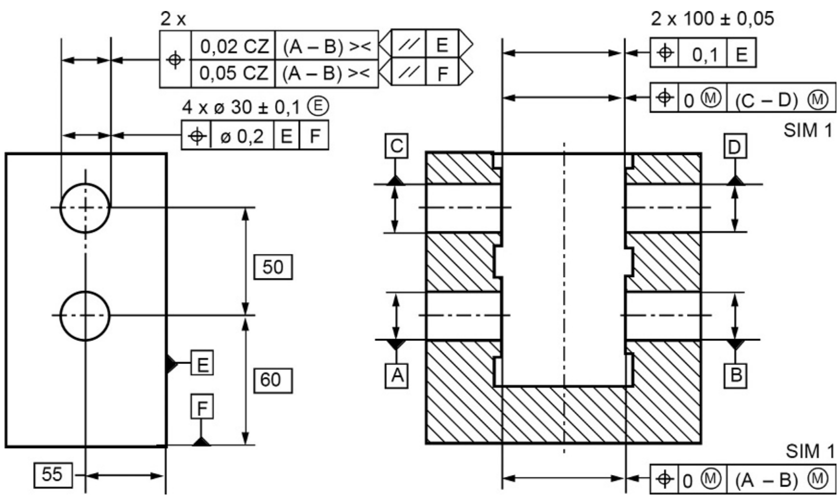


FIG. 5.75 Tolerancing with and without Ⓜ, gearbox, complete tolerancing

5.6.22 Maximum material requirement, application

Figure 5.76 shows the application of the maximum material requirement.

See also 4.5 and Figs 5.45 and 5.74.

maximum material requirement Ⓜ

---

necessary:

with clearance fits for assembly without deformation

(for separate features Ⓔ instead of Ⓜ permissible)

---

forbidden:

with interference fits, transition fits

with kinematics and optics

Ⓜ would allow larger deviations of orientation, location
when mating size deviates from max. mat. size

e.g. for

distance of axes, pitch diameter
inclination of axes
crossing of axes

FIG. 5.76 Maximum material requirement, application

Chapter 6

Profile Tolerancing

6.1 Definitions

Profile tolerances for integral features are defined in ISO 1101, and profile tolerancing is described in ISO 1660. See 3.2.3.1 to 3.2.3.4.

With profile tolerancing, a distinction must be made between tolerancing of the form of (section) lines (symbol ⌒) and of the form of surfaces (symbol ⌓).

The nominal (theoretically exact, geometrically ideal, true) form is to be defined by theoretically exact (rectangular framed) dimensions with (Fig. 6.2) or without (Fig. 6.1) relation to datum(s).

For planes and features of size with variable size (± tolerance), the nominal form (without size) is defined by the drawing outline alone or the CAD model. Then the size (TED, reference feature) is defined by the minimax (Chebyshev) criterion, with or without reference to datum(s) (Figs 6.1 and 6.2).

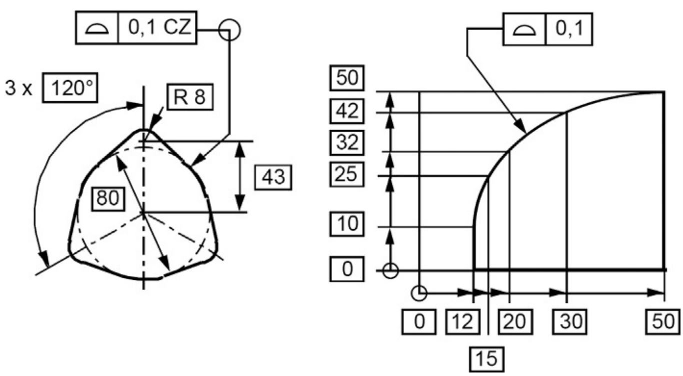


FIG. 6.1 Profile surface tolerances; the nominal form (reference feature) is defined by consecutive planes and cylinders defined by theoretically exact dimensions without reference to datums (left) or by TEDs of splines (right)

The tolerance zone of a profile tolerance is defined by (tangential) envelopes on circles (profile tolerance of a line) or on spheres (profile tolerance of a surface) whose diameters are equal to the tolerance value and centred on the nominal form (Fig. 3.25). Therefore the zone is equally disposed on either side of the nominal profile (reference feature).

Geometrical Dimensioning and Tolerancing for Design, Manufacturing and Inspection
https://doi.org/10.1016/B978-0-12-824061-8.00010-4

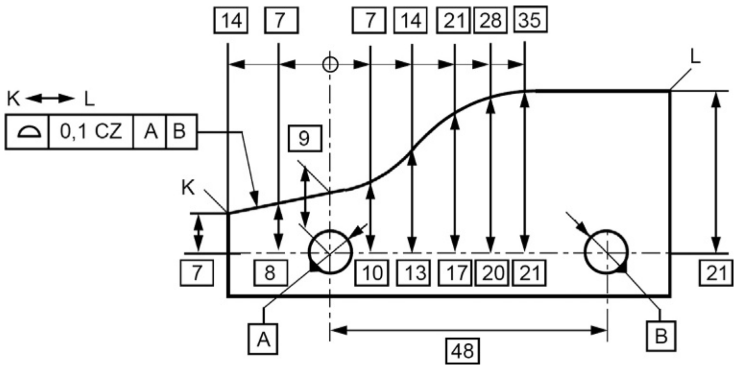
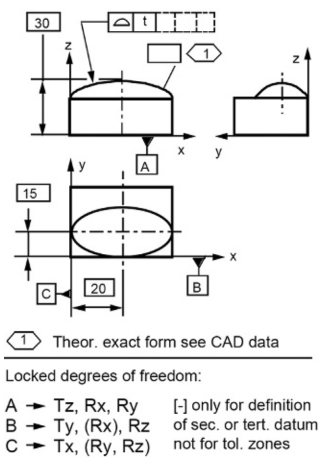


FIG. 6.2 Profile surface tolerance; the nominal form is defined by theoretically exact dimensions with reference to datums (A, B)

The form of the envelope of the circles or spheres (tolerance zone) between the specified points (e.g. splines) is not standardized. Clearly, the envelope shall alter its form in a smooth manner. Programs for CAD (computer aided design) and CMMs (coordinate measuring machines) usually use splines.

Figure 6.3 shows the tolerancing possibilities of a free-form (complex) surface. The upper tolerance (0,3) related to the complete datum system ABC, locking all degrees of freedom of the part, is mandatory. Without this tolerance the part is not completely toleranced.



No.	Drawing indications					Locked degrees of freedom: Translations			Locked degrees of freedom: Rotations		
						Tx	Ty	Tz	Rx	Ry	Rz
(a)	⌓	0,3	A	B	C	X	X	X	X	X	X
(b)	⌓	0,1	A	B ><	C	X		X	X	X	X
(c)	⌓	o,1	A ><	B	C	X	X		X	X	X
(d)	⌓	0,1	A	B			X	X	X	X	X
(e)	⌓	0.1	A ><	B ><	C	X			X	X	X
(f)	⌓	0,1	A ><	B			X		X	X	X
(g)	⌓	0,1	A	B ><				X	X	X	X
(h)	⌓	0,1	A ><	B ><					X	X	X
(J)	⌓	0,1	A [-]	B [-]	C	X				X	X
(k)	⌓	0,1	A [-]	B			X		X		X
(l)	⌓	0,1	A					X	X	X	
(m)	⌓	0,1	A [-]	B [-]	C ><					X	X
(n)	⌓	0,1	A [-]	B ><					X		X
(o)	⌓	0,1	A ><						X	X	
(p)	⌓	0,1	A [-]	B [Rz]							X
(q)	⌓	0,1	A [Rx]						X		
(r)	⌓	0,1	A [Ry]							X	
(s)	⌓	0,1									

FIG. 6.3 Complex feature, possibilities of profile tolerancing

Figure 6.4 shows the possibilities of profile tolerancing of a pattern (group) of planar features. The upper tolerance (0,6) related to the complete datum system RST, locking all degrees of freedom of the part, is mandatory. Without this tolerance the part is not completely toleranced.

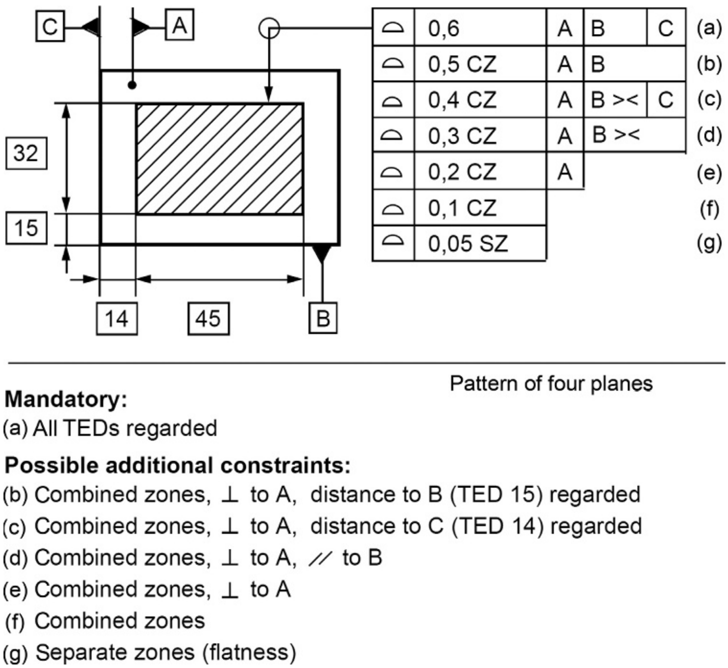


FIG. 6.4 Pattern of planar surfaces, possibilities of tolerancing

Figure 6.5 shows profile tolerancing with UZ (unequally distributed tolerance zones). The value after UZ defines the offset of the zone centre from the TEF (theoretically exact form). Positive values mean offset outward and negative values mean offset inward of the material. When the values are negative and of half the tolerance, part and counterpart have the same CAD model (unilateral tolerance zones); see Fig. 6.6.

Figure 6.7 shows UZ for a complex form. The offset is defined by spheres along the TEF.

Figure 6.8 shows a former practice of UZ tolerancing.

Figure 6.9 shows OZ (offset zone of equal but undefined distance from the TEF). Spheres SØ 0,1 define the width of the tolerance zone. Spheres, with SØ variable but constant along the feature, define the distance from the TEF.

Figure 6.10 shows profile tolerancing where the distance from datum C is variable (>< orientation only). The tolerance zone centre keeps the form of the TEF.

Figure 6.11 shows OZ applied to a cylindrical feature. Above, the OZ zone is centred at datum A; below, the OZ zone can have any location and orientation within the 0.4 zone. See also Figs. 3.28 and 3.46. The OZ zone in Fig. 6.9 is slightly different from the zone in Fig. 6.10.

The difference between OZ with and without a TED for the diameter is shown in Figs 3.29, 3.37 and 3.47.

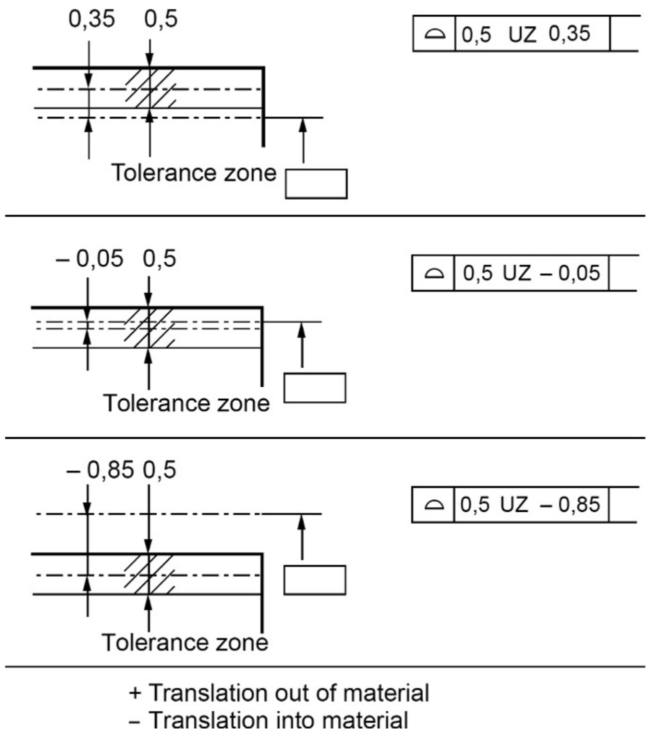


FIG. 6.5 Profile tolerancing with UZ

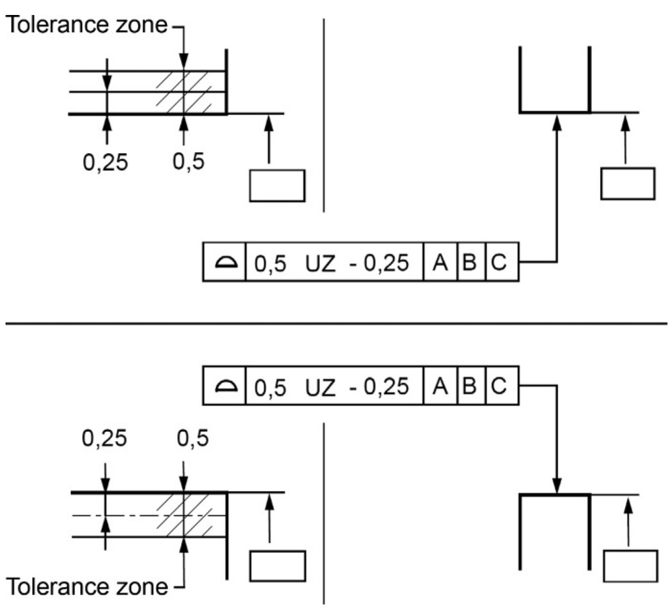


FIG. 6.6 UZ for using the same CAD model for part and counterpart

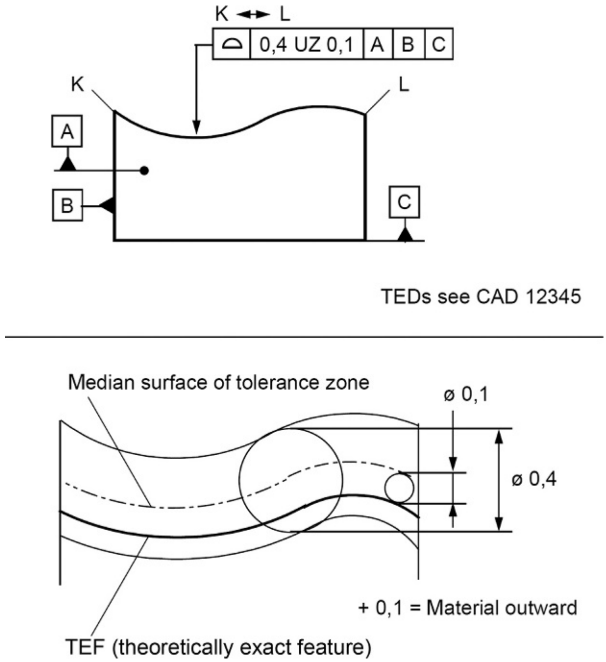


FIG. 6.7 UZ with a complex form

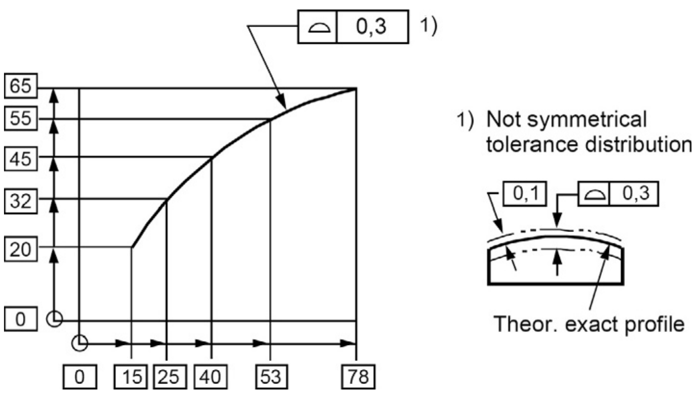


FIG. 6.8 Unequally disposed profile tolerance zone

The difference between OZ and VA in the case of a cone or wedge is shown in Fig. 3.49. With OZ, the angle remains as a TED. With VA, the angle of the tolerance zone is variable and defined by the Chebyshev criterion (minimax).

Figure 6.12 shows left profile tolerancing with variable width. The width varies in a proportional variation, i.e. the distance along the curve (zone width 0,2 left and 0,1 right). On the right is shown tolerancing with restricted length. The reference feature is oriented according to Chebyshev.

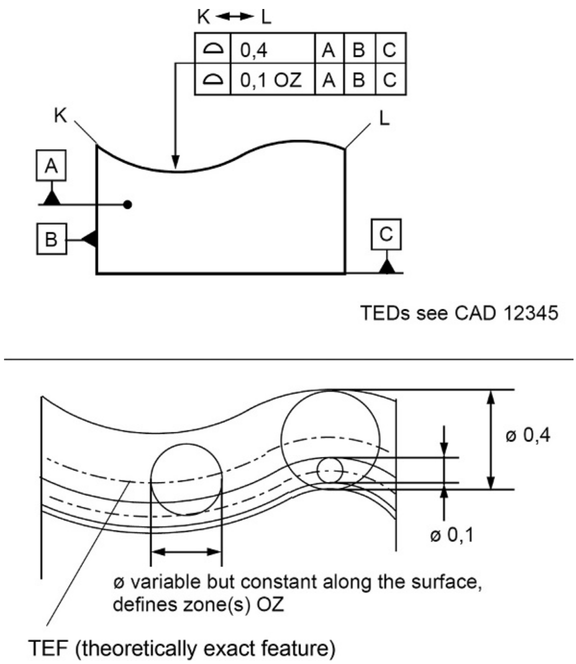


FIG. 6.9 Profile tolerancing with OZ (offset zone of equal but undefined distance from the TEF)

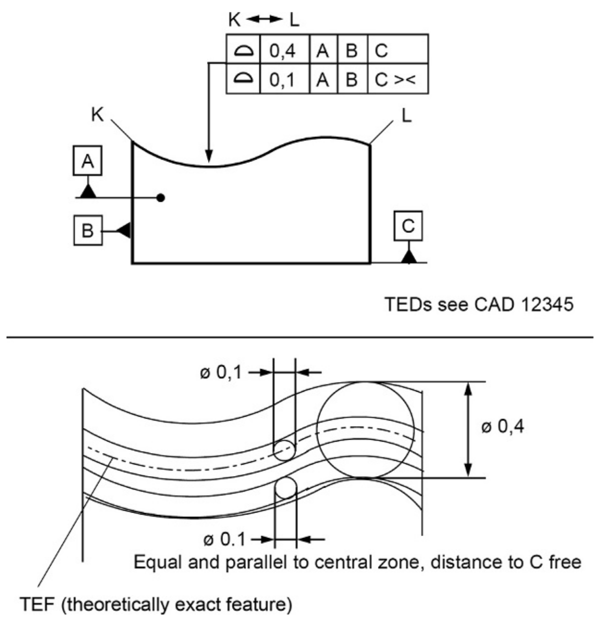


FIG. 6.10 Profile tolerance orientation only (instead of OZ)

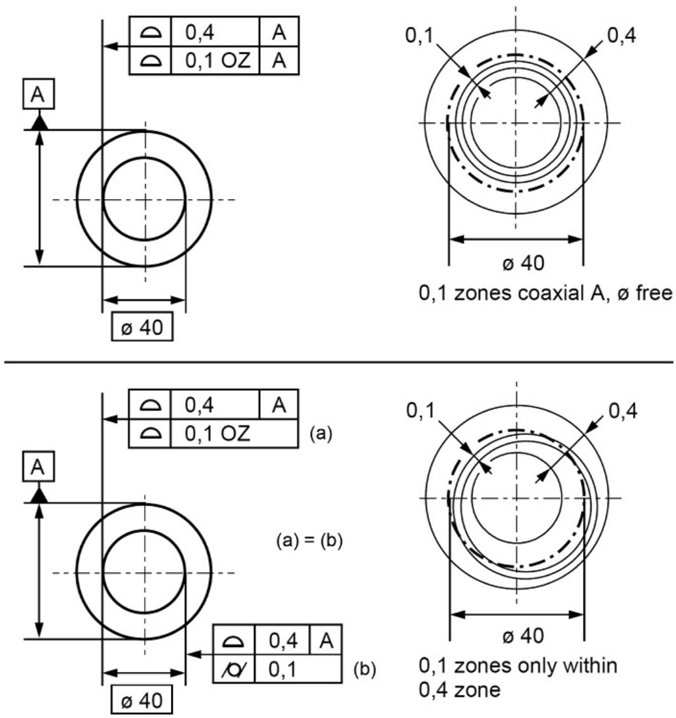


FIG. 6.11 OZ for a cylindrical feature

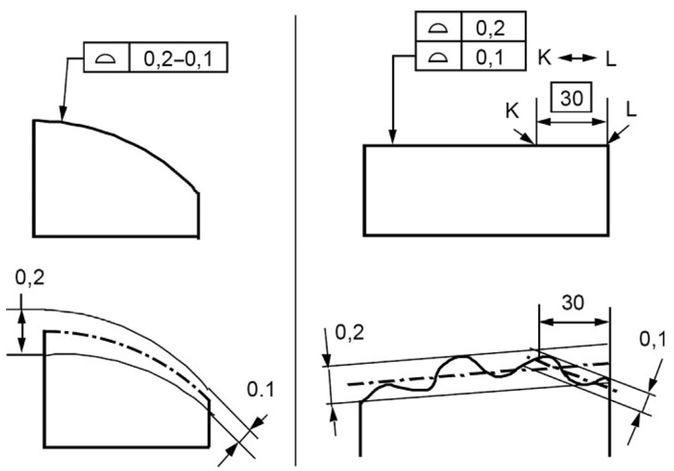


FIG. 6.12 Profile tolerance with variable width and restricted length

Profile tolerancing can be used for tolerancing of form, orientation, location and form with size, depending on the indication of datums and theoretically exact dimensions (Fig. 6.13).

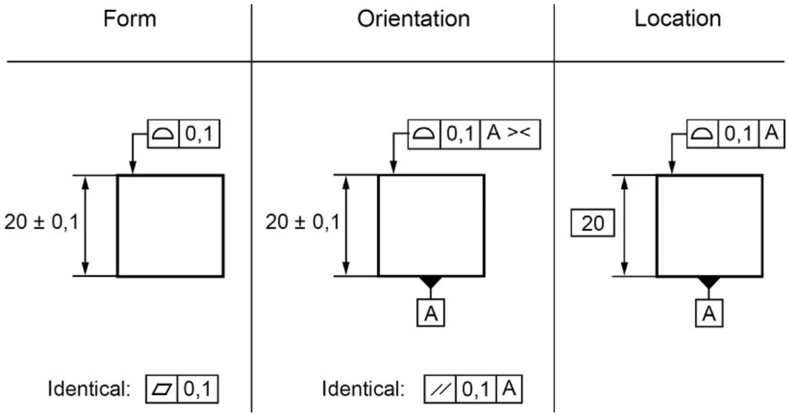


FIG. 6.13 Profile tolerance of surfaces for tolerancing form, orientation and location

The symbol UF (united features) unites the theoretical profiles (reference features). This is in contrast to CZ, which unites the tolerance zones. The difference is that the outer corners of the tolerance zone are rounded. See Fig. 6.14.

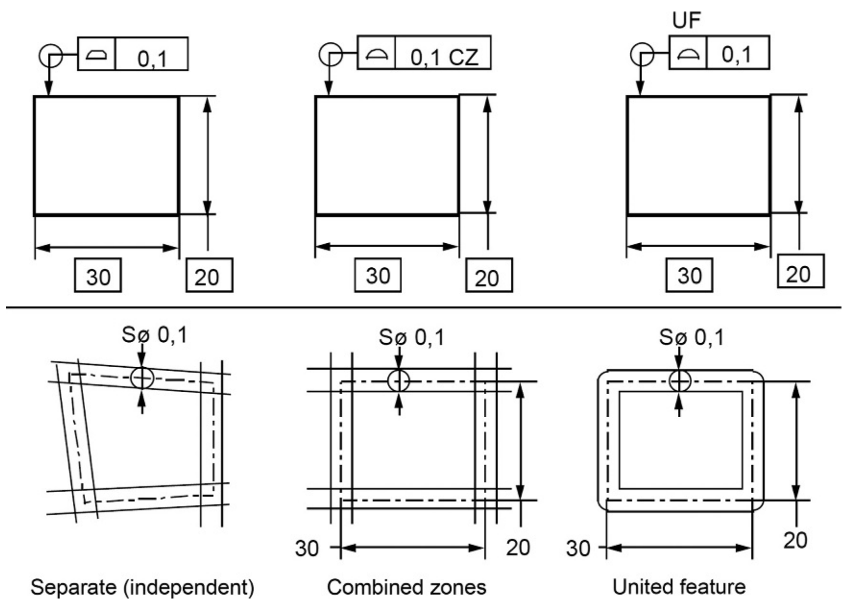


FIG. 6.14 Differences between SZ, CZ and UF

6.2 Further examples

6.2.1 Complex surface

Figure 6.15 shows a profile surface tolerance 0,6 related to a complete datum system (locking all degrees of freedom of the workpiece). Further, possible additional constraints are indicated. The tolerance 0,1 is a combination of orientation (related to A) and location (related to B and C) specification. The tolerance 0,05 is a form specification. The tolerance 0,02 is a specification for section lines in section planes parallel to datum B.

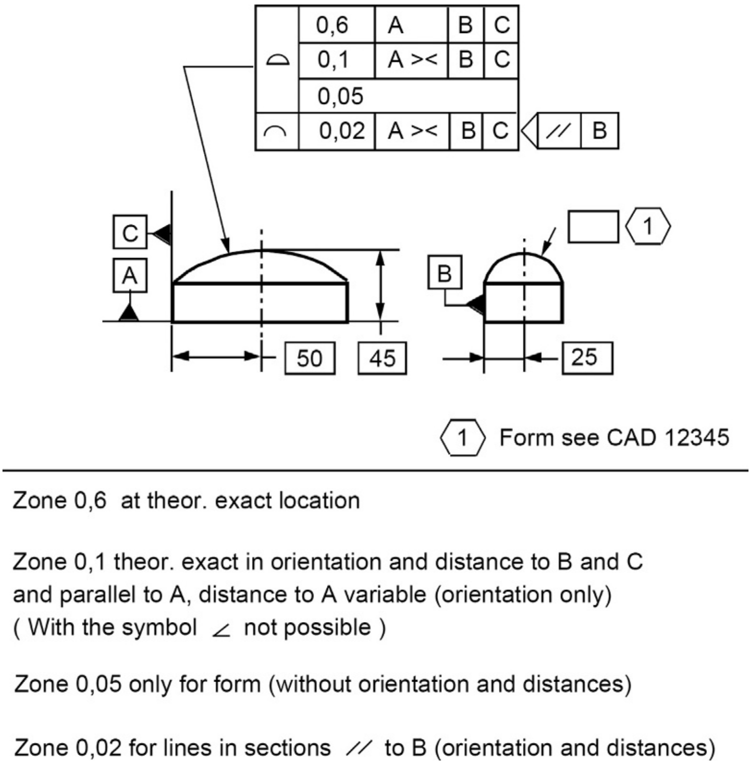


FIG. 6.15 Profile tolerancing for a complex surface with possible additional constraints

ISO 1101 does not explicitly describe whether the datum in the intersection plane indicator is of the same order as in the tolerance indicator. A clear indication is using the future symbol [-] for datums only for the definition of secondary or tertiary datums; see Fig. 6.3. However, the difference between the intersection lines of intersection planes of different order is very small.

6.2.2 Window

Figure 6.16 shows profile surface tolerancing of a window with additional constraints for orientation and form.

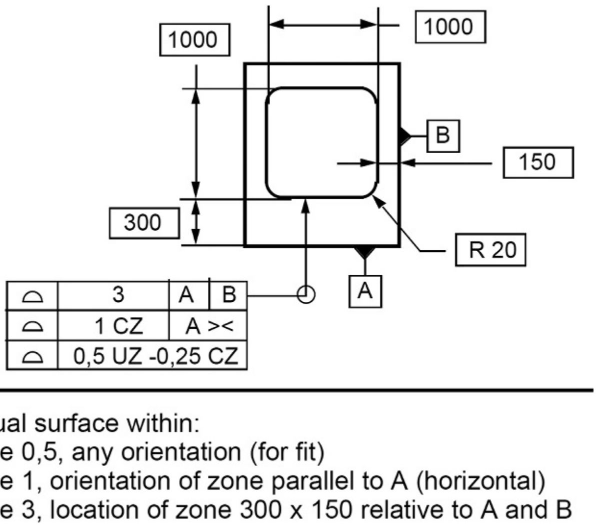


FIG. 6.16 Profile tolerancing of a window with possible additional constraints

6.2.3 Perpendicularity

Figure 6.17 shows a sheet metal part. The upper tolerancing does not limit the deviation of the side faces, but the lower tolerancing does.

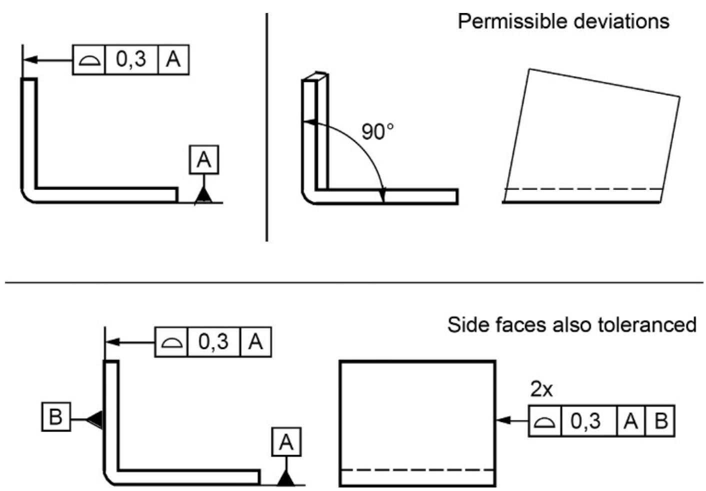


FIG. 6.17 Tolerancing of perpendicularity of a sheet metal part

6.2.4 Groups

Figure 6.18 shows tolerancing of a group of a group. The first CZ or SZ refers to the first order group (group of two rectangular holes) and the second to the second order group (group of four planes).

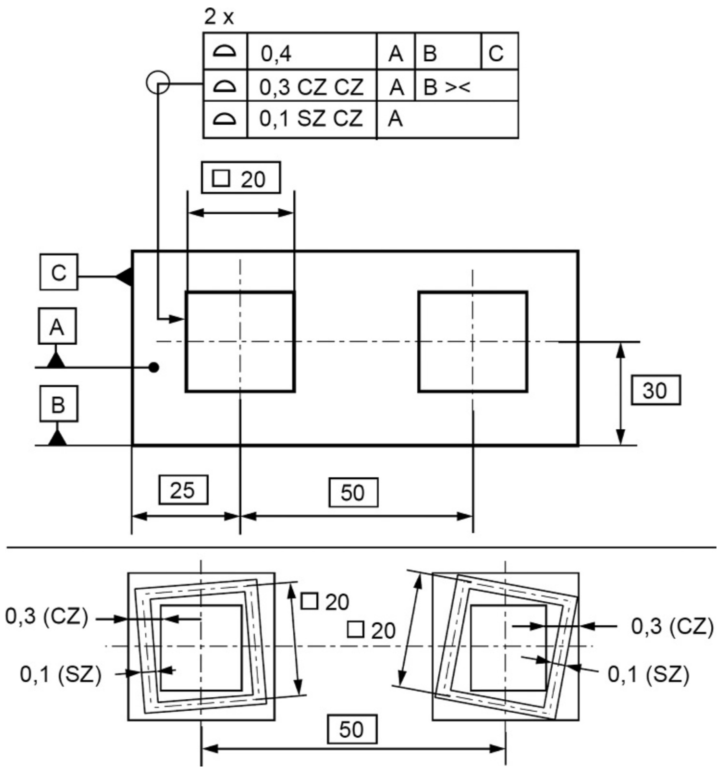


FIG. 6.18 Tolerancing of groups

6.2.5 Inclined surface

Figure 6.19 shows tolerancing of an inclined plane. The tolerancings (b) and (c) are allowed, but not recommended.

6.2.6 Stock material

Figure 6.20 shows tolerancing of stock material.

6.2.7 Wedges

Figure 6.21 shows tolerancing of a wedge (two inclined planes). The theoretically exact position is determined by the datums A and B.

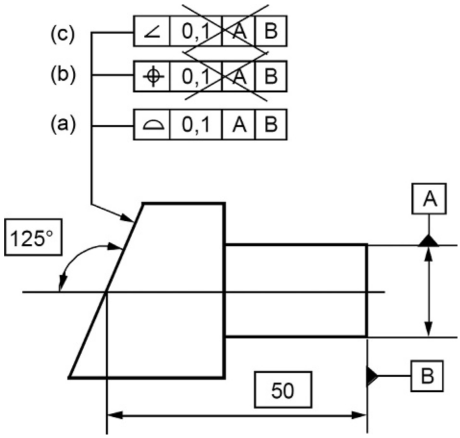


FIG. 6.19 Tolerancing of an inclined plane

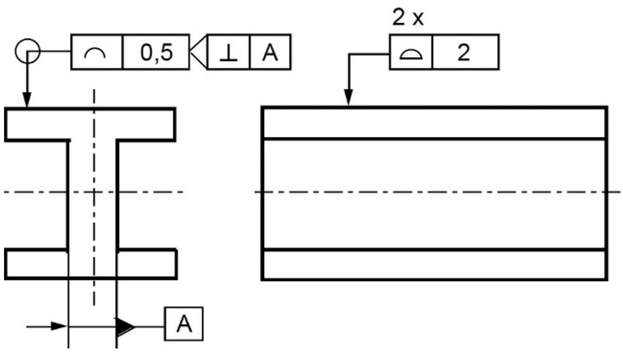
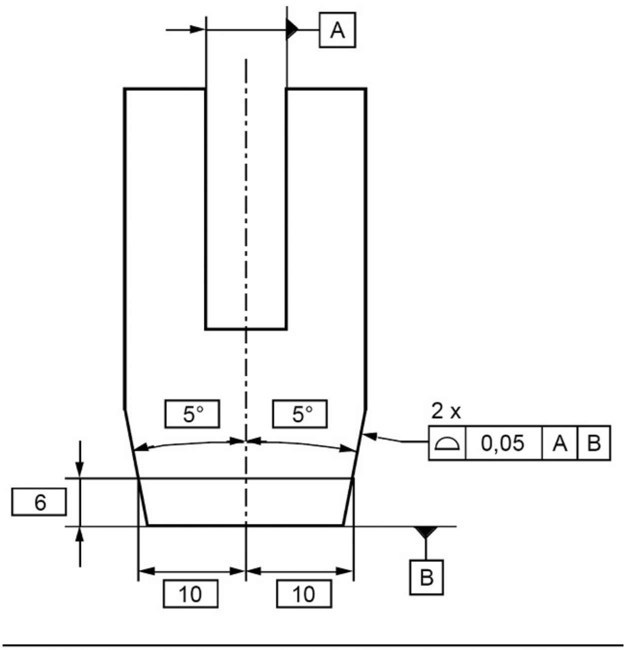


FIG. 6.20 Tolerancing of stock material

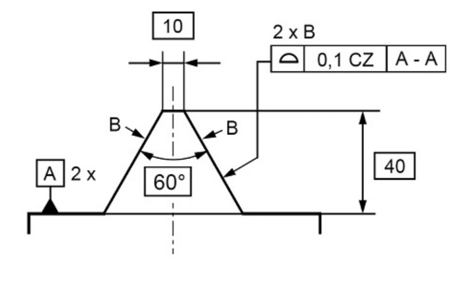
Figure 6.22 shows tolerancing of a wedge for a slide way. CZ includes the TED 10.

Figure 6.23 shows a wedge serving as a datum.



Indication CZ not necessary
because location defined by datum system

FIG. 6.21 Tolerancing of a wedge



No datum for horizontal direction
therefore
CZ

FIG. 6.22 Tolerancing of a wedge for a slide way

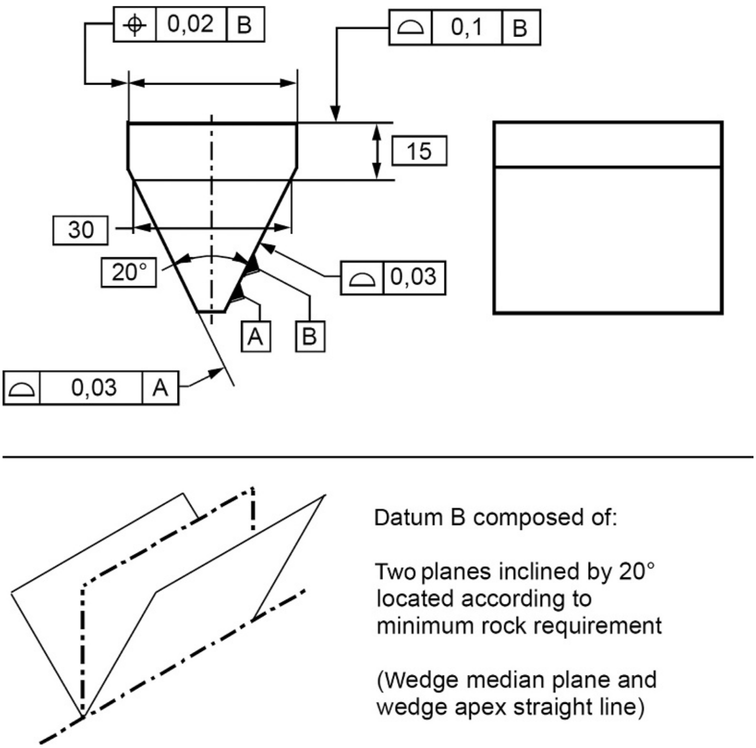


FIG. 6.23 Wedge serving as the datum

6.2.8 Cones

Figure 6.24 shows tolerancing of a cone in location, orientation, and form by one single profile tolerance. Figure 6.25 shows tolerancing when there are further constraints for radial location and for form.

Figure 6.26 shows tolerancing according to Fig. 6.24, and in addition auxiliary diameters and their tolerances for manufacturing (which are easy to measure

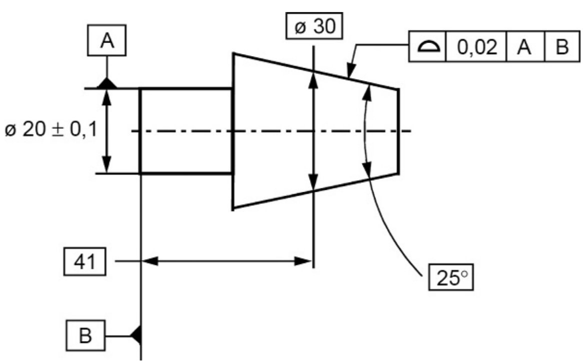


FIG. 6.24 Tolerancing of a cone

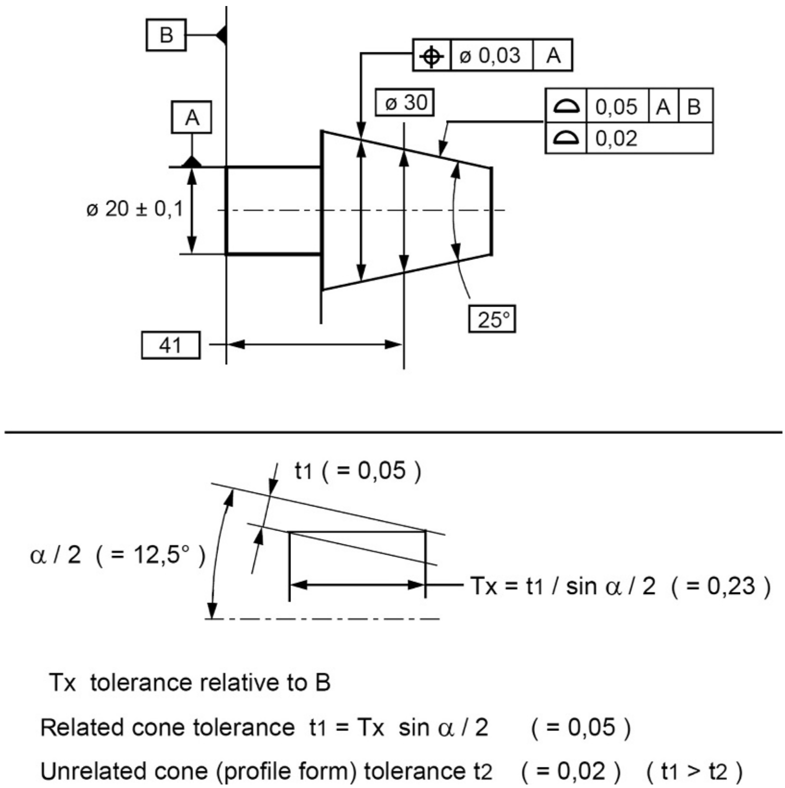


FIG. 6.25 Tolerancing of a cone with constraints

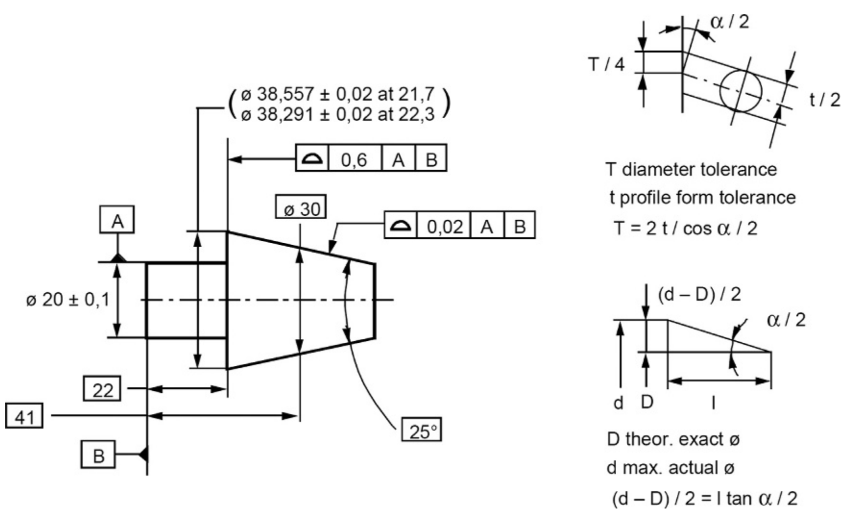


FIG. 6.26 Tolerancing of a cone with auxiliary dimensions for manufacturing purposes

during manufacturing) are indicated in parentheses. The limit dimensions depend on the distance from the datum B. The figure shows the formulae for the calculation.

Figure 6.27 shows a cone serving as the datum (B) for tolerancing the coaxiality deviation of a short cylinder (too short to serve as the datum).

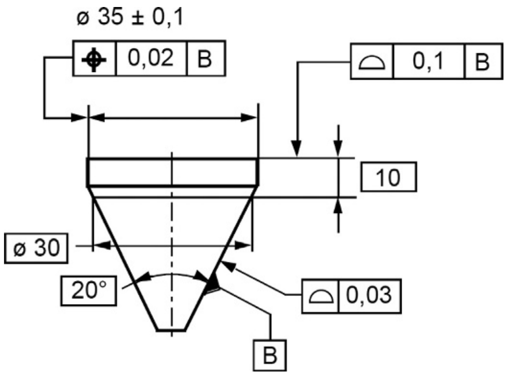


FIG. 6.27 Cone serving as the datum

6.2.9 Fits

Figure 6.28 shows profile tolerancing of a rectangular fit, e.g. a cover sheet in a casing. Compare with Fig. 5.73 showing position tolerancing. The maximum

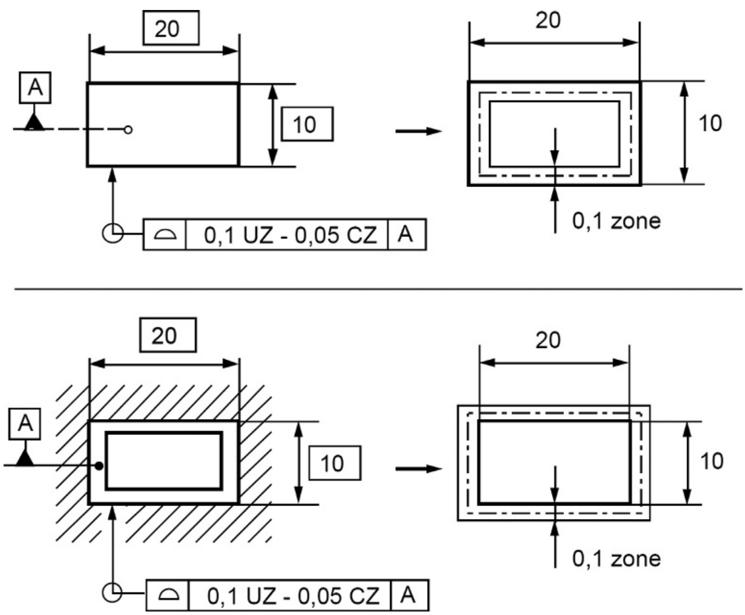


FIG. 6.28 Profile tolerancing of a rectangular fit

possible form deviation is half as much as with position tolerancing. Both parts have the same CAD model.

Figure 6.29 shows tolerancing of a complex fit. The space between centred contours is between 0,005 and 0,105. Figure 6.30 provides the explanation for Fig. 6.29.

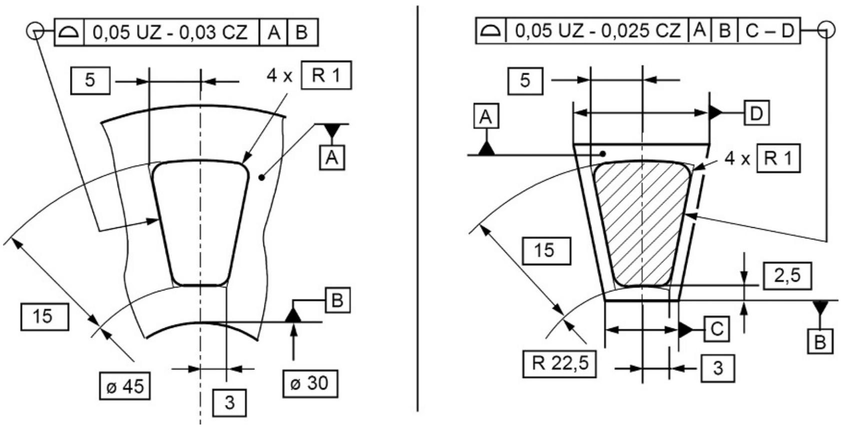


FIG. 6.29 Tolerancing of a complex fit

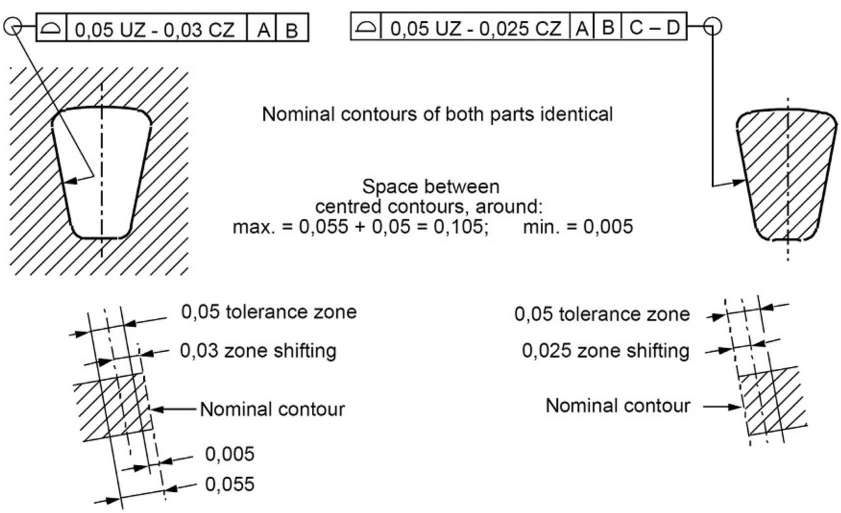


FIG. 6.30 Explanation for Fig. 6.29

This page intentionally left blank

Chapter 7

± Tolerancing, General Tolerancing

7.1 General

The ± tolerancings for sizes **without** modifier (Ⓜ, Ⓛ, Ⓔ, etc.; see ISO 14 405-1) and for non-sizes (steps, distances, roundings, contour dimensions) have large ambiguities and shall no longer be used; see 7.2 to 7.4. They are no longer the state of the art. Instead the following shall be used: profile tolerances for integral features and position tolerances for derived features, the latter together with Ⓜ, Ⓛ, Ⓟ, etc.; see ISO 14 405-1, and ± tolerances for sizes.

Even when large tolerances are permissible, geometrical tolerances (profile, position) shall be used. They are correct and fit into the current tolerancing system with general profile tolerances; see 7.5. Inspection can still be executed with plain measuring instruments, e.g. vernier callipers, when these tolerances are large enough.

Exceptions to this rule are two-point sizes for the lower limit of size with external features of size (e.g. shafts) and for the upper limits of size with internal features of size (e.g. holes).

7.2 Sizes

Cylindrical sizes (holes, bolts) and planar sizes (slots, tabs) toleranced only with ± tolerances can have mating sizes (size of perfect counterpart that just fits) outside of maximum and least material size. For example, see Figs 5.15 and 7.1. These sizes are to be toleranced with Ⓔ, Ⓜ, Ⓛ or other modifiers according to ISO 14 495-1 in order to fit for their function. See for example Figs 5.16 and 7.2.

Figure 7.1 shows the ambiguities in sizes with ± tolerances without a modifier.

Figure 7.2 shows a possible correct tolerancing.

Geometrical Dimensioning and Tolerancing for Design, Manufacturing and Inspection
https://doi.org/10.1016/B978-0-12-824061-8.00014-1

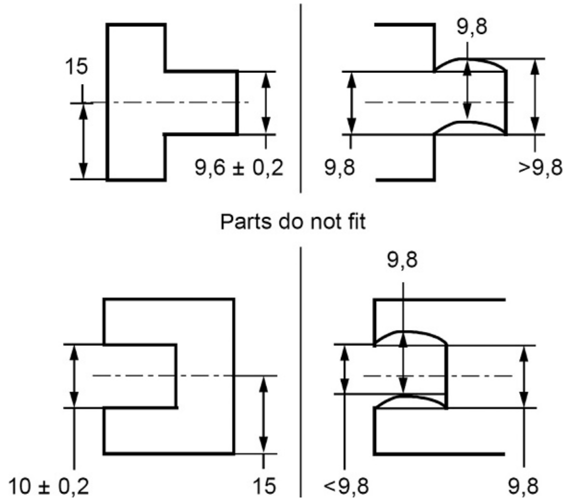


FIG. 7.1 Sizes, ± tolerance without modifier, ambiguities

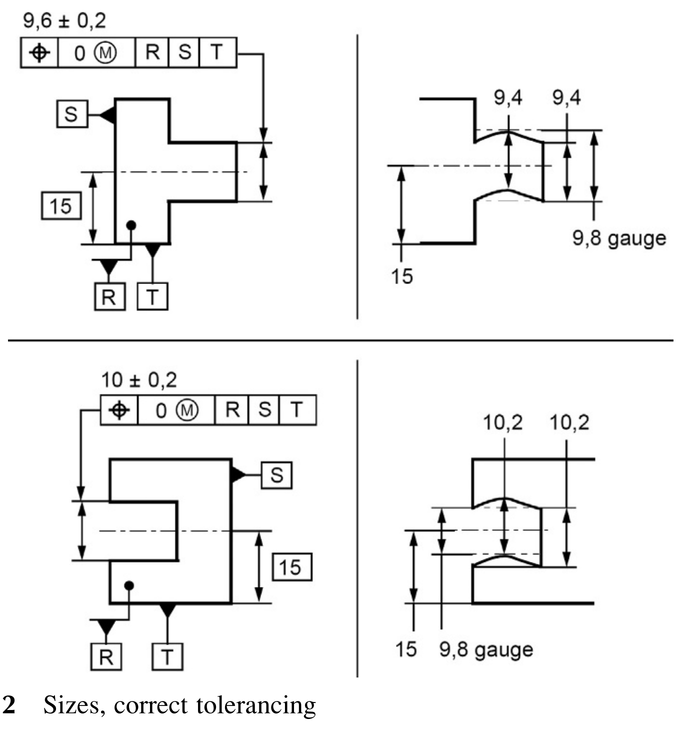


FIG. 7.2 Sizes, correct tolerancing

7.3 Non-sizes

7.3.1 Steps

Figure 7.3 shows ambiguities with ± tolerances for step dimensions. For example, measurements with step gauges may be oriented according to the upper or lower surface, outside material. With CMMs, the orientation may also be according to the upper or lower surface, inside material. It is also possible to measure the distance between two parallel median planes, according to Chebyshev (minimax) or according to Gauss.

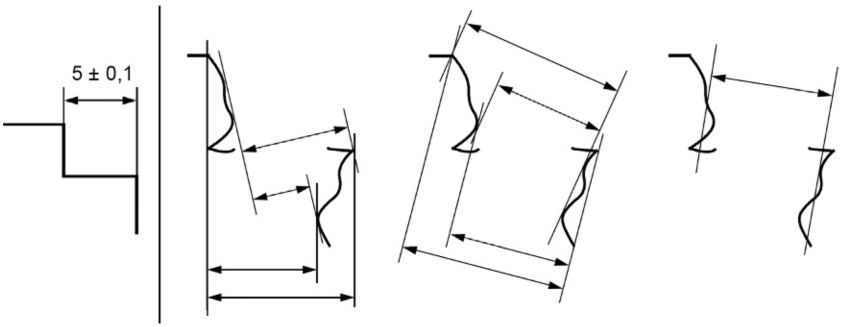


FIG. 7.3 Step dimension, ambiguities

Figures 7.4 and 7.5 show further ambiguities. As the ambiguities depend on the geometrical deviations of the surfaces, it is (without geometrical tolerancing) not clear what the workpiece might look like. Therefore, geometrical tolerancing shall be used; see Fig. 7.6.

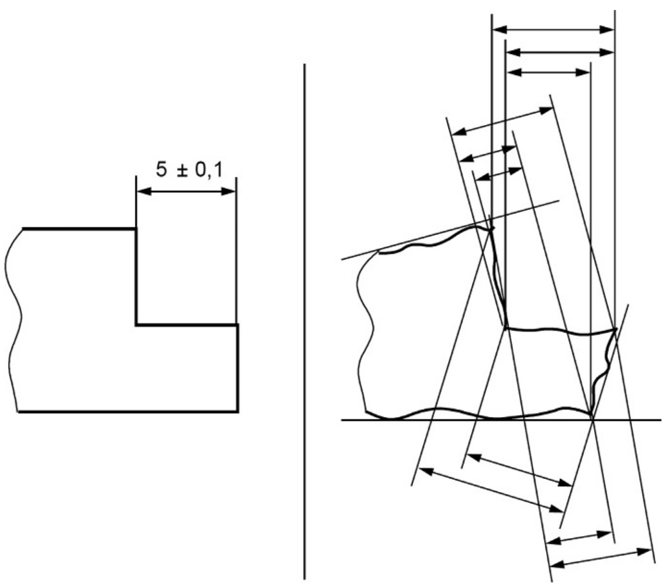


FIG. 7.4 Step dimension, ambiguities

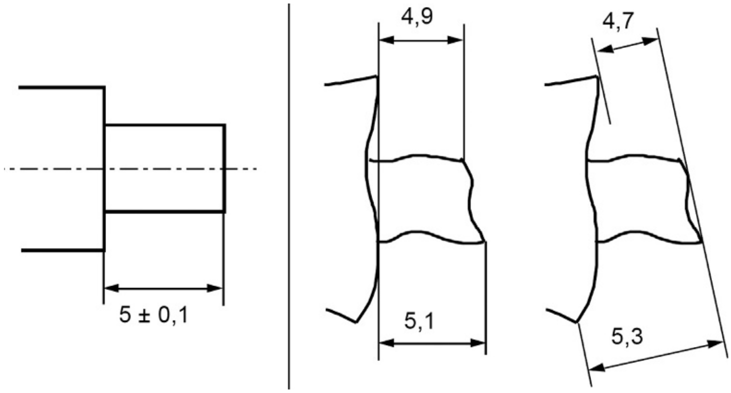


FIG. 7.5 Tolerancing of step dimensions

In order to avoid the rather complicated indications, the indication according to Fig. 7.7 may be used. However, this indication is not standardized and requires the explanation on the drawing (near the title block), as shown in Fig. 7.7.

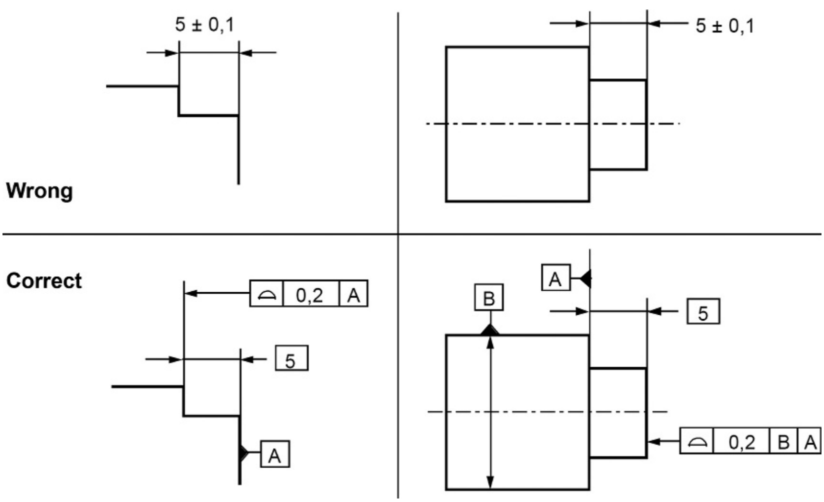


FIG. 7.6 Step dimensions, correct tolerancings

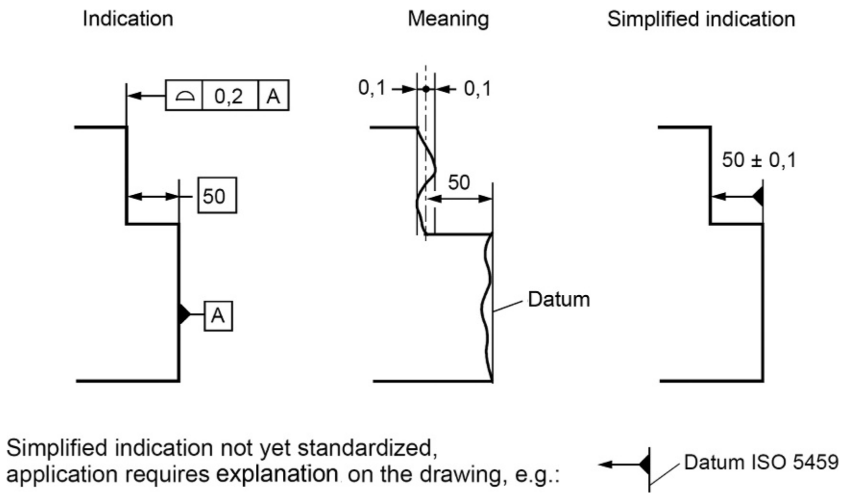


FIG. 7.7 Step dimension tolerancing, simplified drawing indication

7.3.2 Centre distances

It was former practice to use ± tolerances for centre distance, e.g. for holes on a pitch circle (see Fig. 7.8). The cord lengths were indicated with ± tolerances as chain dimensions. Because of the tolerance accumulation, very small (expensive) tolerances were indicated. However, this was not functional, because when all but one chain member were at the limit, the last member exceeded the limit. Also, very small (expensive) ± tolerances were indicated for the pitch diameter. However, even when this small tolerance was respected, the part often could not be assembled, because the pair of opposite holes was eccentric in relation to the other holes.

Also, for other patterns of holes, ± tolerances for the centre distances had large ambiguities; see Fig. 7.9. It was not clear what the centre was (centre of the horizontal or vertical local diameter, centre of the maximum inscribed cylinder, centre of the minimum circumscribed cylinder) and from where to measure (real surface, associated plane, and whether from primary, secondary or tertiary datum plane). Geometrical tolerancing is unique and correct.

Figure 7.10 shows the ambiguities for a centre distance with ± tolerance (same CAD model). Figs. 7.11 and 7.12 show correct tolerancings.

Figure 7.13 shows a ± toleranced angular location. It was not clear where the zenithal line of the angle was located (horizontal: median plane of the outer surfaces or median plane of the inner surfaces or centre of the hole distance; vertical: associated plane of the two horizontal surfaces, median plane of the two hole median lines). There are at least five possible locations for the zenithal line, resulting in five different measurement results. Geometrical tolerancing is unique and correct.

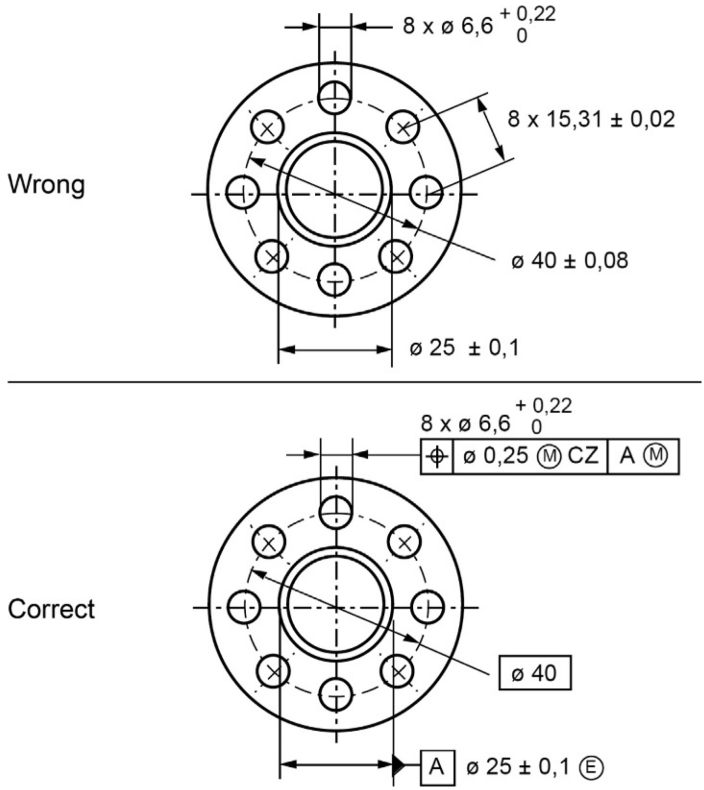


FIG. 7.8 Centre distances, holes on a pitch circle

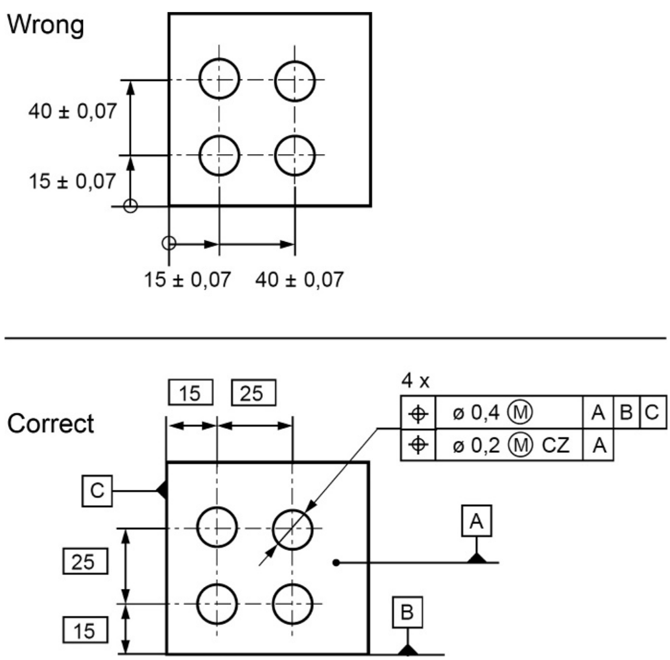


FIG. 7.9 Centre distances for patterns of holes

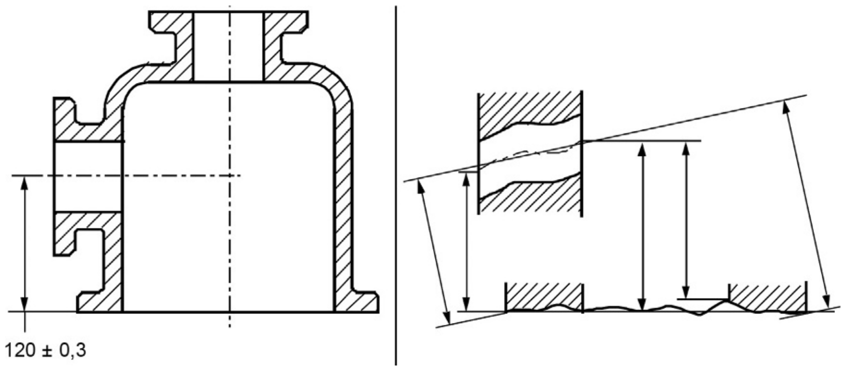


FIG. 7.10 Centre distance, ambiguities

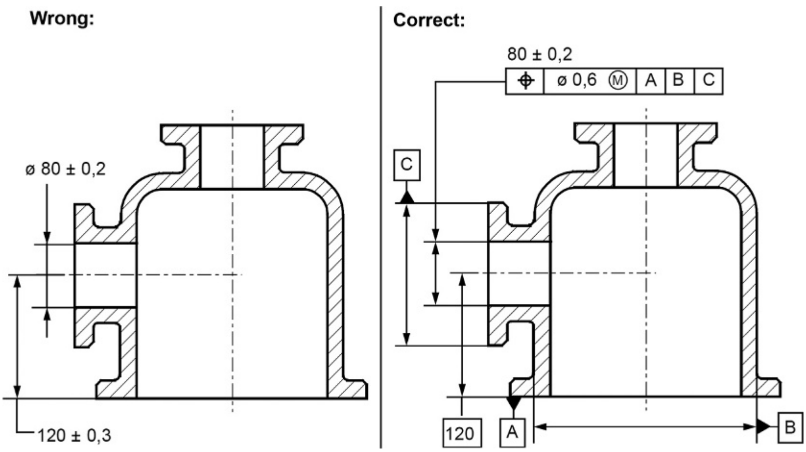


FIG. 7.11 Centre distance, position tolerancing

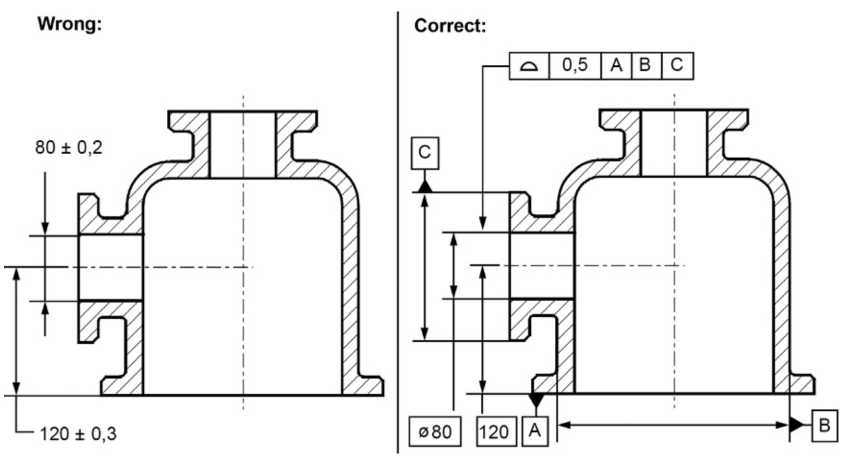


FIG. 7.12 Centre distance, profile tolerancing

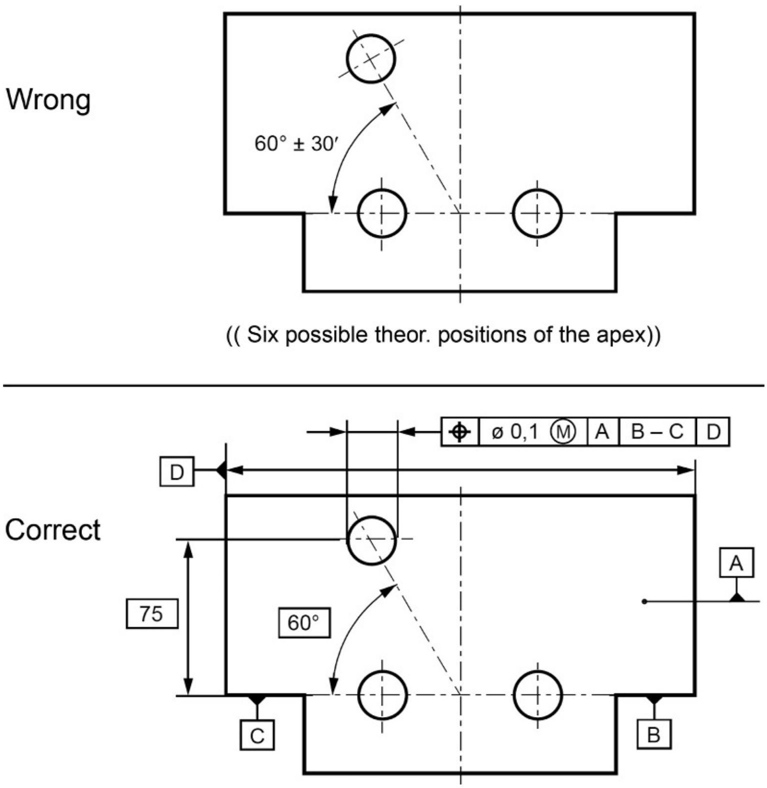


FIG. 7.13 Angular location, ambiguities

Figure 7.14 shows a workpiece with multiple (six) median lines on the centre line. With ± tolerancing, the smallest necessary tolerance must be indicated, perhaps at the two bearing holes. Then this small tolerance applies to all centre distances, and also for the chamber. This is expensive in production.

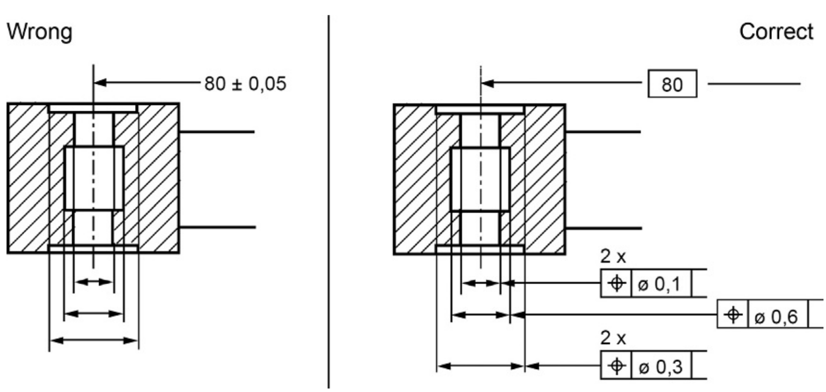


FIG. 7.14 Different centre distances at the same location

Geometrical tolerancing allows specification of different tolerances to each centre line, which may be cheaper in production.

7.3.3 Radii tolerances

According to ISO, ± tolerances are undefined. The real form deviates from the ideal form. According to ISO 14 405-2 they should no longer be used. Instead, profile tolerancing should be used. It has a defined tolerance zone. See Fig. 3.26.

7.3.4 Contours

Figure 7.15 shows a contour of a cover plate that shall fit into a counter part. Here ± tolerancing is inappropriate because the effective length and height can exceed the dimensional limits. The correct choice (according to the design intention) is profile tolerancing.

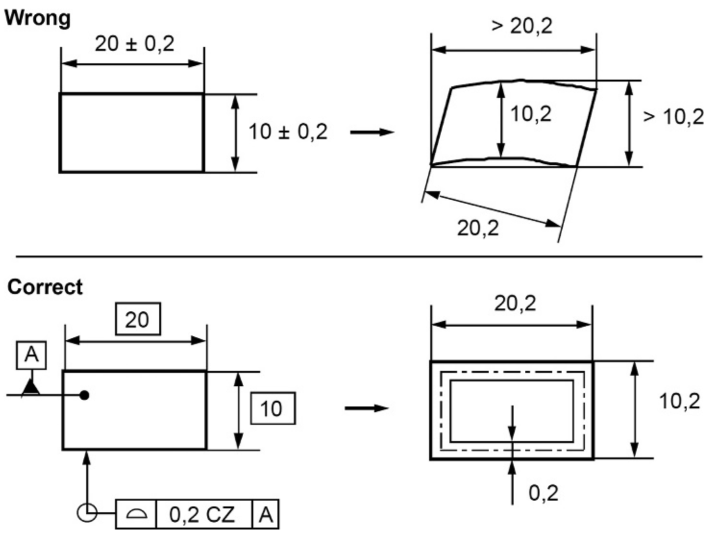


FIG. 7.15 Profile tolerancing of a cover plate (contour)

Figure 7.16 shows a more complicated workpiece that shall fit into a counterpart. Because of the assumed tolerance accumulation, the ± tolerances are very small. However, although the tolerances are so small, the workpieces may not fit, because there are orientation deviations which have not been taken into consideration. The correct choice is profile tolerancing.

When, with profile tolerances, UZ with half of the tolerance is used, the part and the counterpart can have the same theoretical contour.

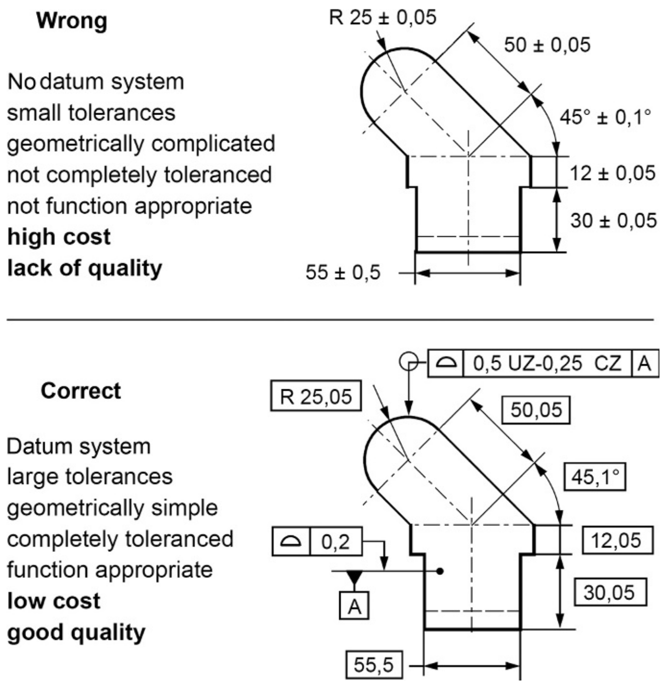


FIG. 7.16 Contour for fit

7.3.5 Chain dimensioning

Figure 7.17 shows chain dimensioning with ± tolerances. Theoretically, the tolerances accumulate between the first and the last member of the chain. However, the accumulation practically does not happen, because the pattern for the casting contains the chain members (approximately) at the exact location. In machining, the CNC machine tool goes (approximately) to the correct location. The correct dimensioning and tolerancing (with profile tolerances related

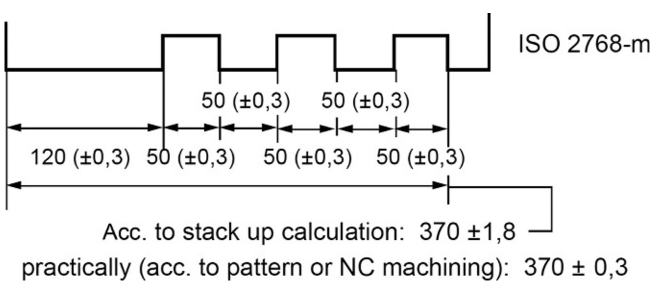


FIG. 7.17 Tolerance accumulation with chain dimensioning

to a datum system) are shown in Fig. 7.18. Figure 7.19 shows another possibility with position tolerancing.

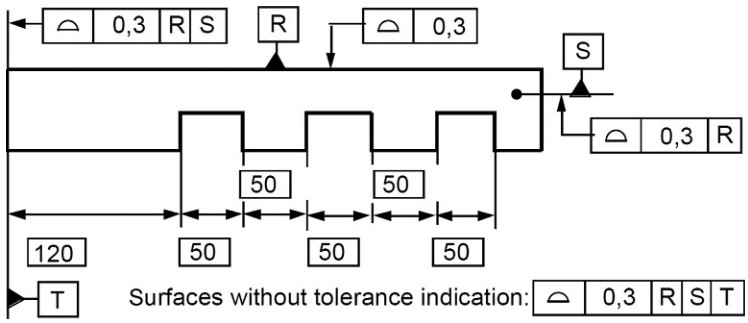


FIG. 7.18 Correct profile tolerancing of a chain-dimensioned contour

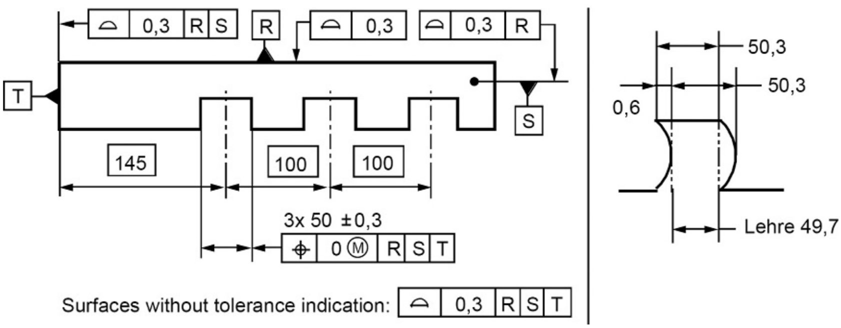


FIG. 7.19 Correct position tolerancing of a chain-dimensioned contour

7.4 Profile tolerances, position tolerances instead of ± tolerances, advantages

Profile tolerancing for integral features and position tolerancing for derived features instead of ± tolerancing for non-sizes (steps, centre distances, radii and contours) have the following advantages:

- tolerance zones can be related to datum systems (Figs 7.11 and 7.12), and unequivocal indication, function-related tolerancing with the largest possible tolerances is possible
- features with the same centre line can have different tolerances (Fig. 7.14)
- cylindrical tolerance zones can be used with 57% more tolerance (Fig. 5.12)
- possibility of taking advantage of Ⓜ, Ⓛ, Ⓟ (Figs 5.16, 4.42 and 5.21), correct functional tolerancing

– no tolerance accumulation with chain dimensioning, larger tolerances (Figs 7.18 and 7.19)
– simple presentation and calculation of functional relationship (Fig. 5.13)
– **better quality, larger tolerances, lower costs**

7.5 General tolerances

General tolerances according to ISO 2768 for metallic machined parts and ISO 8062-3 for castings contain ± tolerances (with the disadvantages described previously) and geometrical tolerances which do not tolerance the workpiece completely. Therefore, these general tolerances shall no longer be used, as they are no longer state of the art; see ISO 14 405-2.

Instead, a profile surface tolerance, related to a general datum system RST, that locks all the degrees of freedom of the workpiece and is rated to the size of the workpiece (space diagonal, diameter of smallest enveloping sphere), ISO 20 170, shall be used for surfaces without dimensional or geometrical tolerance indication.

The datum features shall be toleranced properly (form tolerance for primary datum, secondary datum toleranced in relation to the primary datum, tertiary datum toleranced in relation to the primary and secondary datum).

With that, the workpiece is already completely toleranced.

Figure 7.20 shows an example. It should be noted that the general tolerance applies only to surfaces without tolerance indication. When there is a tolerance indication, a tolerance directly or indirectly related to the general datum system is to be indicated, even when it is equal to the general tolerance.

It is also possible to indicate general tolerances individually when they are necessary for the main function.

Figure 7.21 gives an example in which a tolerance is indirectly related to the general datum system.

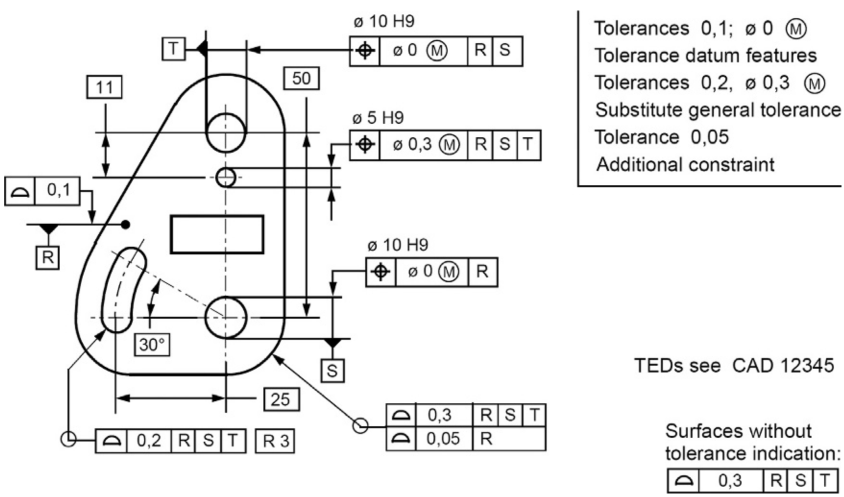


FIG. 7.20 Example of a general toleranced workpiece

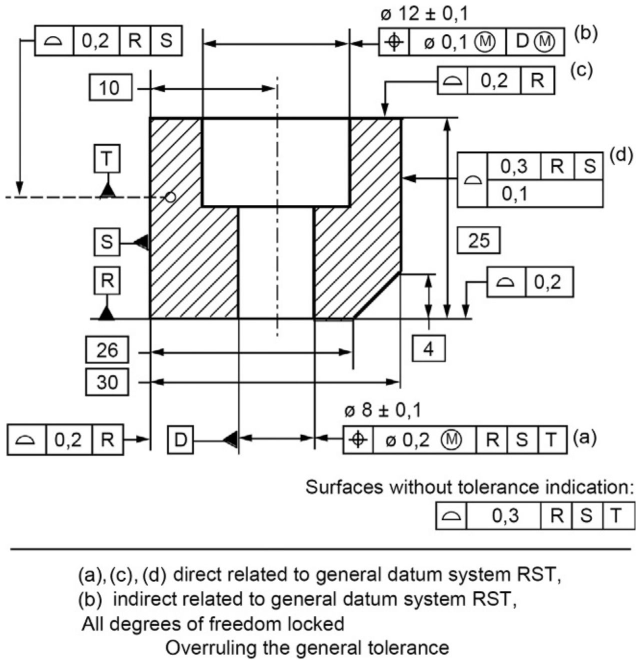


FIG. 7.21 Example with a tolerance indirectly related to the general datum system

When tolerances other than the general tolerances are necessary, they are to be indicated individually; see Fig. 7.20.

The general tolerance shall be large enough to be respected without any additional effort and with a high probability. This can be derived from past measurement results. See Fig. 7.22. Then manufacturing and inspection can focus on the individually indicated tolerances.

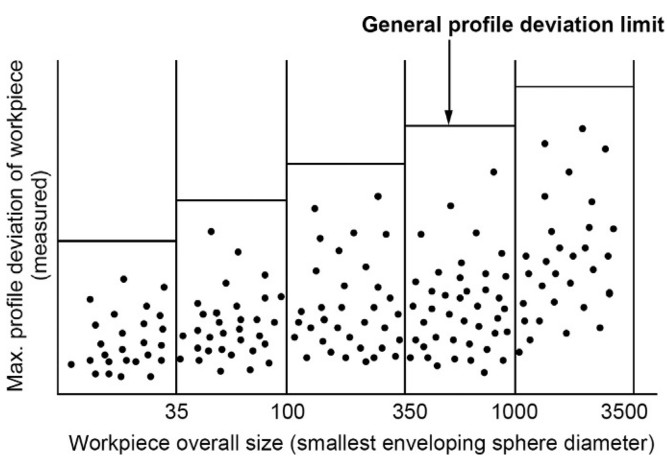


FIG. 7.22 General tolerances derived from past measurements

Figure 7.23 shows a proposal for general tolerances for machined metallic workpieces. The tolerances are derived from ISO 2768-1.

Figure 7.24 shows how the general profile tolerances are derived from the ± tolerances of ISO 2768-1.

Surfaces without tolerance indication: | ⌓ | ... | R | S | T |

General surface profile tolerances

Overall size mm
(ø of smallest envelope sphere of nominal model)
up to

	30	120	400	1000	2000
f	**0,1**	**0,15**	**0,2**	**0,3**	**0,5**
m	**0,2**	**0,3**	**0,5**	**0,8**	**1,2**
c	**0,5**	**0,8**	**1,2**	**2**	**3**
v	**1**	**1,5**	**2,5**	**4**	**6**

FIG. 7.23 Proposal for general profile surface tolerances for metallic machined workpieces

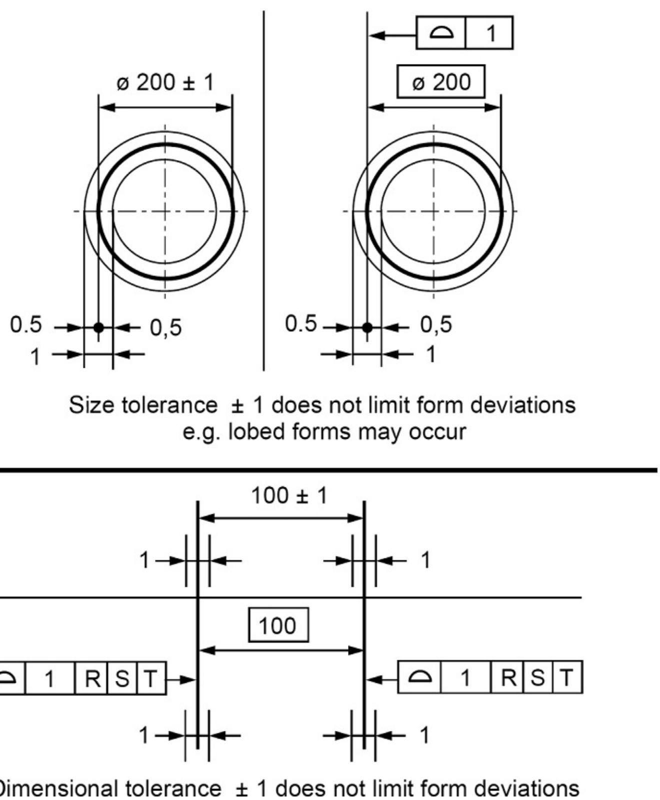


FIG. 7.24 Comparison ± tolerances and profile tolerances

7.6 Further general tolerances

7.6.1 General tolerances for plastic parts

ISO 20 457 gives general (profile) tolerances for plastic parts. They are rated to the Dp dimension; see Table 7.1. This is the nominal maximum special distance of the toleranced surface from the datum system origin (Fig. 7.25). This distance is important for the following reason. Plastic parts deform when cooling after the manufacturing process (warping, distortion). In order to reduce this deviation from the nominal shape, the die is designed deviating from the nominal shape. The more the surface is away from the datum system origin, the more deviation from the nominal shape is needed for the die.

TABLE 7.1 General profile tolerances for plastic parts

ISO 20457:2018 Table 10

	General profile surface tolerances with general datum system				
Dp	≤ 30	> 30 ≤ 100	> 100 ≤ 250	> 250 ≤ 400	> 400 ≤ 1000
Tolerance	0,5	1	2	4	6

Drawing indication:

Surfaces without tolerance indication: | ⌓ | x | R | S | T |

x see ISO 20457:2018 Table 10

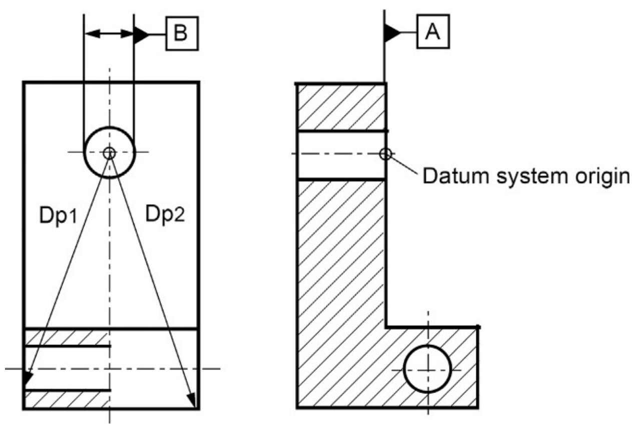


FIG. 7.25 General tolerances for plastic parts Dp dimension for surfaces (not axes) nominally maximum spatial distance of toleranced feature from datum system origin

7.6.2 General tolerances for metallic castings

ISO 8062-4 gives general (profile) tolerances for metallic castings. There are 15 tolerance grades, each rated to the size of the casting (diameter of the smallest envelope sphere); see Table 7.2.

TABLE 7.2 General profile tolerances for metallic castings

Moulded space diagon. above	up to	General profile surface tolerances for tolerance grades P 1	2	3	4	5	6	7	8	9	10	11	12	13	14	15
	10	0,09	0,14	0,19	0,27	0,37	0,53	0,76	1,1							
10	30	0,12	0,16	0,23	0,31	0,44	0,61	0,86	1,3							
30	100	0,14	0,19	0,27	0,38	0,53	0,74	1,1	1,5	2,1	3					
100	300	0,15	0,23	0,35	0,5	0,7	1	1,4	2	2,8	4	5,6	8			
300	1 000			0,42	0,64	0,8	1,3	1,9	2,7	3,8	5,5	7,5	11	15	19	23
1 000	3 000						1,6,	2,4	3,8	5,4	8	10	15	21	26	33
3 000	6 300									7	10	13	19	26	33	41
6 300	10 000										11	16	23	32	40	50

Drawing indication, e.g.:
Surfaces without tolerance indication:
⌓ 2,7 R S T ⦾

ISO/TS 8062-2

In addition, there is a constraint for wall thicknesses; see Table 7.3. It applies only when indicated (see, e.g. Fig. 10.24) and the wall is not machined. Without this constraint, the wall thickness may become too small for the function. See also 10.4.

TABLE 7.3 General tolerances for wall thicknesses (additional constraint)

Dimensions in mm

Thickness *) above	up to	General tolerances t for wall thicknesses, tolerance grades S, (± t / 2) 1	2	3	4	5	6	7	8	9	10	11	12	13	14	15
–	10	0,05	0,06	0,1	0,12	0,17	0,25	0,35	0,5	0,7	1	1,4	2			
10	30	0,06	0,07	0,12	0,15	0,2	0,3	0,4	0,6	0,8	1,2	1,7	2,5	3	4	5
30	100	0,07	0,1	0,14	0,2	0,25	0,4	0,5	0,8	1	1,5	2	3	4	5	7
100	300	0,08	0,12	0,17	0,25	0,35	0,5	0,7	1	1,4	2	3	4	5	7	9

*) Nominal wall thickness of moulded condition

Drawing indication, e.g.:
Wall thicknesses:
25 ± 0,3
35 ± 0,4

Tables 7.4 and 7.5 give information on which tolerance grades may be achievable.

TABLE 7.4 General tolerance grades P and S for short series and single production

	Sand cast hand moulding	
	Clay-bounded	Chemically bounded
Steel	13–14	12–14
Grey iron	13–15	11–14
S.G. iron	13–15	11–14
Malleable iron	13–15	11–14
Copper alloys	13–15	10–13
Zinc alloys		
Light metal alloys	11–13	10–13
Nickel-based alloys	13–15	12–14
Cobalt-based alloys	13–15	12–14

TABLE 7.5 General tolerance grades P and S for long series production

	Sand cast hand moulding	Sand cast machine mould and shell moulding	Metallic permanent mould	Pressure die casting	Investment casting
Steel	11–14	8–12			4–9
Grey iron	11–14	8–12	7–9		4–9
S.G. iron	11–14	8–12	7–9		4–9
Malleable iron	11–14	8–12	7–9		4–9
Copper alloys	10–13	8–10	7–9	6–8	4–9
Zinc alloys	10–13	8–10	7–9	3–6	4–9

Continued

TABLE 7.5 General tolerance grades P and S for long series production—cont'd

	Sand cast hand moulding	Sand cast machine mould and shell moulding	Metallic permanent mould	Pressure die casting	Investment casting
Light metal alloys	9–12	7–9	6–8	6–9	4–9
Nickel-based alloys	11–14	8–12			4–9
Cobalt-based alloys	11–14	8–12			4–9

7.6.3 Multiple general tolerances on the same part

As the general tolerances are applied to the surfaces, it is possible to apply different general tolerances to the same part. The surface texture symbols give the information on which general tolerance applies; see Fig. 7.26 for a machined casting and Fig. 7.27 for a welded part.

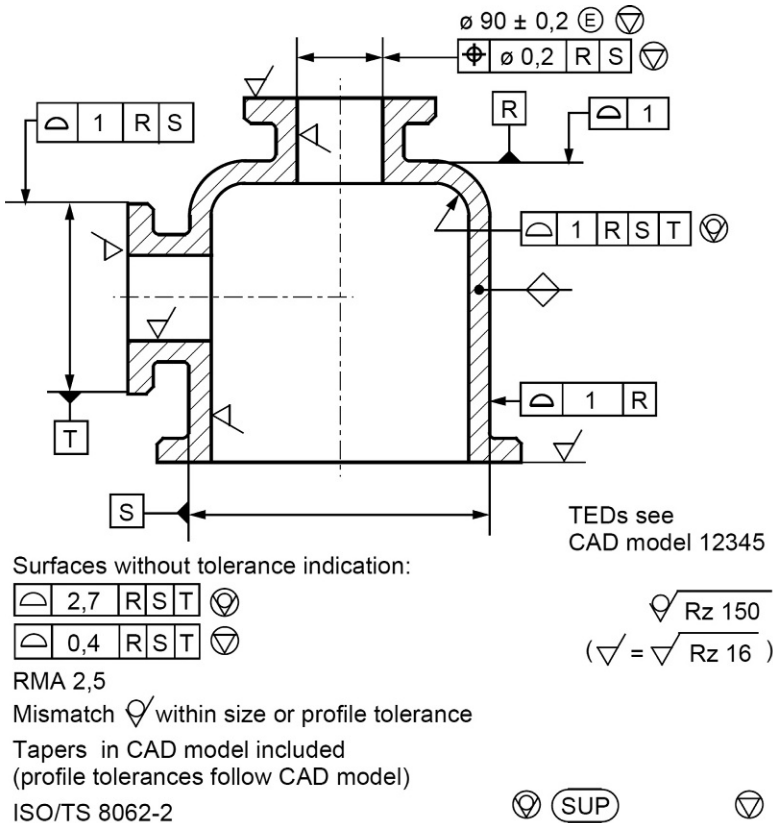


FIG. 7.26 Machined casting with general tolerances for moulded and machined surfaces

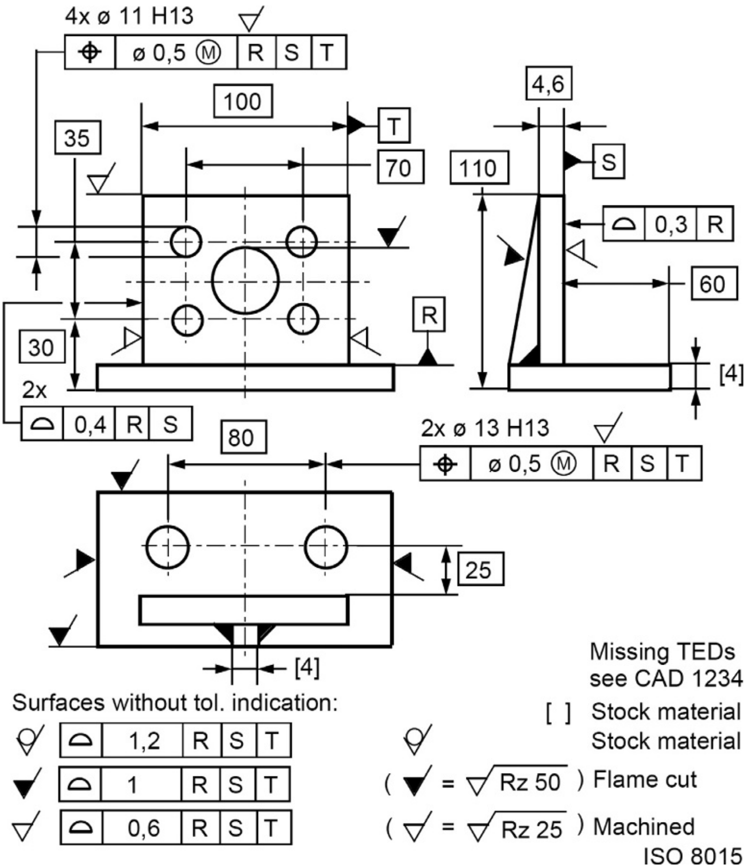


FIG. 7.27 Welded part with general tolerances for stock material, flame cutting and machined surfaces

7.7 General size tolerances

ISO 22 081, on general tolerances, allows, in addition to the general profile tolerance, the use of general linear and/or angular size tolerances. However, in the most functional cases, size tolerances are to be complemented by geometrical tolerances (specifications, e.g. Ⓜ or Ⓔ), and then the general tolerances do not apply. Therefore it is not recommended to use general size tolerances. (See also 5.4.) Instead, general position tolerances can be used; see Fig. 7.28.

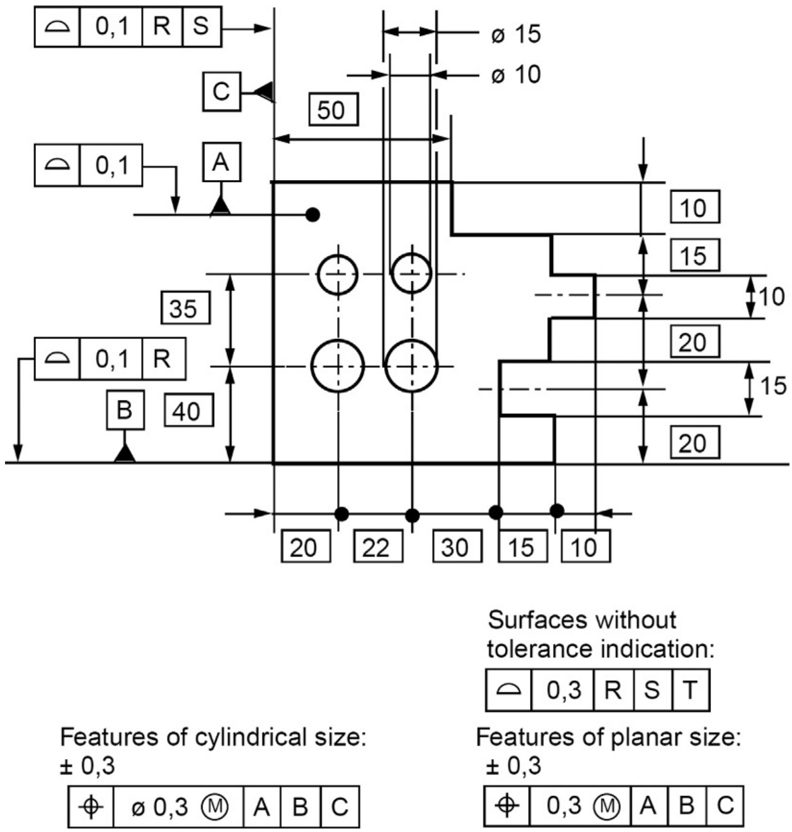


FIG. 7.28 General position tolerances

7.8 General tolerances according to ISO 2768

It was former practice to use general tolerances according to ISO 2768-1 and -2. ISO 2768-2 is now withdrawn, ISO 2768-1 is intended to be withdrawn.

The ± tolerances of ISO 2768-1 leads to ambiguities as shown before and in ISO 14405-2. Further they could only be used when the dimensions are indicated on the drawing. In 3D CAD systems, when no 2D drawing exists and no dimensions are shown in the model, these tolerances cannot be used.

The general geometrical tolerances according to ISO 2768.2 were applied to all possible combinations of features. When the features are too short, they are not inspection appropriate and the general tolerances cannot be used. Further the workpiece was not totally toleranced. Furthermore there was no datum system involved, and it was difficult or impossible, to assess, which permissible total shape was allowed to the workpiece.

Therefore the general tolerances according to ISO 2768 should not be used anymore. Instead, general profile tolerances should be used, as described before.

Chapter 8

Recommended Procedure for Tolerancing

8.1 General

A distinction must be made between integral features (surfaces or lines on surfaces) and derived features (median lines or median surfaces of features of size (e.g. cylinders and widths)). It is recommended to use the indications as shown in Fig. 8.1.

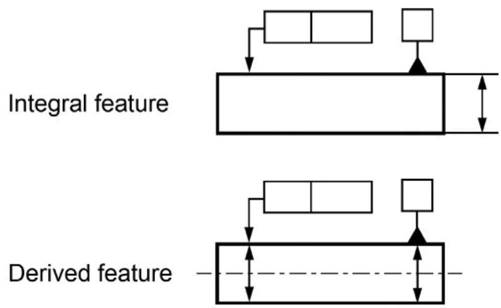


FIG. 8.1 Indication of integral and derived features

Figure 8.2 shows the recommended procedure for tolerancing.

First, the CAD model is established. Then an analysis of the main function is conveyed with establishment of the main datum system. This datum system shall be complete, i.e. locking all degrees of freedom. ISO 8062-4 recommends using the datum letters RST.

Next, the datum features are toleranced, the primary datum feature with a surface profile tolerance without datum, and the secondary and tertiary datum features with surface profile tolerances related to the primary and to the primary and secondary datum.

Then, a surface profile tolerance is indicated as a general tolerance. With that, the part is then completely toleranced.

However, it should be investigated whether surfaces should be toleranced (for functional reasons) with a smaller (or perhaps in special cases with a larger)

Geometrical Dimensioning and Tolerancing for Design, Manufacturing and Inspection.
https://doi.org/10.1016/B978-0-12-824061-8.00003-7

1 **CAD model**
Establishment of the CAD model

2 **Main function and main datum system**
Analysis of the main function and
establishment of the main datum system

3 **Tolerancing of the datum features**

4 **General tolerance** (for surfaces without tol. indication)
Profile surface tolerance, related to general datum system,
to be indicated (with that, part completely toleranced)

5 **Profile tolerancing overruling the gen. tolerance**
If required by the function, other profile surface tolerances,
locking all degrees of freedom, are to be indicated individual-
ly; they overrule the general tolerance

6 **Position tolerancing overruling the gen. tolerance**
If required by the function, for features of size, position tole-
rances, locking all degrees of freedom, together with size
tolerances, are to be indicated individually; they overrule the
general tolerance Ⓔ Ⓜ Ⓛ Ⓟ

7 **Further constraint tolerancing**
If required by the function, additional and more constraint
tolerances are to be indicated individually, e.g.:
⌓ ⌒ ↗ ⌖ ([-] >< UZ OZ SZ CZ SIM ◄► Ⓔ Ⓜ Ⓛ Ⓟ)
Then **all** necessary tolerances to be indicated individually
at the surface, even when identical to the general tolerance.

FIG. 8.2 Recommended tolerancing procedure

surface profile tolerance related to the general datum system. These tolerances overrule the general tolerance.

Further, it should be investigated whether derived features (median lines or median surfaces of features of size (cylinders or widths)) should be toleranced with a position tolerance related to the general datum system and then combined with a ± size tolerance. With that, the combination with Ⓔ, Ⓜ, Ⓛ, Ⓟ may be necessary. Fig. 5.26 shows the combination with Ⓜ for tolerancing of holes for fasteners.

Finally, it should be investigated whether, in addition to these tolerances, further tolerances as constraints are necessary for functional reasons. The integral features shall be profile toleranced and derived features position toleranced.

All these individually indicted tolerances overrule the general tolerance.

When the symbol >< is indicated, only orientation, not location, applies. See Fig. 3.24. The symbol CZ combines the tolerance zones of multiple integral or derived features in orientation and location relative to each other; with >< only orientation applies, not location, relative to the datum. See Table 3.4 and

Figs. 5.8 and 6.4. The symbol SZ defines separate tolerance zones (without relation to other features), e.g. in Figs. 5.8(g) straightness and 6.4(g) flatness.

For surfaces with individually indicated tolerances, the general tolerance does not apply. They must have a tolerance directly or indirectly related to the main datum system; see Fig. 7.20.

It is possible to indicate the general tolerances individually for features that have a main function.

8.2 Dimensioning

There are three types of dimensioning:

– function related (functional requirements directly indicated; the largest tolerances occur)
– manufacturing related (dimensions, to be used by the manufacturer without calculation, indicated)
– inspection related (dimensions, to be used by the inspector without calculation, indicated)

Fig. 8.3 (centre) shows, for the function of the step dimension, function-related dimensioning. The tolerance (±0.1) for the step is as large as possible. The same applies for the length of the adjacent threaded feature (tolerance ±0.3).

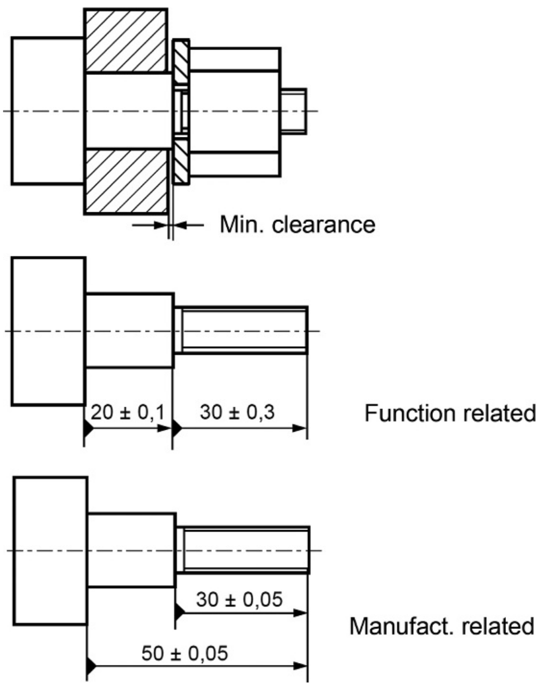


FIG. 8.3 Function-related and manufacturing-related drawings

Figure 8.3 (bottom) shows the manufacturing-related dimensioning for the turning process. The first cutting process starts at the right end, as well as the second. The dimensions correspond to that. The tolerances have become much smaller in order to meet the functional requirement at the step.

Designers should use the function-related dimensioning and tolerancing. All other dimensionings and tolerancings derive from this. When manufacturing-related drawings are necessary, they may be provided in addition. When the tolerance of the manufacturing-related drawing is exceeded, with the function-related drawing it can be investigated as to whether the workpiece is still functioning. However, the contractual drawing, whatever it is, is decisive for the acceptance or rejection of the workpiece.

Chapter 9

Tolerancing of Edges

9.1 Edges of undefined shape

For tolerancing the geometry (sharpness) of edges (within cross sections), a separate standard, ISO 13715, has been developed. Edges are intersections of two surfaces. Edges always exhibit deviations from the ideal geometrical shape (burr or undercut or passing) (Fig. 9.1). It may be necessary to tolerance the size of these deviations for functional reasons (e.g. in hydraulic equipment) or out of safety considerations.

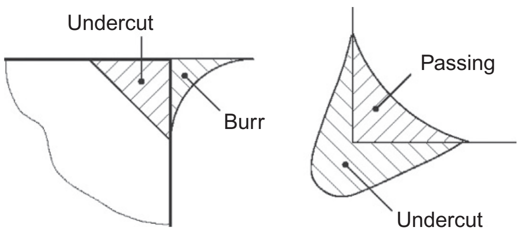


FIG. 9.1 Edge deviations

A **burr** is a rough remainder of material outside the ideal geometrical shape of an external edge, a residue of machining or of a forming process (Fig. 9.1).

An **undercut** is a deviation inside the ideal geometrical shape of an internal or external edge (Fig. 9.1).

A **passing** is a deviation outside the ideal geometrical shape of an internal edge (Fig. 9.1).

The edge deviation is illustrated in Fig. 9.2.

The tolerances for the deviations from the ideal geometrical shape of edges within cross sections are to be indicated by the symbol shown in Fig. 9.3. The indication + at the symbol means only deviations outside the material of geometrical ideal form are allowed (burr or passing permitted, undercut not permitted) (Fig. 9.5). The indication − at the symbol means only deviations inside the material of geometrical ideal form are allowed (undercut permitted, burr or passing not permitted) (Fig. 9.5). The indication ± at the symbol means deviations outside and inside the material of geometrical ideal form are allowed

Geometrical Dimensioning and Tolerancing for Design, Manufacturing and Inspection
https://doi.org/10.1016/B978-0-12-824061-8.00004-9

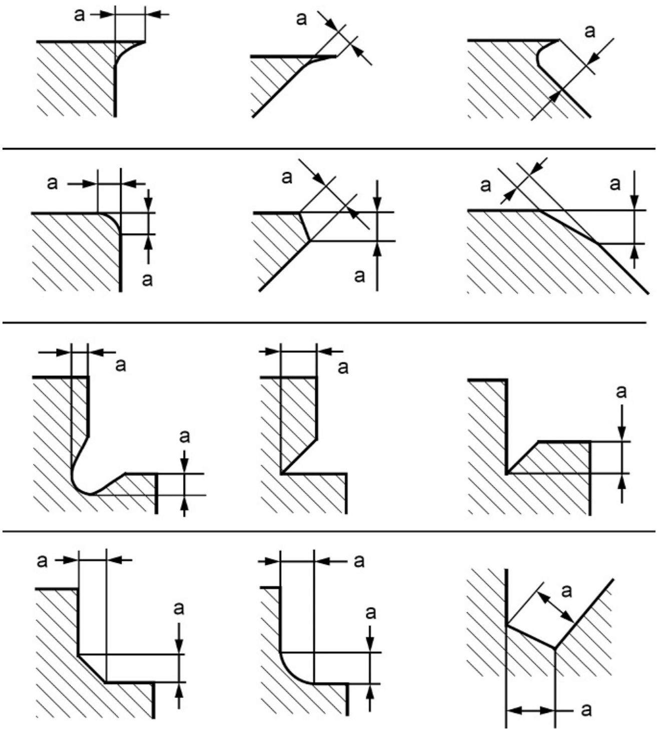


FIG. 9.2 Edge deviations

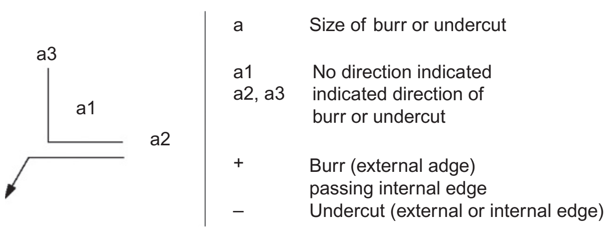


FIG. 9.3 Symbol for edge tolerances

(undercut or burr or passing permitted). When necessary, the upper limit of the size of the deviation or the upper and lower limit of the size of the deviation is to be indicated following the sign (Fig. 9.5). A sharp edge is an edge with an indication of a very small edge tolerance (e.g. −0.05 mm).

When it is needed to specify the direction of the burr on an external edge or the undercut on an internal edge, this indication shall be placed in areas a2 or a3 (as shown in Fig. 9.3) and as shown in the examples in Fig. 9.4.

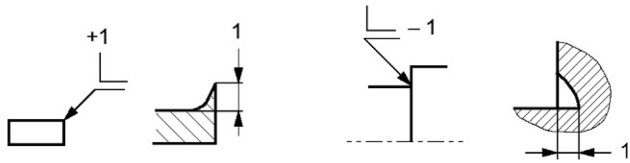


FIG. 9.4 Examples of edge tolerances

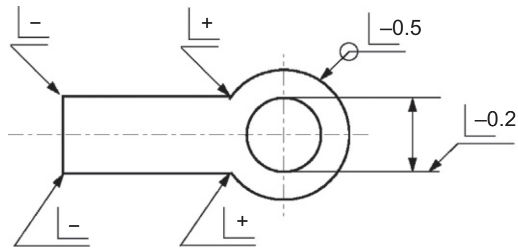


FIG. 9.5 Examples of indication of edge tolerances applied to all edges around the profile on both sides, front and back; applied to the edge of a feature (hole); applied to edges vertical to the projection plane and without tolerance value

When the symbol points to a corner (point), the edge vertical to the projection plane (front view) or the edge of the feature (e.g. hole) is meant (Fig. 9.5). When the symbol points to a line, this edge is meant; if only one view is represented and the outlines of both front and back are the same, the edges of the front and back are meant (Fig. 9.5). When the symbol 'all around' is used, all edges around the profile are meant (Fig. 9.5). In case of ambiguity, the indication may be used at corners. The symbol 'all around' shall not be used in sectional representations.

The edge tolerance indication may be applied

– as an individual indication for a single edge (Figs 9.5 and 9.6)
– as an individual indication for all edges around the represented profile (Fig. 9.5)

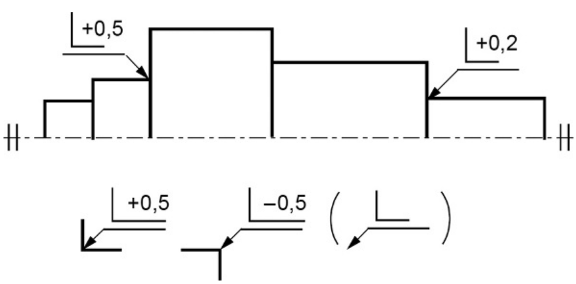


FIG. 9.6 Example of indication of a collective edge tolerance (–0.5)

– as a collective indication near the title block common to all edges without individual edge tolerance indication (Fig. 9.6).

According to ISO 13715, without an indication of the state of the edges, the parts may be delivered direct from the machine without an additional edge treatment.

When the symbols and rules according to ISO 13715 (described previously) are applied, it is recommended that reference be made to ISO 13715 – either within or near the title block or within allocated documents, in the manner shown in Fig. 9.7.

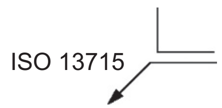


FIG. 9.7 Reference to ISO 13715

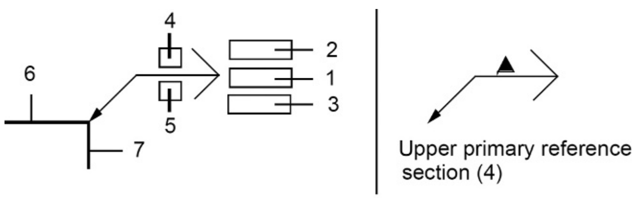
9.2 Transitions (edges of defined shape)

The new standard ISO 21204 provides the possibility of the specification of rather complicated edge forms (tolerancing of transition features) using short symbols.

Figure 9.8 shows the transition specification indicator.

FIG. 9.8 Transition specification indicator

Figure 9.9 shows the indication areas around the transition symbol.



1 Centre indication
2 Upper main indication
3 Lower main indication
4 Upper primary reference section
5 Lower primary reference section
6 Adjacent feature that 2 and 4 refer to
7 Adjacent feature that 3 and 5 refer to

FIG. 9.9 Indication areas around the transition symbol

Table 9.1 shows the symbols used for transition specifications.

TABLE 9.1 Symbols used for transition specifications

C	Chamfer
CF	Chamfer of fixed dimensions
CL	Chamfer of least material boundary
CM	Chamfer of maximum material boundary
D	Length extension of the tolerance feature from the specification origin
E	Ellipse
EF	Ellipse of fixed dimension
EL	Ellipse of least material boundary
EM	Ellipse of maximum material boundary
P	Fixed profile defined by CAD
PL	Profile defined by CAD least material boundary
PM	Profile defined by CAD maximum material boundary
R	Radius
RF	Radius of fixed value
RL	Radius least material boundary
RM	Radius maximum material boundary
T	Profile tolerance value
UZ	Unequally disposed zone

Figure 9.10 shows the dimensioning of a transition edge.

The transition specifications according to ISO 21204 are limited to edges between two planes and between a plane and a cylinder perpendicular to each other; see Fig. 9.11.

The edge transition tolerance defines the tolerance for a section line (line profile tolerance) in a section plane. This section plane is perpendicular to the section (ideal) line of the associated (ideal) features to the adjacent real features. The association is according to Gauss (total least squares). See Fig. 9.11.

According to ISO 21204, in each section plane, the dimensions start from the specification origin point; see Fig. 9.13. This point is the intersection point of

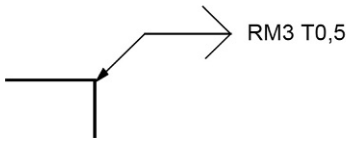
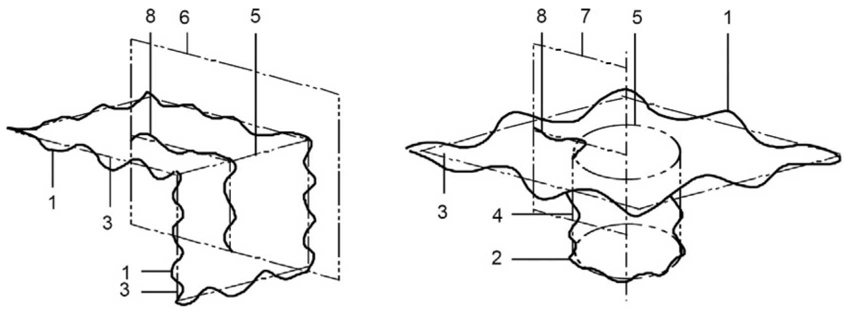


FIG. 9.10 Dimensioning of a transition edge



1 Norminally flat real feature
2 Nominally cylindrical real feature
3 Total least squares (Gaussian) planes associated to 1 independently
4 Total least squares (Gaussian) cylinder associated to 2
5 Intersection straight line or circle between 3 or 3 and 4
6 One of the infinite set of intersection planes perpendicular to 5
7 One of the infinite set of intersection planes perpendicular to 4 (containing the axis of 4)
8 One of the infinite set of profiles containing the toleranced feature

FIG. 9.11 Direction of the section planes

the two associated straight lines on the two adjacent surfaces. The straight lines are to be found by an iteration process. In the first association iteration, a least squares (Gaussian) straight line is associated to each adjacent feature line profile (Fig. 9.12). If the lengths of the features allow, the length of the profile shall be 3D; see Figs 9.16 and 9.14. The intersection between the bisector and the real workpiece is the separation point between the two adjacent features for further association purposes; see Fig. 9.13.

After the first iteration, the piece of the profile between the first separation point and the first intersection is disregarded and the association is repeated.

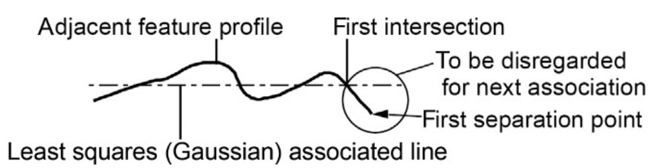


FIG. 9.12 Iteration process to find the specification origin point

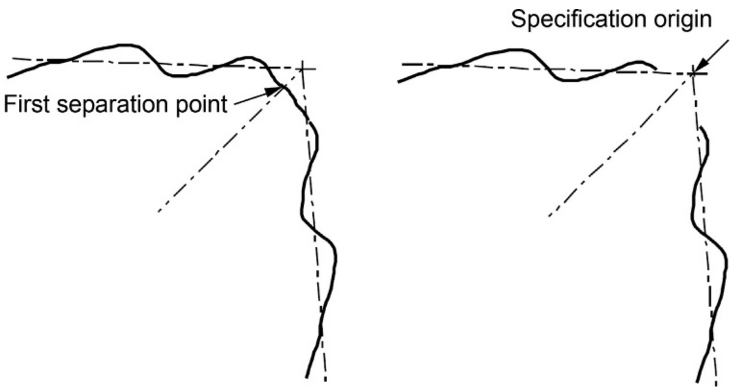


FIG. 9.13 Separation point and specification origin

This iteration continues until it does not result in any further portion of the profile being disregarded. In some types of form deviations, there are always form deviations to be disregarded, e.g. cylindrical form deviations. In these cases, it is recommended to abort after the third iteration. The intersection of the associated lines of the adjacent profile lines is the specification origin; see Fig. 9.13.

Figure 9.14 shows an edge transition specification according to ISO 21204. Figure 9.15 shows an approximately equivalent specification according to ISO 1101. In Fig. 9.14 the references F and B are straight lines not necessarily perpendicular to each other. In Fig. 9.15 they are perpendicular to each other. In both cases, the straightness of the edge (perpendicular to the section

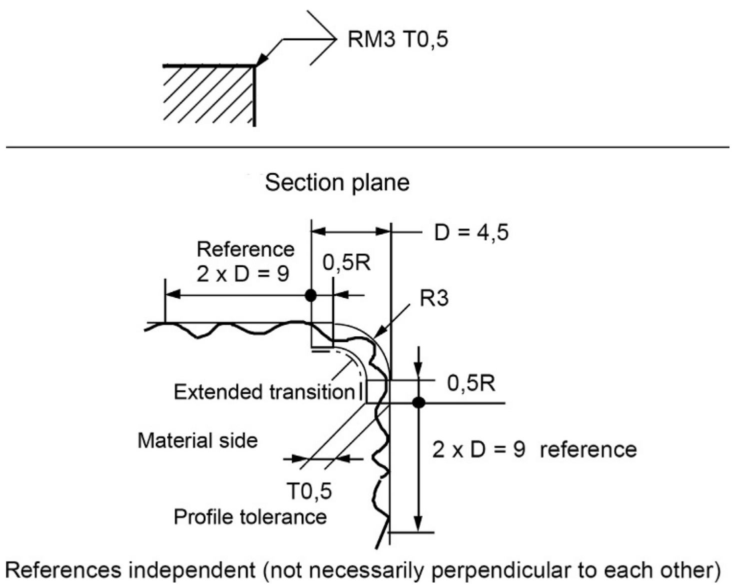


FIG. 9.14 Example of a transition specification according to ISO 21204

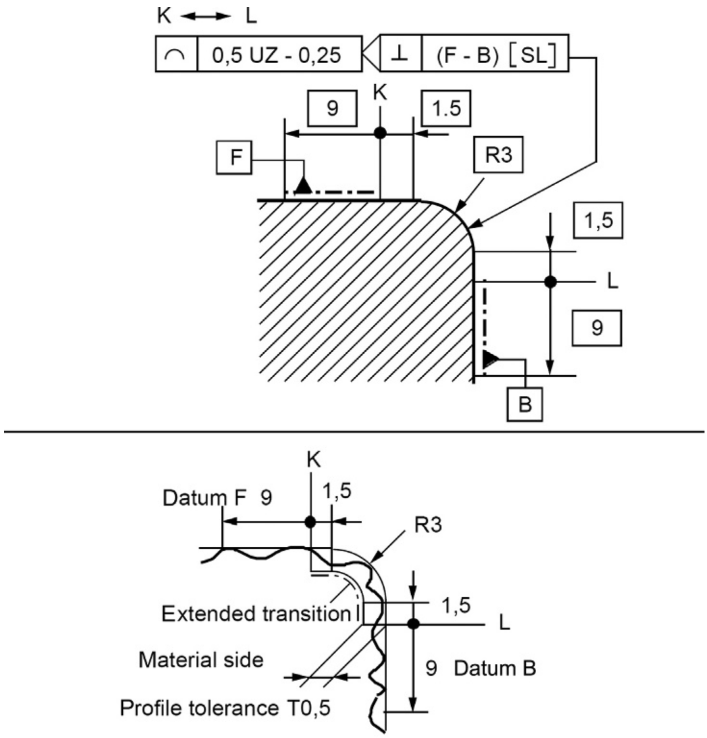


FIG. 9.15 Specification according to ISO 1101 approximately equivalent to Fig. 9.14

planes) is not limited. The datums F and B in Fig. 9.15 are perpendicular to each other, the two lines in Fig. 9.14 not.

Figure 9.16 shows a chamfered transition.

By default, the adjacent reference sections are fitted individually. If one shall be fitted first and the other constrained by it to be in theoretically exact orientation (similar to a primary and secondary datum) at the nominal angle between the two features, a triangle shall be indicated either above or below the transition symbol to indicate which reference section shall be primary; see Fig. 9.9.

According to ISO 21204, various nominal shapes of transitions are possible; see Table 9.1 and ISO 21204. The definitions of the transitions according to ISO 21204 are rather difficult and in some cases difficult to achieve in verification. Therefore it is recommended to use tolerancing according to ISO 1101; for example, see Fig. 9.15.

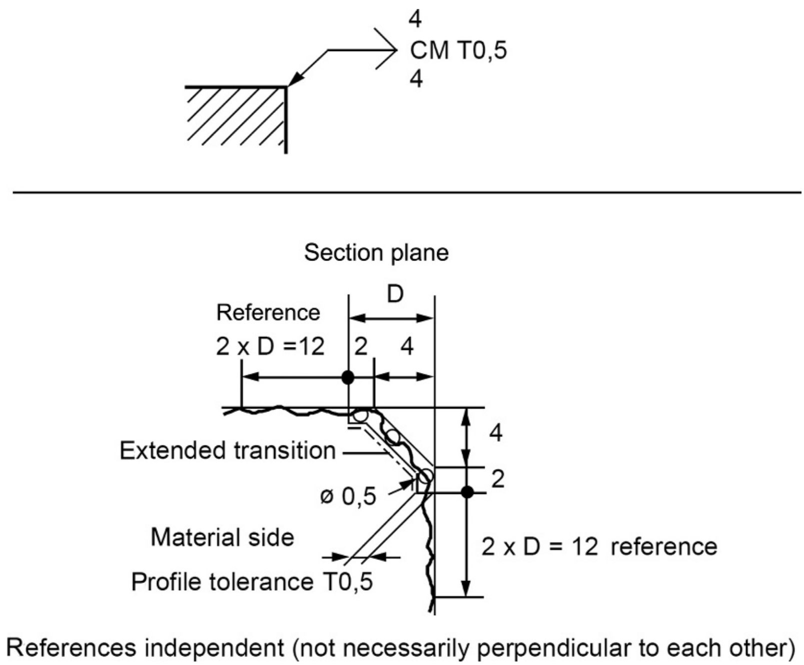


FIG. 9.16 Chamfered transition

This page intentionally left blank

Chapter 10

Tolerancing of Metallic Castings

10.1 General

The tolerancing descriptions already provided in this book also apply to tolerancing of metallic castings. However, there are additional terms, symbols and rules standardized in

ISO 10 135	Symbols for moulded parts
ISO 8062-1	Terms and definitions for metallic castings
ISO/TS 8062-2	Symbols and rules for metallic castings
ISO 8062-4	General tolerances for metallic castings

Another standard, ISO 8062-3, is applicable to general ± tolerances for metallic castings. This standard applies to ± tolerances for non-sizes and is no longer the state of the art.

10.2 Additional symbols and definitions

Table 10.1 shows additional symbols according to ISO 10 135.

Geometrical Dimensioning and Tolerancing for Design, Manufacturing and Inspection
https://doi.org/10.1016/B978-0-12-824061-8.00017-7

TABLE 10.1 Additional symbols for castings according to ISO 10 135, overview

Parting surface	M C S
Tool marking	+ -
Mismatch	SMI M C S
Flash	FL FLF
Specification extend	
Draft angle	TP TM TF
Sink	
Marking	
Undisturbed surface	
Ejector marks	
Porosity	
Surf. enlargem. ribs	

Mould	M	Gate	G	Mismatch	SMI	Taper plus	TP
Core	C	Ejector	E			Taper minus	TM
Slider	S	Riser	R	Flash	FL	Taper to fit	TF
		Vent	V	Flashfree	FLF		

Table 10.2 shows the symbols for the **surface texture** according to ISO 1302. The symbols apply to the most advanced condition of the drawing (moulded, intermediate machined, final machined).

TABLE 10.2 Symbols for the surface texture according to ISO 1302

	Machined
	Moulded
Applies for the most advanced condition (moulded, intermediate machined, final machined)	

Table 10.3 shows the symbols for the identification of the **workpiece conditions** according to ISO/TS 8062-2.

TABLE 10.3 Symbols for workpiece conditions **according to ISO/TS 8062-2**

	Moulded
	Intermediate machined
	Final machined
Without identifier the dimension or tolerance applies for all conditions	

Figure 10.1 shows the application of the workpiece condition symbols to dimensions and tolerances.

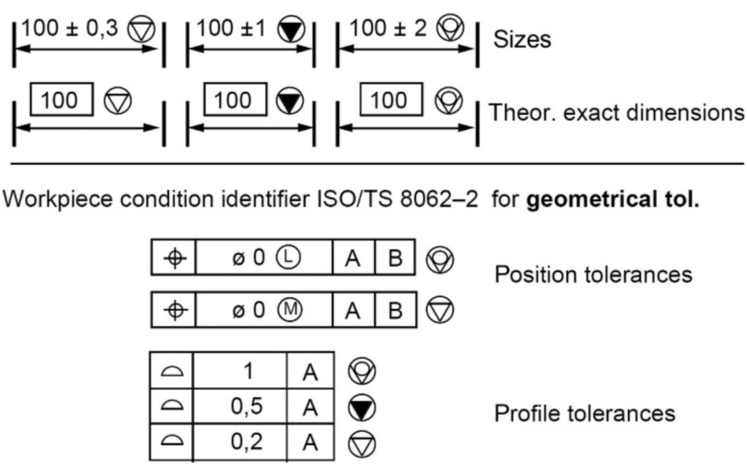


FIG. 10.1 Workpiece condition symbols applied to dimensions and tolerances

On drawings it shall be indicated, in or near the drawing title block, which workpiece conditions are dealt with on the drawing; see Table 10.4, **Drawing type identifiers**.

TABLE 10.4 Drawing type identifiers

		Moulded
		Intermediate machined
	SUP	Intermediate machined by supplier
		Final machined
	SUP	Final machined by supplier

Figure 10.2 shows an example of indicating of **parting surfaces**.

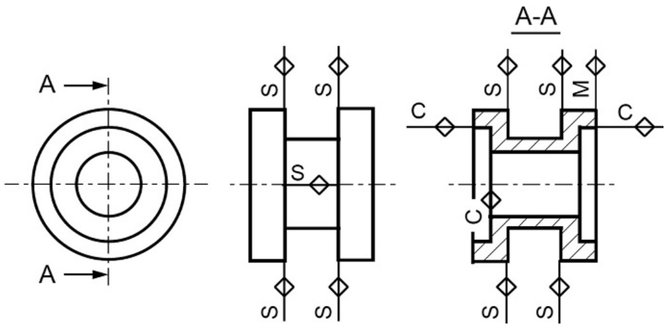


FIG. 10.2 Example of indicating of parting surfaces

Figure 10.3 shows the rules for indicating **tapers** according to ISO 10 135.

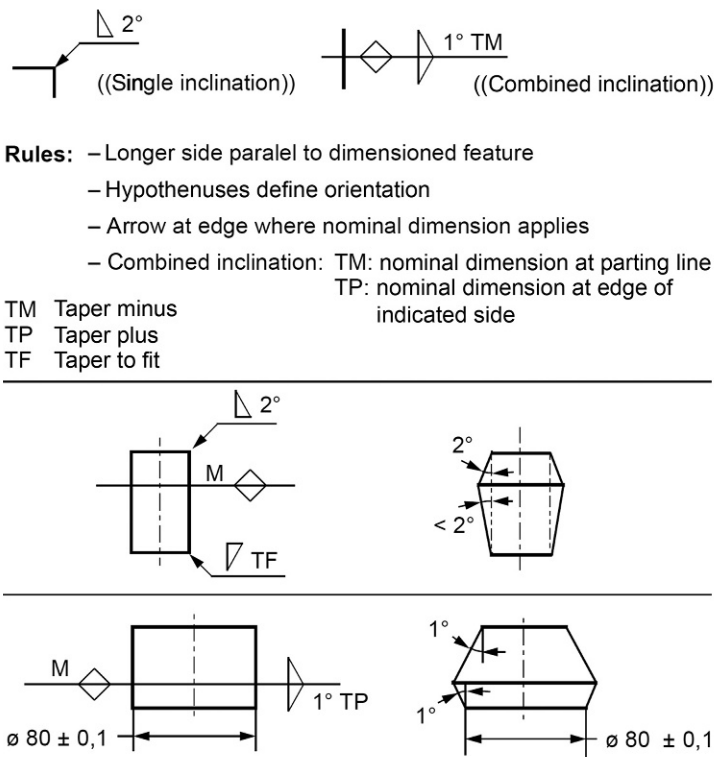


FIG. 10.3 Rules for indicating tapers according to ISO 10 135

Figure 10.4 shows a conical taper.

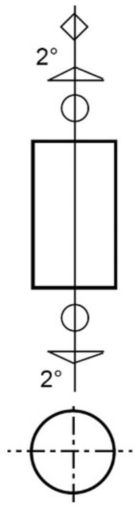


FIG. 10.4 Conical taper

Tapers (draft angles) are necessary to separate the pattern or the casting from the mould. They are nominal values. They increase the tolerance or the nominal shape; see Fig. 10.5.

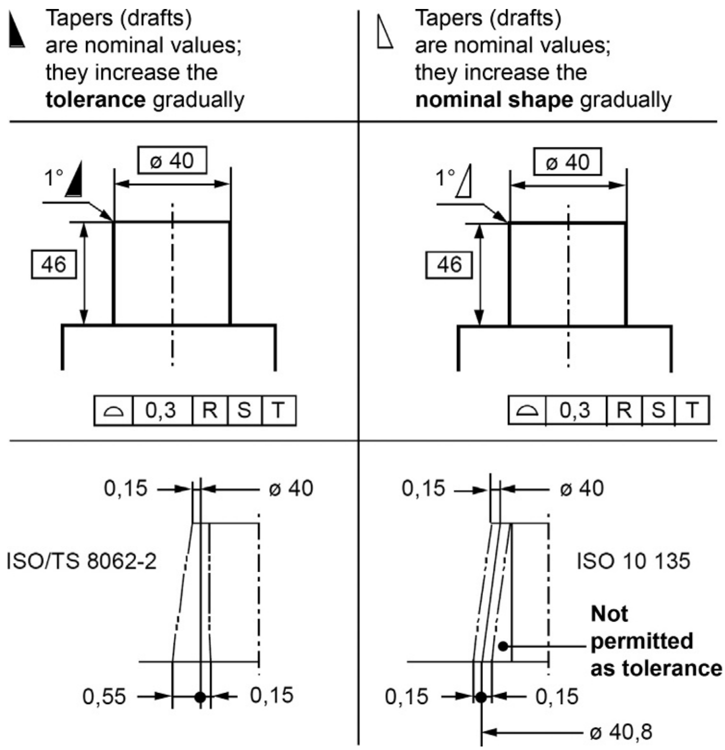


FIG. 10.5 Indicators for tapers

If not otherwise indicated, TP (taper plus, additional material) applies. In the rarely occurring cases of moulded features with Ⓜ, TM (taper minus, material subtracted) applies.

When the taper is included in the CAD model, no taper symbol occurs, but it should be stated on the drawing: "Taper in CAD model included, profile tolerance follows the model".

A distinction must be made between internal and external tapers; see Fig. 10.6. Internal tapers often require larger values than external tapers.

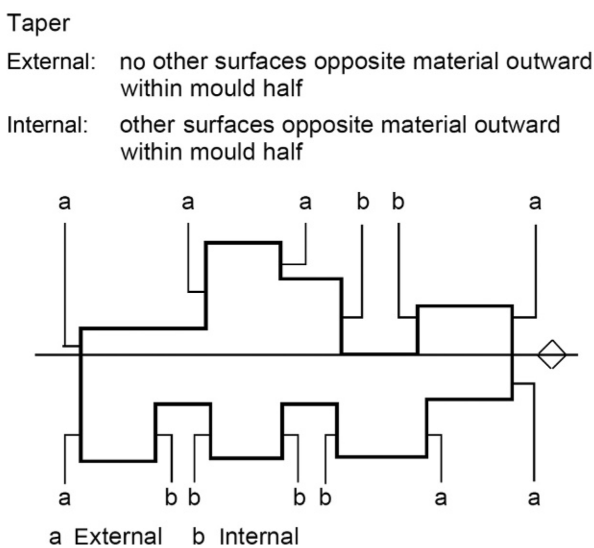


FIG. 10.6 External and internal taper

ISO 8062-4 and ISO 8062-3 give values for recommended tapers, depending on the manufacturing method and the feature length.

Figure 10.7 shows the types of **mismatch** and the indication when individually toleranced. According to ISO 10 135, mismatch must remain within the size tolerance. It is recommended to state on the drawing: "Mismatch within size or profile tolerance".

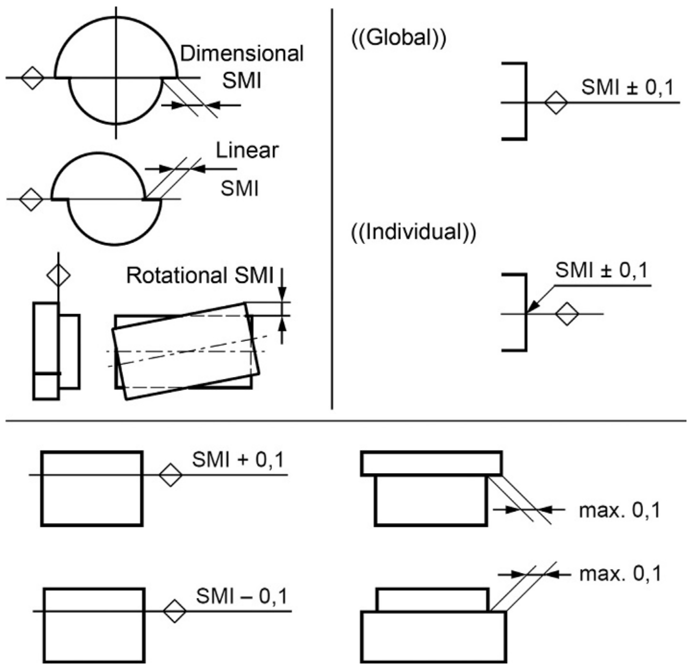


FIG. 10.7 Rules for individual indication of mismatch

Figure 10.8 shows the tolerancing of **flash** according to ISO 10 135.

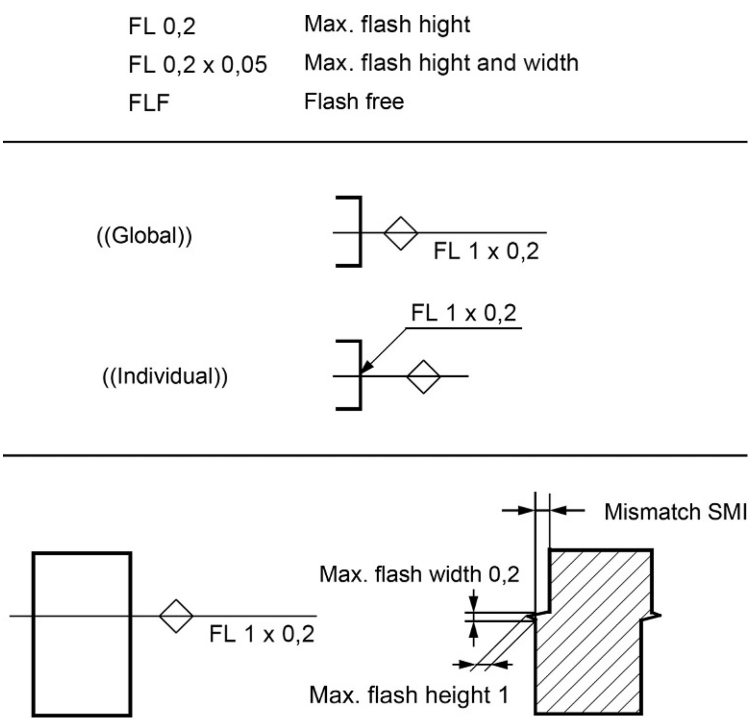


FIG. 10.8 Tolerancing of flash

Figure 10.9 shows the tolerancing of **tool marks** according to ISO 10 135.

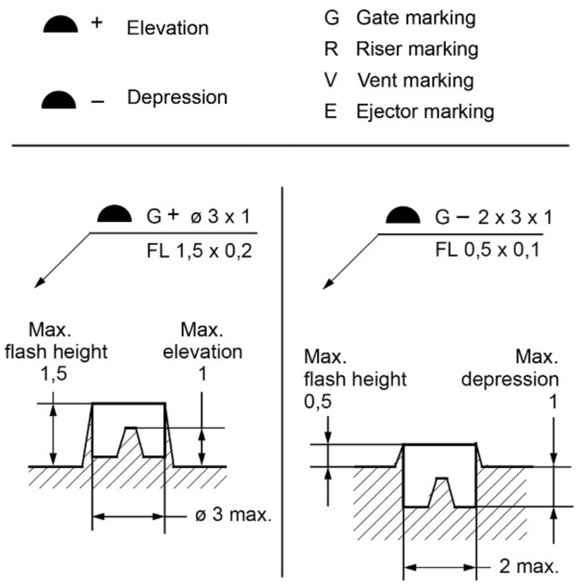


FIG. 10.9 Tolerancing of tool marks

Figure 10.10 shows the drawing indications for **multiple surface applications** according to ISO 10 135. When using these symbols in CAD models (together with the collection plane indicator, see Table 3.6), the interpretation is not easy for complicated parts. In these cases the single indication is recommended.

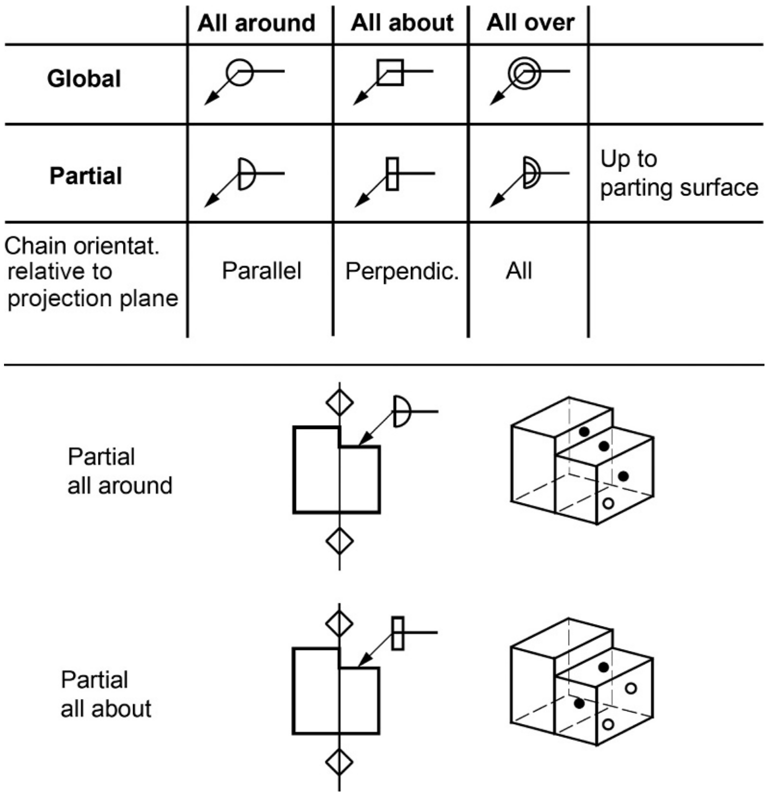


FIG. 10.10 Drawing indications of multiple surface applications

For **datums** the part condition of the toleranced feature applies; see Fig. 10.11.

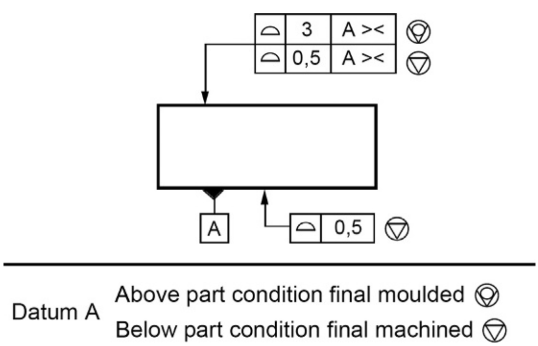


FIG. 10.11 Part conditions for datums

Figures 10.12 to 10.14 show examples of the application of datums.

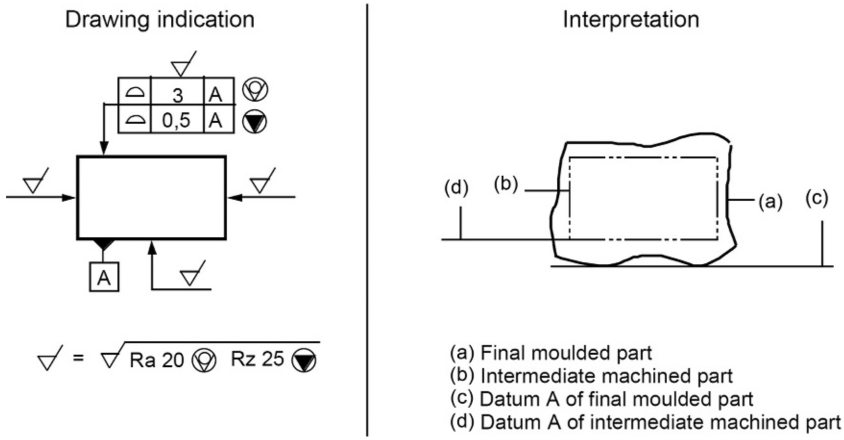


FIG. 10.12 Datums in different conditions

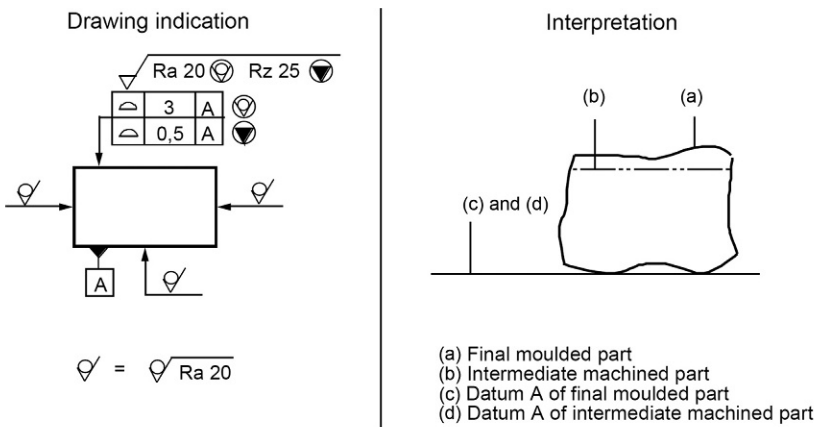


FIG. 10.13 Datums in different conditions

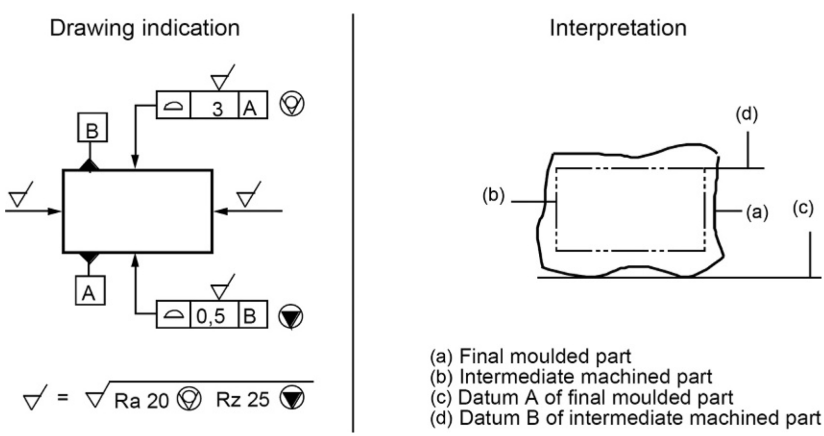


FIG. 10.14 Datums in different conditions

Required machining allowance (RMA) is a layer of material to be removed in order to reach sound material on machined surfaces. Sound material on machined surfaces refers to: layer of machining tolerance without shrinkage cavities and without slag inclusions.

The RMA is a cutting depth: i.e. for a diameter it is to be taken into account twice. The RMA applies to all surfaces to be machined with the same value, if not otherwise indicated according to ISO 1302; see Fig. 10.15.

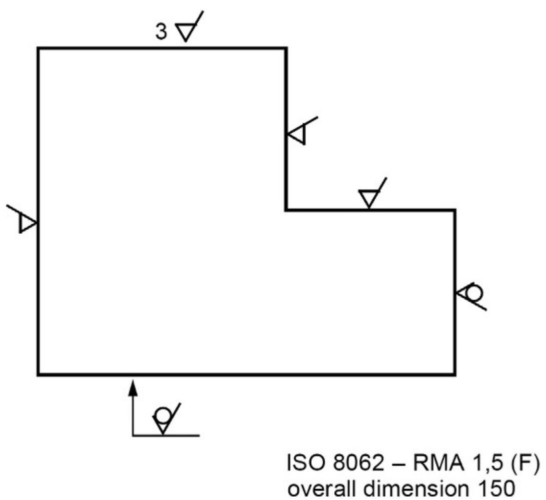


FIG. 10.15 RMA, general and individual

The RMA values are standardized in ISO 8062-4. They are rated in grades according to the overall dimension (diameter of the smallest envelope sphere, space diagonal). See Table 10.5. Table 10.6 provides guidance as to which grades are usually applied.

TABLE 10.5 Required manufacturing allowances (RMA) according to ISO 8062-4

Overall dim.											
Above	Up to	A	B	C	D	E	F	G	H	J	K
–	40	0.1	0.1	0.2	0.3	0.4	0.5	0.5	0.7	1	2
40	63	0.1	0.2	0.3	0.3	0.4	0.5	0.7	1	1.4	3
63	100	0.2	0.3	0.4	0.5	0.7	1	1.4	2	2.8	4
100	160	0.3	0.4	0.5	0.8	1.1	1.5	2.2	3	4	6
160	250	0.3	0.5	0.7	1	1.4	2	2.8	4	5.5	8

Continued

TABLE 10.5 Required manufacturing allowances (RMA) according to ISO 8062-4—cont'd

Overall dim.											
Above	Up to	A	B	C	D	E	F	G	H	J	K
250	400	0.4	0.7	0.9	1.3	1.8	2.5	3.5	5	7	10
400	630	0.5	0.8	1.1	1.5	2.2	3	4	6	9	12
630	1000	0.6	0.9	1.2	1.8	2.5	3.5	5	7	10	14
1000	1600	0.7	1	1.4	2	2.8	4	5.5	8	11	16
1600	2500	0.8	1.1	1.6	2.2	3.2	4.5	6	9	13	18
2500	4000	0.9	1.3	1.8	2.5	3.5	5	7	10	14	20
4000	6300	1	1.4	2	2.8	4	5.5	8	11	16	22
6000	10,000	1.1	1.5	2.2	3	4.5	6	9	12	17	24

TABLE 10.6 RMA grades usually applied

	Sand cast hand moulding	Sand cast machine mould and shell moulding	Metallic permanent mould	Pressure die casting	Investment casting
Steel	G to K	F to H			E
Grey iron	F to H	E to G	D to F		E
S.G. iron	F to H	E to G	D to F		E
Malleable iron	F to H	E to G	D to F		
Copper alloys	F to H	E to G	D to F	B to D	E
Zinc alloys	F to H	E to G	D to F	A to D	
Light metal alloys	F to H	E to G	D to F	B to D	E
Nickel based alloys	G to K	F to H			E
Cobalt based alloys	G to K	F to H			

10.3 Calculating of the moulded nominal dimension

Figures 10.16 to 10.18 show the default calculation of the moulded nominal dimension starting from the machined nominal dimension. In these figures the cutting depths are shown. Other methods of calculation need an agreement between the parties on the casting tolerance and/or the RMA.

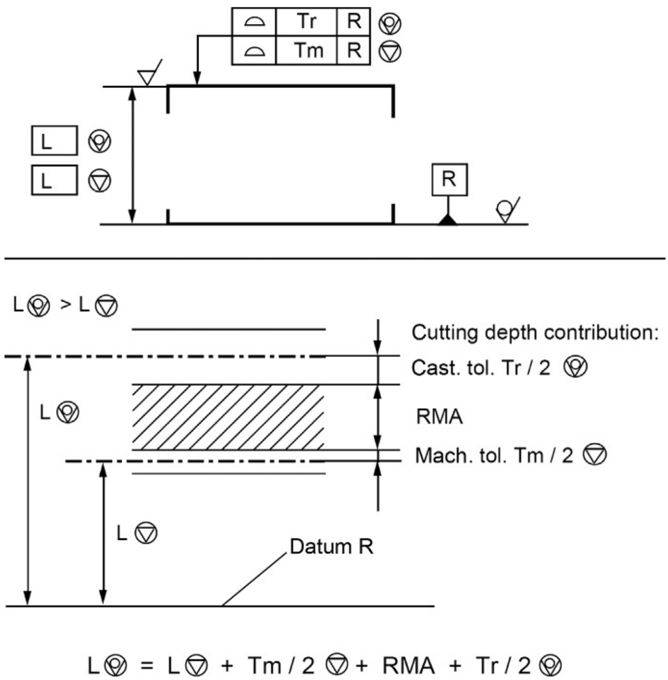


FIG. 10.16 Calculation of the moulded nominal dimension

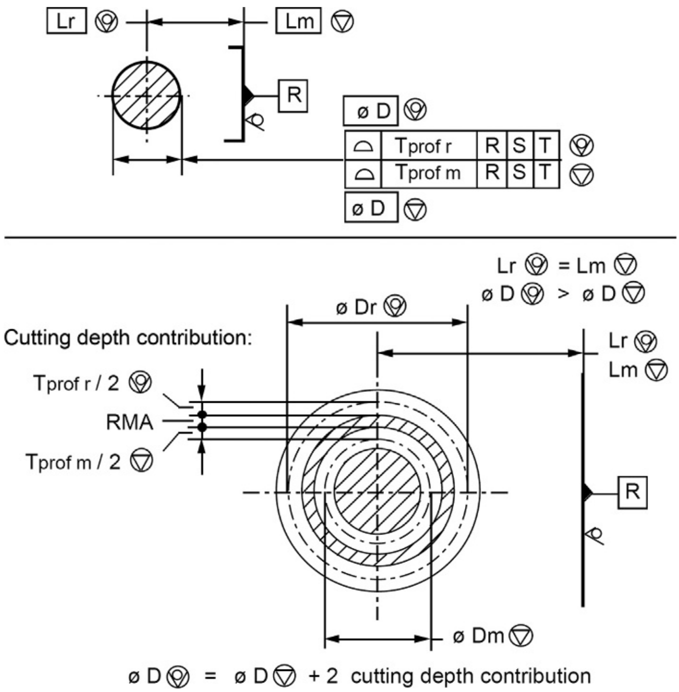


FIG. 10.17 Calculation of the moulded nominal dimension

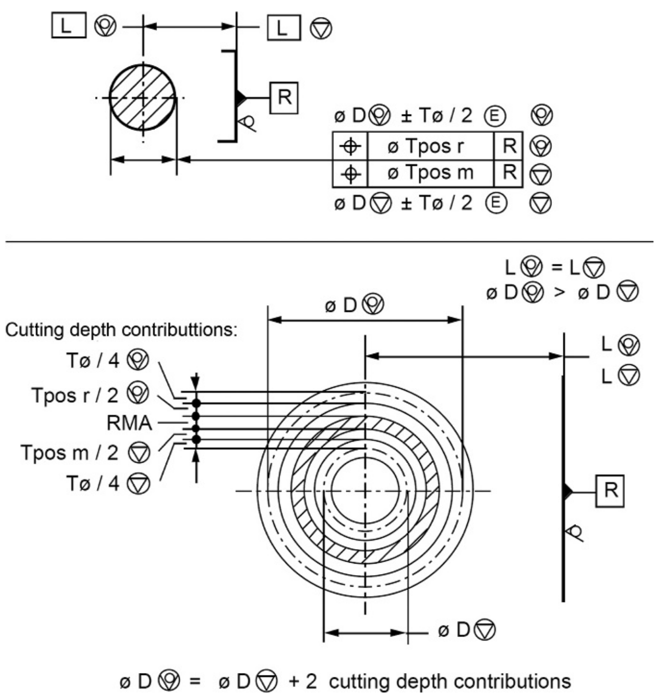


FIG. 10.18 Calculation of the moulded nominal dimension

10.4 Calculation of the minimum wall thickness

Figure 10.19 shows the calculation of the minimum wall thickness when one side is machined. The minimum wall thickness may be increased by increasing the nominal wall thickness or decreasing the casting profile tolerance or by the use of the unequally disposed modifier UZ.

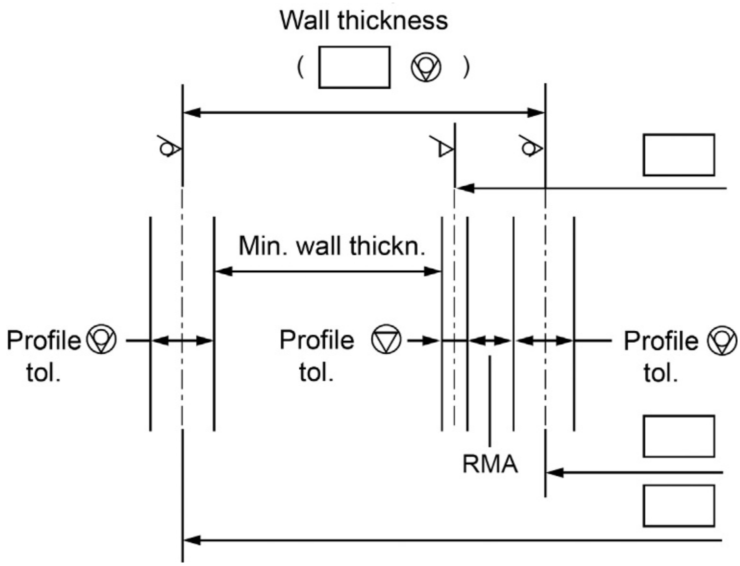


FIG. 10.19 Calculation of the minimum wall thickness when one side is machined

Figure 10.20 shows the minimum wall thickness when both sides are moulded and the wall thickness is limited by an additional ± tolerance.

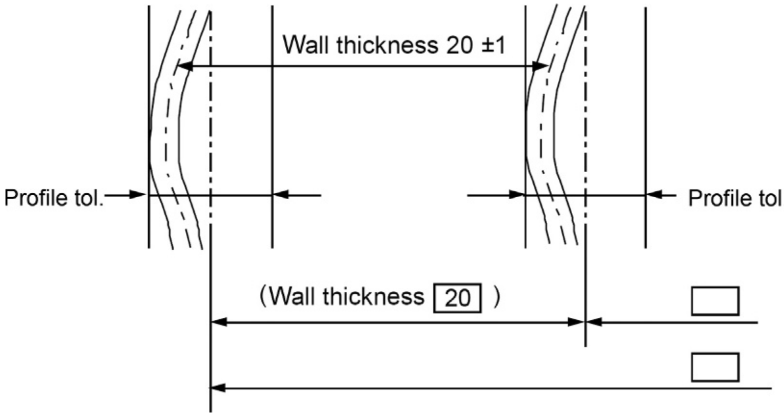


FIG. 10.20 Minimum wall thickness when both sides are moulded and the wall thickness is additionally limited by a ± tolerance

10.5 Calculation of the moulded nominal dimensions when the datum system is changed for the machined surfaces

Figure 10.21 shows the effect of height offset or inclination on the tolerance of the machined primary datum feature A relative to the originally (moulded) primary datum feature R. Additional material is required. Figure 10.22 shows the effect of offset on the tolerance of the machined secondary datum feature B relative to the originally (moulded) secondary datum feature R. Additional material is required.

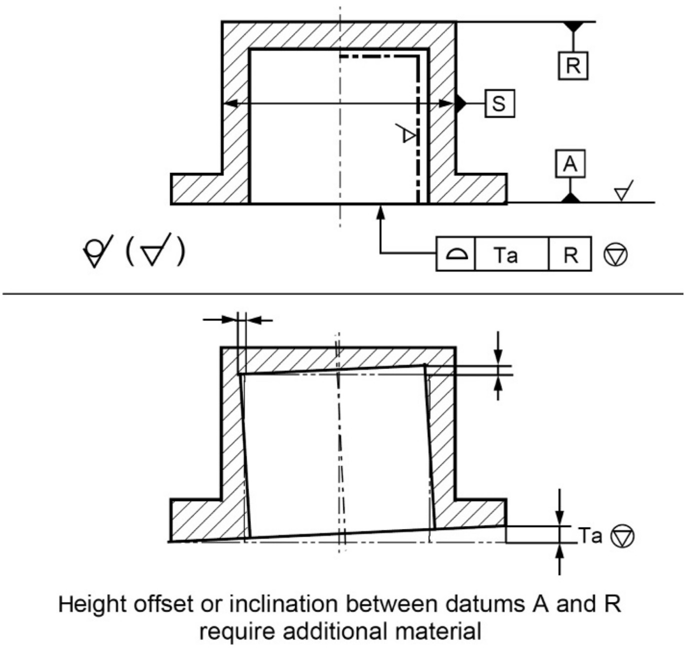


FIG. 10.21 Influence of different primary datums

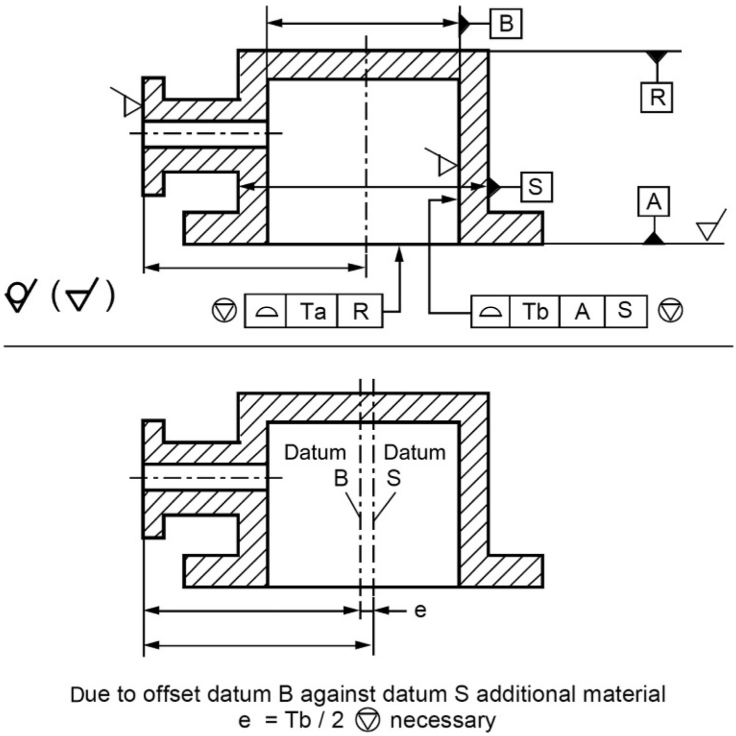


FIG. 10.22 Influence of different secondary datums

Figure 10.23 shows the combined effect of changing both datums.

The tolerances at the machined datum features should be small, because the change of the datum system requires additional material at the other machined surfaces. The calculation of the nominal dimensions of the moulded features (to be machined) can be simplified as follows:

1. Calculate the nominal moulded dimensions as shown in Figs 10.16 to 10.18.
2. Estimate conservatively the required additional material ci due to the change of the datum system.
3. Add to ci
   - for planes: profile tolerance/2 ▽
   - for sizes: profile tolerance/2▽ or
     position tolerance /2▽ + size tolerance/4 ▽
   - Conservatively rounded to the sum C.
4. Calculate the moulded dimension by the following cutting depths:

   C + RMA + moulded profile tolerance/2

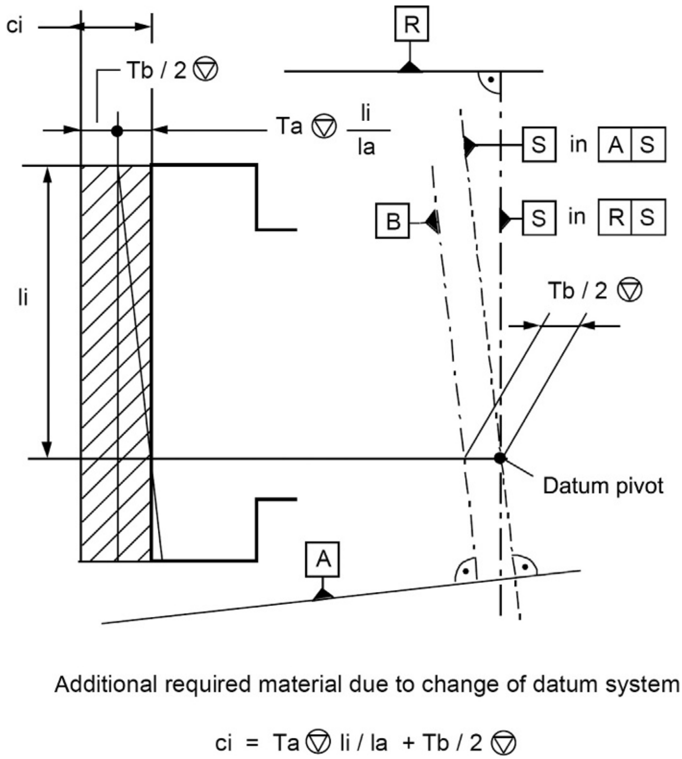


FIG. 10.23 Combined effect of changing the datum system

Figure 10.24 shows an example of dimensioning a casting without changing the datum system. Figure 10.25 shows an example involving changing the datum system, where C is estimated as 1.

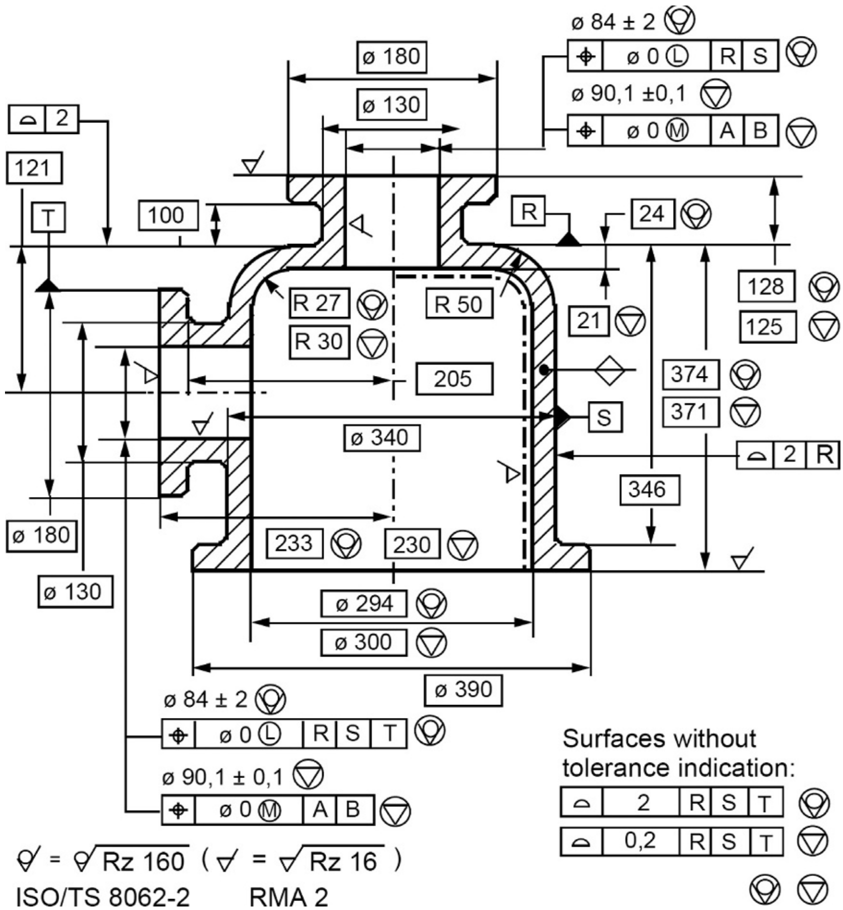


FIG. 10.24 Casting **without** changing the datum system

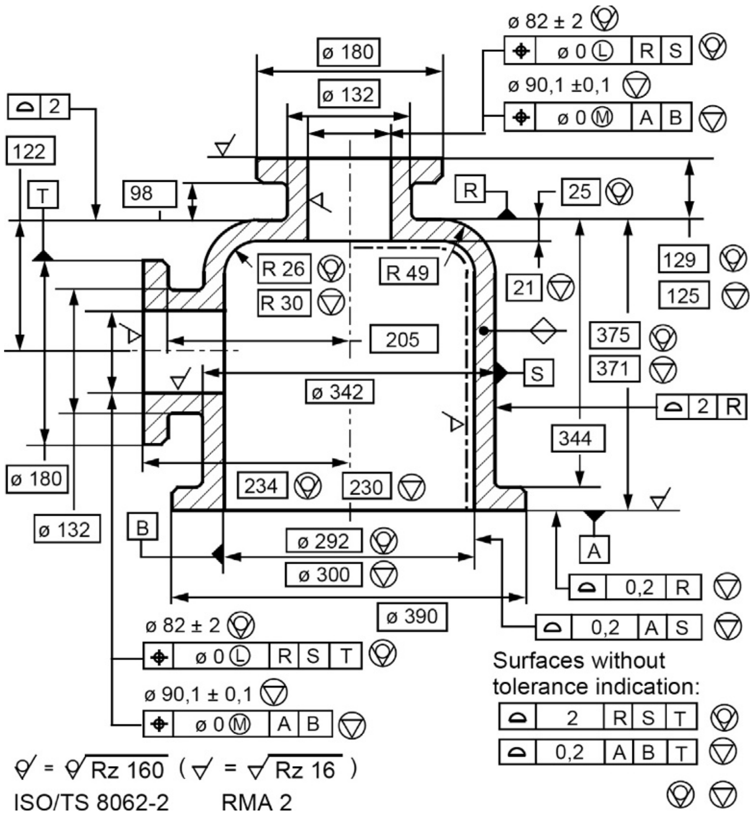


FIG. 10.25 Casting **with** changing the datum system

10.6 General tolerances for castings

See 7.6.2.

10.7 Drawings for ordering castings

10.7.1 Castings remaining as moulded

There is no machining planned. See Fig. 10.26 for an example.

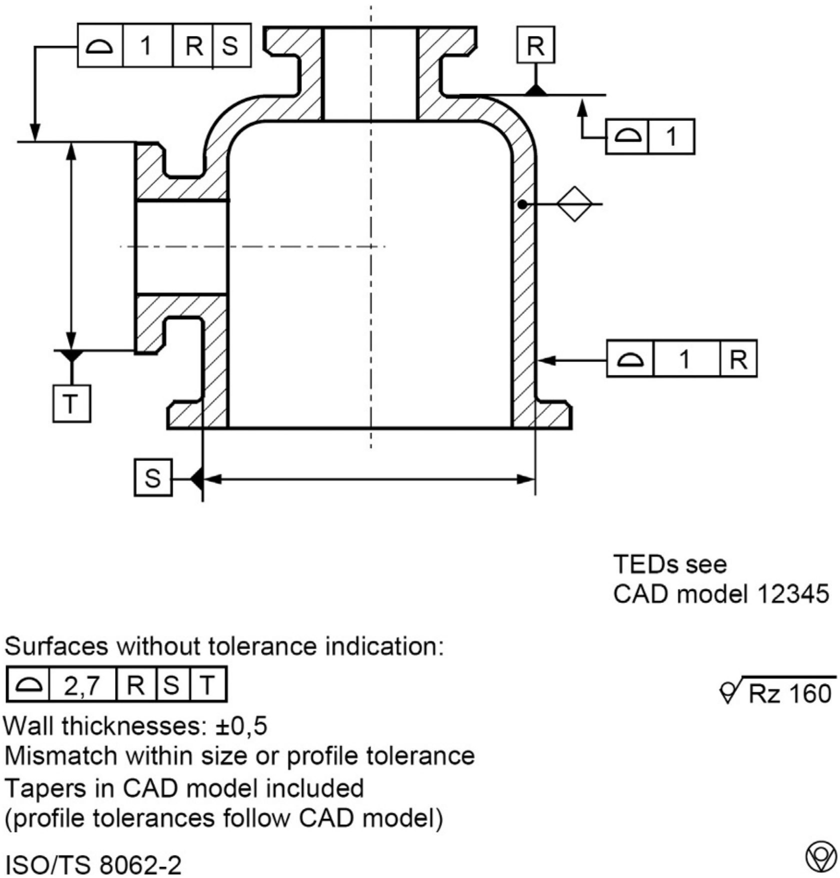


FIG. 10.26 Casting remaining as moulded

10.7.2 Casting ordered as moulded, planned machining later

The surfaces planned to be machined later, e.g. machining by the purchaser, should be identified by phantom lines. The machining allowance, RMA, shall be indicated. See Fig. 10.27 for an example.

The purchaser is responsible for ensuring that the machined part is achievable. The supplier is responsible for ensuring that, after removing the RMA material, the surfaces are on sound material.

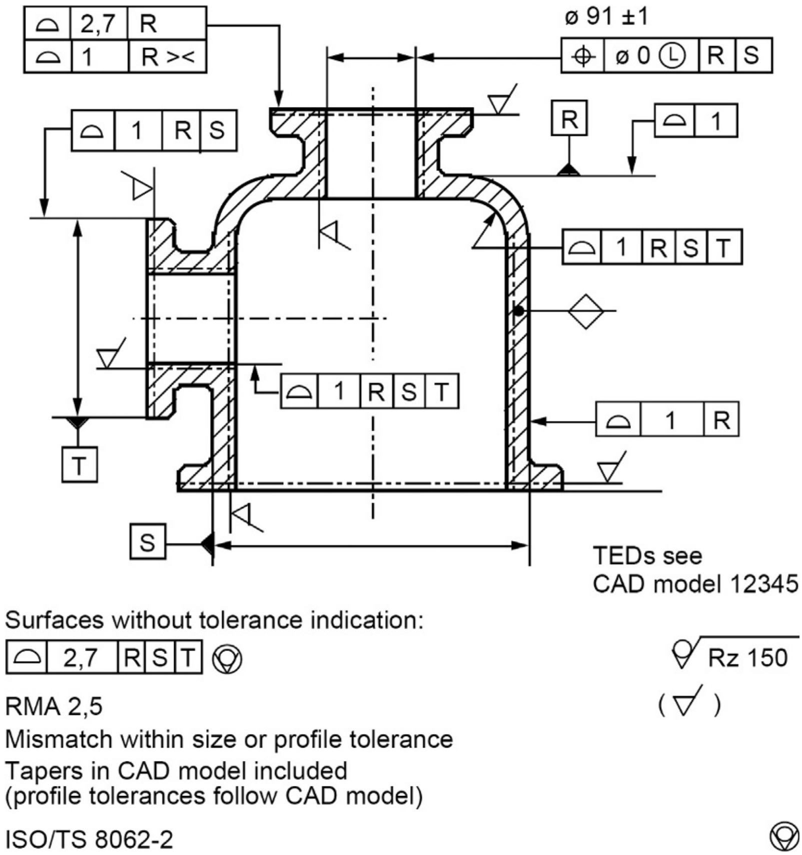


FIG. 10.27 Casting ordered as moulded, planned machining later

10.7.3 Casting ordered as premachined

The drawing shows the premachined condition with tolerances for surfaces remaining as moulded and tolerances for surfaces premachined. See Fig. 10.28 for an example.

The purchaser is responsible for ensuring that the final machined part is possible.

This method is recommended, because it gives the foundry the option of using the experience to minimize the metal to be removed.

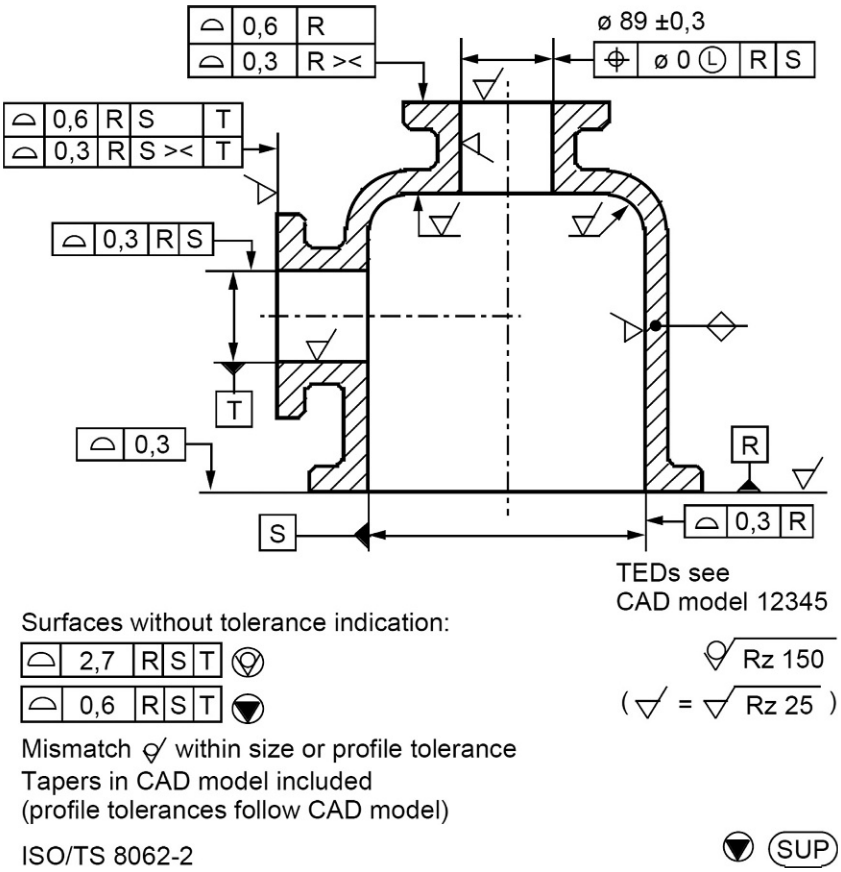


FIG. 10.28 Casting ordered as premachined

10.7.4 Casting ordered as moulded by final machined drawing

The drawing shows the final machined condition, including the tolerances for the surfaces remaining as moulded, the tolerances for the machined surfaces and the machining allowance (RMA). The dimensions for the moulded surfaces to be machined later are to be calculated by the foundry, as described in 10.3. See Fig. 10.29 for an example.

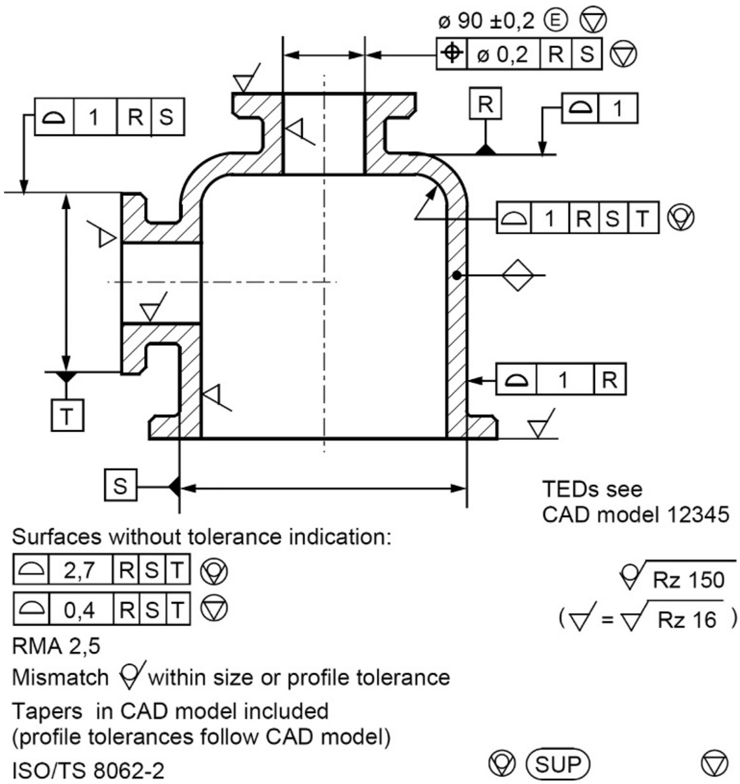


FIG. 10.29 Casting ordered as moulded by final machined drawing

10.7.5 Casting ordered as moulded by combined drawing for moulding and machining

The drawing shows both conditions, moulded and machined; see for example Figs 10.24 and 10.25.

This method leads to very complicated drawings and therefore it is not recommended for industry. Single drawings should be used for the moulded part and the machined part, i.e. for ordering the drawing according to 10.7.2 or 10.7.4.

10.8 General tolerances ISO 8062-3

For old drawings, mainly for drawings without CAD models, there are general tolerances according to ISO 8062-3. This standard uses ± tolerances for centre distances, step dimensions and contour dimensions. The disadvantages of these dimensions are described in 7.3. However, the general tolerances for orientation (parallelism, perpendicularity) together with the ± tolerances give limitations for these dimensions.

The ± tolerances for the linear dimensions are shown in Table 10.7. They are grouped in tolerance grades DCTG (dimensional casting tolerance grade)

TABLE 10.7 General dimensional tolerances

Moulded dim.		DCTG[1]															
Above	Up to	1	2	3	4	5	6	7	8	9	10	11	12	13	14	15	16
–	10	0.09	0.13	0.18	0.26	0.36	0.52	0.74	1	1.5	2	2.8	4.2				
10	16	0.1	0.14	0.2	0.28	0.38	0.54	0.78	1.1	1.6	2.2	3	4.4				
16	25	0.11	0.15	0.22	0.3	0.42	0.58	0.82	1.2	1.7	2.4	3.2	4.6	6	8	10	12
25	40	0.12	0.17	0.24	0.32	0.46	0.64	0.9	1.3	1.8	2.6	3.6	5	7	9	11	14
40	63	0.13	0.18	0.26	0.36	0.5	0.7	1	1.4	2	2.8	4	5.6	8	10	12	16
63	100	0.14	0.2	0.28	0.4	0.56	0.78	1.1	1.6	2.2	3.2	4.4	6	9	11	14	18
100	160	0.15	0.22	0.3	0.44	0.62	0.88	1.2	1.8	2.5	3.6	5	7	10	12	16	20
160	250		0.24	0.34	0.5	0.7	1	1.4	2	2.8	4	5.6	8	11	14	18	22
250	400			0.4	0.56	0.78	1.1	1.6	2.2	3.2	4.4	6.2	9	12	16	20	25
400	630				0.64	0.9	1.2	1.8	2.6	3.6	5	7	10	14	18	22	28
630	1000					1	1.4	2	2.8	4	6	8	11	16	20	25	32
1000	1600						1.6	2.2	3.2	4.6	7	9	13	18	23	29	37
1600	2500							2.6	3.8	5.4	8	10	15	21	26	33	42
2500	4000								4.4	6.2	9	12	17	24	30	38	49

DCTG 16 for wall thicknesses only.
[1] *For wall thickness one grade coarser applies.*

and rated to the nominal dimensions. For wall thicknesses, when they are explicitly dimensioned, one grade coarser applies.

There are general tolerances on form (straightness of median lines, roundness of integral surfaces, flatness of integral surfaces), on orientation (parallelism, perpendicularity, angularity tolerances are not covered of median lines and planar integral surfaces), on coaxiality and symmetry. See Table 10.8. They do not apply to tapered features. Tapered features shall be toleranced individually.

TABLE 10.8 General geometrical tolerances

Feature length 1)		Geometrical tolerance in mm for tolerance grade GCTG									
		Staightness									
		2	3	4	5	6	7	8			
		Flatness									
			2	3	4	5	6	7	8		
		Roundness, parallelism, perpendicularity, symmetry									
				2	3	4	5	6	7	8	
		Coaxiality									
Above	To				2	3	4	5	6	7	8
	10	0,08	0,12	0,18	0,27	0,4	0,6	0,9	1,4	2	3
10	30	0,12	0,18	0,27	0,4	0,6	0,9	1,4	2	3	4,5
30	100	0,18	0,27	0,4	0,6	0,9	1,4	2	3	4,5	7
100	300	0,27	0,4	0,6	0,9	1,4	2	3	4,5	7	10
300	1000	0,4	0,6	0,9	1,4	2	3	4,5	7	10	15
1000	3000	See Tables in ISO 8062-3									
3000	6000										
6000	10000										

1)Raw casting nominal length

The general tolerances on orientation for cylindrical features apply to the median lines.

The datums for orientation tolerances are to be indicated on the drawing. The datums for coaxiality and for symmetry are not indicated. When a cylindrical or plane pair feature extends over the whole workpiece, this is the datum. Otherwise a common datum applies composed of the most outward features. When there are two features, the outer applies. See Figs 10.30 and 10.31.

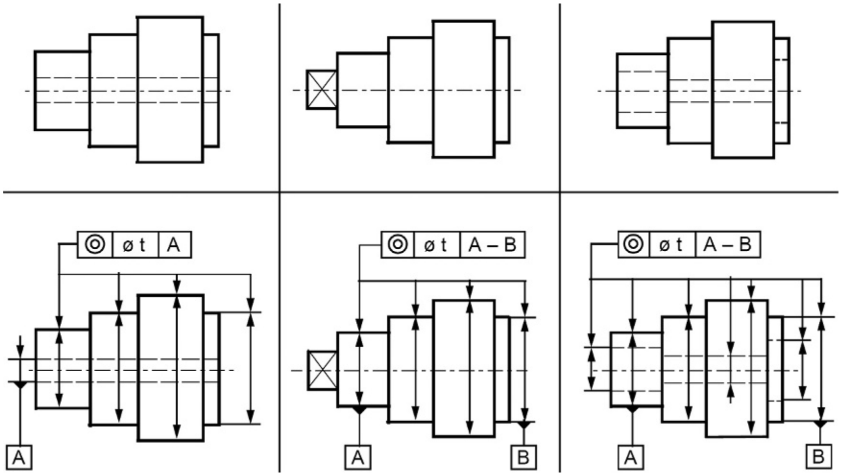


FIG. 10.30 Datums for general coaxiality tolerances

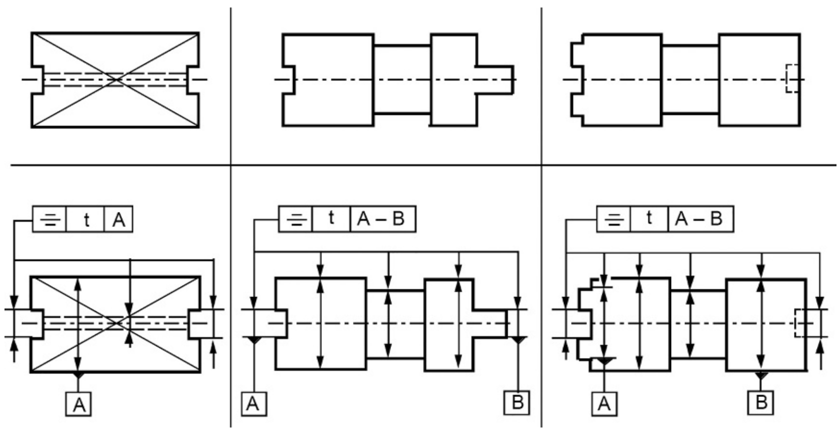


FIG. 10.31 Datums for general symmetry tolerances

The general tolerances for coaxiality and symmetry apply also to the datum features themselves, if a common datum applies. See Figs 10.30 and 10.31.

The nominal dimension to be used in Table 10.8 shall be the longest dimension of the considered integral surface, disregarding fillets or chamfers that are not individually indicated.

For the selection of the tolerance grades, see Tables 10.9 to 10.11. These are possible tolerance grades. The actual tolerance grade is to be agreed upon with the foundry.

TABLE 10.9 General dimensional casting tolerance grades (DCTG) for short series and single production

	Sand cast hand moulding	
	Clay-bounded	Chemically bounded
Steel	13–15	12–14
Grey iron	13–15	11–14
S.G. iron	13–15	11–14
Malleable iron	13–15	11–14
Copper alloys	13–15	10–13
Zinc alloys		
Light metal alloys	11–13	10–13
Nickel based alloys	13–15	12–14
Cobalt based alloys	13–15	12–14

The values in this table apply to nominal dimensions >25 mm.
For smaller dimensions, finer tolerances can be held as follows:
nominal dimensions ≤10 mm, three grades finer;
nominal dimensions >10 mm to ≲16 mm, two grades finer;
nominal dimensions >16 mm to ≲25 mm, one grade finer.

TABLE 10.10 General dimensional casting tolerance grades (DCTG) for long series production

	Sand cast hand moulding	Sand cast machine mould and shell moulding	Metallic permanent mould	Pressure die casting	Investment casting
Steel	11–14	8–12			2)
Grey iron	11–14	8–12	7–9		2)
S.G. iron	11–14	8–12	7–9		2)
Malleable iron	11–14	8–12	7–9		
Copper alloys	10–13	8–10	7–9	6–8	2)

TABLE 10.10 General dimensional casting tolerance grades (DCTG) for long series production—cont'd

	Sand cast hand moulding	Sand cast machine mould and shell moulding	Metallic permanent mould	Pressure die casting	Investment casting
Zinc alloys	10–13	8–10	7–9	3–6	
Light metal alloys	9–12	7–9	6–8	1)	2)
Nickel based alloys	11–14	8–12			2)
Cobalt based alloys	11–14	8–12			2)

1) For the largest overall dimension (diameter of envelope sphere):

≲50 mm		DCTG 6
>50 mm to	≲180 mm	DCTG 7
>180 mm to	≲500 mm	DCTG 8
>500 mm		DCTG 9

2) For the largest over all dimension (diameter of envelope sphere):

	≲100 mm	DCTG 4–6
>100 mm to	≲400 mm	DCTG 4–8
>400 mm		DCTG 4–9

TABLE 10.11 General geometrical casting tolerance grades (GCTG)

	Sand cast hand moulding	Sand cast machine mould and shell moulding	Metallic permanent mould	Pressure die casting	Investment casting
Steel	6–8	5–7			[1)]
Grey iron	5–7	4–6			3–5
S.G. iron	5–7	4–6			3–5
Malleable iron	5–7	4–6			3–5
Copper alloys	5–7	4–6	3–5	2–4	3–5
Zinc alloys	5–7	4–6		2–4	2–4
Light metal alloys	5–7	7–9	3–5	2–4	3–5
Nickel based alloys	6–8	5–7			[1)]
Cobalt based alloys	6–8	5–7			[1)]

[1)]For the largest overall dimension (diameter of envelope sphere):

	$\lesssim$100 mm	GCTG 4–6
>100 mm to	$\lesssim$400 mm	GCTG 4–8
>400 mm		GCTG 4–9

Figure 10.32 shows the limitation of the deviation of the cylinder by the $\pm$ diameter tolerance and the straightness tolerance t_m of the cylinder median line.

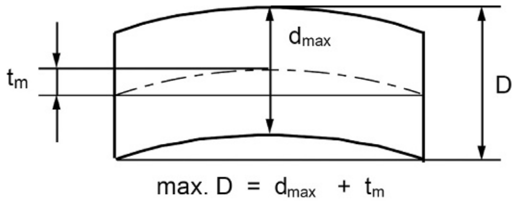


FIG. 10.32 Maximum envelope cylinder D, ± diameter tolerance and straightness tolerance t_m of the median line

Mismatch (see Fig. 10.7) as a default condition is controlled by the tolerances applied to the linear dimensions according to Table 10.7. The size tolerance shall be respected.

For the **required machining allowance (RMA)** the same applies as in ISO 8062-4; see Tables 10.2 and 10.3.

Figure 10.33 gives an example of a casting toleranced according to ISO 8062-3. The parting surface (taper) is not considered in this example.

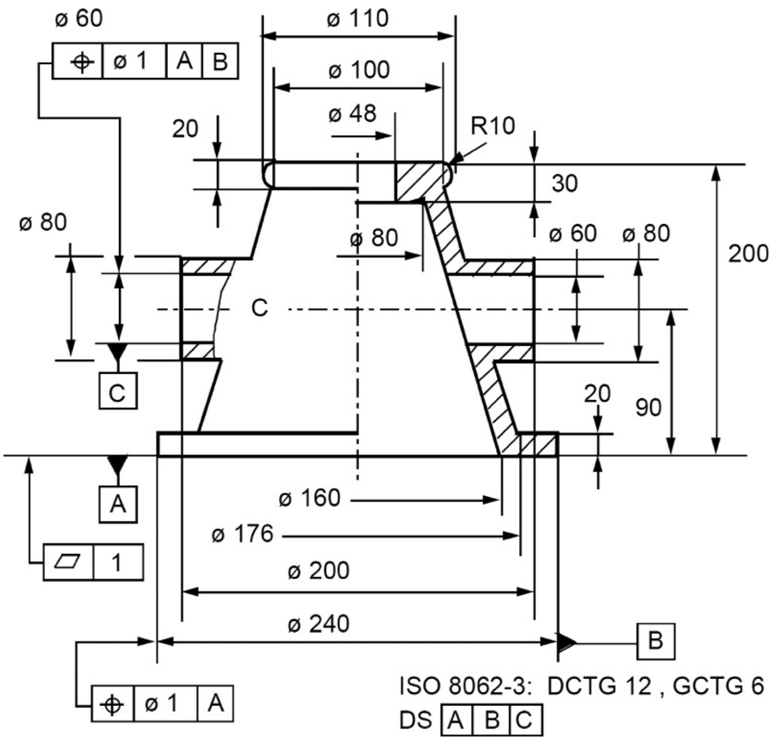


FIG. 10.33 Drawing with general tolerances according to ISO 8062.3

Figure 10.34 shows the datum system. The three situation features (datums A, B, C) form three datum planes perpendicular to each other (Figs 10.38 and 10.39).

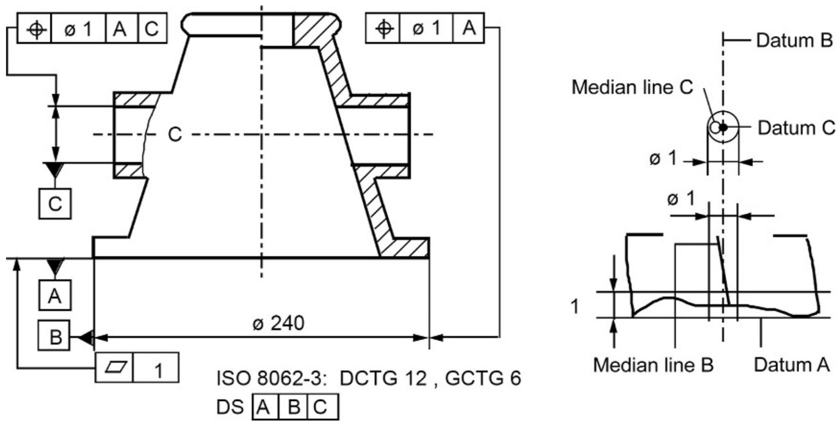
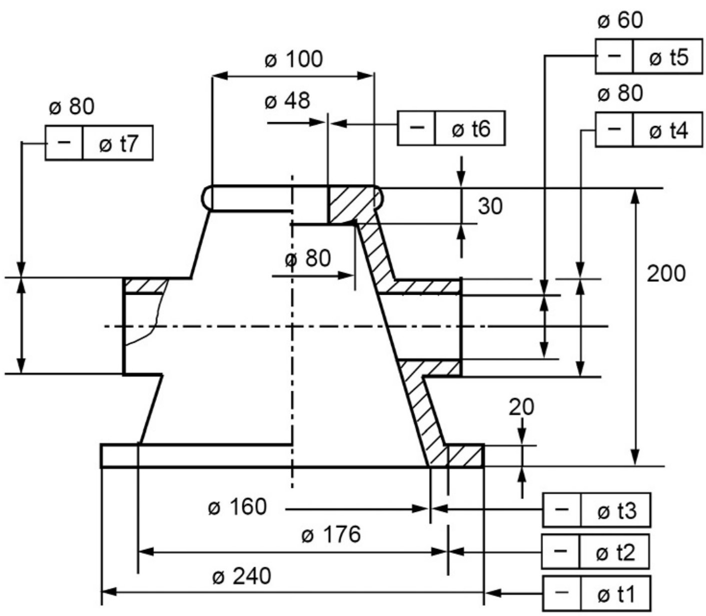
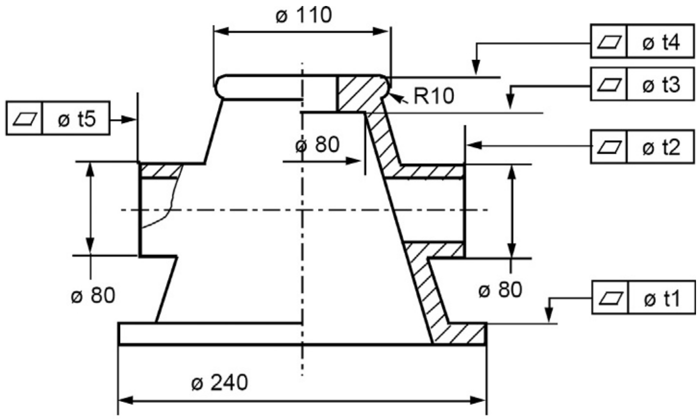


FIG. 10.34 Datum system for orientation tolerances according to ISO 8062-3



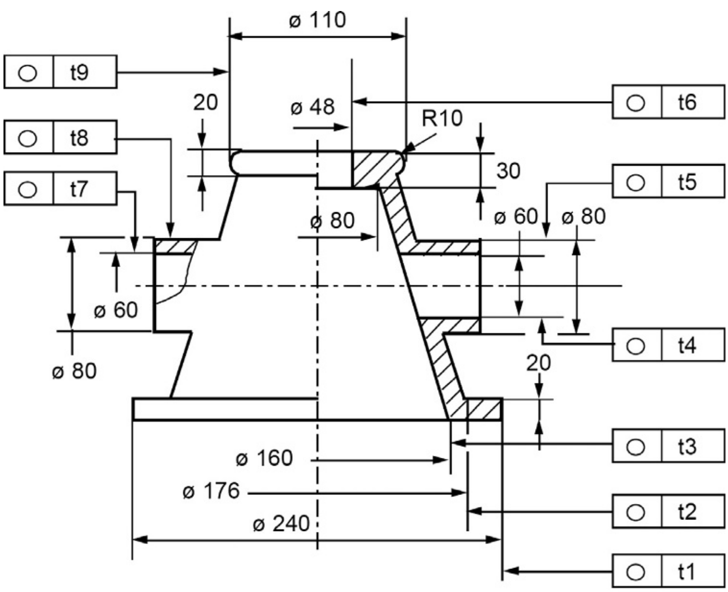
t1 = 1,4 (longest dimension = Ø 240)
t2 = 1,4 (longest dimension = Ø 176)
t3 = 1,4 (longest dimension = 200 – 30 = 170)
t4 = 0,9 (longest dimension = Ø 80)
t5 = 0,9 (longest dimension = Ø 60)
t6 = 0,9 (longest dimension = Ø 48)
t7 = 0,9 (longest dimension = Ø 80)

FIG. 10.35 General tolerances for straightness of median lines



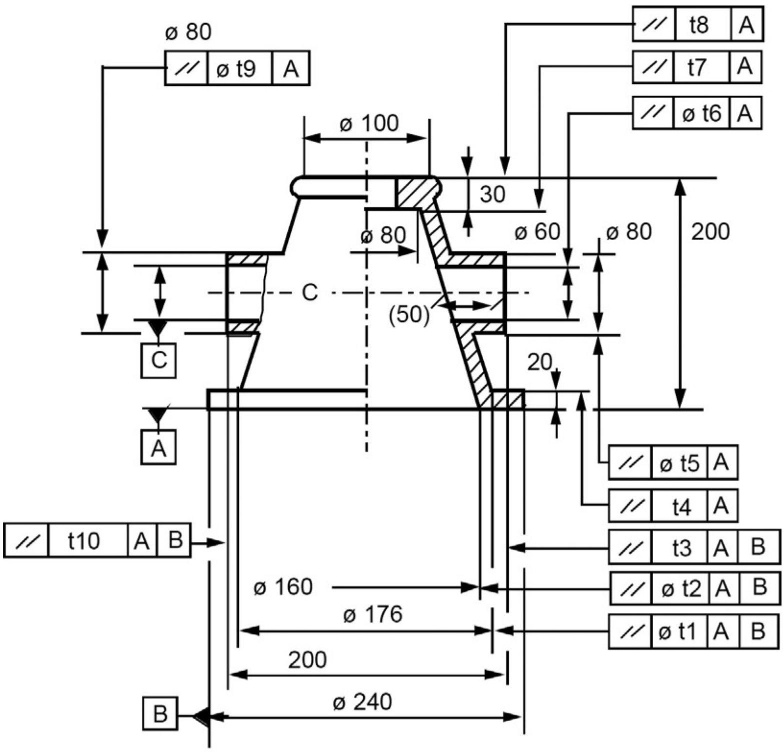
t1 = 2 (longest dimension = Ø 240)
t2 = 2 (longest dimension = Ø 80)
t3 = 1,4 (longest dimension = Ø 80)
t4 = 1,4 (longest dimension = Ø 110 -20 = Ø 90)
t5 = 1,4 (longest dimension = Ø 80)

FIG. 10.36 General tolerances for flatness of integral surfaces



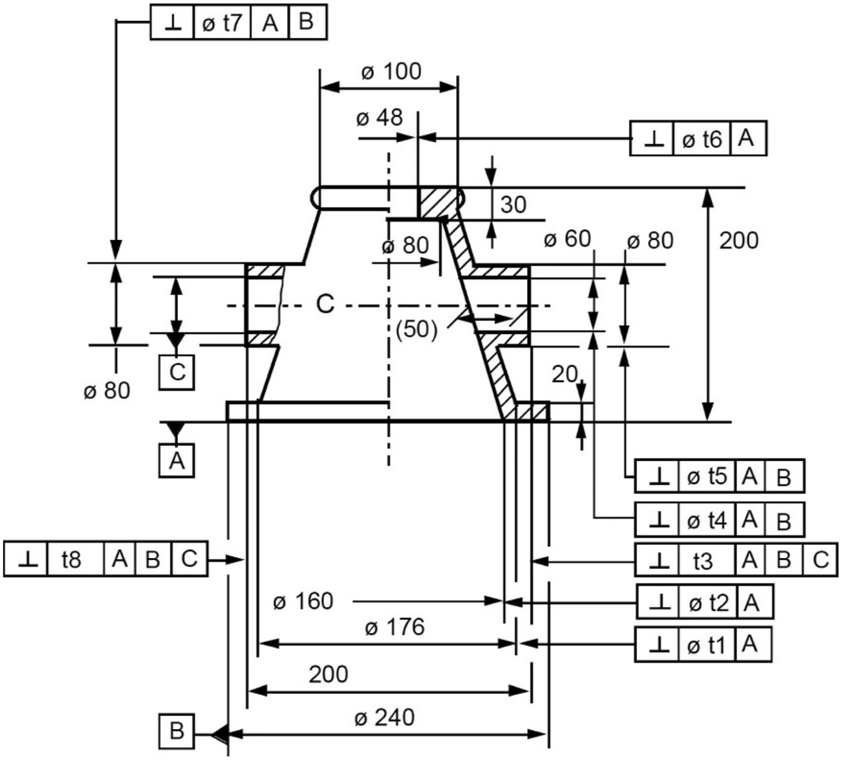
t1 = 3 (longest dimension = Ø 240)
t2 = 3 (longest dimension = Ø 176)
t3 = 3 (longest dimension = Ø 160)
t4 = 2 (longest dimension = Ø 60)
t5 = 2 (longest dimension = Ø 80)
t6 = 2 (longest dimension = Ø 48)
t7 = 2 (longest dimension = Ø 60)
t8 = 2 (longest dimension = Ø 80)
t9 = 3 (longest dimension = Ø 110)

FIG. 10.37 General tolerances for roundness



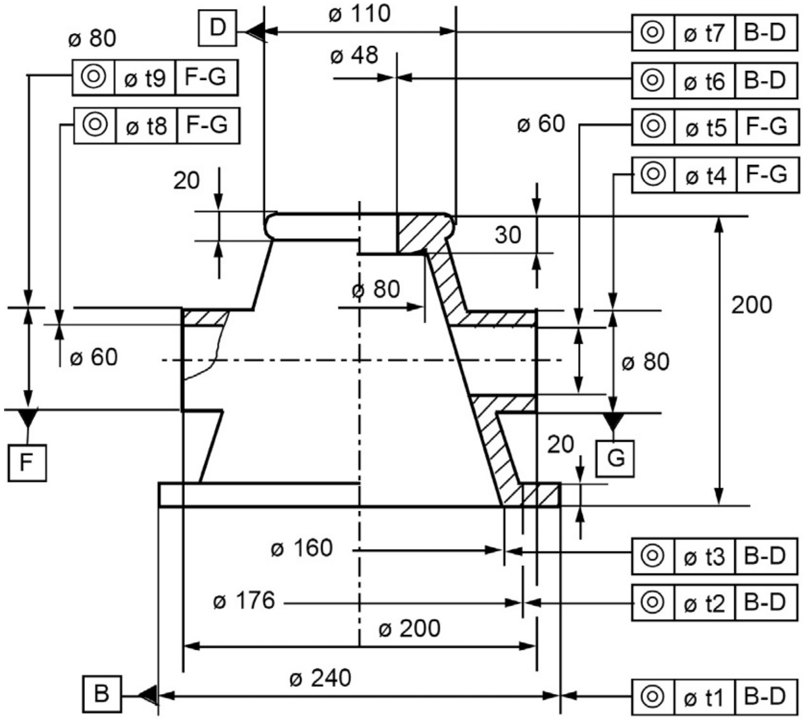
t1 = 3 (longest dimension = Ø 176)
t2 = 3 (longest dimension = 200 – 30 = 170)
t3 = 2 (longest dimension = Ø 80)
t4 = 3 (longest dimension = Ø 240)
t5 = 2 (longest dimension = Ø 80)
t6 = 2 (longest dimension = Ø 60)
t7 = 2 (longest dimension = Ø 80)
t8 = 2 (longest dimension = Ø 100)
t9 = 2 (longest dimension = Ø 80)
t10 = 2 (longest dimension = Ø 80)

FIG. 10.38 General tolerances for parallelism



t1 = 3 (longest dimension = Ø 176)
t2 = 3 (longest dimension = 200 – 30 = 170)
t3 = 2 (longest dimension = Ø 80)
t4 = 2 (longest dimension = Ø 60)
t5 = 2 (longest dimension = Ø 80)
t6 = 2 (longest dimension = Ø 48)
t7 = 2 (longest dimension = Ø 80)
t8 = 2 (longest dimension = Ø 80)

FIG. 10.39 General tolerances for perpendicularity



t1 = 4,5 (longest dimension = Ø 240)
t2 = 4,5 (longest dimension = Ø 176)
t3 = 4,5 (longest dimension = 200 – 30 = 170)
t4 = 3 (longest dimension = Ø 80)
t5 = 3 (longest dimension = Ø 60)
t6 = 3 (longest dimension = Ø 48)
t7 = 4,5 (longest dimension = Ø 110)
t8 = 3 (longest dimension = Ø 60)
t9 = 3 (longest dimension = Ø 80)

FIG. 10.40 General tolerances for coaxiality

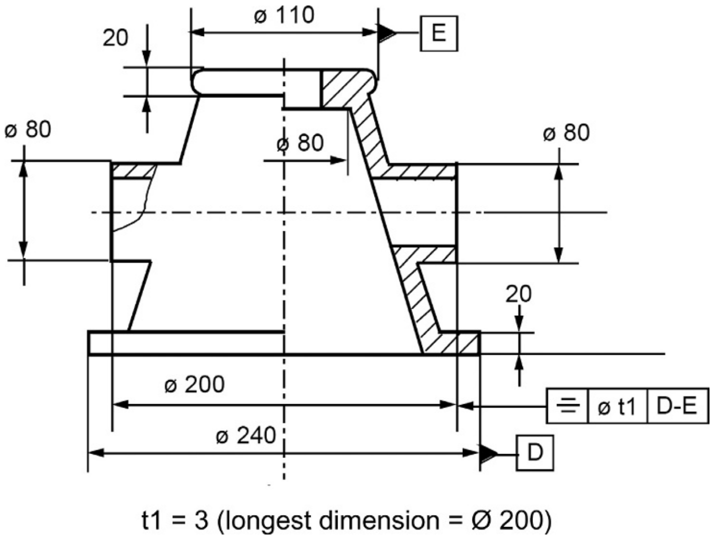


FIG. 10.41 General tolerance for symmetry

As the datums are perpendicular to each other and the tolerances for parallelism and perpendicularity are the same amount, they are equal.

This page intentionally left blank

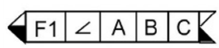
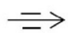
Chapter 11

Tolerancing of Movable Assemblies

11.1 Symbols

For movable assemblies (parts which can move relative to each other), e.g. roller bearings, additional symbols have been standardized according to ISO 17863; see Table 11.1.

TABLE 11.1 Symbols for movable assemblies

Description	Symbol
Force indicator with direction of the force	
Complementary force indicator	
Gravity direction	
Flag note	
Mobility, translational	
Mobility, rotational	
Movable part	MP
Fixed part	FP
Translational mobility, positive direction	T+
Translational mobility, negative direction	T−
Rotational mobility, positive direction	R+
Rotational mobility, negative direction	R−

Geometrical Dimensioning and Tolerancing for Design, Manufacturing and Inspection
https://doi.org/10.1016/B978-0-12-824061-8.00009-8

11.2 Rules

Force shall be applied on the movable part. (Force may also be applied on the fixed part to maintain a stable location.)

The direction of the force is to be indicated by the symbols //, ⊥, <, ≡ in the force indicator.

All degrees of freedom that are not necessary for the concerned characteristic shall be locked.

By default, the part with tolerance indications is considered the movable part and the part with datum(s) is considered the fixed part.

Locking degrees of freedom shall be realized by application of forces.

By default, the force is evenly distributed at the integral feature. The force can be the force of gravity; see the symbol in Table 11.1.

If the force is restricted to a point, the force indicator line terminates at a cross, dimensioned by TEDs relative to a datum system.

If the force is restricted to a straight line, the line is indicated by a long dashed narrow line between crosses, dimensioned by TEDs relative to a datum system.

If the force is restricted to a restricted area, this area shall be indicated as a hatched area surrounded by long-dashed narrow lines, dimensioned by TEDs relative to a datum system. The force indicator line terminates at a point within the area.

By default, the force direction is perpendicular to the concerned feature.

If necessary, the direction of mobility shall be indicated; see Table 11.1.

The velocity of mobility may be indicated:

translational mobility with the symbol ν, e.g.: $\nu = 15\,\text{m/s}$,
rotational mobility with the symbol ω, e.g.: $\omega = 3\,\text{rad/s}$,
or with the symbol RPM, e.g.: $\text{RPM} = 500\,\text{min}^{-1}$.

11.3 Examples

Figure 11.1 shows the indication of the direction of force by using a force indicator in combination with a complementary force indicator. Figure 11.2 shows an example of the indication of the direction of rotational mobility. Figure 11.3 shows an example of the indication of force indicators for a roller bearing.

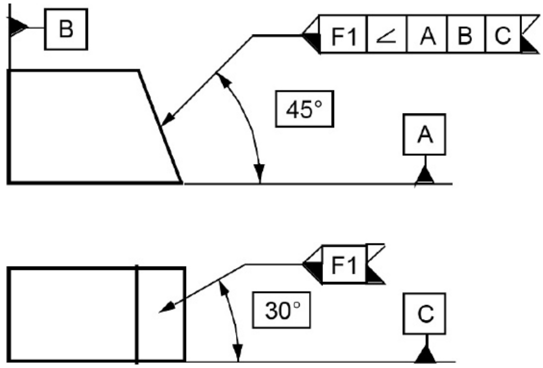


FIG. 11.1 Direction of force indicated by using a force indicator in combination with a complementary force indicator

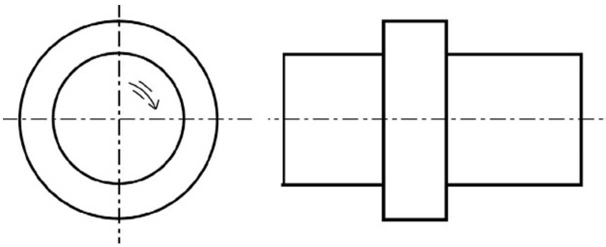


FIG. 11.2 Example of indication of the direction of rotational mobility of the movable part relative to the fixed part

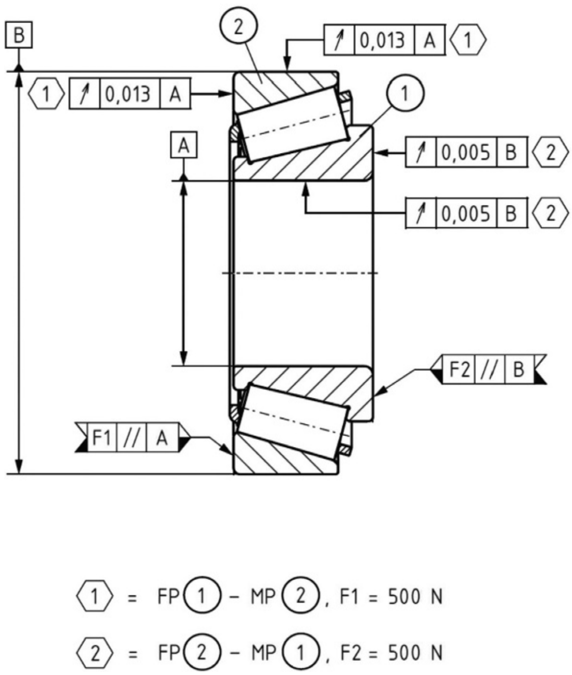


FIG. 11.3 Example of indication of forces in the case of a roller bearing, according to ISO 17863

Chapter 12

Respecting Geometrical Tolerances During Manufacturing

12.1 Manufacturing influences

Geometrical deviations are influenced by the following, which are sometimes referred to as the '5 Ms'.

Material	– Rigidity of the workpiece (shape)
	– Material
	– Stress in the material
Machine (tool)	– Precision of the machine tool, bearing play
	– Static and dynamic rigidity of the machine tool
	– Thermal properties of the machine tool
	– Maintenance
	– Environment (e.g. vibrations)
Method	– Tool
	– Chuck, fixing, clamping method
	– Processing data (e.g. cutting speed, thickness of cut, cutting pressure, cooling)
Measuring	– Uncorrected systematic measuring deviations
	– Random measuring deviations
Manufacturer	– Education, skill, precision of re-chucking
	– Environment

The following figures show some results of investigations in the field of metal removal processes. These results are only examples and are not general.

Figure 12.1 shows the mean values x and the variations of measured roundness deviations of workpieces made from steel, ø 20 mm, manufactured with certain machine tools.

Figure 12.2 shows the ability of manufacturing devices to respect tolerances of roundness, cylindricity and coaxiality by turning and grinding.

The machine tools are to be classified according to their abilities. The abilities related to the workpiece material and shape and process data (e.g. cutting

Geometrical Dimensioning and Tolerancing for Design, Manufacturing and Inspection
https://doi.org/10.1016/B978-0-12-824061-8.00019-0

speed, thickness of cut, chucking method) are to be recorded for manufacturing planning.

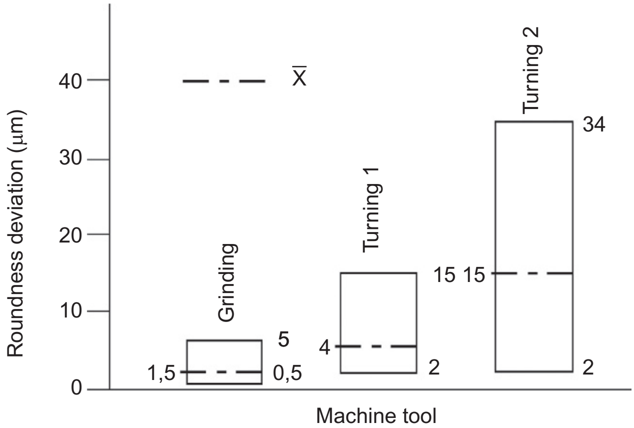


FIG. 12.1 Ability of machine tool, mean values and variations of roundness deviations of workpieces made out of steel ø 20 mm, example

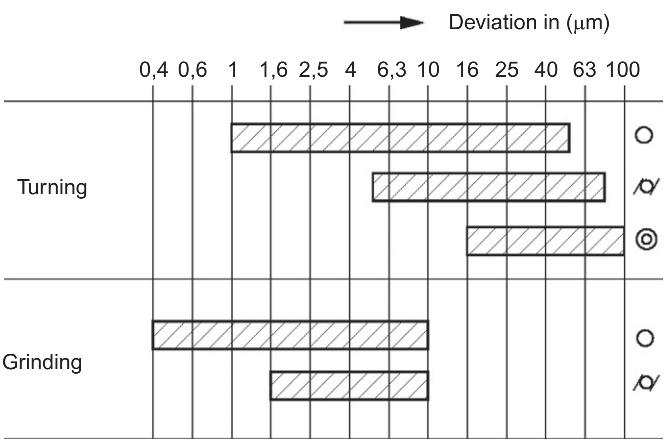


FIG. 12.2 Ability of manufacturing devices in a particular workshop, example

Figure 12.3 shows, as an example, the typical dependence of roundness deviation on cutting speed.

Figure 12.4 shows the result of chucking and cutting pressure influence on the workpiece form.

The largest geometrical deviations occur in general as a result of re-chucking the workpiece. Therefore special care should be taken in re-chucking. Whenever possible, toleranced features and datum features related by tolerances of orientation or location should be manufactured without re-chucking.

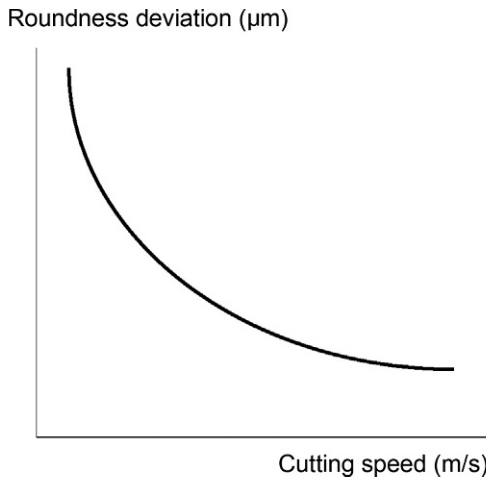


FIG. 12.3 Roundness deviation as a function of cutting speed

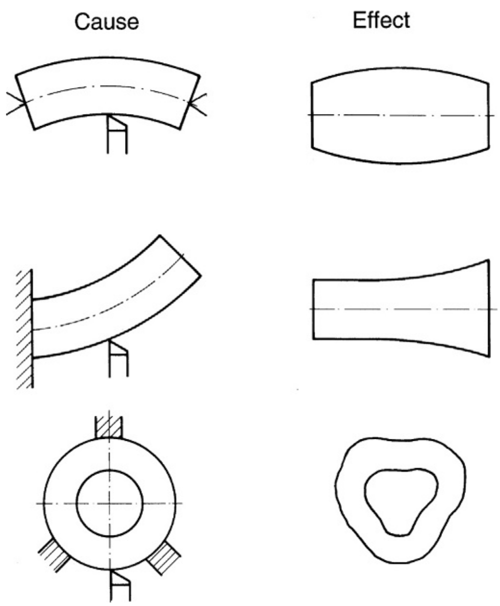


FIG. 12.4 Influence of chucking and cutting pressure on the workpiece form

12.2 Recommendations for manufacturing

In order to respect **geometrical tolerances**, the abilities of the machine tools should be assessed and classified. In order to respect narrow geometrical tolerances by the classification, the process data (e.g. cutting speed) and workpiece properties (e.g. wall thickness) must be taken into account. Because re-chucking strongly influences the geometrical deviations, it should be specified in the

manufacturing planning whether the workpiece is to be re-chucked and, if so, then when, where, how and with what precision.

With regard to respecting the **general geometrical tolerances**, see 13.

In order to respect the **envelope requirement** Ⓔ, it is recommended that the tolerance specified in the drawing at the maximum material limit be reduced by the expected form deviation, i.e. the actual sizes must differ from the maximum material size at least by the amount of the expected form deviation.[1)]

In order to respect the **maximum material requirement Ⓜ at the toleranced feature**, the following are recommended:

– with drawing indication 0 Ⓜ: the same applies as for Ⓔ
– with drawing indication t Ⓜ (where $t > 0$): Manufacture as if Ⓜ is not indicated. Only when the specified geometrical tolerance is exceeded, it is necessary to check according to the maximum material requirement. [1), 2)]

In order to respect the **maximum material requirement Ⓜ at the datum feature**, the following is recommended:

when at the datum letter in the tolerance indicator the symbol Ⓜ is indicated but the datum feature itself has no geometrical tolerance with specified Ⓜ:
– when it is a primary: datum the same applies as for Ⓔ.
– when it is a secondary or tertiary datum: reduce the size tolerance at the maximum material limit by the expected perpendicularity deviation.

When the datum feature itself has a geometrical tolerance with specified Ⓜ: Manufacture as if Ⓜ is not indicated. Only when the specified geometrical tolerance of the datum feature is exceeded, it is necessary to check according to the maximum material requirement.

In order to respect the **least material requirement** Ⓛ, proceed in the same way as with the maximum material requirement, but reduce the size tolerance at the least material limit (see 4.6.3).

In order to respect the **projected tolerance zone** Ⓟ, it is recommended that the positional tolerance be reduced by twice the expected angularity deviation related to the projected length (Fig. 12.5).

With the **reciprocity requirement** Ⓡ, it is assumed that in the manufacturing process selected, the tolerances are respected without any special effort.

Figure 12.6 gives a synopsis of these recommendations. However, for the decision as to whether the workpiece meets the drawing specifications, the definitions of the requirements must be observed.

1) If appropriate, a manufacturing drawing should be issued using the reciprocity requirement (see 4.4.3.6 and 4.6.4).

2) Features related by relatively small tolerances of orientation or location (toleranced feature(s) and datum feature(s)) should be manufactured without re-chucking.

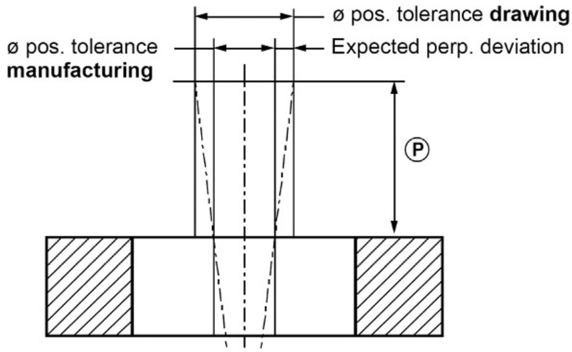


FIG. 12.5 Recommendation for manufacturing in order to respect the projected tolerance zone Ⓔ

10 H8 Ⓔ	For the toleranced feature reduce the size tolerance at max. mat. limit by the expected form deviation
| | 0 Ⓜ | | | 0 Ⓜ | |	For the toleranced feature reduce the size tolerance at max. mat. limit by the expected form deviation, by the expected orientation, or location deviation
| | | A Ⓜ |	For the datum A feature see Ⓔ or see 0,1Ⓜ when at the datum feature a geom. tolerance followed by Ⓜ is indicated
| | | A | B Ⓜ |	For the datum B feature reduce the size tolerance at max. mat. limit by the expected perpendicularity deviation related to A
| | 0,1 Ⓜ | | | 0,1 Ⓜ | |	Manufacture the toleranced feature as without Ⓜ
| | 0,1 Ⓜ Ⓡ | | | 0,1 Ⓜ Ⓡ | |	Manufacture the toleranced feature as without Ⓜ Ⓡ
| | 0 Ⓛ | | | 0 Ⓛ | |	For the toleranced feature reduce the the size tol. at least mat. limit by the expected form deviation, by the expected orientation, or location deviation
| | | A Ⓛ |	For the datum A feature reduce the size tolerance at least mat. limit by the expected form deviation
| | | A | B Ⓛ |	For the datum B feature reduce the size tolerance at least mat. limit by the expected perpendicularity deviation related to A
| | 0,1 Ⓛ | | | 0,1 Ⓛ | |	Manufacture the toleranced feature as without Ⓛ
| | 0,1 Ⓛ Ⓡ | | | 0,1 Ⓛ Ⓡ | |	Manufacture the toleranced feature as without Ⓛ Ⓡ

Features related by relatively small tolerances of orientation or location (toleranced feature(s) and datum feature(s)) should be manufactured without re-chucking

FIG. 12.6 Recommendations for manufacturing in order to respect Ⓔ, Ⓜ, Ⓛ, Ⓡ

The **drawing tolerance** is to be diminished by the **measurement uncertainty**; see Fig. 2.20.

12.3 Process capability

It is assumed that the considered characteristic within a lot (population) has a Gaussian distribution. Then for the lot, the following are defined according to ISO 22514-2:

$$\text{Arithmetical mean}: \mu = \Sigma x/\text{n}$$

$$\text{Standard deviation}: \sigma = \sqrt{\left([\text{x} - \mu]^2/\text{n} - 1\right])}$$

where

x = single values of the characteristic,
n = number of parts.

When samples are used, calculations should be made, according to the rules of statistics, of μ and σ, from $\bar{x}$ (arithmetical mean of the sample) and s (standard deviation of the sample).

Then the following can be defined:

$$\text{Process capability}: C_p = T/6\sigma$$

where T = tolerance.

$$\text{Process capability index}: C_{pk} = D_{min}/3\sigma$$

where D_{min} is the smaller distance of μ to the upper or lower tolerance limit.

When $D_{min} \geq T/3$ (i.e. offset $\mu \lesssim \pm 1.5\sigma$) and $C_{pk} \geq 1.33$:

$$\sigma \lesssim T/(3 \bullet 3 \bullet 1.33) = T/12$$

which is to be proofed (in addition to the proof of $D_{min} \geq T/3$). See Fig. 12.7.

In order to assess s or σ, the characteristic must be a variable. Therefore, when Ⓔ is used, the mating size can be used, in order to proof the process capability. The mating size is the diameter of smallest circumscribed cylinder with shafts, the diameter of largest inscribed cylinder with holes, the width of smallest circumscribed cuboid with tabs, and the width of largest inscribed cuboid with slots.

When the mating size cannot be measured, an approximation can be taken. In the case of a cylindrical feature of size, the component two-point size deviations and the cylindricity deviations can be measured. This cylindricity deviations are the location deviations (points) of the toleranced feature from the reference cylinder. The standard deviation of the convolution (sum of distributions) can be estimated according to the propagation law by:

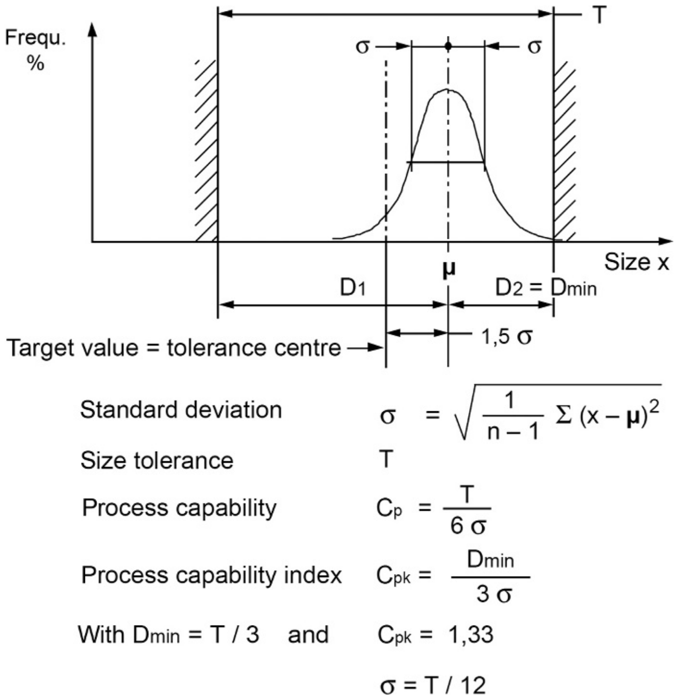


FIG. 12.7 Process capability

$$\sigma = \sqrt{\left(\sigma_{size}^2 + \sigma_{cyl}^2\right)}$$

In the case of a planar feature of size, the component two-point size deviations and the flatness deviations of the derived features can be measured. This flatness deviations are the location deviations (points) of the derived tolerance feature from the reference plane. The standard deviation of the convolution (sum of distributions) can be estimated according to the propagation law by:

$$\sigma = \sqrt{\left(\sigma_{size}^2 + \sigma_{flat}^2\right)}$$

When one of the members in this formula is much smaller than the other, e.g. $\sigma_{size} = 0.5$ and $\sigma_{cyl} = 0.1$, the smaller one can be omitted, because $0.5^2 = 0.25$, $0.1^2 = 0.01$ and $\sqrt{0.26}$ is practically equal to $\sqrt{0.25}$. Then the proof of C_{pk} becomes easier.

When a related position tolerance with Ⓜ is indicated, the related mating size can be measured, in order to proof the process capability. See Fig. 4.59.

With geometrical deviations (tolerances of form, orientation, location), there is one specification limit = 0. The deviations are close to this limit. The farther away from this limit the deviation is, the less often it occurs. The

distribution of the deviations is theoretically log-normal (not normal, Gaussian), which has to be considered when estimating C_{pk}.

Computer programs are available which transform the data (deviations) to a normal distribution. They use the normalized data for the calculation of C_{pk} as defined earlier.

Chapter 13

Inspection of Geometrical Deviations (Verification)

13.1 General

This section deals with generalities of the inspection of geometrical deviations. A synopsis of inspection methods is given in ISO TR 5460. More detailed descriptions of inspection methods are given in the former East German Standards TGL 39 092 to TGL 39 098, TGL 43 041 to 43 045, TGL 43 529 and TGL 43 530.

Geometrical tolerances are geometrically exactly defined. They determine geometrical zones within which the surface of the feature must be contained. There are several methods for inspecting whether the geometrical tolerances have been respected. These methods are more or less precise. The drawing indications according to ISO 1101 do not prescribe a particular (certain) inspection method. According to ISO 14 253-2, the Procedure for Uncertainty Management (PUMA) method should be used in order to optimize manufacturing and inspection (see 13.11.3).

In industrial practice the inspection of geometrical deviations is often only economical with less precise inspection methods, which are normally less time consuming and less costly. In contrast, more precise inspection methods are normally more time consuming and more costly. Therefore it is often advisable to start the inspection with a cheaper but less precise method and to switch to a more precise (and more expensive) method only in those cases where the measurement result is near the limit given by the tolerance (see, for example, 13.7.9.3.6).

A prerequisite is that the possible errors of the inspection methods must be known. Information on systematic errors of inspection methods are given in ISO 14 253-2 (see 13.11).

Specifications for the necessary sample sizes (number of traces, number of probed points) are not internationally standardized. They depend on the size of the feature to be inspected and on the ratio between form deviation and geometrical tolerance. The measurement result shall be representative of the feature. Some hints are given in 13.9.

Geometrical Dimensioning and Tolerancing for Design, Manufacturing and Inspection
https://doi.org/10.1016/B978-0-12-824061-8.00012-8

Particular inspection methods should not be prescribed by the drawing indication for the following reasons:

- The type and frequency of inspection to be used depend on the control of the manufacturing process (reliability).
- There are often different but equivalent correct inspection methods. Prescription of particular inspection methods would force the manufacturer to provide inspection devices prescribed by the customer, even though other sufficient inspection devices may already be available.
- Prescribing inspection methods that differ in assessment from the precise tolerance zone requires further specifications of the measuring conditions. Inspection methods that differ from the precise tolerance zone and different measuring conditions would make the inspection of geometrical deviations obscure and prone to mistakes.

Therefore the ISO standards do not recommend the indication of a particular inspection method. The drawing shall only specify the geometrically exact defined requirement (tolerance zone) (see 3).

However, ISO has prepared standards on the measurement of form deviations. The standards specify the ideal operators (nominal properties of the reference measuring process) that are as close as practically possible to the geometrically exact definition of the deviation. These standards are:

ISO 12 780-1	Measurement of straightness deviations, terms and definition
ISO 12 780-2	Measurement of straightness deviations, measurement
ISO 12 781-1	Measurement of flatness deviations, terms and definition
ISO 12 781-2	Measurement of flatness deviations, measurement
ISO 12 181-1	Measurement of roundness deviations, terms and definition
ISO 12 181-2	Measurement of roundness deviations, measurement
ISO 12 180-1	Measurement of cylindricity deviations, terms and definition
ISO 12 180-2	Measurement of cylindricity deviations, measurement

These methods are then the bases to which the measurement uncertainties refer. A measurement with an ideal operator would have a measurement uncertainty of zero. (But this measurement is impossible to achieve in reality because each realization of a measuring instrument has deviations from the ideal operator (properties) and therefore produces a measurement uncertainty).

Form measuring instruments measure the form of cylindrical features but no diameters. This measurement (verification method) is (depending on the number of extracted points and the tracing strategy) near to the correct definition of the deviation (tolerances). See 3 to 6.

Coordinate measuring systems (CMSs) extract (probe) points of the workpiece surface. These points are stored in a computer with their coordinates x, y, z. Everything that follows is performed by the computer. This measurement (verification method) is (depending on the number of extracted points and the tracing strategy) near to the correct definition of the deviation (tolerances). See 3 to 6.

In the following paragraphs, other inspection (verification) methods are described. They can be used when CMSs are not available or too costly. However, often these alternative inspection methods have relatively large measurement uncertainties, which must be observed.

13.2 Terms

Embodiment: Measuring is comparing. In order to measure geometrical deviations, the workpiece surface must be compared with a geometrically ideal feature. But geometrically ideal features cannot be manufactured. Therefore, almost geometrically ideal embodiments of geometrically ideal features (e.g. straight lines, planes, circles, cylinders, spheres) are used. They are called embodiments in the following descriptions.

These embodiments can be surfaces of measuring devices (e.g. straightedges, measuring tables, solid angles, sliding guides of measuring devices) or can be established by the movements of precision guides (e.g. by rotating the workpiece or the measuring device).

Datum: Theoretically exact geometrical reference (such as an axis, plane or straight line) to which toleranced features are related. Datums may be based on one or more datum features of a workpiece (ISO 5459); see 3.3.

Datum system: Group of two or more separate datums, used as a combined reference for a toleranced feature (ISO 5459); see 3.3.

Datum feature: Real feature of a workpiece (such as a straight edge, a planar surface or a cylindrical hole) that is used to establish the location of a datum (ISO 5459); see 3.3.

Datum target: Point, line or limited area on the workpiece to be used for contact with the manufacturing and inspection equipment, to define the required datums in order to satisfy the functional requirements (ISO 5459); see 3.3.2.

Simulated datum feature: Real surface of sufficiently precise form (such as a surface plate, a bearing, or a mandrel) contacting the datum feature(s) and used to establish the datum(s). Simulated datum features are used as the **practical embodiment** of the datums during manufacture and inspection (ISO 5459); see 3.3.

Reference feature: Geometrically ideal element (straight line, circle, line of any form defined by theoretically exact dimensions, plane, cylinder, surface of any form defined by theoretically exact dimensions) relative to which the geometrical deviations are evaluated.

The reference feature is established by or derived from the embodiment at the measured (toleranced) feature. When the reference feature is calculated (e.g. using a coordinate measuring system), the reference feature may differ from the embodiment in orientation, location and size.

The orientation and location of the reference feature depend on the characteristic to be measured (deviations of unrelated form, related form, orientation, circular or total axial run-out, circular or total radial run-out), location; see 3.2.

Coordinate measuring systems and form measuring instruments often approximate the reference feature by the (Gaussian) least-squares substitute element. The substitute element (see 3.5) intersects or contacts the workpiece surface, but the reference feature need not (see Table 13.1). In cases of a straight line or plane, the minimum zone (Chebyshev) substitute element and the reference feature have the same orientation but not necessarily the same location. In cases of a circle or cylinder, the minimum zone (Chebyshev) substitute element and the reference feature have the same location and orientation but not necessarily the same size.

New coordinate measuring systems and form measuring instruments can normally work to the ISO defaults, i.e. according to the Chebyshev criterion.

Geometrical deviation: The definitions of geometrical deviations are not yet internationally standardized (international standards on this subject are planned). However, they can be derived from the definitions of geometrical tolerances according to ISO 1101. The geometrical deviation is the deviation of a workpiece feature (axis, section line, edge, surface, median surface) from the (geometrically ideal) reference feature (embodiment).

The reference feature is

- oriented according to the minimum requirement (minimax, Chebyshev) (13.3.2) for the measurement of deviations of unrelated form;
- oriented according to the datum (13.3.4) or datum system (3.3) for the measurement of deviations of orientation;
- oriented and located according to the datum (13.3.4) or datum system (3.3) and to the theoretically exact dimensions for the measurement of deviations of location.

In the cases of circular radial run-out or total radial run-out, the theoretically exact dimension between the axes of the toleranced feature and the datum feature is zero.

For the evaluation of the geometrical deviation to be compared with the geometrical tolerance, see 13.6.

With profile deviations (of form and location of integral features) and with position deviations (of form and location of derived features), a distinction must be made between local deviations (deviation of a point of the surface or derived feature from the reference feature) and the global deviation (peak-to-valley deviation T, corresponding to the tolerance zone). Further, there are global orientation and location deviations in specified directions of x, y, z; see 3.5, Decomposition.

There are further standardized form deviations:

- reference-to-peak deviation, P
- reference-to-valley deviation, V
- root-mean-square deviation, Q

See ISO 12 780-1, ISO 12 781-1, ISO 12 181-1, ISO 12 180-1 and Table 3.8.

Coordinate measuring system, CMS, is a measuring device that measures the geometry of physical objects by sensing discrete points on the surface. The former term used was coordinate measuring machine (CMM).

13.3 Alignment of the workpiece

13.3.1 General

Most important for measurement is the alignment of the workpiece in the measuring device. Misalignment can cause large errors in measurement, so that workpieces that actually comply with the specification may appear as not complying, or, more seldom but possible in principle, workpieces that actually do not comply with the specification may appear as complying. E.g. a cylindricity measurement measured in sections inclined to the median line.

For the measurement of form deviations, the minimum requirement (minimax, Chebyshev) must be respected (see 13.3.2 and 13.3.3).

For the measurement of deviations of orientation, location and run-out (related geometrical deviations), the minimum rock requirement must be respected for the datum(s) (see 13.3.4).

For the measurement of roundness or cylindricity, additional alignment requirements must be respected (see 13.3.3).

13.3.2 Minimum requirement

The minimum requirement (minimax, Chebyshev) defines the orientation or location of the form tolerance zone. With straightness and flatness, this requires that the parallel straight lines or parallel planes enclosing the toleranced feature be directed so that their distance is a minimum (Fig. 13.1, δ_1); ISO 1101. With roundness and cylindricity, this requires that concentric circles and coaxial cylinders enclosing the toleranced feature be located so that their radial distance is a minimum (Fig. 13.2, δ_1); ISO 1101.

Deviations from the minimum requirement (e.g. alignment according to the Gaussian regression line) simulate larger form deviations.

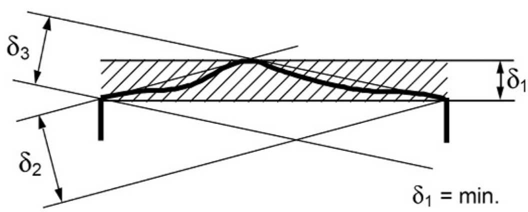


FIG. 13.1 Minimum requirement for straight lines (Chebyshev)

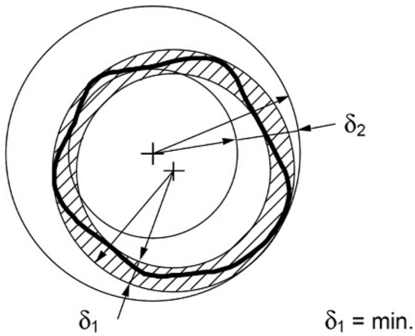


FIG. 13.2 Minimum requirement for circles

13.3.3 Additional alignment requirements for the measurement of roundness or cylindricity deviations

13.3.3.1 Axis of measurement

The **axis of measurement** is a straight line relative to which the measurement is performed and established by the measuring device. The measurements are considered to be perpendicular and centred on the axis of measurement.

This axis may be defined (established) as follows:

a) A straight line such that the root-mean-square value of the distances from it to the defined centres of a representative number of cross sections has a minimum value.

The defined centres may be the centres of the

- (total) least-squares circle (LSC) (according to Gauss, the circle for which the sum of the squares of the radial distances to the circumference is a minimum, regression circle);
- minimum zone circle (MZC) (according to the Chebyshev circle, for which the maximum radial distance to the circumference is a minimum);
- contacting circle (maximum inscribed circle for holes (MIC); minimum circumscribed circle for shafts (MCC), sometimes referred to as plug gauge circle (PGC); and ring gauge circle (RGC)).

b) A straight line passing through the defined centres of two separated and defined cross sections. For the defined centres see a).

c) A straight line passing through the defined centre of one defined cross section and perpendicular to a defined shoulder. For the defined centres see (a). The defined shoulder may be determined by the.

- (total) least-squares plane (according to Gauss, plane for which the sum of the squares of the distances to the actual feature (shoulder) is a minimum, regression plane)

– minimum zone plane (according to Chebyshev, plane for which the maximum distance to the actual feature (shoulder) is a minimum)
– datum plane (according to the minimum rock requirement; see 13.3.4)

d) A straight line passing through two support centres (centre bores).

If not otherwise specified (agreed upon) according to ISO 1101, the definition a) LSC applies.

The straightness tolerance indication of a cylinder axis is shown in Fig. 13.3.

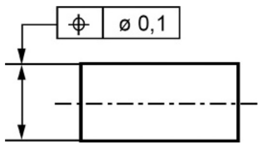


FIG. 13.3 Straightness tolerance indication

13.3.3.2 Inclination of the axis

If the axis of measurement is inclined to the ideal reference axis, a measurement of a round workpiece feature may appear oval and an oval workpiece feature may appear round (Fig. 13.4).

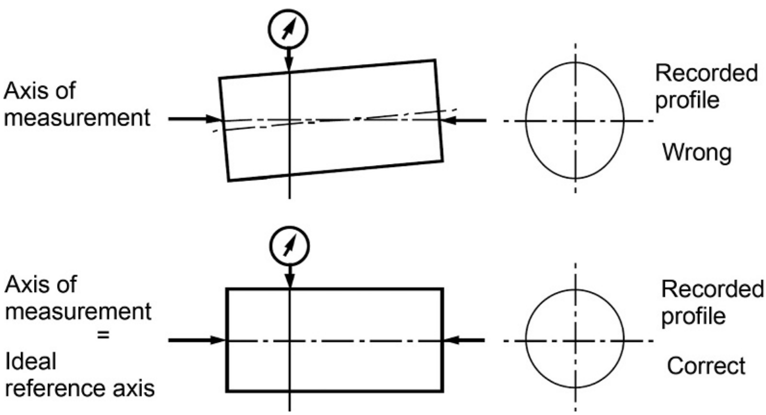


FIG. 13.4 Roundness measurement: false measurement caused by a plane of measurement not perpendicular to the ideal reference axis

It is good practice for the measurement of roundness and cylindricity that the levelling (alignment) be performed at the top and bottom of the workpiece feature until the least-squares centres (LSCs) at these two levels coincide with a value better than 10% of the roundness or cylindricity tolerance.

13.3.3.3 Offset of the workpiece axis

Some form measuring instruments strongly magnify the form deviations but not the radius itself. When these instruments are used and when in the measuring plane the centre of the workpiece cross section line (profile) differs from the axis of measurement, distortions of the profile occur (limaçon effect) (Fig. 13.5).

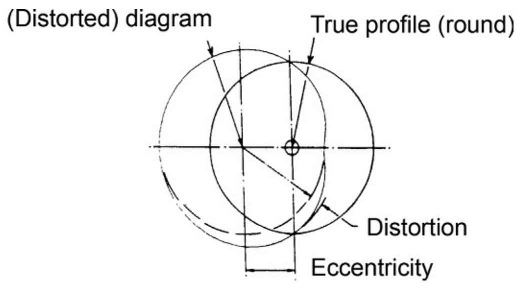


FIG. 13.5 Distortion of the profile diagram caused by an offset of the true workpiece centre (ideal reference circle) from the axis of measurement (limaçon effect)

It is good practice to centre the workpiece at each level that is measured individually until a measuring range can be used (i.e. the profile is fully contained in this range) that results in a resolution of the instrument smaller than 1% of the roundness tolerance.

13.3.4 Minimum rock requirement

In the measurement of deviations of orientation, location and run-out, the minimum zone (minimax, Chebyshev) outside material criterion applies to the alignment of the datum feature. In the past was and often today still is the minimum rock requirement used.

When the minimum rock requirement is used and the datum is not stable (e.g. a convex datum feature) relative to the contacting surface (simulated datum feature, e.g. measuring table or mandrel), it shall be arranged so that the possible movement (inclination) in any direction is equalized, i.e. that the maximum possible inclination to the extreme position is a minimum (minimum rock requirement) (Fig. 3.81). In other words, the datum feature shall be aligned relative to the simulated datum feature into a median position. See also 2.4, Datums.

In the case of a planar datum feature, an approximation to the Chebyshev or minimum rock requirement is to place three equal-height supports between the datum feature and the simulated datum feature at the same distances from the end of the datum feature.

In the case of a cylindrical datum feature, the datum is the axis of the contacting cylinder (the maximum inscribed simulated datum cylinder for a datum feature that is a hole, and minimum circumscribed simulated datum cylinder for a datum feature that is a cylindrical shaft) arranged according to the minimum rock requirement, i.e. in a median position.

When the datum feature is the common axis of two cylindrical shafts of different nominal sizes, the common axis is the axis of two contacting coaxial cylinders where the maximum distance to the filtered datum feature is minimized; see Fig. 13.6.

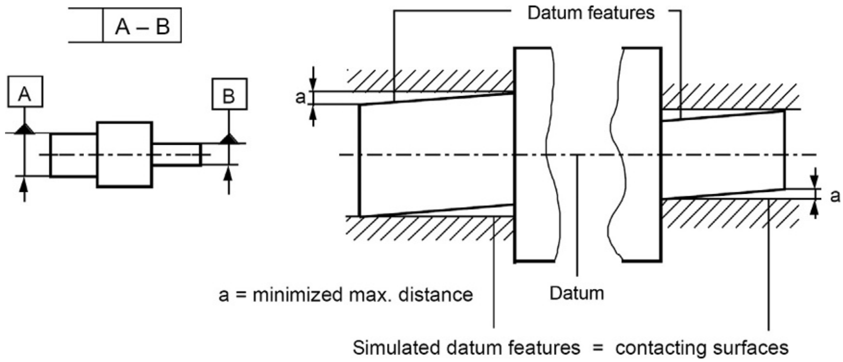


FIG. 13.6 Common axis as a datum, ISO 5459

An approximation of a common axis of two or more datum features is the axis of the smallest imaginary cylinder containing the actual axes of the datum features.

Often the connection of the centres of the cross sections at half the length of the datum features is used to establish the common axis (Fig. 13.7) (measurement with form measuring instrument or support in edge V-blocks). See also 13.7.10.1.

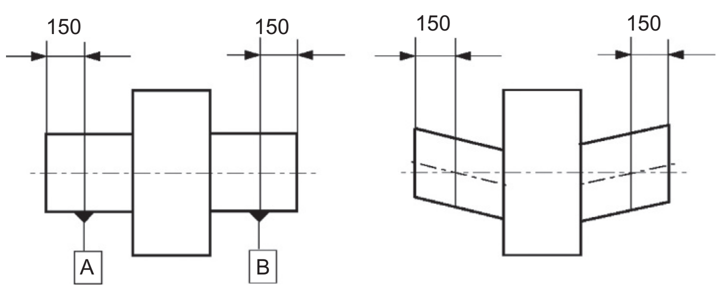


FIG. 13.7 Common datum axis established by the centres of two specified sections

For more information about datums, see 2.4.

For the alignment of the datum features according to the three-plane concept, see 3.3.

ASME Y14.5-2009 defines the candidate datum method. This method selects contacting planes which contact the actual datum feature to a certain extent and give possible orientations relative to which all related tolerances are to be respected.

13.4 Interchanging of the datum feature and toleranced feature

With related geometrical tolerances, the drawing distinguishes between toleranced features and datum features. When, for the inspection, the toleranced

feature and datum feature are interchanged, completely different values of the geometrical deviation may be obtained from the same workpiece (Fig. 13.8).

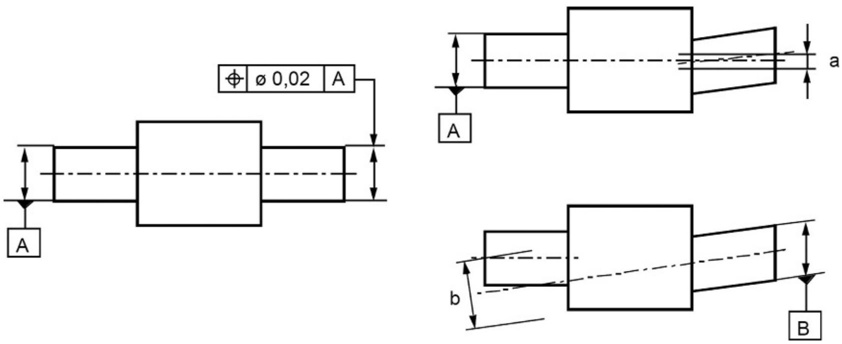


FIG. 13.8 Change in geometrical deviation caused by interchanging of toleranced feature and datum feature

Therefore, for the inspection the toleranced feature and datum feature must not be interchanged. When the datum feature indicated on the drawing is not suitable for inspection purposes (e.g. because it is too short), a change in the drawing is necessary.

13.5 Simplified inspection method

The inspection of certain types of geometrical tolerances (e.g. coaxiality tolerance) is relatively costly. Often in such cases, in the first step a "quick" (but less precise) inspection is chosen and only in case of doubt is the more precise (and more costly) inspection executed in a second step. For example, in the cases of a coaxiality tolerance or a common straightness tolerance zone of axes, the inspection in the first step is performed as if there were a run-out tolerance of the same value. Only when this tolerance is exceeded is a more precise method used to determine whether the coaxiality tolerance or the straightness tolerance of the axis is exceeded.

A similar situation applies in the case of a cylindricity tolerance. In the first step, the check of total run-out may be executed.

A similar situation also applies in the case of a roundness tolerance. In the first step, the check of run-out may be executed.

Often with related geometrical tolerances, the precise verification of the Chebyshev or minimum rock requirement is very costly. Therefore approximate inspection methods are used with the aid of V-blocks, mandrels, centre bores, etc. Using V-blocks, the form deviation of the datum feature leads to simulation of a larger related geometrical deviation than actually exists (according to the definition). Depending on the shape of the form deviation and the angle of the V-block, the increase can be equal to or less than the form deviation of the datum feature (see 13.7.4.3).

A similar consideration applies with the use of centre bores. There the eccentricity of the centre bore relative to the datum feature increases the result of the measurement of the related geometrical deviation.

Mandrels for inspection purposes are normally rated in diameter in units of 0,01 mm. The largest mandrel that fits into the hole is to be used. In cylindrical holes, the mandrel can be inclined by 0.01 mm at most (Fig. 13.9), and thereby can give an incorrect measuring result. In very rare cases of very detrimental form deviations, larger inclinations may occur (Fig. 13.10).

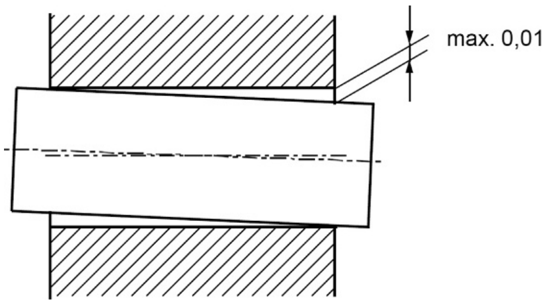


FIG. 13.9 Mandrel, possible inclination

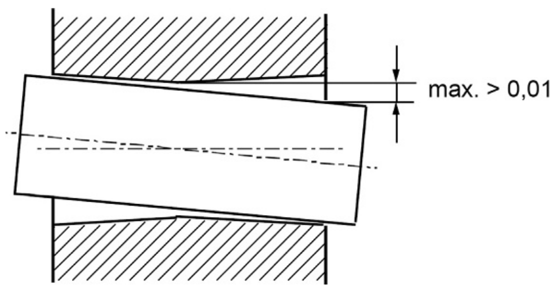


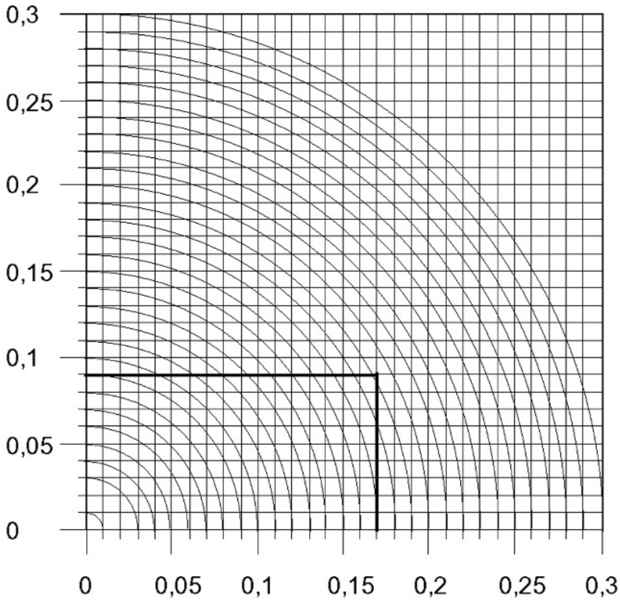
FIG. 13.10 Mandrel, possible deviation from the minimum rock requirement

Pneumatic mandrels are self-centring, as are expanding mandrels and conical mandrels.

13.6 Evaluation of measurement

Evaluation of a measurement can be executed manually by calculation, or graphically or automatically by suitable computer programs (e.g. in a form measuring instrument or in a coordinate measuring system). In the following sections the principles for the evaluation by calculation are described.

In many cases a cylindrical tolerance zone is derived from the function and is indicated in the drawing. When during inspection the deviations in Cartesian (rectangular) coordinates are assessed, they can be compared with the tolerance diameter with the aid of the diagram in Fig. 13.11.



Example:
measured coordinates e_x = 0,17 and e_y = 0,09
are within r = 0,2 ; ø t = ø 0,4

Example: measured coordinates 0,17 and 0,09 lie within ø 0,4 (r = 0,2)

FIG. 13.11 Diagram to compare assessed coordinates with cylindrical or circular tolerances

Figure 13.12 shows a typical method for the assessment of related geometrical deviations (here, parallelism deviations of an axis relative to a datum axis) by detecting rectangular coordinates δ_x, δ_y and calculating the cylindrical coordinate δ_o.

Often the tolerance applies over the length l of the feature, whereas for practical reasons the measurement was applied to the length l_m. The measured values are to be corrected by the ratio l/l_m, i.e.

$$\delta_o = \delta_l \,(l/l_m)$$

where

δ_o = orientation deviation
δ_l = measured deviation over the length l_m
l = length of the feature.

The geometrical tolerances t are defined as widths or diameters of tolerance zones within which the toleranced feature must be contained. Geometrical deviations δ, however, are measurable deviations from the geometrical ideal form δ_m, orientation δ_o or location δ_s.

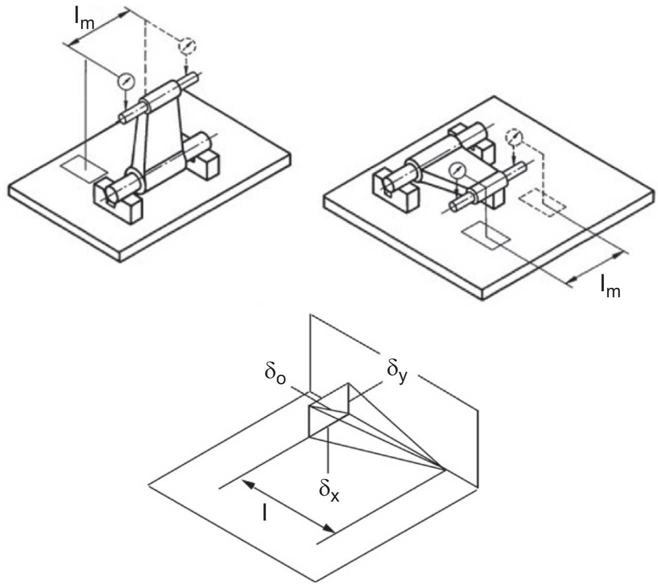


FIG. 13.12 Measuring of the radial deviation of parallelism by detecting the deviations in rectangular coordinates

The maximum permissible radial deviation from orientation can be calculated from the deviations in the x and y directions; see Figs 13.13 and 13.48 to 13.50. A similar technique applies to deviations of location, but here δ_x and δ_y must be multiplied by 2; see Table 13.1, Figs 13.13 and 13.52 to 13.54.

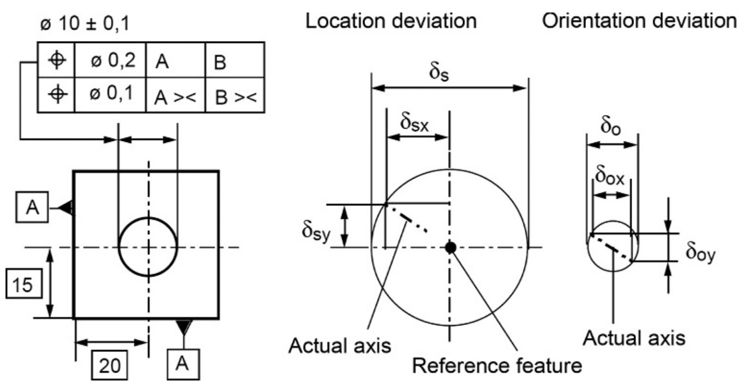
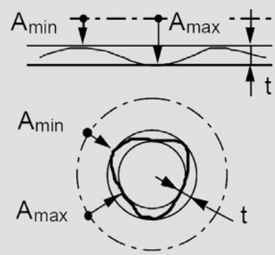
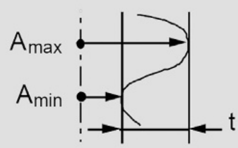
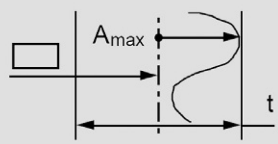
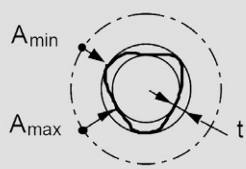


FIG. 13.13 Radial deviation of location deviation δ_s and orientation deviation δ_o by measuring deviations in rectangular coordinates δ_x and δ_y

Table 13.1 gives a synopsis of geometrical deviations δ in comparison to geometrical tolerances t.

TABLE 13.1 Synopsis of geometrical deviations δ, comparison with geometrical tolerances t

	Reference: — - — - —	Toleranced element: ———
Form	$t \geq \delta = A_{max} - A_{min}$	
Orientation	$t \geq \delta = A_{max} - A_{min}$	
Location	$t \geq 2\,\delta = 2\,A_{max}$	
Run-out	$t \geq \delta = A_{max} - A_{min}$	

13.7 Methods of inspection

The following provides a survey of the most relevant inspection methods when coordinate measuring systems are not available.

13.7.1 Assessment of straightness deviations of lines of surfaces

13.7.1.1 Definition

The deviations of the line of the workpiece surface from an (almost) geometrically ideal reference straight line (embodiment, e.g. established by a straightedge) are measured. The line of the workpiece surface and the reference line are contained in a section plane (Fig. 3.18).

When the reference line does not intersect (but eventually touches) the workpiece line, the straightness deviation δ_m according to ISO 1101 is the difference between the largest and smallest distances between the workpiece line and the reference line (Fig. 13.16). When the reference line does intersect the workpiece line, the straightness deviation according to ISO 1101 is the sum of the largest distances of the workpiece line from the reference line on both sides of the reference line (Table 13.1), i.e. the range of the local deviations of the workpiece line from the reference line (Fig. 13.15).

The reference line is to be aligned according to the minimum requirement (Fig. 13.1).

The straightness deviation δ_m must not exceed the straightness tolerance t_m: $\delta_m \leq t_m$.

There are standards for straightness deviations: ISO 12 780-1 for terms and definitions and ISO 12 780-2 for measurement.

13.7.1.2 Type of detecting

The deviations can be detected by

1. continuously probing and recording;
2. consecutive probing (sampling, approximation) and recording

while

a) measuring the distance to the reference line (embodiment)
b) measuring the inclination of a two-point bridge on the surface relative to the reference line (embodiment)1

13.7.1.3 Measuring methods

The reference line (embodiment) may be established by:

– straight guides	(of a form measuring instrument) (of a coordinate measuring instrument);
– measuring plate	(with length measuring instrument, e.g. dial gauge) (optical flat with evaluating of contour lines);
– straightedge	(with length measuring device, e.g. dial gauge or feeler gauge);
– measuring wire	(with measuring microscope);
– straight line marking	(within profile projector);
– optical axis	(collimating microtelescope with targets)[1)] (autocollimator and test mirror)[2)] (laser beam with photoelectric detector)[1)] (laser interferometer with two-point bridge)[2)]
– earth curvature	(liquid surface of a hose levelling instrument)[1),3)] (two-point bridge with inclinometer)[2)]

[1)]*Height measurement.*
[2)]*Inclination measurement.*
[3)]*With long reference lines, the curvature of the earth must be taken into account; see Fig. 13.14 and Table 13.2.*

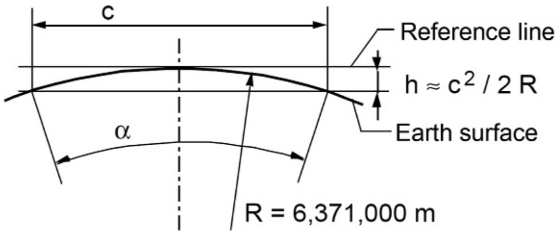


FIG. 13.14 Height correction h in the distance c for eliminating the effect of the earth's curvature

TABLE 13.2 Height correction h over a distance c for eliminating the effect of the earth's curvature (see Fig. 13.14)

c (m)	h (μm)	c (m)	h (μm)	c (m)	h (μm)
1	0,08	11	9,5	25	49
2	0,31	12	11	30	71
3	0,71	13	13	35	96
4	1,3	14	15	40	126
5	2,0	15	18	45	159
6	2,8	16	20	50	196
7	3,8	17	23	60	283
8	5,0	18	25	70	385
9	6,4	19	28	80	507
10	7,9	20	31	100	785

When the reference line of the profile is cut, the minimum requirement (Chebyshev, minimax, minimum zone) applies and the form deviation is the sum of the distances to the highest and the lowest points of the profile (Fig. 13.15).

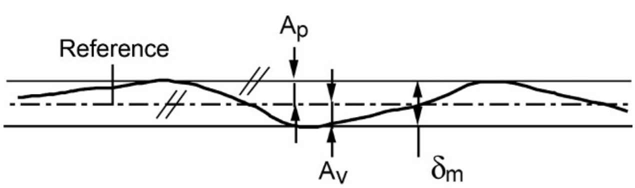


FIG. 13.15 Measurement of straightness deviation, reference element within the tolerance zone

When the workpiece has two highest points far apart (Fig. 13.16) or two lowest points far apart (Fig. 13.17), the longer distance applies; these two points define the orientation of the reference line of a form deviation measurement, according to the minimum requirement.

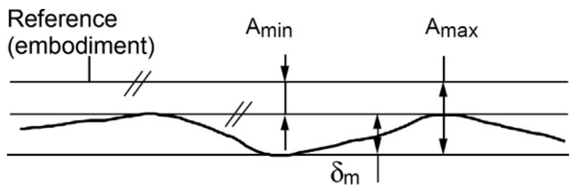


FIG. 13.16 Measurement of straightness deviation; alignment according to the minimum requirement (minimax, Chebyshev), two highest point

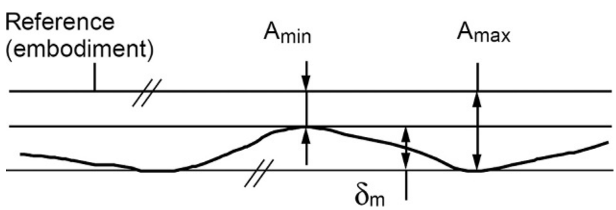


FIG. 13.17 Measurement of straightness deviation; alignment according to the minimum requirement, two lowest points

When form measuring instruments or coordinate measuring systems are used, the alignment, as an approximation, is often performed according to the Gaussian regression line (sum of squares of the distances to the regression line is a minimum). Then the obtained value of the straightness deviation can be larger than the value according to ISO 1101 (minimax, Chebyshev).

When the toleranced length is longer than the straightness embodiment, the embodiment (e.g. straightedge with height adjustable supports) is used in consecutive measuring positions without a) or with b) overlapping.

a) When the straightness embodiment is used in positions without overlapping, it is to be adjusted according to a levelling instrument. The first height indication of the following position is to be adjusted according to the last height indication of the preceding embodiment position (Figs 13.18 and 13.20).

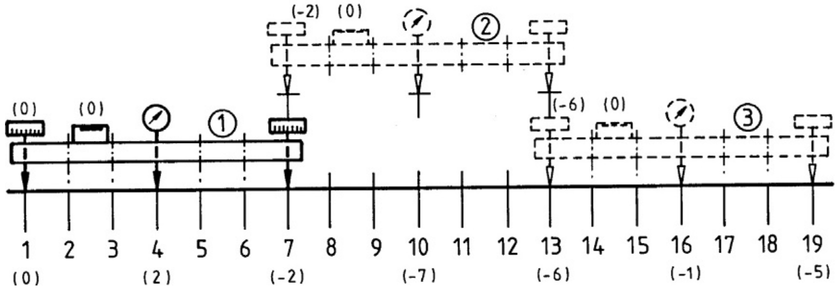


FIG. 13.18 Consecutive measurement of straightness deviation with adjusted straight edge without overlapping, example with measured values in parentheses, according to TGL 39 093

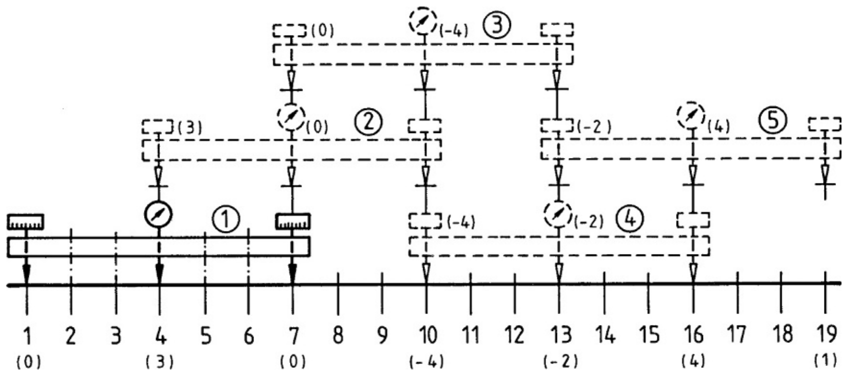
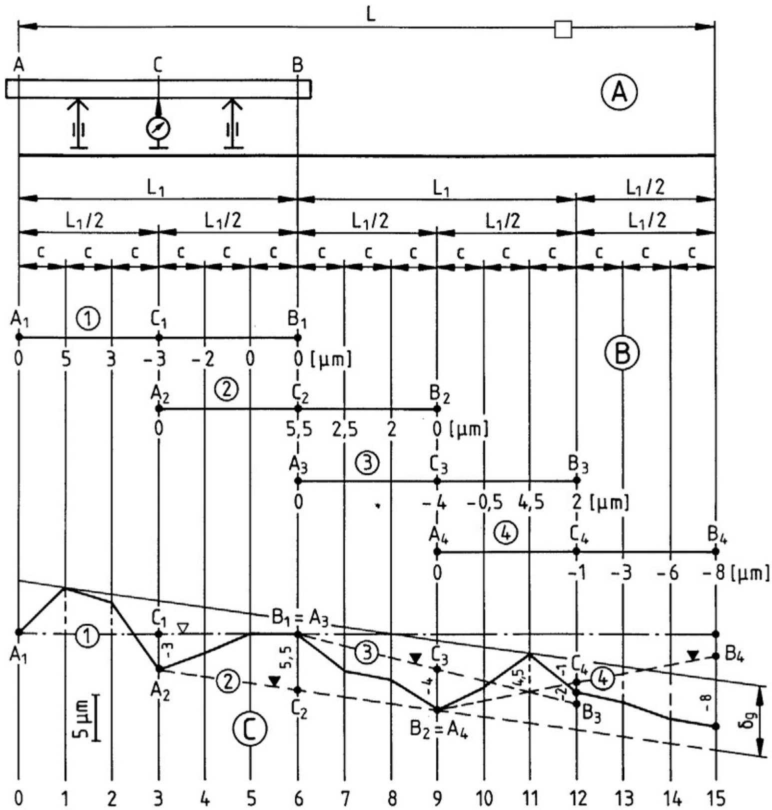


FIG. 13.19 Measurement of straightness deviation with overlapping straightedges, example with measured values in parentheses, according to TGL 39 093

b) When the straightness embodiment is used in overlapping positions, at least two measuring positions must overlap. The two overlapping height indications of the following position must be adjusted according to the height indications of the preceding embodiment position (alignment of the straightedge) (Figs 13.19 and 13.20). The measurement uncertainty is lower the more the measuring positions overlap.

When the angle α (inclination) of the connection of adjacent measuring positions relative to the reference line (straightness embodiment) is measured in seconds, the height difference is $h = c \tan \alpha$ in µm, where c is the distance in m between the measuring positions ($\tan 1'' = 4{,}848\,\mu m/1000\,mm \approx 5\,\mu m/1000\,mm$) (Figs. 13.21 and 13.22).



Ⓐ measuring scheme and length of measuring steps

Ⓑ positions of straightedge and examples of distances to the straightedge related to position A of the straightedge

Ⓒ examples of graphical evaluation of the measuring result

FIG. 13.20 Graphical evaluation of the straightness deviation for consecutive measurement of the deviations using overlapping positions of the straightedge, example with measured values, according to TGL 39 093

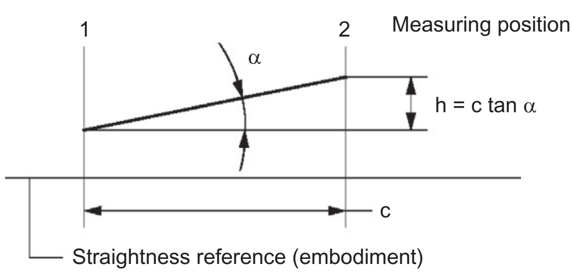


FIG. 13.21 Inclination α along a distance c converted to height deviation h

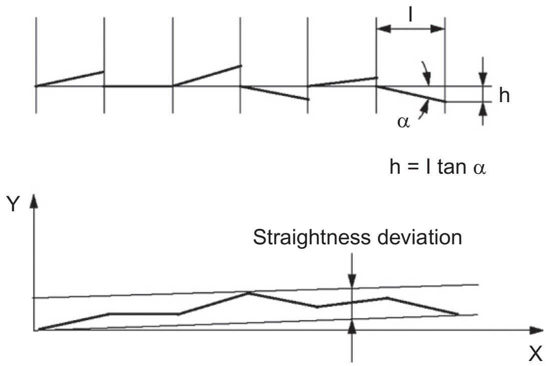


FIG. 13.22 Graphical evaluation of the straightness deviation form consecutive measurement of the deviation from the horizontal orientation

13.7.2 Assessment of straightness deviations of axes

13.7.2.1 Definition

With cylindrical features, the **actual axis** can be taken as a sequence of centres of circles that can be defined as follows:

a) (total) least-squares circle (LSC) (according to Gauss, a circle of which the sum of the squares of the radial distances to the circumference is a minimum, regression circle)
b) minimum zone circle (MZC) (according to Chebyshev, a circle of which the maximum radial distance to the circumference is a minimum)
c) contacting circle (maximum inscribed circle for holes (MIC), minimum circumscribed circle for shafts (MCC), sometimes referred to as plug gauge circle (PGC), and ring gauge circle (RGC)

in cross sections perpendicular to the axis of the following reference cylinders:

a) (total) least-squares cylinder (according to Gauss, a cylinder of which the sum of the squares of the radial distances to the actual cylinder surface is a minimum)
b) median cylinder of minimum zone cylinder (according to Chebyshev, a cylinder of which the maximum radial distance to the actual cylinder surface is a minimum)
c) contacting cylinder (maximum inscribed cylinder for holes, minimum circumscribed cylinder for shafts)
d) two cross section cylinders, defined by the least-squares circles or the minimum zone circles or the contacting circles of two cross sections near the ends of the feature

ISO 17 450-3 defines as a default the Gauss solution a) a); see Fig. 4.8.

Coordinate measuring systems normally apply the least-squares cylinder and the least-squares circle and, if optionally available, the minimum zone circle and the contacting circles. Form measuring instruments normally apply the two cross sections cylinder defined by the least-squares circles or minimum zone circles or contacting circles. (But so far with coordinate measuring systems, the reference cylinder is predominantly the least-squares cylinder, because the mathematical techniques were developed early. With form measuring instruments, it is normally the two cross sections cylinder.)

The least-squares circle and the least-squares cylinder are the only methods that are always unique and that need the least number of measurements (probes, measured points) to be sufficiently stable in size and in location. Therefore they have become the default case (to be applied if not otherwise specified) according to ISO 17 450-3.

The deviations of the actual axis from an (almost) geometrically ideal reference straight line (embodiment) are measured. The deviations have to be calculated in relation to an (imagined) reference line to which the minimum requirement applies, i.e. the maximum local deviation e shall be a minimum (Figs 13.1). The straightness deviation δ_m is the range of the local deviations e; $\delta_m = 2\ e_{max}$ (Fig. 13.23). The straightness deviation must not exceed the straightness tolerance t_m, i.e. the maximum local straightness deviation e_{max} must not exceed one-half of the straightness tolerance t_m:

$$\delta_m = 2\, e_{max} \leq t_m$$

There are standards for straightness deviations: ISO 12 780-1 for terms and definitions and ISO 12 780-2 for measurement.

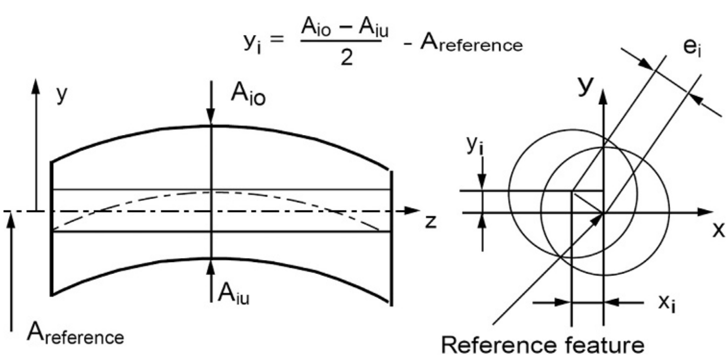


FIG. 13.23 Assessment of coordinates of an actual axis (centres of cross sections)

13.7.2.2 Assessment of the coordinates of axes

For the cylindrical tolerance zone, i.e. to assess the straightness deviation (curvature) in the space, it is normally measured in each cross section in two mutually perpendicular directions (coordinates x_i, y_i, z_i, as in Fig. 13.23).

The smallest cylinder containing all assessed centres is to be determined. There are approximations for this purpose.

One approximation is to define a reference line (z direction) to align the workpiece in this coordinate system, to assess the actual axis (coordinates) and to calculate the distances between the coordinates of the actual axis and the reference line. The local distance in space is $e_i = \sqrt{(x_i^2 + y_i^2)}$, where

x_i is the distance in the xz plane (x-direction)
y_i is the distance in the yz plane (y-direction)

The diameter of the smallest cylinder containing all centres (points of the actual axis) is $\delta_m = 2\ e_{max} \leq t_m$. Its axis is the reference line (Fig. 13.23).

Often the reference line is established by the centres of the first and last cross sections. With coordinate measuring systems, the reference line is the axis of the substitute cylinder (least-squares cylinder) (see 13.7.2.1 and 13.3.3).

When the reference line is not exactly parallel to the z axis, but only approximately so, the measured coordinates x_i can be recorded in a xz coordinate system and y_i in a yz coordinate system (Fig. 13.24). For an approximation in each coordinate system, the reference line is to be determined so that the maximum deviation is a minimum (parallel to the two parallel straight lines of minimum distance that enclose all points of the actual axis; see 13.24). The deviations p and q of the points from the reference line are to be determined. The local actual deviations in space from the reference line are $e_i = \sqrt{(p_i^2 + q_i^2)}$, and the diameter of the smallest cylinder that contains all centres e_i (points of the actual axis) is the straightness deviation δ_m. The straightness deviation δ_m must not exceed the straightness tolerance t_m: $\delta m < t_m$.

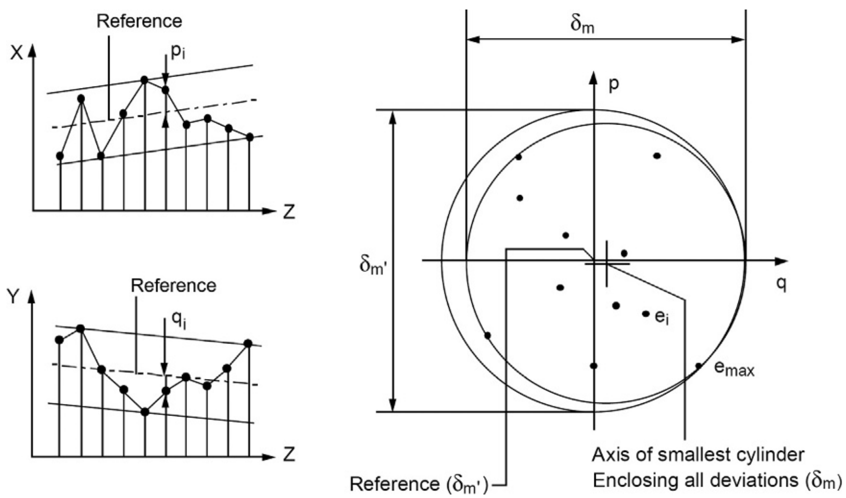


FIG. 13.24 Evaluation of the straightness deviation of an axis in space

An approximation, easy to calculate, is $\delta_m \approx \delta_m' = 2\ e_{max} \leq t_m$ (Fig. 13.24).

13.7.2.3 Assessment with one dial indicator

A simple approximate method for assessing the straightness deviation of an axis is shown in Fig. 13.25. The workpiece has to be supported centrally (e.g. in the centres of the measuring positions z_1 and z_n).

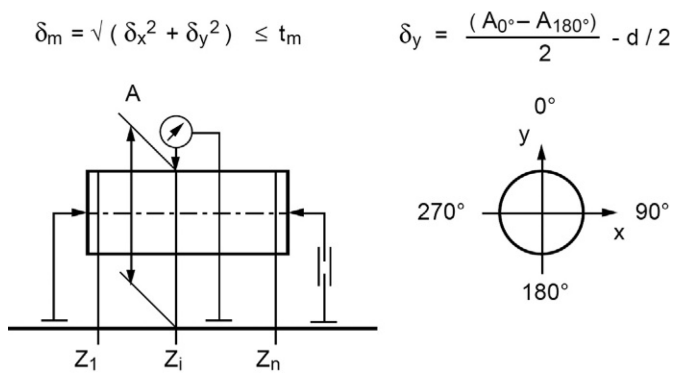


FIG. 13.25 Assessment of the straightness deviation of an axis with one dial indicator

In each cross section (measuring position), the coordinates of the centres are to be determined by probing at the angle positions 0° and 180°, 90° and 270°. The coordinates of the centre (observing the signs + and −) are to be introduced into a polar diagram (in the x direction $x_i = (A_0 - A_{180})/2$ and in the y direction $y_i = (A_{90} - A_{270})/2$) (Fig. 13.23). The diameter of the smallest circle enclosing the centres of all cross sections corresponds to the straightness deviation δ_m.

With this method, problems arise with positioning the axis of revolution (reference line) according to the minimum requirement and with the assessment of the actual centres according to the definition given in 13.7.2.1.

13.7.2.4 Assessment with two dial indicators

Another simple approximation method for assessing the straightness deviation of an axis is shown in Fig. 13.26. The workpiece axis has to be aligned parallel to the reference line (embodiment, e.g. measuring table), for example through the centres of the measurement positions z_1 and z_n with the same distance from the measuring table.

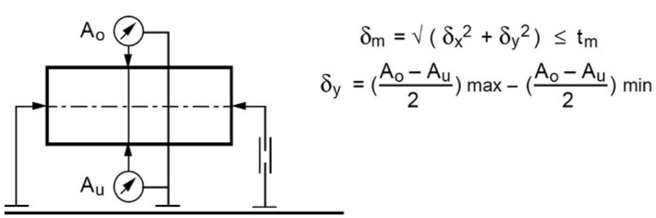


FIG. 13.26 Assessment of the straightness deviation of an axis with two dial indicators

In each longitudinal section (containing the axis) of, for example, four sections, the values $R=(A_o-A_u)/2$ are to be determined at several (at least three) measuring positions. The difference between R_{max} and R_{min} within one section represents the straightness deviation of the axis in this section. The straightness deviation of the axis of the cylindrical feature is the maximum of the straightness deviations of the sections (Fig. 13.27). The straightness deviation of the axis δ_m is shown in Fig. 13.26. As the directions of measurement are opposite, it is the difference of half the sums of the distances from the measuring plate that is determined.

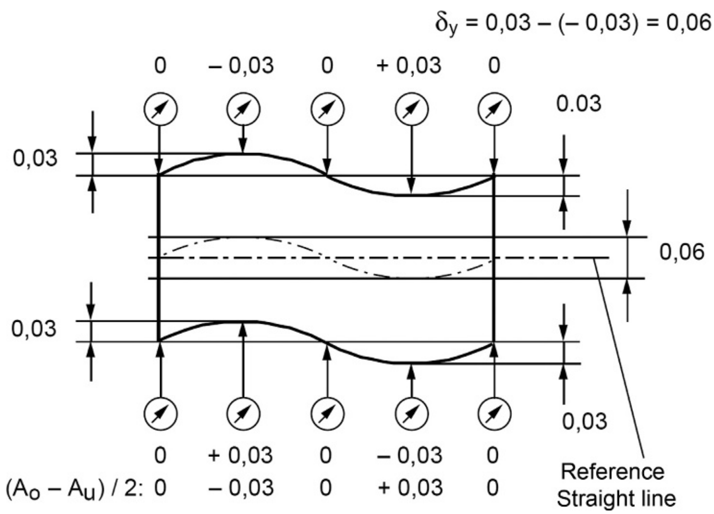


FIG. 13.27 Example of the assessment of the straightness deviation of an axis with two dial indicators

For the measurement of the straightness deviations of generatrixes (generator lines) or axes of cylindrical or conical shafts (or edges), cylindrical or flat anvils should be used (see Fig. 13.117c).

13.7.2.5 Assessment with form measuring instrument or with coordinate measuring system

The instruments assess the coordinates of points of the actual circumference in cross sections approximately perpendicular to the axis and calculate the coordinates of the actual centre points of which the actual axis is composed. From the points of the actual axis, they calculate the local straightness deviations, taking into account the minimum requirement. Twice the value of the maximum local straightness deviation corresponds to the straightness deviation δ_m according to ISO 1101 (see 13.7.2.2).

The instruments allow measurements close to the definitions. For the definition of the actual axis, see 13.7.2.1.

13.7.3 Assessment of flatness deviations

13.7.3.1 Definition

The deviations of the surface from an (almost) ideal reference plane (embodiment, e.g. established by a measuring plate) are measured. When the reference plane does not intersect (but eventually touches) the workpiece surface, the flatness deviation δ_m according to ISO 1101 is the difference between the largest and smallest distances between the workpiece surface and the reference plane (Table 13.1). When the reference plane does intersect the workpiece surface, the flatness deviation δ_m according to ISO 1101 is the sum of the largest distances between the reference plane and the workpiece surface above and below the reference plane (Table 13.1).

The reference plane is to be aligned according to the minimum requirement (Fig. 13.1). The flatness deviation δ_m must not exceed the flatness tolerance t_m: $\delta_m \leq t_m$.

There are standards for flatness deviations: ISO 12 781-1 for terms and definitions and ISO 12 781-2 for measurement.

13.7.3.2 Type of detection and measurement methods

These are similar to those used for the assessment of straightness deviations (see 13.7.1.2). When the deviations are assessed in lines with the aid of straightness embodiments (e.g. with a straightedge), the embodiment must remain in the same plane (inclination, height level) or the measured values must accordingly be corrected by calculation.

Figure 13.28 shows a measuring device in which the alignment is controlled by an inclination measuring instrument.

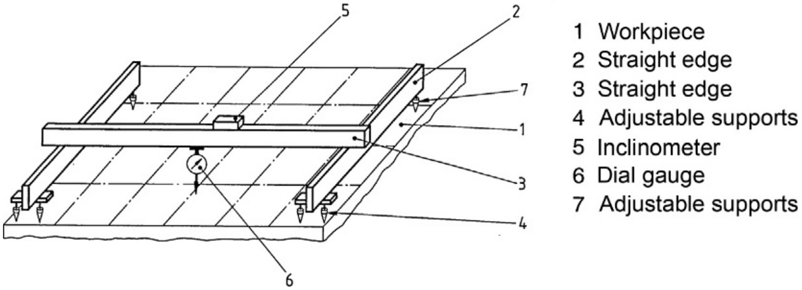


FIG. 13.28 Assessment of flatness deviation of a workpiece (1), with straightedges (2,3); adjustable supports (4); inclinometer (5); and dial gauge (6)

However, it is difficult to align the workpiece surface relative to the plane embodiment so that the largest measured deviation is a minimum. Often, as an approximation, the distances between workpiece surface and plane embodiment are equalized at three ends (points) of the surface (e.g. three supports of equal height). Another approximation, when a computer is used (e.g. with coordinate measuring systems) is alignment parallel to the least-squares plane (plane for which the sum of the squares of the distances to the actual surface is a minimum). With the approximations, there are always larger estimations of the deviations than according to the minimum requirement according to ISO 1101.

The plane embodiment according to the minimum requirement has one of the following positions (alignments):

a) it touches the three highest points (concave form)
b) it touches the three deepest points (convex form)
c) it touches the two highest points and is parallel to the straight line touching the two deepest points (saddle form, Fig. 13.29).

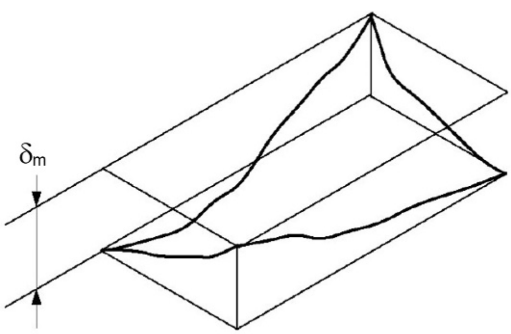


FIG. 13.29 Flatness deviation δ_m of a saddle form surface

The former East German Standard TGL 39 094 describes a graphical method ZNIITM for the assessment of the flatness deviation close to the definition (minimum requirement) as follows:

The distances of the workpiece surface from the plane embodiment are measured. The inclination of the embodiment relative to the workpiece surface should be small. The measurement positions (points) are enumerated and plotted on a scale, Ⓐ in Fig. 13.30, measuring points A1 to C5. At each point, the measured value is indicated. In the view Ⓑ the height differences from the plane embodiment are plotted on the scale. The straight line P_x touches the points from above and is aligned with respect to the points according to the minimum requirement. P_x determines one direction of the reference plane.

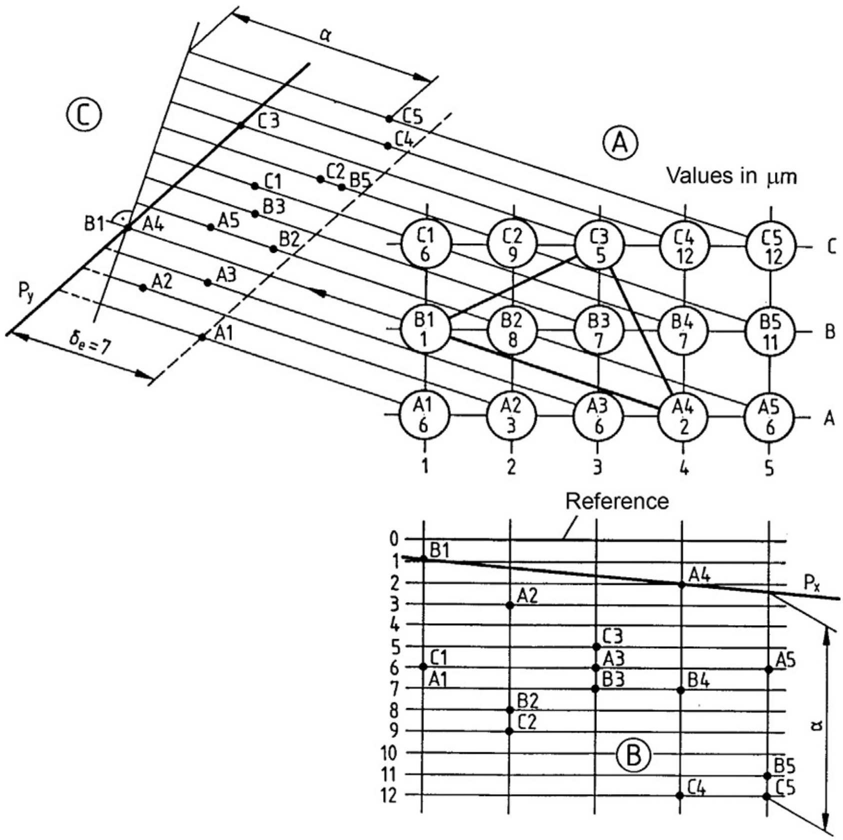


FIG. 13.30 Assessment of the flatness deviation according to the method ZNIITM

In an inclined plane ©, perpendicular to the straight line P_x (connection B1–A4), the distances of the surface points from P_x (in view Ⓑ) are plotted (e.g. a). The straight line P_y touches the highest point and is directed according to the minimum requirement ($\delta_m = \delta_e = \min$). P_y determines the other direction of the reference plane. The point with the largest distance (in the direction of P_x, B1–A4) to P_y corresponds to the flatness deviation $\delta_m = \delta_e$ (point A1). (The reference plane in Fig. 13.30 is determined by the points B1, C3, A4).

Another method according to TGL 39 094 for the assessment of the flatness deviation close to the definition uses analogue mechanical devices (Fig. 13.31).

The pins are movable and are to be aligned on a larger scale according to the measured deviations. As all pins are of equal length, the other side exhibits a mirror image of the measured form. By inclining the device on a measuring plate, the position is to be determined in which the pin with the largest distance to the measuring plate has the least distance. This distance corresponds to the flatness deviation δ_m.

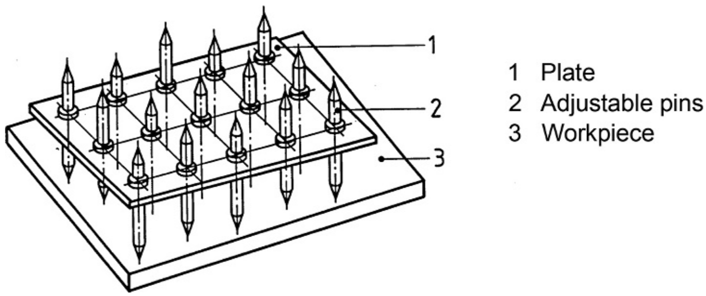


FIG. 13.31 Analogue mechanical device for the assessment of the flatness deviation

13.7.3.3 Assessment of the flatness deviation with straightedge and dial indicator

The procedure for the assessment of the flatness deviation with straightedge and dial indicator (Fig. 13.32) is as follows:

1. straightedge in position C1–A5, supports adjusted to 0;
2. centre point B3 measured and registered;
3. straightedge in position A1–C5, value of B3 from step 2 aligned and straightedge at the ends aligned to equal distances; the two diagonals define the plane embodiment; measurements A1 and C5 registered;
4. straightedge in position A1–C1 aligned to the already registered measured values A1 and C1, B1 measured and registered, etc.

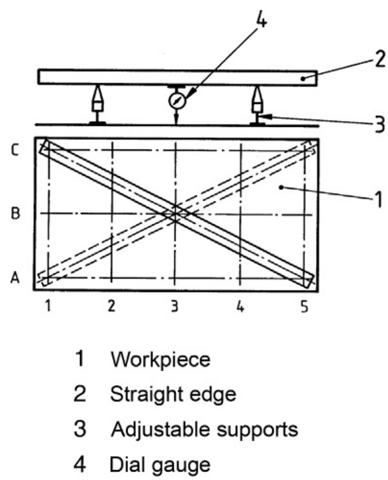


FIG. 13.32 Assessment of the flatness deviation with straightedge and dial indicator

13.7.4 Assessment of roundness deviations

13.7.4.1 Definition

The deviations of the workpiece circumference from an (almost) ideal reference circle (embodiment, e.g. established by a circular movement) are measured in cross sections perpendicular to the axis (see 13.3.2 and 13.3.3).

When the reference circle does not intersect (but eventually touches) the workpiece circumference line, the roundness deviation δ_m according to ISO 1101 is the difference between the largest and smallest radial distance of the workpiece circumference from the reference circle (Fig. 13.33 and Table 13.1).

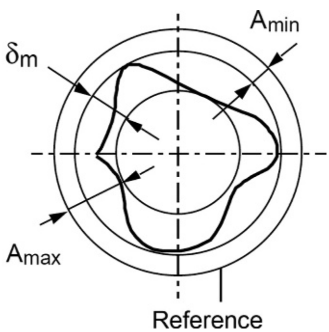


FIG. 13.33 Roundness measurement with reference circle that does not intersect the workpiece circumference line: $\delta_m = A_{max} - A_{min} \leq t_m$

When the reference circle does intersect the workpiece circumference line, the roundness deviation δ_m according to ISO 1101 is the sum of the largest radial distances of the workpiece circumference line from the reference circle on both sides of the reference circle (Table. 13.1), i.e. the range of the local deviations e of the workpiece circumference line from the reference circle (Fig. 13.34).

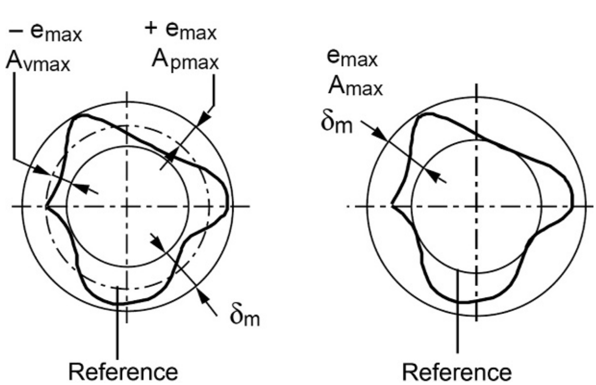


FIG. 13.34 Roundness measurement with reference circles that intersect or touch the workpiece circumference line: $\delta_m = A_{max} + A_{min} \leq t_m$

The reference circle is to be aligned according to the minimum requirement (Fig. 13.2). See also 13.3.2 and 13.3.3.

The roundness deviation δ_m must not exceed the roundness tolerance t_m: $\delta_m \leq t_m$.

For the assessment of the roundness deviation with coordinate measuring systems or with form measuring instruments, the following reference circles are standardized according to ISO 4291:

a) (total) least-squares circle (LSC), basis for tolerance or deviation zone Z_q
b) minimum zone circle (MZC), basis for tolerance or deviation zone Z_z
c) contacting circle (MIC, MCC), basis for tolerance or deviation zone Z_i or Z_c

These reference circles intersect or touch the workpiece circumference line.

According to ISO 1101, the minimum zone circle (minimax, Chebyshev) applies. However, for practical reasons the other definitions are sometimes applied.

The minimum zone circle leads to the smallest values of roundness deviation. According to TGL 39 096, the (random) differences in the values of the roundness deviation caused by the different definitions of the reference circle are up to +15%.

The least-squares circle needs fewer measuring points than the other reference circles to be sufficiently stable. Therefore the least-squares circle is preferred in the measurement technique.

For the definition of the cross sections perpendicular to the axis, see 13.7.2.1 and 13.3.3.

There are standards for roundness deviations: ISO 12 181-1 for terms and definitions and ISO 12 181-2 for measurement.

13.7.4.2 Measuring methods

The reference circle (embodiment) may be established by:

– high-precision circular movement	(form measuring instrument)
	(measurement while workpiece mounted in centres or in a chuck);
– high-precision straight guides perpendicular to each other and calculation of circles	(coordinate measuring systems);
– revolving on a measuring plate	(measurement of diameters);
– revolving in a V-block	(three-point measurement);
– circular device	(measurement in a ring or on a plug);
– circular marking	(profile projector).

When polar diagrams are used, it should be noted that the form in the profile diagram looks quite different from the form of the real profile, because of the amplification of radial distances (Fig. 13.35).

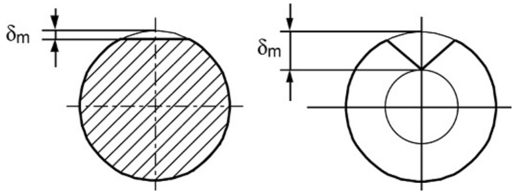


FIG. 13.35 Distortion of a profile diagram caused by the amplification in the diagram: left, without amplification and without distortion; right, with amplification and distortion

13.7.4.3 Two-point measurement, three-point measurement

For the methods, see Figs 13.36 and 13.37.

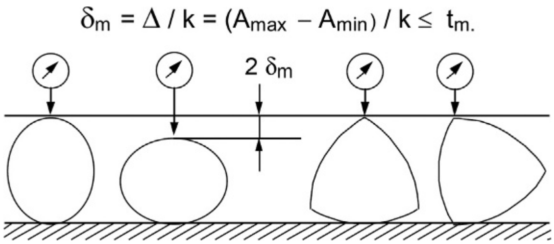


FIG. 13.36 Oval and lobed forms

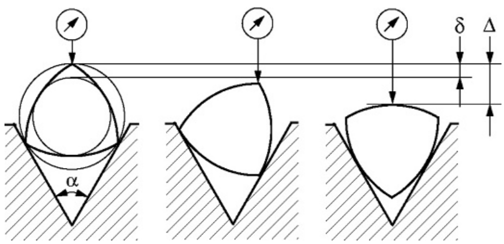


FIG. 13.37 Assessment of form deviations of lobed forms in V-blocks; summit method

With two-point measurements (measurements of diameter), lobed forms cannot be detected (Fig. 13.36). When lobed forms are possible (e.g. with centreless grinding, reaming), in addition to the two-point measurement other inspections (e.g. three-point measurements, shown in Fig. 13.37) are necessary. With three-point measurements (e.g. in V-blocks, as in Fig. 13.37), the possibility of detecting the different types of lobed forms and other forms differs, depending on the type of form deviation (e.g. type of lobed form) and on the type of measuring method. Table 13.3 gives correction values k. The measured values $\Delta = A_{max} - A_{min}$ are to be divided by k in order to correspond to the roundness deviation δ_m:

$$\delta_m = \Delta / k = (A_{max} - A_{min}) / k \leq t_m.$$

TABLE 13.3 Correction values k for the measurement of form deviations by two-point measurements and three-point measurements n is the order of the harmonics (number of undulations) to be assessed

		Angles α and α /β of measuring methods (Figs 13.38 to 13.40)											
							Summit method		Ryder method				
n	60°	72°	90°	108°	120°	180°	60°/30°	120°/60°	60°	72°	90°	108°	120°
2	–	0,47	1	1,4	1,6	2	1,4	2,4	2	1,5	1	0,62	0,42
3	3	2,6	2	1,4	1	–	2	2	3	2,6	2	1,4	1
4	–	0,38	0,41	–	0,42	2	1,4	1	2	2,4	2,4	2	1,6
5	–	1	2	2,2	2	–	2	2	–	1	2	2,2	2
6	3	2,4	1	–	–	2	0,73	0,42	1	0,38	1	2	2
7	–	0,62	–	1,4	2	–	2	2	–	0,62	–	1,4	2
	I								I				
8	–	1,5	2,4	1,4	0,42	2	1,4	1	2	0,47	0,4	0,62	1,6
9	3	2	–	–	1	–	2	2	3	2	–	–	1
10	–	0,70	1	2,2	1,6	2	1,4	2,4	2	2,7	1	0,24	0,42
11	–	2	2	–	–	–	–	–	–	2	2	–	–
12	3	1,5	0,41	0,38	2	2	0,73	1	1	0,47	2,4	0,62	–
13	–	0,62	2	1,4	–	–	–	–	–	0,62	2	1,4	–
14	–	2,4	1	–	1,6	2	1,4	0,42	2	0,38	1	2	0,42

15	3	1	–	2,2	1	–	2	2	3	1	–	2,2	1
16	–	0,38	2,4	–	0,42	2	1,4	1	2	2,4	0,41	2	1,6
17	–	2,6	–	1,4	2	–	2	2	–	2,6	–	1,4	2
18	3	0,47	1	1,4	–	2	0,73	2,4	1	1,5	1	0,62	2
19	–	–	2	–	2	–	2	2	–	–	2	–	2
20	–	2,7	0,41	2,2	0,42	2	1,4	1	2	0,7	2,4	0,24	1,6
21	3	–	2	–	1	–	2	2	3	–	2	–	1
22	–	0,47	1	1,4	1,6	2	1,4	0,42	4	1,5	1	0,62	0,42

180° is two-point measurement.

With three-point measurements, a distinction must be made between the summit method (V-support, two fixed anvils on one side and the measuring anvil, indicator, on the other side of the workpiece; see Figs 13.37 to 13.39) and the rider method (two fixed anvils and the measuring anvil, indicator, on the same side of the workpiece; see Fig. 13.40).

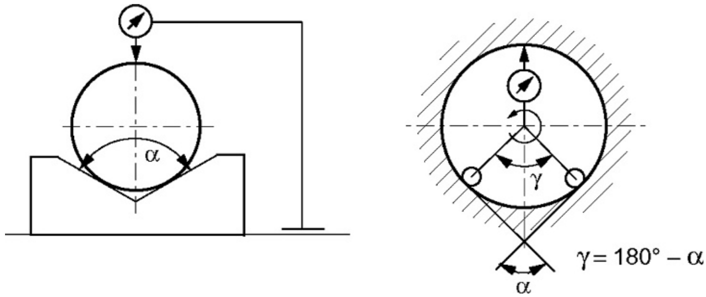


FIG. 13.38 Three-point measurement: summit method, symmetrical setting

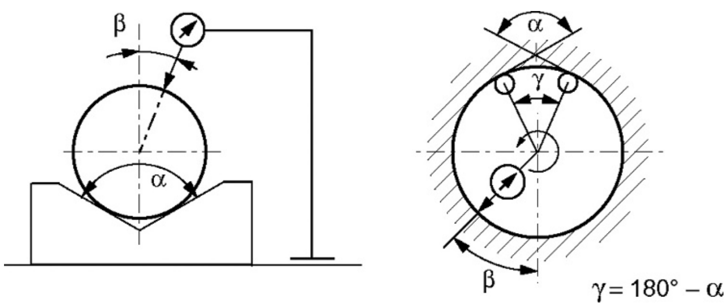


FIG. 13.39 Three-point measurement: summit method, asymmetrical setting

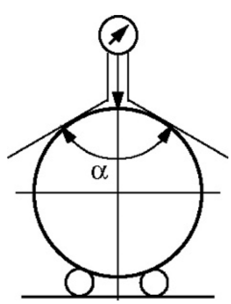


FIG. 13.40 Three-point measurement: rider method, symmetrical setting

The number of sides n of the lobed form (number of undulations) may be detected by counting the significant maximum (or minimum) indications during one revolution of the workpiece in the V-block (e.g. in the 72-degree V-block). When there is a superposition of harmonics (undulations) (see Figs 13.113 and 13.115), selection of the proper correction value is impossible to achieve realistically. The correction value must be estimated. For this case, TGL 39 096 recommends the asymmetrical summit method (Fig. 13.39) $\alpha = 120°/\beta = 10°$ and the average correction value 1.2.

In order to cover all possible form deviations and numbers of undulations, BS 3730: Part 3: 1982 recommends that one two-point measurement and two three-point measurements be taken at different angles between fixed anvils and angles be selected from the following:

symmetrical setting: $\alpha = 90°$ and $120°$ or $\alpha = 72°$ and $108°$;
asymmetrical setting: $\alpha = 120°$, $\beta = 60°$ or $\alpha = 60°$, $\beta = 30°$;

where

α = the angle between fixed anvils
β = the angle between the direction of measurement and the bisector of the angle between fixed anvils (Fig. 13.39).

The measuring anvil shall be selected from Table 13.4. ISO 4292 has a similar table but does not specify spherical or cylindrical anvils.

TABLE 13.4 Types of anvil

Surface form	Anvil radius in mm	Surface radius in mm
Convex surface	Spherical: 2,5	All
Convex edge	Cylindrical: 2,5	All
Concave surface	Spherical: 2,5	≥ 10
Concave edge	Cylindrical: 2,5	≥ 10
Concave surface	Spherical: 0,5	<10
Concave edge	Cylindrical: 0,5	<10

It is further recommended that the following fixed anvils be used:
Measurement:

- for external measurement: V-support; the median plane of the V-support should be in the same plane as the plane of measurement
- for internal measurement: spheres with a small radius; the median plane of the spheres should be in the same plane as the plane of measurement.

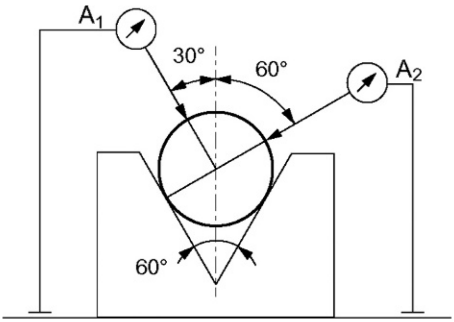


FIG. 13.41 Combination of two-point and three-point measurement according to U. Barth for the determination of the smallest actual size, the mating size and the roundness deviation

With a combination of two-point measurement, three-point measurement and evaluation by calculation according to U. Barth (see Ref. [3]) (Fig. 13.41), approximations of the smallest diameter, the mating size and the roundness deviation can be made. This method can assess form deviations up to the 10th harmonic (e.g. number of sides of the lobed form). It is not necessary to know the number of the harmonic.

The actual sizes are detected by the dial indicator A_2. The mating size P and the roundness deviation δ_m[1)] are to be calculated as follows:

When the indicator reading is $\Delta A_1 \leq A_2$:

$$P \approx A_{2min} + \Delta A_2 = A_{2max}$$

$$\delta_m \approx \Delta A_2/2 \leq t_m$$

When the indicator reading is $\Delta A_1 > \Delta A_2$:

$$P \approx A_{2min} + \Delta A_1/2$$

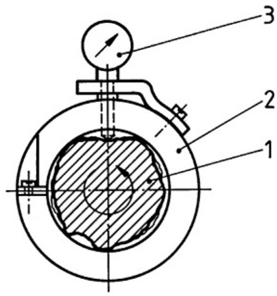
$$\delta_m \approx \Delta A_1/2 \leq t_m$$

This method is relatively precise for short (disk-shaped) workpieces (which are practically not bent along the axis).

13.7.4.4 Assessment with measuring ring or measuring plug

Figure 13.42 shows a measuring ring according to the former East German Standard TGL 39 096. The inner ring surface establishes the circle embodiment. The diameter is adjustable within the range of the size tolerance. For the assessment

1) Applicable to disk-shaped workpieces or when the straightness and the parallelism deviations are negligible.



1 Workpiece
2 Embodiment (ring)
3 Dial gauge

FIG. 13.42 Assessment of roundness deviations with a measuring ring

of the roundness deviations of holes, a similar method with measuring plugs can be used. These assessments are close to the definition of the roundness deviation according to ISO 1101.

13.7.4.5 Simplified inspection

The deviations are typically much smaller than the tolerances. Then, instead of roundness, the easy-to-measure circular radial run-out deviations can be measured. When this is within the tolerance, the roundness deviation is also within the tolerance.

13.7.5 Assessment of cylindricity deviations

13.7.5.1 Definition

When the reference cylinder does not intersect the workpiece surface (but eventually touches it), the cylindricity deviation $\delta_m = \delta_z$, according to ISO 1101, is the difference between the largest and smallest radial distances of the workpiece surface from the reference cylinder. It is calculated using $\delta_m = \delta_z = A_{max} - A_{min}$, similarly to the roundness measurement shown in Fig. 13.33.

When the reference cylinder does intersect the workpiece surface, the cylindricity deviation, according to ISO 1101, is the sum of the largest radial distances of the workpiece surface from the reference cylinder on both sides of the reference cylinder (Table 13.1), i.e. the range of the local deviations *e* of the workpiece surface from the reference cylinder are measured (Fig. 13.34).

The reference cylinder is to be aligned according to the minimum requirement (Fig. 13.2). When methods a), b), c), e) of 13.7.5.2 (measurement in sections) are used, the alignment of the workpiece according to 13.3.2 and 13.3.3 must be observed.

The cylindricity deviation $\delta_m = \delta_z$ must not exceed the cylindricity tolerance t_m: $\delta_m = \delta_z \leq t_m$

For the assessment of the cylindricity deviation with coordinate measuring systems or form measuring instruments, the following reference cylinders are used:

a) (total) least-squares cylinder (by Gauss, cylinder for which the sum of the squares of the radial distances to the surface is a minimum, regression cylinder);

b) minimum zone cylinder (by Chebyshev, cylinder for which the maximum radial distance to the surface is a minimum);

c) contacting cylinder (maximum inscribed cylinder for holes, minimum circumscribed cylinder for shafts).

These reference cylinders intersect or touch the workpiece surface. According to ISO 1101, the minimum zone cylinder (minimum requirement) applies. However, for practical reasons sometimes the other definitions are applied.

The minimum zone cylinder leads to the smallest values of cylindricity deviation δ_m.

The least-squares cylinder needs fewer measuring points than the other reference cylinders to be sufficiently stable. Therefore the least-squares cylinder is preferred in the measurement technique.

There are standards for cylindricity deviations: ISO 12 180-1 for terms and definitions and ISO 12 180-2 for measurement.

13.7.5.2 Measuring strategies

The following measuring strategies are to be distinguished:

a) radial section method;	d) helical method;
b) generatrix method;	e) extreme positions method;
c) generatrix and radial section method;	f) point method.

In all methods the measured profiles or points must be related to the same coordinate system. The workpiece (feature) axis must be aligned parallel to the straight-line guidance of the measuring device. This alignment can also be replaced by calculation. For the definition of the axis, see 13.7.2.1. See also 13.3.3.

(a) Radial section method

The profile lines (circumferences) of several cross sections perpendicular to the reference cylinder (axis of measurement) are to be plotted in one common polar diagram and evaluated according to the minimum requirement (Fig. 13.43).

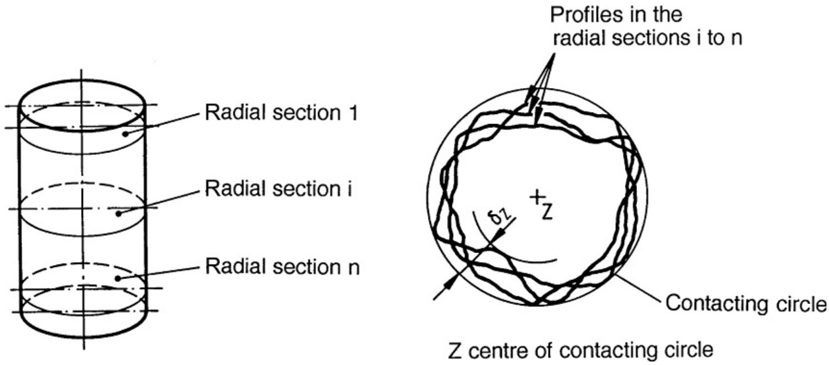


FIG. 13.43 Radial section method

b) Generatrix method

In several sections containing the reference cylinder axis (axis of measurement), the two always-opposite profile lines (generatrixes) are assessed, plotted in one common diagram and evaluated according to the minimum requirement (Fig. 13.44).

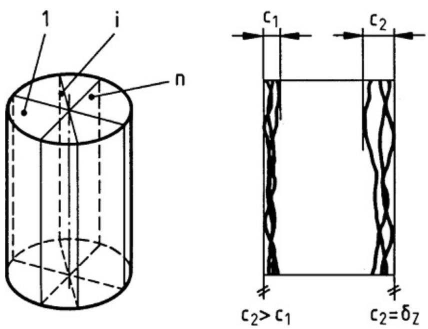


FIG. 13.44 Generatrix method

c) Generatrix and radial section method

This is a combination of the generatrix method and the radial section method. It gives the smallest measurement uncertainty, because in both directions (axial and radial) the deviations are assessed with small sampling intervals (e.g. according to the Nyquist theorem) (see 13.9.1). Therefore this method is recommended.

d) Helical method

The profile lines assessed by a helical probing trace, probing perpendicularly to the reference cylinder axis (axis of measurement), are plotted in one common diagram and evaluated according to the minimum requirement (Fig. 13.45). Eventually the profiles of two cross sections at the features end perpendicular

to the reference cylinder (axis of measurement) will also be assessed and included in the diagram and evaluation.

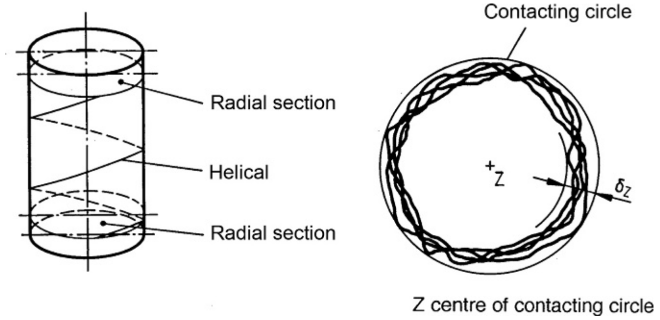


FIG. 13.45 Helical method

e) Point method

Randomly distributed points of the workpiece surface are assessed, plotted in one common polar diagram perpendicular to the reference cylinder (axis) and evaluated according to the minimum requirement (Fig. 13.46).

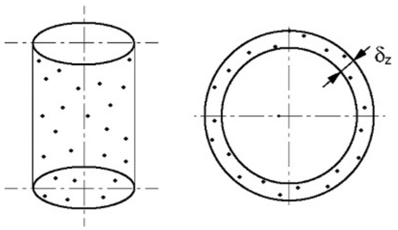


FIG. 13.46 Point method

f) Extreme positions method

According to TGL 39 097, the following are assessed and plotted:

– generatrix in one section containing the reference cylinder axis (generator lines at 0° and 180°)
– circumference lines in cross sections perpendicular to the reference cylinder axis in the positions where the largest (max) and the smallest (min) distances of the generator lines occur (cross sections a) and b).

The evaluation is shown in Fig. 13.47. Z_1, c_{max} and Z_2, c_{min} in the section 0° to 180° are the same in both top and bottom sections of Fig.13.47. The cylindricity deviation is Γ_{max} - Γ_{min}.

The cylindricity deviation is

$$\delta_m = \Gamma_{max} - \Gamma_{min} \leq t_m$$

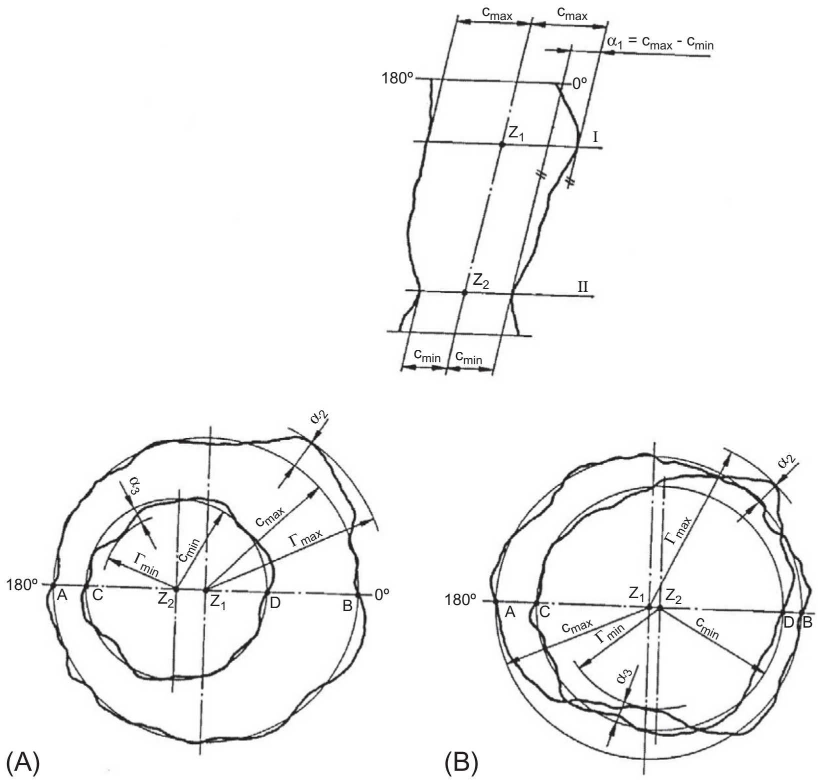


FIG. 13.47 Extreme position methods, evaluation, (A) and (B) show two possibilities of the radial sections

13.7.5.3 Measuring methods

13.7.5.3.1 Measuring the cylindricity deviation

The reference cylinder (embodiment) may be established by:

- high precision circular movement and high precision straight guides perpendicular to the circular movement	(form measuring instrument) (measuring while workpiece is in centres or in a chuck)
- high precision straight guides perpendicular to each other and cylinder calculation	(coordinate measuring system)
- revolving in V-block and parallel straight guides	(three-point measurement with V-block, straight edge, dial indicator and measuring table)

The three-point measurement does not allow establishing a stable workpiece coordinate system (the workpiece axis has no stable position in the coordinate system of measurement). Therefore this method is a rough approximation (see 13.7.4.3).

13.7.5.3.2 Assessment by measurement of components

a) Approximate evaluation of the cylindricity deviation δ_z from the roundness deviation δ_m and the parallelism deviation δ_o[2)] of the generator lines: $\delta_z \approx \delta_m + \delta_o \leq t_z = t_m$

b) Approximate evaluation of the cylindricity deviation δ_z from the total run-out deviation δ_r relative to another cylindrical feature: $\delta_z \approx \delta_r \leq t_z = t_m$

The total run-out deviation δ_t is always larger than the cylindricity deviation δ_z, because δ_t comprises the cylindricity deviation and the coaxiality deviation.

13.7.5.3.3 Simplified inspection

The deviations are mostly much smaller than the tolerances. Then instead of cylindricity the easy to measure total radial run-out deviations can be measured. When this is within the tolerance, the cylindricity deviation is also within the tolerance.

13.7.6 Assessment of profile deviations of lines

13.7.6.1 Definition

The deviations of the workpiece section (profile) line from an (almost) ideal reference line (embodiment) are measured in cross sections, the orientation of which is defined by the datum system (related profile tolerance) by the section plane indicator (see Fig. 6.15) or for surfaces of revolution containing the workpiece established axis (of another feature), as described in 13.3.3.1.

The location and orientation of the reference line are defined relative to the datum system (related profile tolerance), as explained in 13.7.9, or are defined by the minimum requirement, i.e. the maximum distance of the surface line from the reference line shall be a minimum (unrelated profile tolerance). (Regarding the alignment of the workpiece established axis, in principle similar effects occur as explained in 13.3.3.2 and 13.3.3.3.)

The profile deviation δ_m is the maximum distance of the workpiece section (profile) line from the reference line, measured perpendicular to the reference line (Table 13.1).

The profile deviation δ_m must not exceed half of the profile tolerance t_m.

$$\delta_m \leq t_m/2$$

2) The cylindricity deviation comprises deviations of roundness, straightness and parallelism As the parallelism deviation δ_p, according to ISO 1101, comprises the deviations and the inclinations of the generator lines relative to each other, the cylindricity deviation δ_z cannot be larger than the sum of the roundness deviation δ_m and the deviation δ_o. In most cases it is smaller.

13.7.6.2 Measuring methods

The reference line (embodiment) may be established by:

- profile template	copying system[3)]
- profile marking	profile projector
- high precision straight guides perpendicular to each other and calculation of the reference line	coordinate measuring system

13.7.7 Assessment of profile deviations of surfaces

13.7.7.1 Definition

The deviations of the workpiece surface from an (almost) ideal reference surface (embodiment, e.g. established by a form template or by calculation in a coordinate measuring system) are measured. For the location and orientation of the reference feature (alignment of the workpiece), the same applies as to the assessment of profile deviations of lines (see 13.7.6.1).

The profile deviation δ_m is the maximum distance of the workpiece surface from the reference surface, measured perpendicular to the reference surface (Table 13.1).

The profile deviation δ_m must not exceed half of the profile tolerance t_m: $\delta_m \leq t_m/2$.

13.7.7.2 Measuring methods

The reference line (embodiment) may be established by:

- surface template copying system[3)]
- high-precision straight guides coordinate measuring system perpendicular to each other and calculation of the reference line
- profile template rotating profile template device (for surfaces of revolution only)

13.7.8 Assessment of orientation deviations

13.7.8.1 Definition

The deviations of the workpiece surface or the workpiece line (axis, generator line) from an (almost) geometrically ideal reference feature (plane or straight-line embodiment) are measured. The reference feature is to be aligned according to the datum or datum system (parallel, perpendicular, in the specified angle).

When the reference feature does not intersect the workpiece feature (but eventually touches it), the orientation deviation δ_o according to ISO 1101 is

3) The copying system probes the workpiece and the template with the same tip radius and records the deviations of the workpiece from the template

the difference between the largest and smallest distances of the workpiece feature from the reference feature (Table 13.1).

When the reference feature does intersect the workpiece feature, the orientation deviation δ_o according to ISO 1101 is the sum of the largest distances of the workpiece feature from the reference feature on both sides of the reference feature (Table 13.1).

The orientation deviation δ_o must not exceed the orientation tolerance t_o: $\delta_o \leq t_o$.

Note that this definition is in accordance with ISO 1101 and ISO 5459. The orientation deviation encloses the flatness deviation of the surface to be measured or the straightness deviation of the axis or line to be measured.

13.7.8.2 Combination of features

There are possible deviations of parallelism, perpendicularity or angularity:

- of a plane surface relative to a plane surface;
- of a plane surface relative to a straight line (axis, generator line);
- of a straight line (axis, generator line) relative to a plane surface;
- of a straight line (axis, generator line) relative to a straight line (axis, generator line).

13.7.8.3 Measurement methods

13.7.8.3.1 General

For the reference straight line (embodiment of a straight line) see 13.7.1.3, and for the reference plane (embodiment of a plane) see 13.7.3.2.

The reference angle (embodiment of angle) may be established by:

– high-precision divided circle	(dividing head)
– high-precision straightness	(form measuring instrument)
– guides, perpendicular	(coordinate measuring system)
– to each other	
– solid angle	(measuring plate with measuring angle)
	(sine bar rule)
– optical beam	(pentaprism)

With the use of mandrels, the cylindricity deviation of the workpiece hole to be measured is eliminated from the measurement. Therefore this method is an approximation according to ISO 1101.

13.7.8.3.2 Assessment of orientation deviations of straight generator lines or plane surfaces by measuring distances

For the measuring method, see Fig. 13.48.

The orientation deviation δ_o (deviation of parallelism, deviation of perpendicularity) is the difference between the largest distance A_{max} and the smallest distance A_{min} (indicator reading) of the workpiece feature from the measuring table or from the solid angle, and must not exceed the orientation tolerance t_o: $\delta_o \leq t_o$.

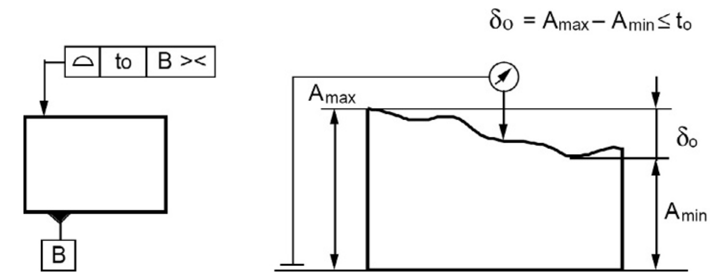


FIG. 13.48 Assessment of the orientation deviation δ_o of a surface or generator line

13.7.8.3.3 Assessment of the orientation deviation of an axis by measuring distances

For the measuring method, see Fig. 13.49.

The differences of the distances from the measuring plate are to be measured. During the measurements, the indicators must keep their adjustment, but need not be calibrated in height. As the measuring directions are opposite, the difference of half the sums of the distances from the measuring plate is assessed.

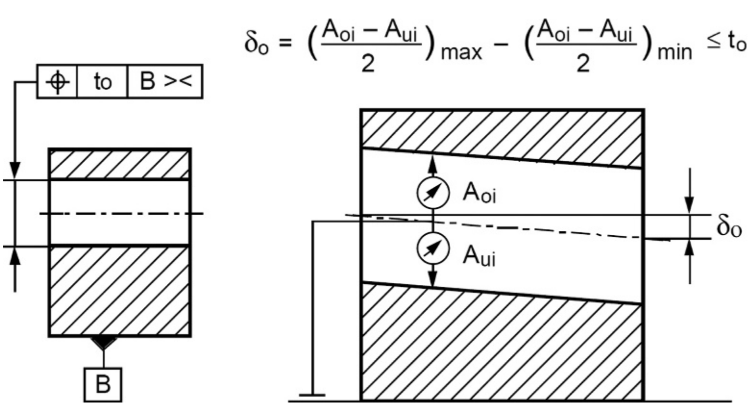


FIG. 13.49 Assessment of the parallelism deviation δ_o of an axis, evaluated from measurement of distances

Figure 13.50 shows the assessment of the orientation deviation with two parallel measuring plates. Fig. 13.51 shows the assessment by distance and diameter measurements.

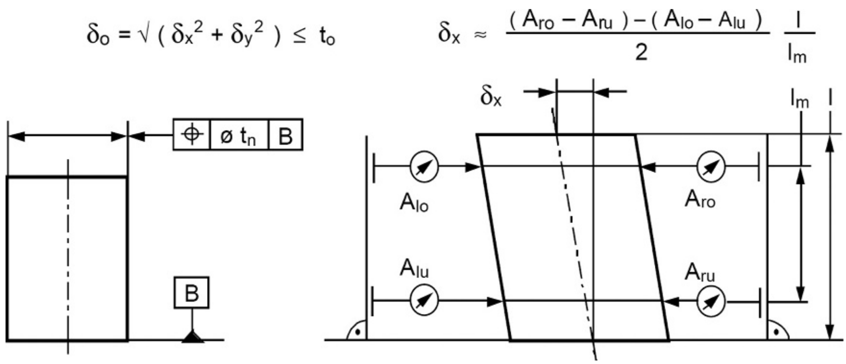


FIG. 13.50 Assessment of the perpendicularity deviation δ_o of an axis, evaluated from measurements of distances from parallel planes

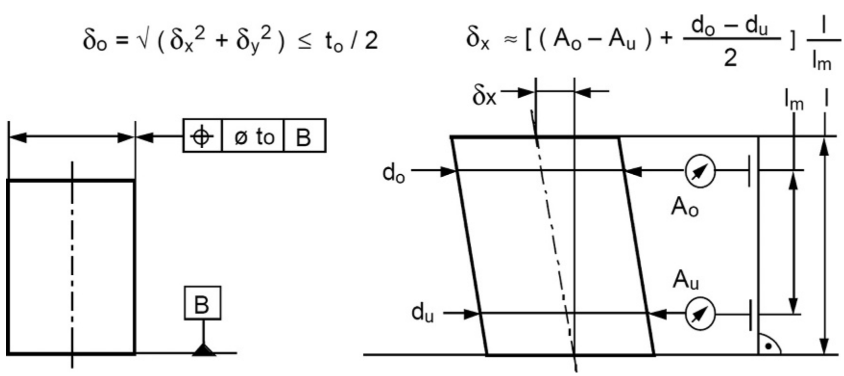


FIG. 13.51 Assessment of the perpendicularity deviation δ_o of an axis, evaluated from measurements of distances and diameters

When the tolerance zone is cylindrical, the orientation deviations are to be assessed in two mutually perpendicular axial sections. The cylindrical orientation deviation in this case is

$$\delta_o = \sqrt{\left(\delta_x{}^2 + \delta_y{}^2\right)}$$

(see 13.6).

13.7.8.3.4 Assessment of orientation deviations with form measuring instruments or coordinate measuring systems

For the assessment of the orientation deviation of a plane surface or a straight generator line, see 13.7.8.3.2.

The orientation deviation of an axis can be assessed by measurements of distances, as described in 13.7.8.3.3, or by the centres of cross sections assessed from the circumference lines in cross sections perpendicular to the axis.

For the assessment of cross section centres, see 13.7.2.1.

13.7.9 Assessment of location deviations

13.7.9.1 Definition

The deviations of the workpiece surface (planar surface) or the workpiece line (axis) from an (almost) ideal reference feature (plane or straight line embodiment) are measured. The reference feature (embodiment) is to be aligned according to the datum or datum system (parallel, perpendicular, in the specified angle). The distance between the reference feature and the datum (theoretically exact location of the embodiment) is zero (coaxiality, symmetry) or specified by theoretically exact dimensions (in rectangular frames). The same applies to the distances of the reference features between each other, if applicable.

The embodiment can be located apart from the theoretically exact location, but then the measured values have to be corrected accordingly.

The location deviation δ_s according to ISO 1101 is the largest distance of the workpiece feature (surface, axis) from the reference feature (which is or is considered to be in the theoretically exact location and orientation) (Figs 13.52 and 13.53, Table 13.1), and must not exceed half of the location tolerance t_s: $\delta_s \le t_s/2$.

Note that this definition is in accordance with ISO 1101 and ISO 5459. The location deviation encloses the form and the orientation deviation of the surface to be measured or the straightness deviation of the axis to be measured.

13.7.9.2 Combination of features

There are possible deviations of location:

- of a point relative to a plane surface;
- of a point relative to a straight line (axis, generator line);
- of a point relative to a point;
- of a straight line (axis, generator line) relative to a plane surface;
- of a straight line (axis, generator line) relative to a straight line;
- of a plane surface relative to a plane surface;
- of a surface relative to a datum system;
- of a line relative to a datum system;
- of a point relative to a datum system;
- of a straight line (axis) relative to a straight line (axis) (coaxiality);
- of a centre point relative to a straight line (axis) (coaxiality);

– of a straight line (axis) relative to a plane surface (symmetry surface[4)]);
– of a plane face (symmetry face) relative to a plane face (symmetry surface[4)]).

13.7.9.3 Measuring methods

13.7.9.3.1 General

For the reference feature (embodiment of straight line or plane), see 13.7.8.1. For the embodiment of the orientation, see 13.7.8.1.

The embodiments are to be located in the theoretically exact distance (zero or specified theoretically exact dimension) from the datum, or the measured values are to be corrected accordingly.

13.7.9.3.2 Assessment of the location deviation of a straight generator line or a plane surface by measuring distances

For the measuring method, see Fig. 13.52. The location deviation δ_s is the absolute value of the maximum difference between the measured distance A and the theoretically exact distance A_{th} and must not exceed half of the location tolerance t_s: $\delta_s \leq t_s/2$.

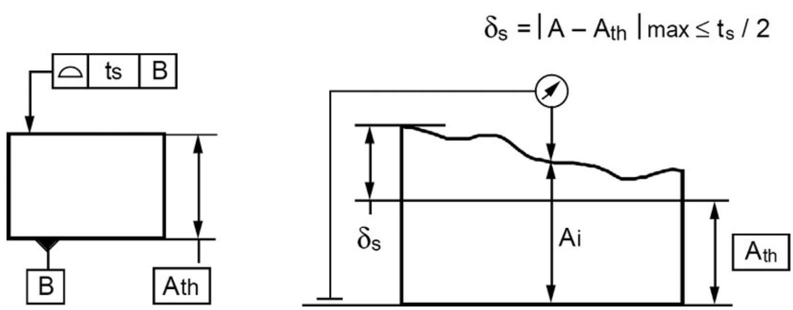


FIG. 13.52 Assessment of location deviation of a generator line or a surface

13.7.9.3.3 Assessment of the location deviation of an axis relative to plane surface by measuring distances

For the measuring method, see Figs 13.53 and 13.54. Both dial indicators are calibrated to the distance from the measuring plate. In each section, the arithmetical mean of the distances from the measuring plate $(A_{oi}+A_{ui})/2$ is assessed. The location deviation δ_s is the largest absolute value of the difference between the arithmetical mean and the theoretically exact distance, and must not exceed half of the location tolerance t_s: $\delta_s \leq t_s/2$.

4) Median surface of a plane pair

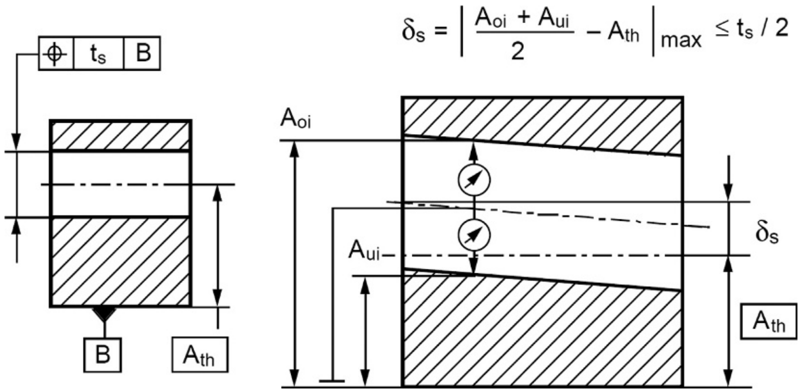


FIG. 13.53 Location deviation δ_s of the axis of the hole calculated from distance measurements

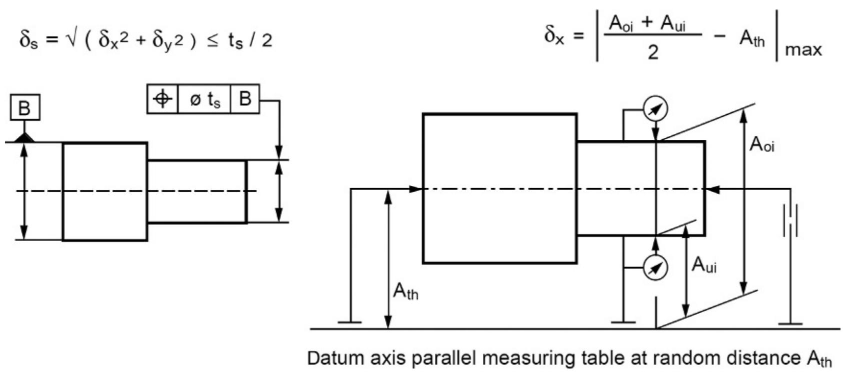


FIG. 13.54 Assessment of the location deviation δ_s (here coaxiality deviation) of a shaft by measuring distances

When the tolerance zone is cylindrical, the values $A_i = (A_{oi} + A_{ui})/2 - A_{th}$ (Fig. 13.54) must be assessed in each axial location in sections perpendicular to each other, A_{ix} and A_{iy}. The cylindrical location deviation δ_s is the largest value of $\sqrt{(A_{ix}^2 + A_{iy}^2)}$ and must not exceed half of the location tolerance t_s: $\delta_s \leq t_s/2$.

When the theoretically exact distance from the datum is specified by theoretically exact dimensions arranged in a chain, the sum of these dimensions applies to the distance of the tolerance zone and reference feature from the datum (Fig. 13.55).

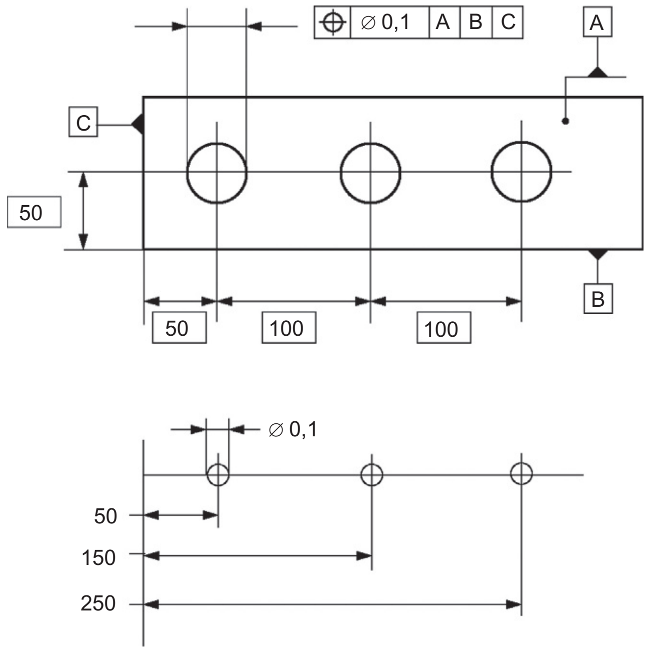


FIG. 13.55 Addition of theoretically exact dimensions arranged in a chain

13.7.9.3.4 Assessment of location deviations with form measuring instruments or coordinate measuring systems

For the assessment of the location deviation of a plane surface or a straight generator line, see 13.7.9.3.2.

The location deviation of an axis can be assessed by measurements of distances, as described in 13.7.9.3.3, or by the centres of cross sections assessed from the circumference lines in cross sections perpendicular to the axis.

For the assessment of cross section centres, see 13.7.2.1.

13.7.9.3.5 Assessment of coaxiality deviations or symmetry deviations by measuring distances

The coaxiality deviation δ_s and the symmetry deviation δ_s are half of the largest absolute value of the difference of opposite distances (Figs 13.56 and 13.57), and must not exceed half of the coaxiality tolerance t_s or half of the symmetry tolerance t_s:

$$|A_1 - A_2|_{max} / 2 \leq t_s / 2$$

$$|A_1 - A_2|_{max} \leq t_s$$

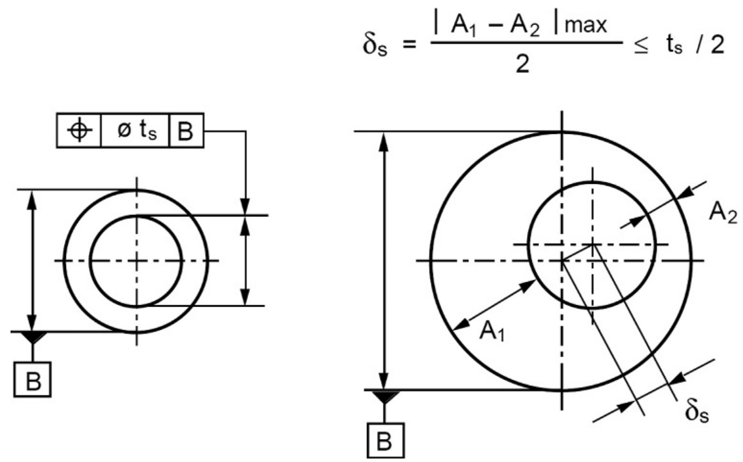


FIG. 13.56 Assessment of the coaxiality deviation δ_s by measuring distances

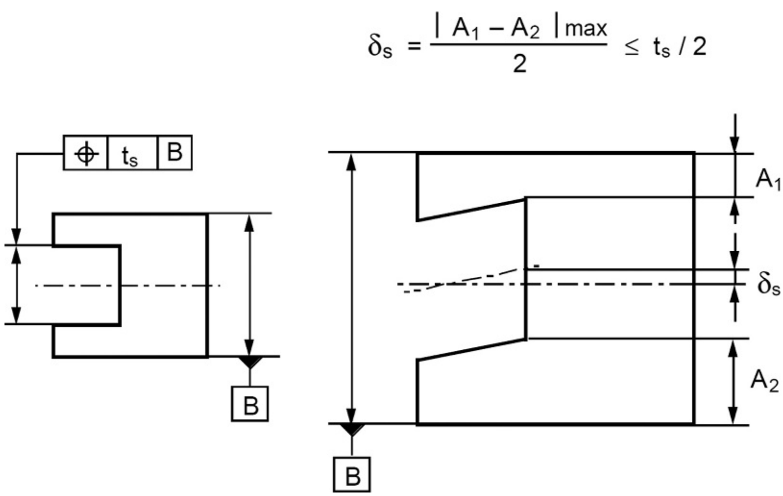


FIG. 13.57 Assessment of the symmetry deviation δ_s by measuring distances

Figure 13.58 shows the inspection of a keyway for a key assembly with interference fits at the key and at the shaft and hub (see also 13.7.8.1).

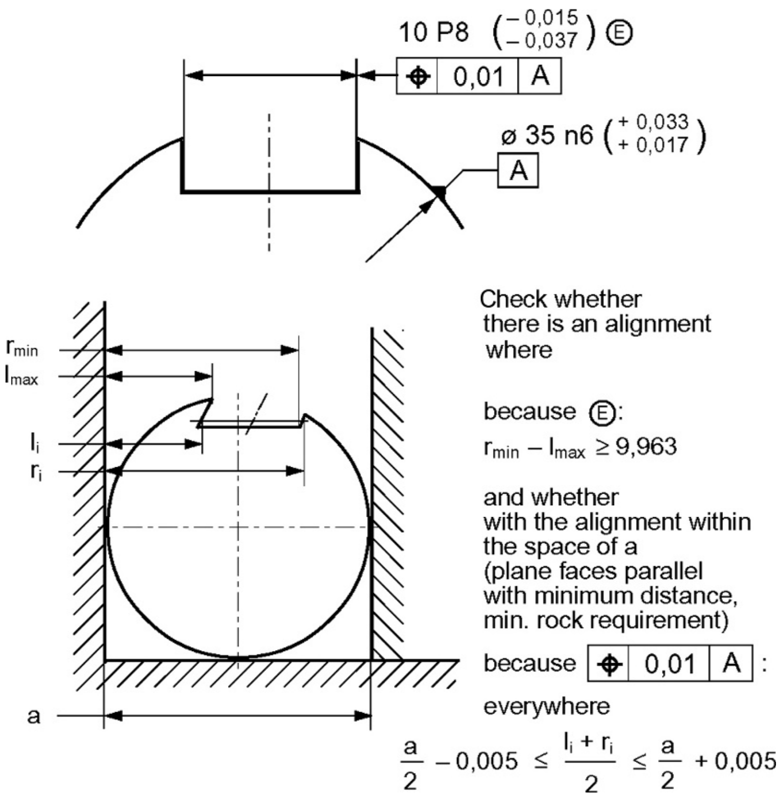


FIG. 13.58 Measurement of the symmetry deviation of a keyway (toleranced regardless of feature size)

13.7.9.3.6 Inspection of coaxiality deviations by measuring run-out deviations

Measuring the coaxiality deviation is often complicated and costly. Therefore, for the inspection of the workpiece, often the simpler run-out deviations are assessed (see 13.7.10.2.1) and compared with the value of the coaxiality tolerance. Only when the run-out deviation exceeds the coaxiality tolerance is it checked to determine whether twice the coaxiality deviation exceeds the coaxiality tolerance (or whether the larger run-out deviation is caused by roundness deviations only, which is permissible).

13.7.9.3.7 Assessment of coaxiality deviations or symmetry deviations with form measuring instruments or coordinate measuring systems

The coaxiality deviations and symmetry deviations can be assessed by measurements of opposite distances, as described in 13.7.9.3.5. The coaxiality deviation can also be assessed by the centres of cross sections assessed by the

circumference lines in cross sections perpendicular to the axis. For the assessment of cross section centres, see 13.7.2.1.

For the definition of median surfaces[5)] see Fig. 4.9.

13.7.10 Assessment of run-out deviations

13.7.10.1 Datums

For the assessment of the run-out deviation, the workpiece is to be aligned according to the datum axis, or the coordinates are to be transformed accordingly by calculation.

For the datum, the minimum rock requirement according to ISO 5459 applies (see 3.3.1).

The following practical solutions are available:

a) Mandrel

The mandrel should fit into the hole without clearance. If the mandrel rocks, the minimum rock requirement applies.

b) Centre bores

This is according to the definition when the drawing indicates the centre bores as datum. When the drawing indicates the axis of a cylindrical feature (not the centre bores) as the datum, the measurement is made incorrect by the form deviations and the eccentricity of the centre bores relative to the correct datum axis (of the cylindrical feature).

c) Chuck

The chuck must have a small run-out deviation in comparison with the run-out tolerance. The suitability can be checked prior to the measurement by measuring the run-out deviation of an almost-perfect cylinder embodiment in the chuck. If necessary and possible, the workpiece datum feature is to be aligned within the chuck with the aid of an indicator to indicate the least possible indication difference.

d) Revolving table

The datum feature is to be aligned with the aid of an indicator in two cross sections, (e.g. 1/8 of the datum feature length apart from the ends) according to the least possible indication difference. This alignment deviates from the theoretically exact datum according to ISO 5459, when the datum feature axis deviates from straightness.

---

5) Similar to the way in which the actual axis deviates from a straight line according to the form deviations of the cylindrical surface, the actual median surface deviates from a plane according to the form deviations of the planar surfaces establishing the plane pair

e) Coordinate measuring system

From the assessed (probed) points of the datum feature surface, the contacting cylinder (see 13.7.2.1) must be determined. When the contacting cylinder can rock, the minimum rock requirement according to ISO 5459 applies (see Fig. 3.81). However, in a practical sense the contacting cylinder is aligned coaxial to the regression cylinder (because the appropriate mathematical technique has been developed).

f) Form measuring instrument

Depending on the type of instrument procedure, d) or e) applies.

g) V-blocks, measuring table

This method should be applied only when the cylindricity deviation of the datum feature is small in comparison with the run-out tolerance. In other cases the measurement is made considerably inaccurate by the form deviations (see 13.7.4.3).

13.7.10.2 Measuring methods of circular run-out

The deviations of the workpiece surface section line from an (almost) geometrically ideal reference circle (circle embodiment) coaxial (concentrical) with the datum axis are measured in sections which are plane and perpendicular to the datum, or conical or cylindrical and coaxial to the datum. The circle embodiments are established by revolving the workpiece (Fig. 13.59) or the indicator (Fig. 13.60) using:

- length measuring instrument and workpiece support by
  - mandrel between centres,
  - centres,
  - chuck,
  - revolving table;
- form measuring instrument.

In coordinate measuring systems, the circle embodiment is established by calculation relative to the almost straight and perpendicular axes of the coordinate measuring system.

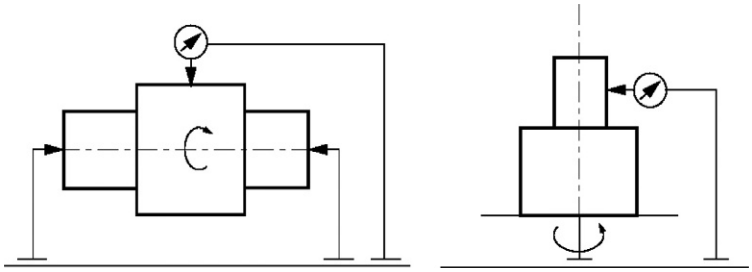


FIG. 13.59 Measurement of the run-out deviation with revolving workpiece

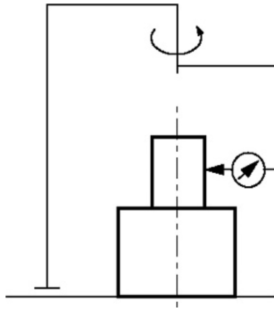


FIG. 13.60 Measurement of the run-out deviation with revolving indicator

13.7.10.2.1 Assessment of radial run-out deviations

The deviations of the workpiece circumference line of cylindrical features in sections perpendicular to the datum axis from an (almost) geometrically ideal reference circle (circle embodiment, simulated circle, ISO 5459) are measured. The circle embodiment is coaxial (concentrical) to the datum axis (Fig. 13.61).

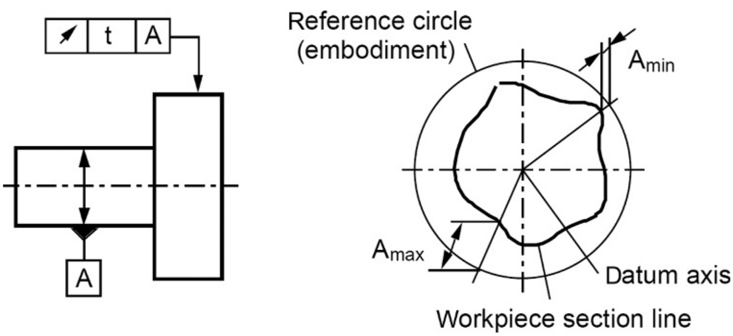


FIG. 13.61 Assessment of circular radial run-out deviation

When the reference circle does not intersect the workpiece circumference line, the circular radial run-out deviation δ_r is the difference between the largest, A_{max}, and the smallest, A_{min}, distances of the workpiece circumference line from the reference circle, and must not exceed the run-out tolerance t_r. Using dial indicators, this is the difference between the largest and smallest indication during one revolution:

$$\delta_r = A_{max} - A_{min} \leq t_r$$

Each section is to be considered independently from the others.

The radial run-out tolerance and deviation is only applicable to cylindrical or sectors of cylindrical features relative to cylindrical datum features.

Note that the circular radial run-out deviation comprises the eccentricity and parts of the roundness deviation of the feature to be measured. When (theoretically) the roundness deviation is zero, the circular radial run-out deviation is twice the eccentricity (Fig. 13.62).

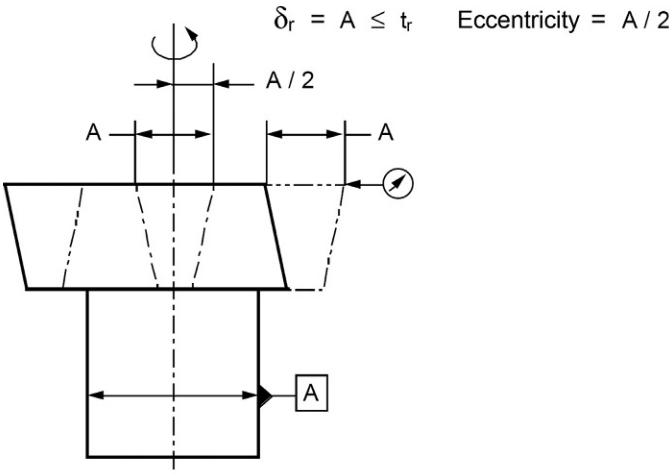


FIG. 13.62 Assessment of circular radial run-out deviation

When the workpiece feature to be measured is not very short (so that one measurement is not sufficient), the circular radial run-out deviation should be measured at least in the centre and near the ends of the workpiece feature.

13.7.10.2.2 Assessment of circular axial run-out deviations

The deviations of (circular) section lines of the (planar) workpiece surface in cylindrical sections coaxial with the datum axis (measuring cylinders) from an (almost) geometrically ideal reference plane (with the datum concentric circle in the reference plane, circle embodiment) are measured. The circle embodiment is coaxial (concentric) with the datum axis (Fig. 13.63).

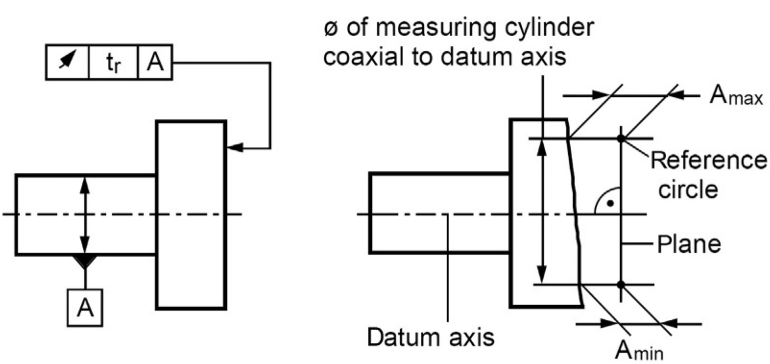


FIG. 13.63 Assessment of circular axial run-out deviation

When the reference plane does not intersect the workpiece section line, the circular axial run-out deviation δ_r is the difference between the largest, A_{max}, and smallest, A_{min}, distances of the workpiece section line from the reference plane (reference circle within the reference plane concentric to the datum), and must not exceed the run-out tolerance t_r. Using dial indicators, this is the difference between the largest and smallest indications during one revolution:

$$\delta_r = A_{max} - A_{min} \le t_r$$

Each cylindrical section is to be considered independently from the others.

Note that the circular axial run-out deviation is equal to the perpendicularity deviation of the circular section line of the surface, but may be smaller than the flatness deviation and the perpendicularity deviation of the entire surface (Fig. 3.67).

The circular axial run-out deviation should be measured at ≈ 1, 0,75 and 0,5 times the outer diameter. With surfaces manufactured by metal removing, the measurement near the outer diameter is in general sufficient, because here the largest run-out deviation occurs.

During measurement of the circular axial run-out deviation, the workpiece and indicator must be fixed in the axial direction by using.

- a chuck,
- an axial support against an auxiliary datum surface that is plane, perpendicular to the axis and with a diameter no less than the measured surface,
- an axial support coaxial to the datum axis,
- a support at the surface to be measured (Fig. 13.64).

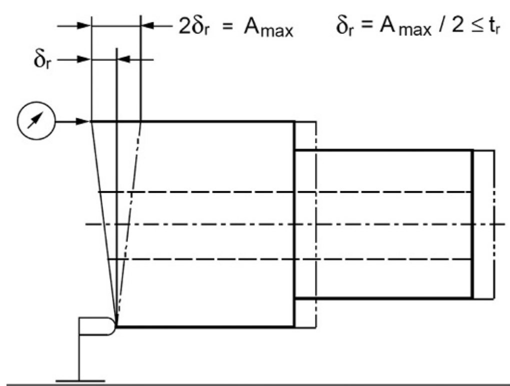


FIG. 13.64 Assessment of the circular axial run-out deviation δr with axial support at the outer diameter of the surface to be measured

Support at a surface different from the surface to be measured, where the support is apart from the datum axis, should be avoided, because deviations of form and orientation of the surface invalidate the measured result. When the support is at the surface to be measured and near the outer diameter, twice the value of the circular axial run-out deviation is indicated (Fig. 13.64).

13.7.10.2.3 Assessment of circular run-out deviations in any or in a specified direction

The deviations of (circular) section lines of the workpiece surface (of rotationally symmetric features) in conical sections coaxial with the datum axis from an (almost) geometrically ideal reference circle (circle embodiment) within the conical section and coaxial with the datum are measured. If not otherwise specified, the conical section (measuring cone, measuring direction) is perpendicular to the surface to be measured (Figs 13.65 and 3.69).

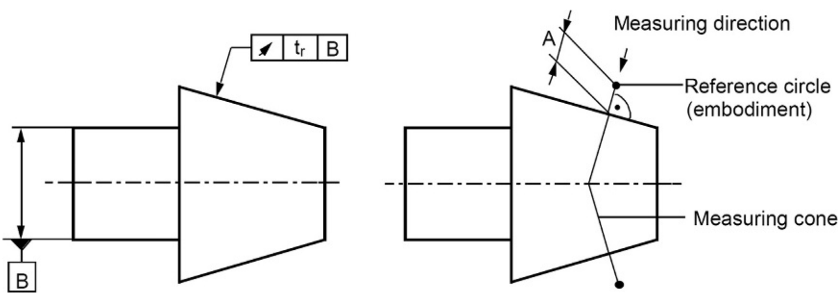


FIG. 13.65 Assessment of circular run-out deviation δ_r in any directions

When the reference circle does not intersect the workpiece section line, the circular run-out deviation (in any direction or in the specified direction) δ_r is the difference between the largest, A_{max}, and smallest, A_{min}, distances of the workpiece section line from the reference circle, and must not exceed the run-out tolerance t_r.

Using dial indicators, this is the difference between the largest and smallest indication during one revolution:

$$\delta_r = A_{max} - A_{min} \leq t_r$$

Each conical section is to be considered independently from the others.

When the workpiece feature to be measured is not very short (so that one measurement is not sufficient), the circular run-out deviation (in any direction or in the specified direction) should be measured in the centre and near the ends of the workpiece feature. For axial support of the workpiece the same applies as with the circular axial run-out deviation (see 13.7.10.2.2).

Note that the circular run-out deviation in any or in the specified direction is composed of the eccentricity and parts of the roundness deviations of the workpiece feature to be measured.

13.7.10.3 Assessment of total run-out deviations

The total run-out deviations

– total radial run-out deviation
– total axial run-out deviation

are to be distinguished from the circular run-out deviations. With total run-out deviations, the measuring sections are to be considered as dependent from each other. The circle embodiments establish the following:

– with the assessment of the total radial run-out deviation, one reference cylinder coaxial with the datum axis (i.e. using length measuring instruments, the instrument is to be guided along a straight line parallel to the datum axis) (Fig. 13.66);
– with the assessment of the total axial run-out deviation, one reference plane perpendicular to the datum axis (i.e. using length measuring instruments, the instrument is to be guided along a straight line perpendicular to the datum axis) (Fig. 13.67).

When the reference feature (cylinder, plane) does not intersect the workpiece surface, the total run-out deviation δ_r is the difference between the largest, A_{max}, and smallest, A_{min}, distances of the workpiece surface from the reference feature, and must not exceed the total run-out tolerance t_r. Using dial indicators, this is the difference between the largest and smallest indications during several (all) revolutions along the workpiece feature:

$$\delta_r = A_{max} - A_{min} \leq t_r$$

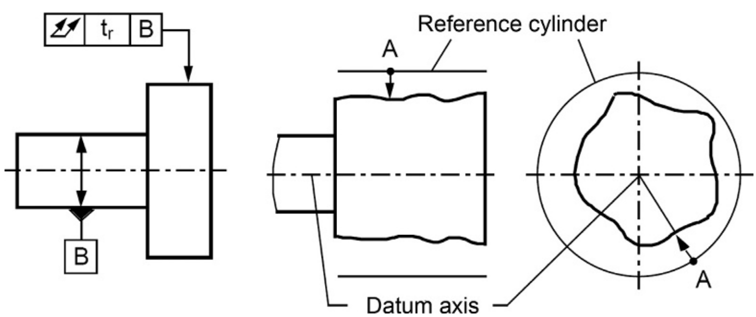


FIG. 13.66 Assessment of the total radial run-out deviation

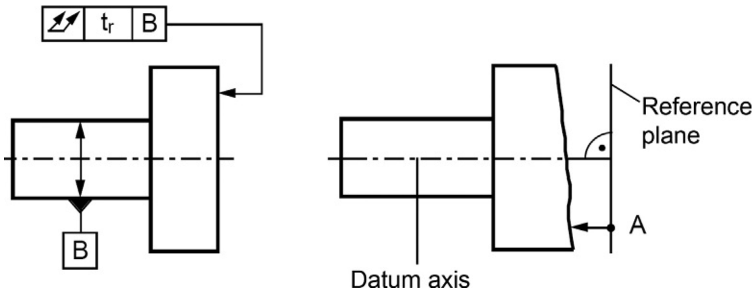


FIG. 13.67 Assessment of the total axial run-out deviation

13.7.11 Inspection of the envelope requirement

13.7.11.1 Definition

It must be determined if the surface of the workpiece feature (cylindrical surface or plane-pair surfaces) is contained within the (almost) geometrically ideal envelope of maximum material size (embodiment, reference feature).

13.7.11.2 Inspection methods

The embodiment of the envelope of maximum material size may be established by:

- functional gauge (gauge covering the entire length of the feature, e.g. plug, ring, squared block out of gauge blocks);
- calculation of the rectangular coordinates related to the straight and perpendicular guides of the coordinate measuring system (simulated gauge);
- calculation of the cylinder coordinates related to the straight and perpendicular guides of the form measuring instrument (simulated gauge);
- revolutions on a revolving table, (solid) straight and perpendicular guides and length measuring instrument;
- revolutions between centres, measuring table and length measuring instrument;
- revolutions in a chuck, measuring table and length measuring instrument;
- revolutions in V-block, measuring table and length measuring instrument;
- measuring table and length measuring instrument (for plane pairs).

13.7.11.2.1 Inspection with functional gauge

When the gauge covers the entire surface of the workpiece feature, this is an inspection close to the definition (Fig. 13.68).

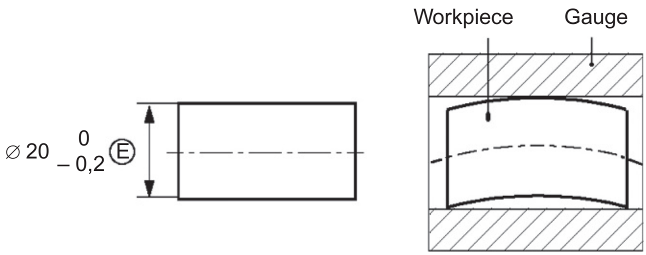


FIG. 13.68 Inspection of the envelope requirement with a functional gauge

13.7.11.2.2 Inspection with coordinate measuring system

When computer programs are applied to simulate the functional gauge, and sufficient points of the feature's surface are assessed (probed), this is an inspection close to the definition.

13.7.11.2.3 Inspection with form measuring instrument

When computer programs are applied to simulate the functional gauge, and sufficient sections of the workpiece feature are assessed, this is an inspection close to the definition. But form measuring instruments normally cannot measure diameters and therefore cannot inspect the envelope requirement.

13.7.11.2.4 Inspection with length measuring instrument and measuring table, centres or chuck or revolving table

The inspection method is shown in Fig. 13.69. The reference cylinder is established by the axis of revolution parallel to the measuring table (e.g. established by centres, one of which is adjustable) and a length measuring instrument calibrated to a distance (radius) from the axis of revolution of half the maximum material size (e.g. calibrated with a disc of known diameter).

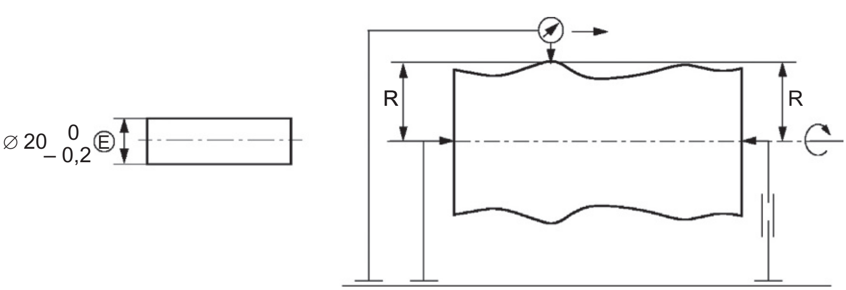


FIG. 13.69 Inspection of the envelope requirement: length measuring instrument, measuring table and centres

When sufficient points of the workpiece feature surface are assessed and the workpiece is correctly adjusted (e.g. with an adjustable support in order to avoid the effect shown in Fig. 13.70), this is an inspection close to the definition.

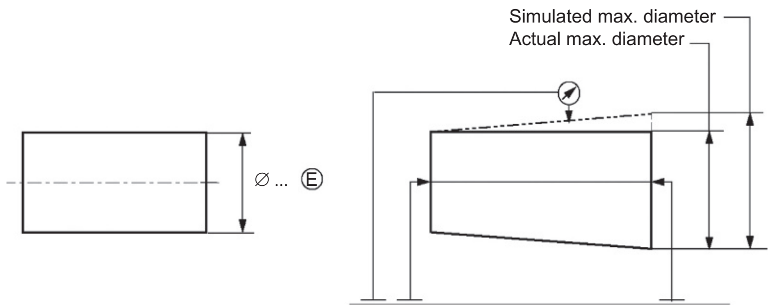


FIG. 13.70 Inspection of the envelope requirement: failure caused by insufficient adjustment of the workpiece feature

13.7.11.2.5 Inspection with length measuring instrument, measuring table and/or V-block

Revolutions of the workpiece feature in V-blocks or revolutions (rolling) on a measuring table cannot detect certain types of form deviations (oval and lobed) and should be applied only when these types of form deviations are not dominant (see 13.7.4.3).

13.7.11.2.6 Inspection with measuring table or measuring plate and length measuring instrument

The inspection method is shown in Figs 13.71 and 13.72, which show that it is necessary to inspect from both sides. With outer dimensions (Fig. 13.71), the method with the smaller maximum indication applies and with inner dimension (Fig. 13.72), the method with the larger minimum indication applies.

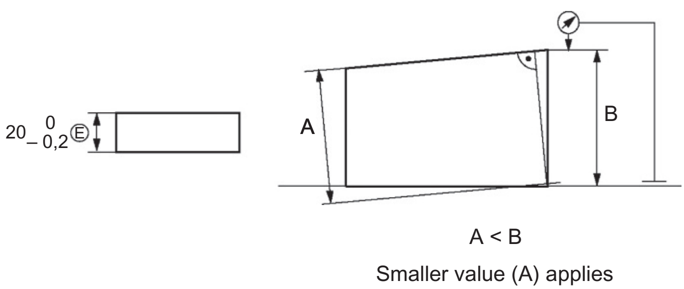


FIG. 13.71 Inspection of the envelope requirement of two parallel opposite plane surfaces on a measuring table with length measuring instrument: A < B, smaller value (A) applies

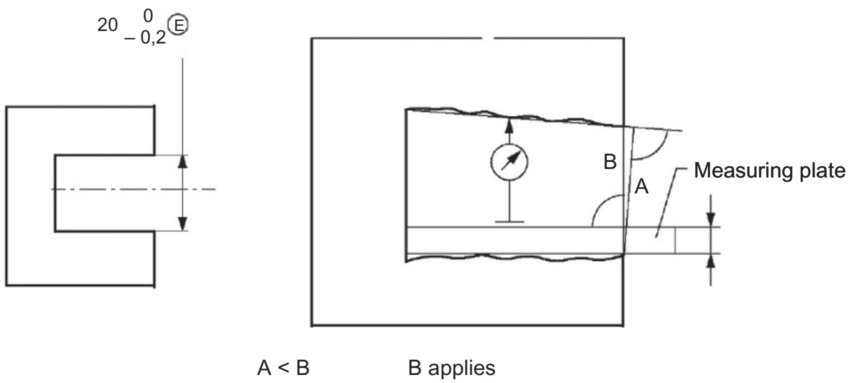


FIG. 13.72 Inspection of the envelope requirement of a plane-pair with a measuring plate (e.g. gauge block) and a length measuring instrument: A < B, larger value B) applies

When sufficient points of the workpiece feature surfaces are assessed, this is an inspection close to the definition.

13.7.12 Inspection of the maximum material requirement

13.7.12.1 Definition

It must be determined whether the surface of the workpiece feature (cylindrical surface or plan-pair surfaces) is contained in the (almost) geometrically ideal boundary of maximum material virtual size (embodiment, reference feature).

The reference features at the toleranced feature and at the datum feature have the (almost) geometrically ideal orientation (parallel, perpendicular, in specified angle) and, when the maximum material requirement is applied to a location tolerance, are located (almost) at the theoretically exact (geometrically ideal) location.

The reference feature at the toleranced feature is to be adjusted according to the datum or datum system (parallel, perpendicular, in specified angle). The orientation of the datum is defined by the minimum rock requirement at the datum feature of the workpiece (see Fig. 3.81) or by the adjustment of the workpiece in the datum system (see 3.3). The distance between reference feature and datum (theoretically exact location of the reference feature) is zero (coaxiality, symmetry) or specified by a theoretically exact dimension. The same applies to the distances between the reference features.

When the maximum material requirement is applied to the datum feature, the reference feature at the datum feature has (almost) geometrically ideal form at maximum material virtual size (see 4.4.3.5) and replaces the datum.

13.7.12.2 Inspection methods

The reference feature (embodiment of the boundary of maximum material virtual size) may be established by

– functional gauge;
– calculation of the rectangular coordinates related to the straight and perpendicular guides of the coordinate measuring system;
– calculation of the cylinder coordinates related to the straight, perpendicular and circular guides of the form measuring instrument (occurs with coaxiality tolerances to which the maximum material requirement is applied). Normally form measuring instruments cannot measure diameter and therefore cannot proof the maximum material requirement.

As an approximation, the actual sizes of the features and the actual distances of the features can be assessed for the inspection.

13.7.12.2.1 Inspection with functional gauge

For sizes and forms of the gauges, see 4.4.3.5. When the gauge covers the entire surfaces of the workpiece features, this is an inspection close to the definition (Fig. 13.73).

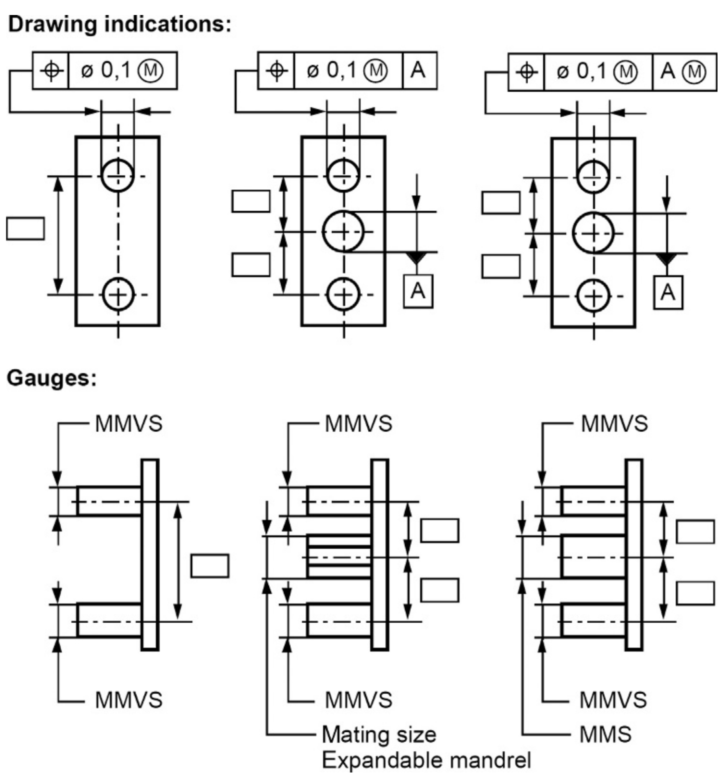


FIG. 13.73 Inspection of the maximum material requirement with functional gauges

13.7.12.2.2 Inspection with form measuring instrument or with coordinate measuring system

When computer programs are applied to simulate the functional gauge, and sufficient sections or points of the feature surfaces are assessed (probed), this is an inspection close to the definition.

13.7.12.2.3 Approximate inspection by measuring sizes and distances

When the deviations of form and orientation of the features are negligible compared with the location tolerances, the maximum material requirement applied to the location tolerance can be inspected by measuring the actual sizes of the toleranced feature(s), the actual sizes of the datum feature(s) and the actual distance(s) of the features.

The location tolerance may be exceeded by the difference between maximum material size and actual size (when the actual size does not take full advantage of the size tolerance).

13.7.12.2.3.1 Position tolerances without datum

Figure 13.74 shows an example of four holes related to each other by position tolerances but without datum. Fig. 13.75 shows the relevant functional gauge (embodiments). Fig. 13.76 shows the tolerance zones for the case when all actual sizes are at the maximum material size (ø 8,1) (and the holes almost of geometrically ideal form and orientation related to each other). Fig. 13.77 shows the dynamic tolerance diagram. It shows the permissible deviation of the actual axes from the geometrically ideal location depending on the (for all holes equal) actual sizes. The same information is given in Table 13.5.

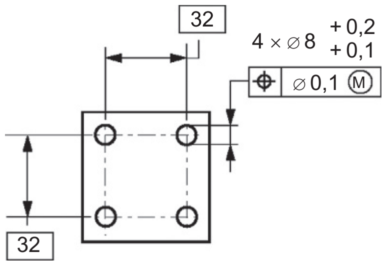


FIG. 13.74 Example of drawing indication, position tolerancing without datum

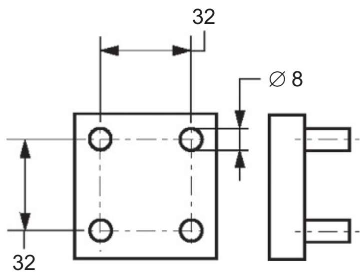


FIG. 13.75 Functional gauge according to the example in Fig. 13.74

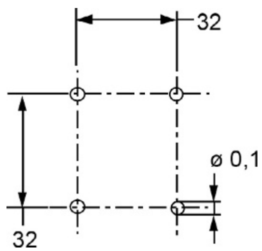


FIG. 13.76 Tolerance zones for the maximum material condition according to the example in Fig. 13.74

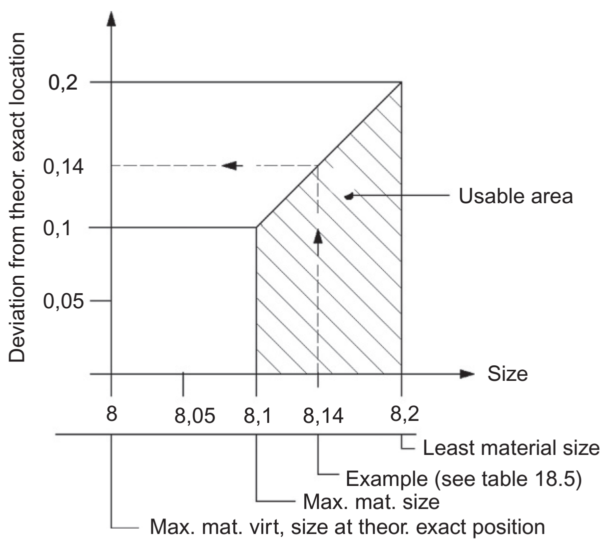


FIG. 13.77 Dynamic tolerance diagram for the example in Fig. 13.74

TABLE 13.5 Position tolerances for the example in Fig. 13.74

Diameter	Position tolerance
8,10	0,10
8,12	0,12
8,14	0,14
8,16	0,16
8,18	0,18
8,20	0,20

Measurements and graphical evaluations are as follows:

– specify the coordinate system for the measurement, e.g. the actual axis (centre) of the hole, left below, as coordinate origin, and the actual axis (centre) of the hole, right below, to determine the x axis (Fig. 13.78);
– assess the position deviations of the holes in this coordinate system;
– plot the tolerance zone in this coordinate system
– if a point remains outside the tolerance zone, measure the actual size of the feature (hole), enlarge the tolerance zone concentrically by the difference between maximum material size and actual size, and check whether the point is contained within this zone (Fig. 13.79).

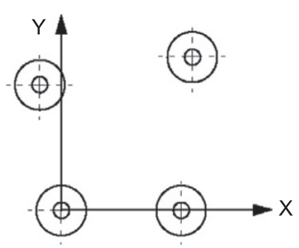


FIG. 13.78 Coordinate system for the measurement of the workpiece according to Fig. 13.74

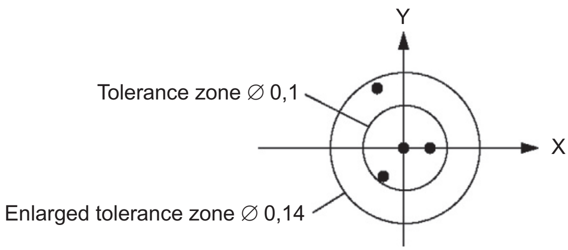


FIG. 13.79 Measured points (position deviations) and tolerance template of the workpiece according to Fig. 13.74

13.7.12.2.3.2 Position tolerances with datum

When the maximum material requirement applies to the datum, the pattern of tolerance zones (as a whole) may deviate from the theoretically exact location (relative to the datum axis) by the difference between the maximum material size and the actual size of the datum feature (this does not influence the location of the tolerance zones relative to each other).

Figure 13.80 shows an example of four holes related to each other by position tolerances and related to a datum hole. The maximum material requirement applies to all five holes. Fig. 13.81 shows the relevant functional gauge (embodiment). Fig. 13.82 shows the location and magnitude of the tolerance zones when

the toleranced holes	are at maximum material size,
	are at least material size;
the datum hole	is at maximum material size,
	is at least material size.

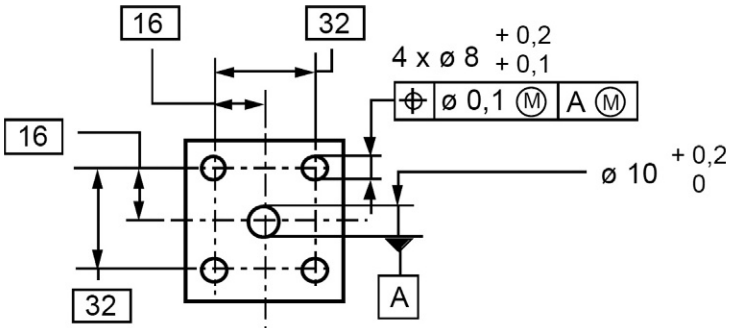


FIG. 13.80 Example of drawing indications, position tolerancing with datum

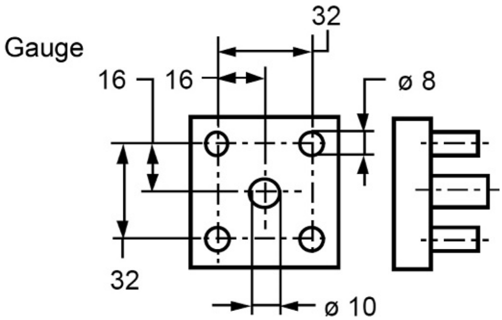


FIG. 13.81 Functional gauge according to the example in Fig. 13.80

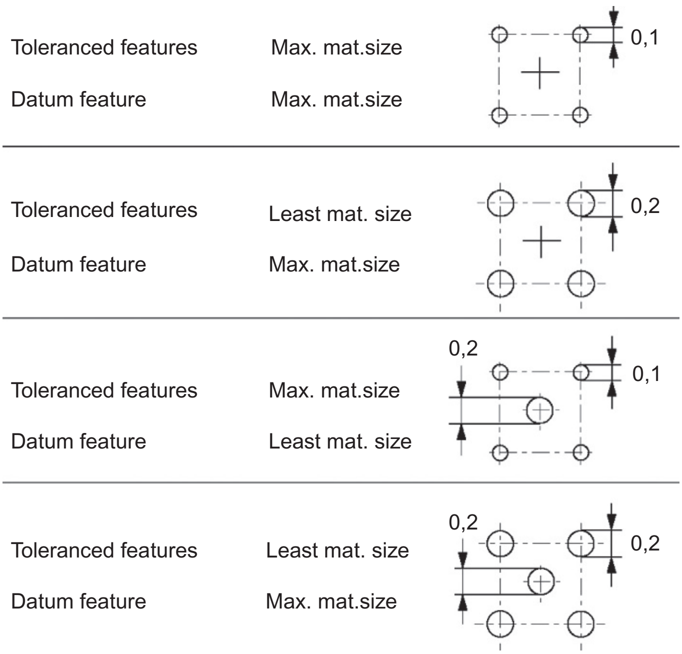


FIG. 13.82 Tolerance zones for the maximum material condition and for the least material condition of the example in Fig. 13.80

Table 13.6 gives the position tolerances of the toleranced features (four holes) depending on their actual sizes, and the floating zone of the datum feature, depending on its actual size.

TABLE 13.6 Position tolerances[1] for the example in Fig. 13.80

Diameter of toleranced hole		Position tolerance of all holes	Diameter of datum hole		Floating zone of datum
8,1	MMS	0,1	10	MMS	0
8,12		0,12	10,05		0,05
8,14		0,14	10,1		0,1
8,16		0,16	10,15		0,15
8,18		0.18	10,2	LMS	0,2
8,2	LMS	0,2			

[1]Each combination of the values in the 2. and 4. columns is possible. The values of the 2. and 4. columns cannot be added, because they have different effects. Some extreme combinations are shown in Fig. 13.82.

Measurements and graphical evaluations are as follows:

- specify the coordinate system for the measurement, e.g. the actual axis (centre) of the datum feature as coordinate origin, and the connection of the actual axes (centres) of the two holes below as the direction of the x axis (Fig. 13.83);
- assess the position deviations of the holes in this coordinate system (in an enlarged scale), Fig. 13.84
- use a transparent sheet (template) in the same scale with concentric;
  - tolerance zone;
  - floating zone (difference between maximum material size and actual size of the datum feature);
- and move it around until, if possible, all points are enclosed in the tolerance zone and the coordinate origin is still within the floating zone (Fig. 13.84);
- if a point remains outside the tolerance zone, measure the actual size of the feature (hole), enlarge the tolerance zone concentrically by the difference between maximum material size and actual size, check whether the point is contained within this enlarged zone while the other points are contained in the (concentric) original tolerance zone and the coordinate origin in the floating zone (Fig. 13.84).

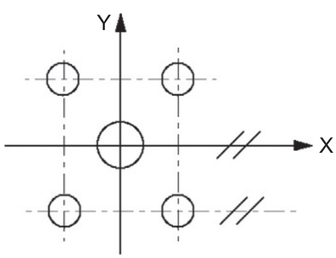


FIG. 13.83 Coordinate system for the measurement of a workpiece according to Fig. 13.80

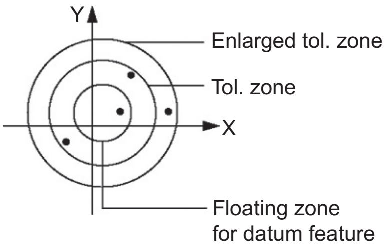


FIG. 13.84 Position deviations and tolerance template for a workpiece according to Fig. 13.80; the point is to be accepted

13.7.12.2.3.3 Position tolerances with and without datum

Figure 13.85 shows an example of four holes related to each other by rather small position tolerances and related to the datum features A and B (datum system) by rather large position tolerances.

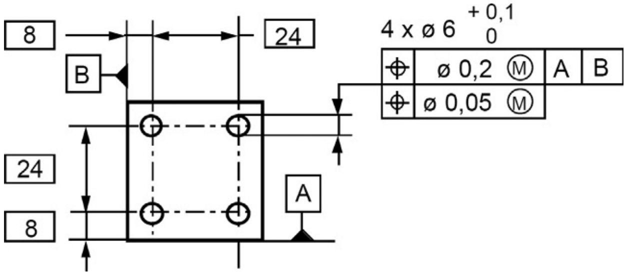


FIG. 13.85 Example of drawing indication, position tolerances with and without datum

Figure 13.86 shows the relevant functional gauge (embodiment).

Figure 13.87 shows the tolerance zones when all actual sizes are at the maximum material size (ø 6). Dynamic tolerance diagrams or tables containing the position tolerances depending on the actual sizes of the holes, similar to Fig. 13.77 and Table 13.6, could also be shown.

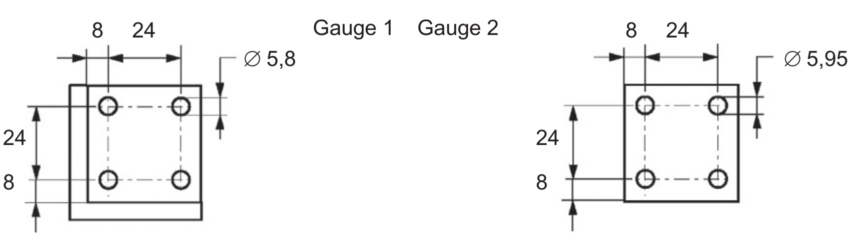


FIG. 13.86 Functional gauges according to the example in Fig. 13.85

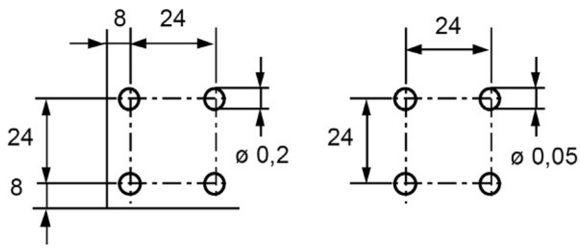


FIG. 13.87 Tolerance zones for the maximum material condition of the example in Fig. 13.85

Measurements and graphical evaluations are as follows:

- assess the position deviations of the holes in the coordinate system AB;
- plot the position deviations in an enlarged scale (Fig. 13.88);
- use a transparent sheet (template) in the same scale with the tolerance zone ø 0,05 (Fig. 13.88); the procedure is the same as described in 13.7.12.2.3.1;
- use a transparent sheet (template) in the same scale with the tolerance zone ø 0,2 with centre located at the coordinate origin, check whether all points are contained within this zone; if not, measure the actual size of the concerned hole, enlarge the tolerance zone by the difference between the maximum material size and the actual size, and check whether the point is within this zone (Fig. 13.88).

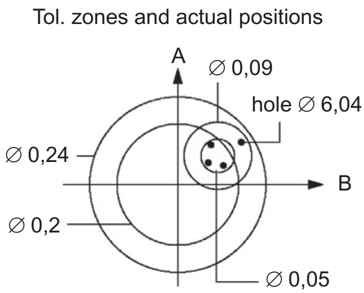


FIG. 13.88 Position deviations and tolerance templates for a workpiece according to Fig. 13.85; the part is to be accepted

13.7.12.2.3.4 Position tolerances for key ways

When gauges or coordinate measuring systems are not available, the part may be inspected similarly to the method shown in Fig. 13.89, by simulating the gauge using a measuring plate, a measuring right angle and a dial gauge.

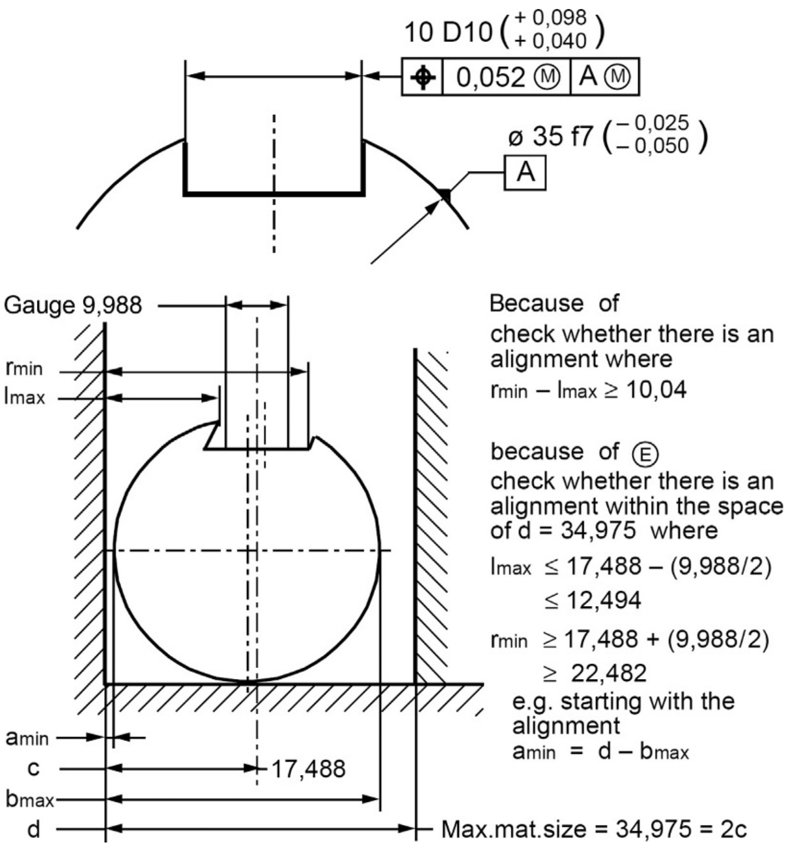


FIG. 13.89 Inspecting a keyway by simulating the gauge

13.7.12.2.4 Simplified inspection

When no functional gauges are available and coordinate measuring systems are not currently used, the workpieces may be inspected in the first step as if the maximum material requirement were not applied. Only if in this inspection the location tolerance is exceeded should it be checked in the second step, by a suitable method (e.g. by a coordinate measuring system), whether the maximum material requirement is violated.

A prerequisite for this procedure is that the total tolerance is split into a size tolerance and a location tolerance (and is not indicated as size tolerance together with a zero location tolerance).

13.7.13 Inspection of the least material requirement

13.7.13.1 Definition

This inspection is made to determine whether the surface of the workpiece feature (cylindrical surface or plane-pair surfaces) violates the (almost) geometrically ideal boundary of least material virtual size (embodiment, reference feature).

The reference features of the toleranced feature and of the datum feature have the (almost) geometrically ideal orientation (parallel, perpendicular, in specified angle) and, when the least material requirement is applied to a location tolerance, are located (almost) at the theoretically exact (geometrically ideal) location.

The reference feature at the toleranced feature is to be adjusted according to the datum or datum system (parallel, perpendicular, in specified angle). The orientation of the datum is defined by the minimum rock requirement at the datum feature of the workpiece (see Fig. 3.81) or by the adjustment of the workpiece in the datum system (see 3.3). The distance between reference feature and datum (theoretically exact location of the reference feature) is zero (coaxiality, symmetry) or is specified by a theoretically exact dimension. The same applies to the distances between the reference features.

When the least material requirement is applied to the datum feature, the reference feature at the datum feature has (almost) geometrically ideal form at least material virtual size (see 11.3). The surface of the datum feature must not violate this boundary (Fig. 13.90).

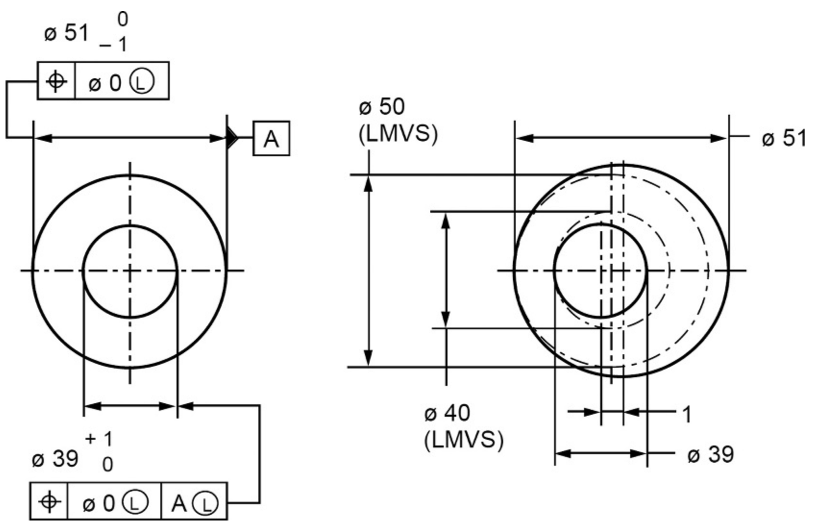


FIG. 13.90 Least material requirement applied to the toleranced feature and to the datum feature

13.7.13.2 Inspection methods

The reference feature (embodiment of the boundary of least material virtual size) may be established by

– calculation of the rectangular coordinates related to the straight and perpendicular guides of the coordinate measuring system;
– calculation of the cylinder coordinates related to the straight, perpendicular and circular guides of the form measuring instrument (with coaxiality tolerances to which the least material requirement is applied). Normally form measuring instruments cannot measure diameter and therefore cannot proof the least material requirement.

As an approximation, the actual sizes of the features and the actual distances of the features can be assessed for the inspection.

Since it must be checked as to whether the reference feature is entirely within the material of the workpiece feature (whether the workpiece feature's surface violates the reference feature), gauging is not possible.

13.7.13.2.1 Inspection with coordinate measuring systems

When computer programs are applied to simulate the geometrically ideal boundary of least material virtual size, and sufficient points or sections of the feature's surfaces are assessed, this is an inspection close to the definition.

13.7.13.2.2 Approximate inspection by measuring sizes and distances

When the deviations of form and orientation of the features are negligible compared with the location tolerances, the least material requirement applied to the location tolerance can be inspected by measuring the actual sizes of the toleranced feature(s), the actual sizes of the datum feature(s) and the actual distances of the features.

The location tolerance may be exceeded by the difference between the least material size and the actual size (when the actual size does not take full advantage of the size tolerance).

Figure 13.91 shows an example of application of the least material requirement. Fig. 13.92 shows the relevant geometrically ideal boundary at the least material virtual size that is coaxial to the datum A (reference feature).

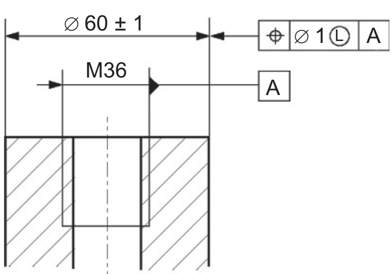


FIG. 13.91 Example of drawing indication, position tolerancing applied to the least material requirement

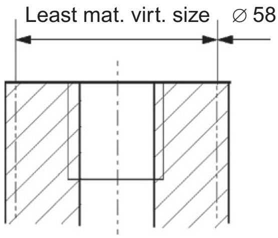


FIG. 13.92 Geometrically ideal boundary at least material virtual size for the example in Fig. 13.91

Figure 13.93 shows the position tolerance zone at least material size and at maximum material size. Table 13.7 lists the diameters of the position tolerance zones of the outer cylindrical feature depending on its actual sizes.

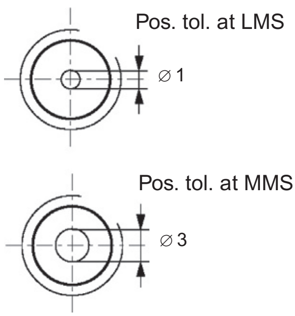


FIG. 13.93 Position tolerance zones at maximum material size and at least material size according to the example in Fig. 13.91

TABLE 13.7 Position tolerances for the example in Fig. 13.91

Diameter of cylinder	Position tolerance of cylinder
59 LMS	1
60	2
61 MMS	3

13.7.14 Assessment of the position deviation for projected tolerance zones

13.7.14.1 Definition

The deviations of a specified extension (projection) of the feature axis from an (almost) geometrically ideal reference straight line (embodiment, simulated straight line, ISO 5459) that is in the (almost) geometrically ideal orientation and location are measured. The position deviation δ_p is the largest distance (within the specified length) of the extension of the axis from the reference straight line and must not exceed half the position tolerance t_p: $\delta_p \leq t_p/2$.

13.7.14.2 Measuring methods

For the measurement, the extension of the axis to be assessed can be established, for example by.

- fitting, as far as possible without clearance, with an (almost) geometrically ideal counterpart (conical threaded mandrel, screw, bolt);
- simulation of the fitted counterpart (by calculation) in the coordinate measuring machine.

The position deviation along the specified extension of the feature axis is measured as described in 13.7.9.3.

See also 5.5.

13.8 Assessment of geometrical deviations of threaded features

If not otherwise specified, the tolerances of orientation, location or run-out of threaded features apply to the pitch diameter (ISO 1101). For the measurements, the following are used:

- form measuring instruments[6)]
- coordinate measuring systems[6)]
- special devices (see, e.g. Figs 13.94 to 13.97).

When measuring deviations of orientation, location or run-out, special measuring devices are to be applied for the support in the thread with or without clearance (with or without application of the maximum material requirement) or probing or tracing of threaded features.

6) If suitable probes and computer programs are available or if measured together with special devices as described in this section

For the support in the thread without clearance (datum without application of the maximum material requirement), the following may be used:

– conical threaded mandrel (e.g. with a cone angle of 0,5°);
– mandrel or ring with two parts of thread that can be adjusted in radial or axial direction in order to contact the thread flanks without clearance;
– two thread ring gauges screwed against each other

with surfaces to be supported and sufficiently cylindrical and coaxial with the thread.

The same devices may be used at the toleranced threaded features to assess the deviations of orientation or location or run-out by measurement at the cylindrical surfaces (see 13.7.8, 13.7.9 and 13.7.10).

Instead of expandable or conical or parted thread devices (mandrels), the revised standard ISO 4759-1 on tolerances for fasteners will probably recommend the use of (lubricated) spring washers to support the proper alignment of go gauge thread devices along the workpiece thread flanks (see Figs 13.94, 13.95, 13.97 and 13.99).

For probing at the pitch diameter by a self-centring probing mode, the following may be used:

– ball of diameter suitable for the thread pitch (Fig. 13.98 and Table 13.8);
– notch and/or measuring cone suitable for the thread pitch (median diameter of the truncated cone = P/2) (see Fig. 13.98);
– thread segments.

When deviations of orientation or location are to be measured with a probing ball by a self-centring probing mode, the probing ball should contact the thread flanks near the pitch diameter. Table 13.8 gives the ball diameters when the contact points are theoretically.

– between 0,75H and 0,375H of the thread profile (see Fig. 13.98);
– at the pitch diameter (0,5H).

D_{rec} in Table 13.8 refers to probing ball diameters that ensure contacts within this range.

When the deviation is measured (probed) directly at the pitch diameter, the stylus must be guided parallel and symmetrical with the pitch diameter axis, i.e. the probing direction line (axis) must meet the pitch diameter axis. With coordinate measuring systems, this guidance is not necessary when it is replaced by appropriate calculations.

When coordinate measuring systems and a small ball probe (smaller than according to Table 13.8) are used, the thread flanks are to be assessed in order to calculate the associated thread flanks. The line of intersection of two adjacent associated thread flanks is used in order to calculate the pitch diameter line according to the theoretical thread configuration (Fig. 13.98).

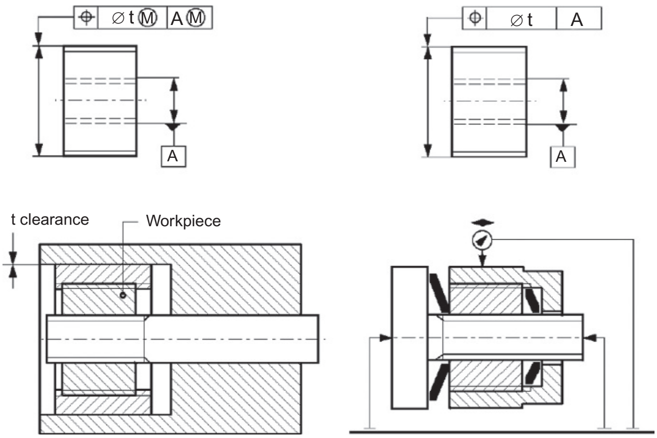


FIG. 13.94 Assessment of the coaxiality deviation of a threaded ring

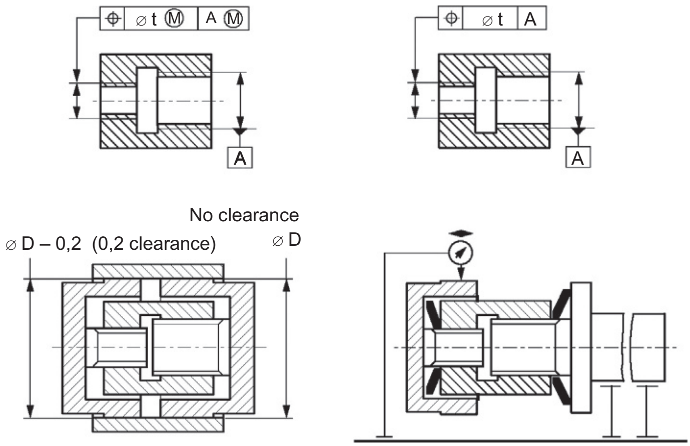


FIG. 13.95 Assessment of the coaxiality deviation of a threaded ring

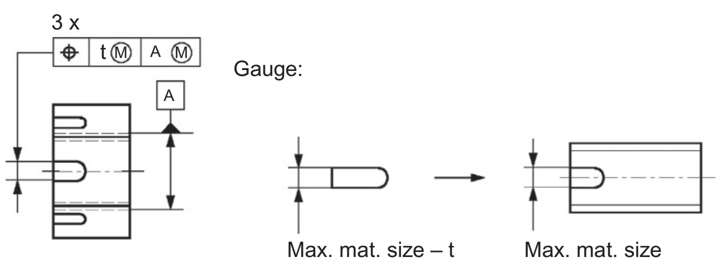


FIG. 13.96 Gauge for the inspection of the symmetry deviation of the slot relative to the thread of a nut

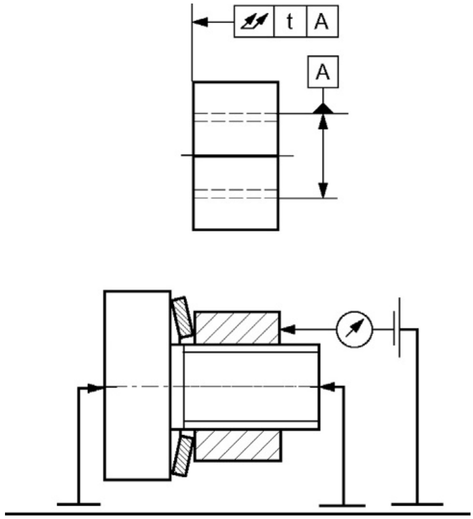


FIG. 13.97 Assessment of the axial run-out deviation of a nut

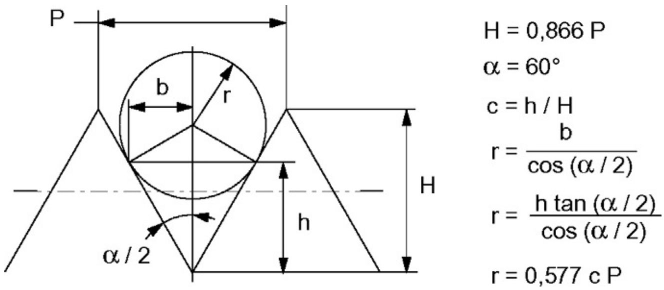


FIG. 13.98 Metrical thread, configuration, probing ball radius r for probing near the flanks (using a self-centring probing mode)

TABLE 13.8 Recommended probing ball diameters D_{rec} in millimetres for probing at the pitch diameters of metric threads according to ISO 261 (P pitch in mm)

P	D0.5H	D0.375H	D0.75H	Drec	ISO metric screw threads
0,5	0,29	0,22	0,43	0,3	M3
0,6	0,35	0,26	0,52	0,3	M3.5
0,7	0,40	0,30	0,61	0,5	M4
0,75	0,43	0,32	0,65	0,5	M4.5
0,8	0,46	0,35	0,69	0,5	M5
1	0,58	0,43	0,87	0,5	M6 M7
1,25	0,72	0,54	1,08	0,8	M8 M9
1,5	0,87	0,65	1,30	0,8	M10 M11
1,75	1,01	0,76	1,51	1	M12

TABLE 13.8 Recommended probing ball diameters D_{rec} in millimetres for probing at the pitch diameters of metric threads according to ISO 261 (P pitch in mm)—cont'd

P	D0.5H	D0.375H	D0.75H	Drec	ISO metric screw threads
2	1,15	0,87	1,73	1	M14M16
2,5	1,44	1,08	2,16	1,5	M18M20M22
3	1,73	1,30	2,60	1,5	M24M27
3,5	2,02	1,52	3,03	2	M30M33
4	2,31	1,73	3,46	2	M36M39
4,5	2,60	1,95	3,89	3	M42M45
5	2,89	2,17	4,33	3	M48M52
5,5	3,18	2,38	4,76	3	M56M60
6	3,46	2,60	5,19	3	M64 to M120
8	4,62	3,46	6,92	5	M125 to M180

For the inspection of threaded features to which the maximum material requirement is applied, gauges with threads (almost) at the maximum material size are to be applied (Fig. 13.99), or, when coordinate measuring systems are applied and appropriate software is available, the gauges may be simulated.

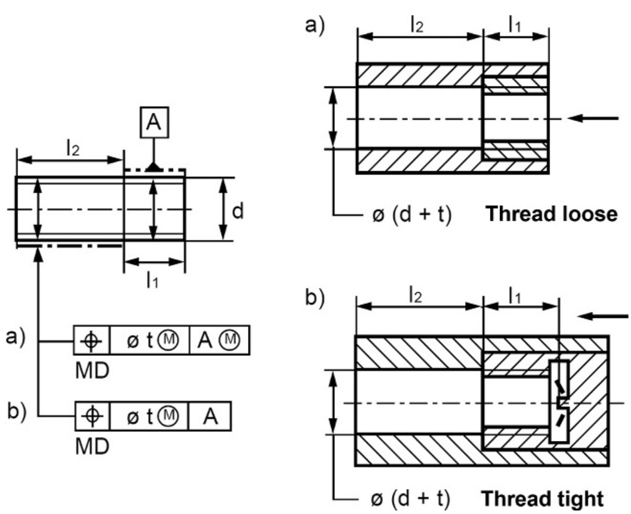


FIG. 13.99 Threaded part with maximum material requirement applied to achieve suit-ability for a stud function, drawing indications and gauges

13.9 Tracing and probing strategies

13.9.1 General

The workpiece features can be inspected by

– gauging, e.g. with a functional gauge (covering the entire feature);
– areal contacting, e.g. with a measuring plate or mandrel;
– tracing, scanning (continuously, consecutively), e.g. with a dial gauge or form measuring instrument;
– probing (assessment of a number of points) (discontinuously), e.g. with a coordinate measuring system.

With tracing, scanning and probing, the surface is assessed by sampling at selected sections or points. Therefore these methods are approximate. The more sections or points assessed, the more realistically are the geometrical deviations assessed and the smaller is the measurement uncertainty.

Figure 13.100 shows form deviations whose amplitude cannot be fully assessed because the spacing of probing (assessed points) is too wide.

In order to assess a certain bandwidth of deviations, the Nyquist theorem should be respected in all directions of the surface, i.e. the spacing of the assessed points including the distances of sections to be traced or scanned should be not more than one-half (better not more than one-seventh) of the deviation wavelength to be assessed.

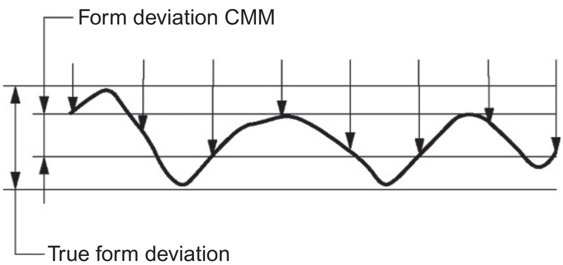


FIG. 13.100 Incomplete assessment of form deviations because of too wide spacing of assessed points

Figure 13.101 shows the effect of the spacing c of the assessed points on a surface exhibiting sinusoidal form deviations of the wavelength λ. When the spacing c is greater than half the wavelength λ, the wavelength of the sine wave cannot be assessed (Fig. 13.101a)). When the spacing is 0,4 λ, the wavelength of the sine wave will be assessed, but some amplitudes will be considerably reduced (Fig. 13.101b)). When the spacing c is 0,2 λ, the true shape of the sine wave (wavelength and amplitudes) is almost obtained (Fig. 13.101c)).

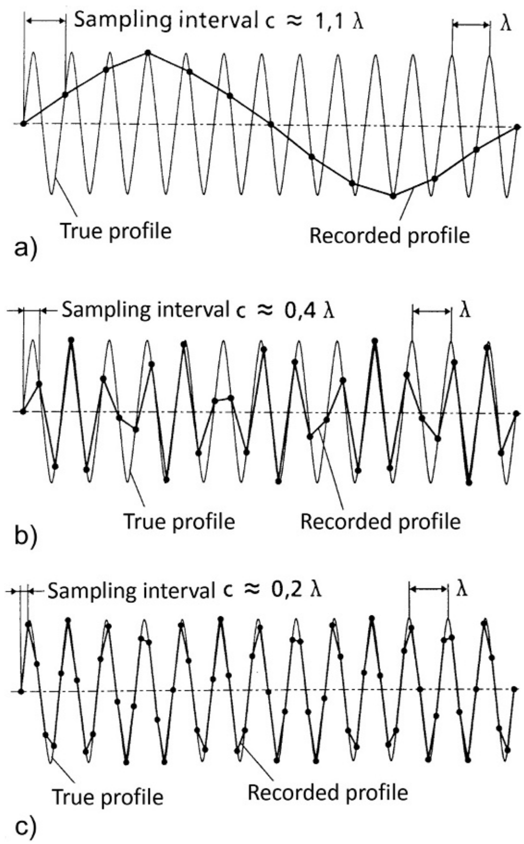


FIG. 13.101 Assessment of sinusoidal form deviations depending on the spacing c of assessed points relative to the wavelength λ of the sine wave

With present techniques, this would lead to very time-consuming and costly inspections. Therefore, for economic reasons, the number of selected sections and the number of probed points are normally reduced, but should be distributed in an optimized way. The necessary numbers of points and sections to be assessed and their optimized distribution depend on:

- the type (shape) of the form deviation (depending on the type of manufacturing process);
- the magnitude of the form deviation;
- the ratio form deviation to geometrical tolerance;
- the geometrical characteristic to be assessed.

Normally for the assessments of, for example, a datum axis or circular run-out deviations, fewer sections are necessary than for the assessments of cylindricity or total run-out deviations. International Standards on this subject do not yet

exist. However, the British Standard BS 7172 and former East German Standards TGL 39 093 to TGL 39 098 and TGL 43 041 to 43 045 give some recommendations, which are included in the following summary. They may serve as a guide in cases where there is no specific information available on the features to be measured that would lead to different strategies (e.g. information on the manufacturing process and on the type and magnitude of the form deviations).

13.9.2 Tracing strategies

Flatness: Fig. 13.102 shows recommended traces for the assessment of flatness deviations.

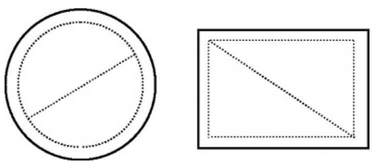


FIG. 13.102 Traces for the assessment of flatness deviations

Roundness: Table 13.9 gives the recommended number and location of radial sections (measuring sections) for the assessment of roundness deviations, according to TGL 39 096.

TABLE 13.9 Recommended number and location of measuring sections for the assessment of roundness

Length 1 of surface mm	≤50			> 50 ≤ 250			> 250		
Ratio l/d of length / diameter	≤ 1	>1 ≤ 3	> 3	≤ 1	>1 ≤3	>3	≤1	>1 ≤ 3	>3
Number nc of sections	1	2	3	2	3	4	3	4	5

Cylindricity: Table 13.10 gives the recommended number of sections and number of traces for the assessment of cylindricity deviations, according to TGL 39 097.

TABLE 13.10 Minimum number of sections and traces for the assessment of cylindricity deviations

Measuring strategy	Minimum number of sections	Lines
Radial section method	3 Radial sections	3
Generatrix method	3 Axial sections	6
Helical method	2 Radial sections +1 helical line of 2 pitches	3
Extreme positions method	1 Axial section +2 radial sections	4

Datum axis: Table 13.11 gives the recommended number and location of radial sections (measuring sections) for the assessment of datum axes, according to TGL 43 043.

TABLE 13.11 Recommended number and location of the measuring sections for the assessment of datum axes

Length L_B of datum cylinder mm	≤ 50		> 50 ≤ 250			> 250		
Ratio L_B / d_B of datum cylinder	≤ 3	>3	≤ 1	> 1 ≤3	> 3	≤1 ≤ 3	>1	>3
Number of radial sections nc	2	3	2	3	4	3	4	5
Face distance of radial sections mm	$L_B/8$	$L_B/12$	$L_B/8$	$L_B/12$	$L_B/16$	$L_B/12$	$L_B/16$	$L_B/20$
Distance between radial sections mm	$3L_B/4$	$5L_B/12$	$3L_B/4$	$5L_B/12$	$7L_B/24$	$5L_B/12$	$7L_B/24$	$9L_B/40$

13.9.3 Probing strategies

The points should be distributed over the entire surface, but not so that they could follow systematic periodic form deviations. For example, six points equally spaced on the circumference of a cylinder cannot detect three lobed form deviations (Fig. 13.103). However, seven points equally spaced on the circumference of a cylinder can assess these form deviations by 79% of their amplitudes.

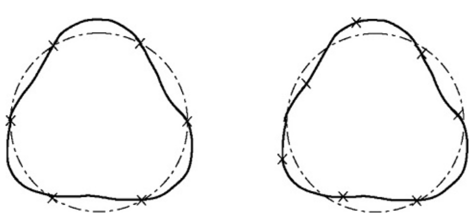


FIG. 13.103 Distribution of points with lobed form deviations

Straight line: The length is to be divided into 3n − 2 subintervals, with points placed in the 1st, 4th, 7th, …, (3n − 2) subintervals, at random positions (Fig. 13.104), where n is the number of points.

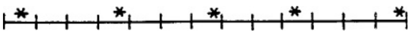


FIG. 13.104 Distribution of n = 5 points on a straight line

Plane: The area is to be divided into $N_1 \times N_2$ rectangles (squares, if possible). Within each rectangle, one point is placed at a random position (Fig. 13.105).

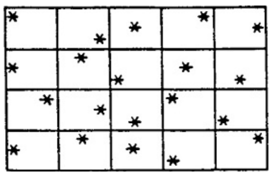


FIG. 13.105 Distribution of n = 20 points on a plane surface

When only a small number of points are to be measured, these points may be placed in alternate rectangles in a "chessboard" fashion (Fig. 13.106).

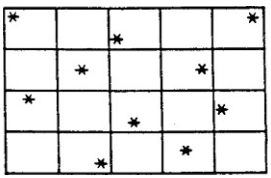


FIG. 13.106 "Chessboard" fashion distribution of n = 10 points on a plane surface

Circle: n equally spaced points are used, where n is, if possible, a prime number and greater than the expected number of lobes (if a lobed form is to be expected) (Fig. 13.103).

Sphere: For a sphere sector between two parallel planes, n points are distributed over n_c sections parallel to the end faces and equally spaced with np points each (Fig. 13.107), where

h	height of sphere section in millimetres,
r	sphere radius in millimetres,
nc	number of sections parallel to the end faces (including the latter),
np	number of points in each section,
n	total number of points,

$$n_c \approx \sqrt{[n_h/(2\pi r)]}$$

$$n_p \approx n/n_c$$

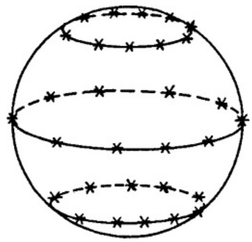


FIG. 13.107 Distribution of $n = 30$ points on a spherical surface of $r = 100\,mm$ and $h = 150\,mm$, so that $n_c = 3$ and $n_p = 10$

For a complete sphere, single points at each pole are also to be measured.

Cylinder:	a) Dividing similarly to the plane b) Dividing similarly to the sphere

It is recommended to alternate between odd and even numbers np of points on the circles, in order to detect lobings (Fig. 13.108).

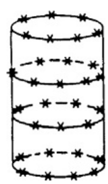


FIG. 13.108 Distribution of 30 points on a cylindrical surface of $r = 10\,mm$ and $h = 30\,mm$ ($n_c = 4$, $n_p = 7$ or 8)

Cone: n points are used, distributed over n_c sections perpendicular to the axis and equally spaced, where.

h	height of truncated cone in millimetres,
r_1	radius at the smaller end in millimetres,
r_2	radius at the larger end in millimetres,
k	length of circumference in millimetres,
n_c	number of sections (end faces included),
n_p	number of points of a section (this should decrease towards the vertex of the cone by the number s; Fig. 13.109).

$$k = \sqrt{\left[h^2 + (r_2 - r_1)^2\right]}$$

$$n_c \approx \sqrt{[kN/\pi\,(r_1 + r_2)]}$$

$$s \approx 2\pi\,(r_2 - r_1)/k$$

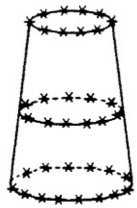


FIG. 13.109 Distribution of $n=35$ points over the surface of a truncated cone of $r_1=10\,mm$, $r_2=15\,mm$, $h=20\,mm$ ($k=20.6\,mm$), so that $n_c=3$, $s=2$, $n_p=10$, 12 and 14

13.9.4 Number of points

In the absence of form deviations, a minimum number of points would be sufficient to determine the geometrical feature in a coordinate system (Table 13.12). Because there are always form deviations, more points have to be assessed. Table 13.12 gives the minimum numbers recommended by BS 7172. Fewer points should not be chosen. More points decrease the measurement uncertainty caused by the form deviations.

TABLE 13.12 Recommended number of points

Feature	Number of probed points mathematically recommended		Remarks
Straight line	2	5	
Plane	3	9	Distributed on 3 lines
Circle	3	7	For assessment of 3-lobed forms
Sphere	4	9	Distributed on 3 parallel sections
Cylinder	5	12	For assessment of straightness distributed on 4 radial sections
		15	For assessment of roundness distributed on 3 radial sections
Cone	6	12	For assessment of straightness distributed on 4 radial sections
		15	For assessment of roundness distributed on 3 radial sections

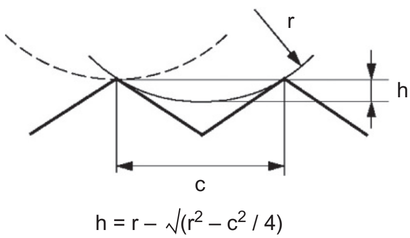
In order to assess periodic deviations, the next greater prime number should be chosen.

For curved features with changing curvature (e.g. turbine blades) the spacing of the points should be closer for regions of small radii of curvature than for regions of larger radii of curvature.

13.10 Separation of roughness and waviness

The separation of roughness and waviness from geometrical deviations is not yet completely internationally standardized. Normally the peaks of the surface roughness contribute fully to the geometrical deviation. How much the surface roughness valleys contribute to the geometrical deviation depends on the measuring method; see Fig. 2.8.

In practice, the roughness is filtered out by the effect of the stylus tip (ball) of the measuring instrument (with dial gauges, the ball radius is normally 1.5 mm). The effect on the assessment of the geometrical deviation depending on the ball radius r and on the spacing c of the deviations (irregularities) of the workpiece surface is shown in Fig. 13.110. With surfaces manufactured by metal removal, normally the waviness depth W_t is smaller than 5 μm and the waviness spacing wider than 0.3 mm. In these cases, the waviness is part of the geometrical deviations.



r mm	h in μm with c in mm			
	c = 0,01	c = 0,1	c = 0,3	c = 1
0,25	0,05	5	50	
0,5	0,025	2,5	23	
1	0,0125	1,25	11	134
1,5	0,0083	0,83	7,5	86
2,5	0,0050	0,50	4,5	51
5	0,0025	0,25	2,3	25
10	0,0013	0,13	1,1	13

FIG. 13.110 Effect of the stylus ball radius r and of the workpiece deviation spacing c on the assessment of geometrical deviations

However, for some functions, e.g. the ball race surface on rolling bearings, the waviness must be toleranced and measured separately by the use of appropriate filters; see Fig. 13.115.

Form measuring instruments usually allow the use of a low pass filter to separate (filter out) roughness (and parts of waviness) in the assessment of geometrical deviations. Fig. 13.111 shows the composition of the instruments. Fig. 13.112 shows the filter characteristics of these instruments.

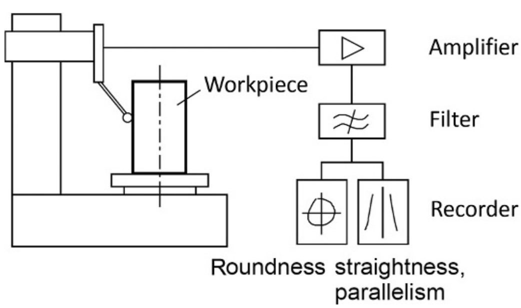


FIG. 13.111 Form measuring instrument

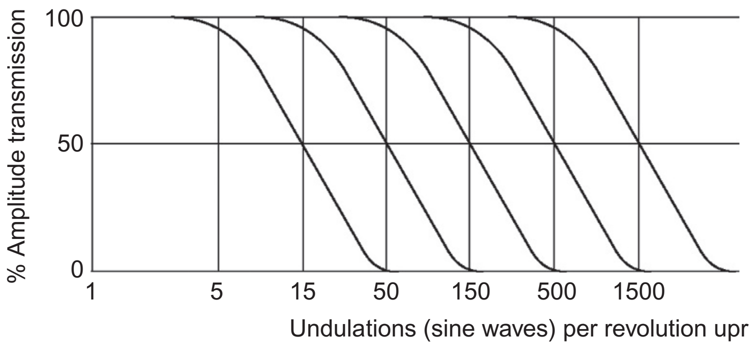


FIG. 13.112 Filter characteristics

The roundness profile of a workpiece can be considered as a superposition of sine waves. Fig. 13.113 shows such a Fourier analysis of a roundness profile.

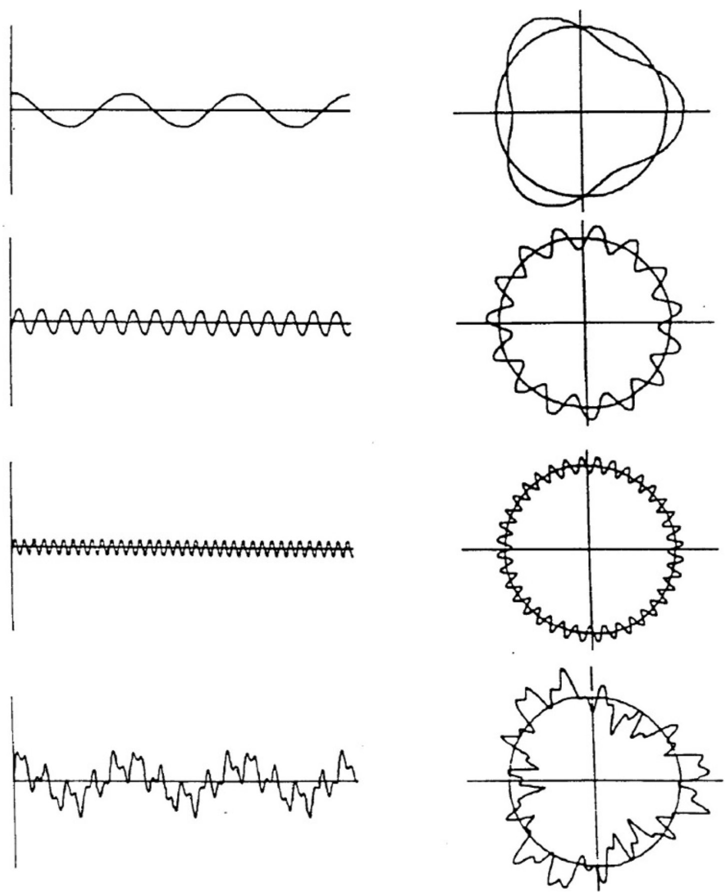


FIG. 13.113 Analysis of a roundness profile according to Wirtz (see Ref. [4])

Each sine wave portion will be transmitted by the filter, according to the filter characteristic, with full or attenuated amplitudes. Waves of short wavelengths (several waves per revolution) (e.g. roughness) will not be transmitted by the filters (or will only be transmitted with strong attenuation).

The filters are named according to the limiting number of waves (cut-off) ng or according to the limiting wavelength (cut-off) λ_R. These are the number of sine waves (undulations) per revolution (circumference) UPR or the sine wave length where the amplitudes are transmitted by 75% (old RC filter according to ISO 3274:1975) or by 50% (new phase correct Gaussian filter according to ISO 16610-21:2011). The measuring results may be significantly different depending on whether the RC filter or the Gaussian filter has been used.

Figure 13.114 shows an example of the effect of the different filters on the same workpiece profile to be measured. It shows that the result of the measurement is also significantly different depending on the filter size (cut-off) used.

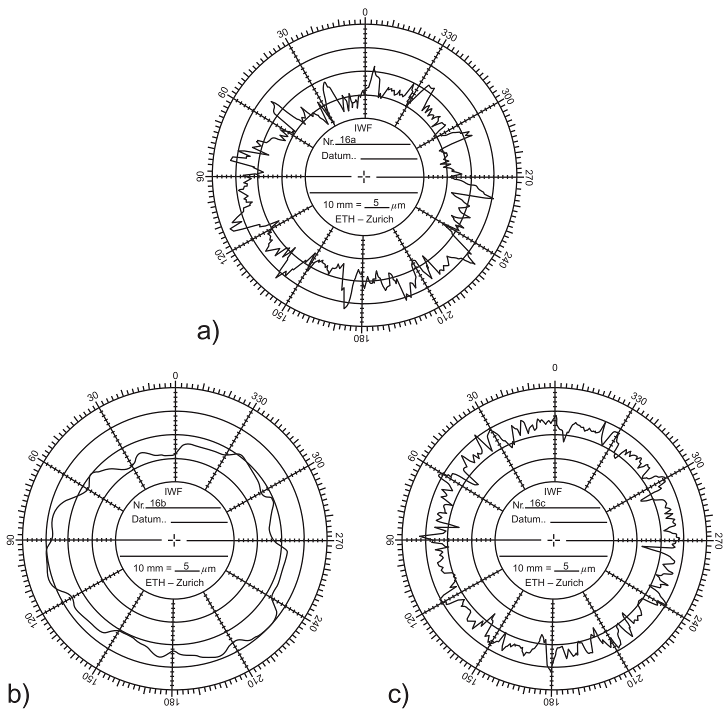


FIG. 13.114 Effect of different filters: a) without filter; b) with filter 50; c) with a filter combination (bandpass) (according to Wirtz [4])

Using narrow bandpass filters (combination of low-pass and high-pass filters), deviations of certain wavelengths can be selected. Thereby eventually the reasons for certain form deviations and the reasons for the functional behaviour become evident. Fig. 13.115 shows such an analysis. The three-lobed form indicates the effect of the three-point chuck).

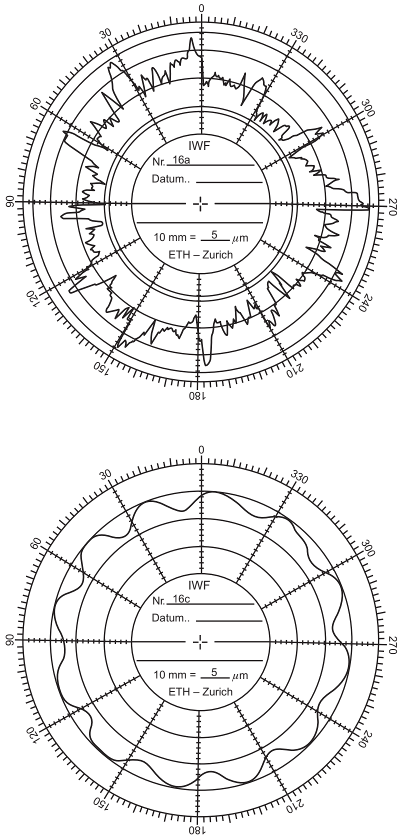


FIG. 13.115 Measurement of the same profile with different bandpasses (according to Wirtz [4])

In cases of dispute as to whether a workpiece does or does not comply with the geometrical tolerance, the rate of included waviness may be decisive. In these cases it is recommended to agree on the following:

– when measuring instruments with filter devices are available, to apply a stylus tip ball radius of 0,5 mm and a Gauss filter of 0,8 mm for straight features and according to Table 13.13 for round features;

(Then the height of narrow peaks may be considerably attenuated by the Gauss filter. In order to limit the height of these peaks on the workpiece, an additional specification of waviness may be necessary. Application of stylus tips without further filtering (see the following) should assess the full peak height and may be preferable.)

– when only measuring instruments without filter devices are available or shall be applied, a stylus tip ball radius of 1,5 mm. (For small holes and slots, a smaller stylus tip ball radius must be agreed upon.)

TABLE 13.13 Filter (ISO 11 562, ISO 16 610-61) in UPR for the measurement of roundness and cylindricity according to ISO 12 181 and ISO 12 180

Workpiece diameter (mm)	Filter UPR
0–8	15
Above 8–25	50
Above 25–80	150
Above 80–250	500
Above 250	1500

However, there is no filter standardized as an ISO default. If, for example in international trade, filters shall be applied, this should be specified in the drawing or in related documents.

The filters according to Table 13.13 have been calculated so that for a median diameter a filter has a similar effect as an 0,8-mm filter for straight features. The 0,8-mm filter has been chosen because it is the filter that is most frequently used for roughness measurements according to ISO 4288 and for waviness measurements. The tip radius of 0,5 mm is recommended in order to leave its filter effect beyond the 0,8-mm filter effects, i.e. what is filtered out by the tip radius will be filtered out by the 0,8-mm filter anyway, so the tip radius has no filtering effect on the measurement result.

ASME B89.3.1 states that, if not otherwise specified, for round features a 0,25-mm tip radius and the 50 UPR filter applies (see 14.17).

When the stylus tip (ball) penetrates into the grooves of the roughness (e.g. with turned surfaces), it may occur that the groove is inclined to the section plane of probing so that the stylus tip goes from the peak to the valley within, for example, half of a revolution. Thereby an eccentricity is simulated that does not in fact exist (Fig. 13.116). This eccentricity remains even when there is a filter involved that separates the roughness from the geometrical deviation. This error can be avoided by the use of a suitable stylus tip form (e.g. a hatched form with radius r = 10 mm; Figs 13.116 and 13.117).

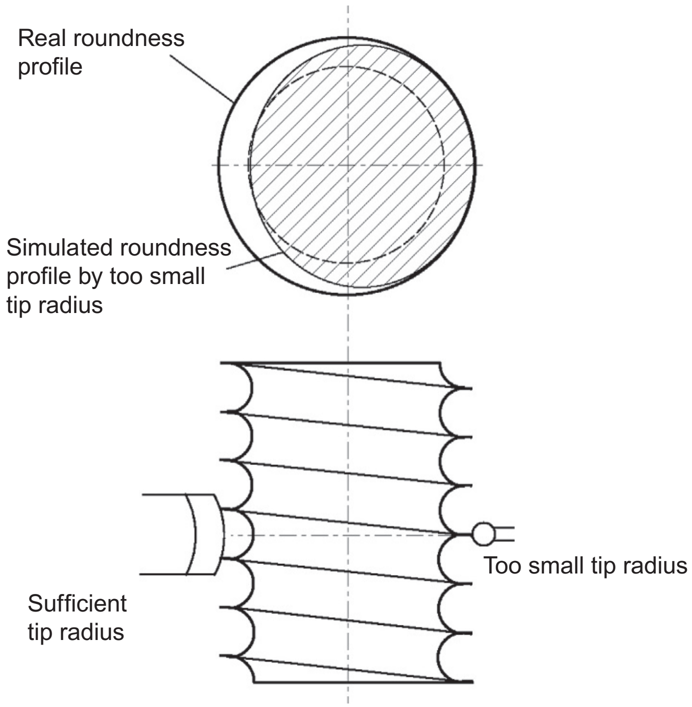


FIG. 13.116 Simulated eccentricity of a turned surface

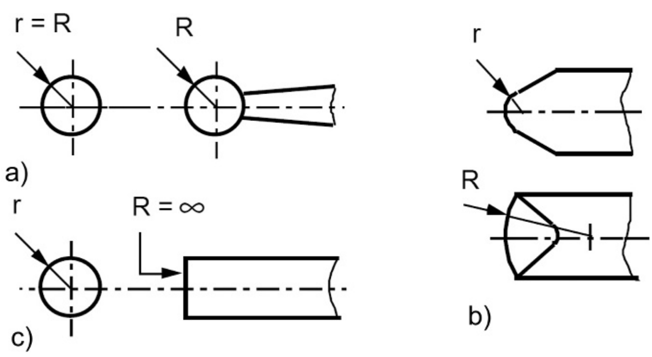


FIG. 13.117 Stylus tip forms: a) ball; b) hatched (toroidal); c) cylindrical form

13.11 Measurement uncertainty

13.11.1 Definition

According to the International Vocabulary of Basic and General Terms in Metrology (VIM), the measurement uncertainty is an estimate characterizing the range of values within which the true value of a measurand lies. The measurement uncertainty comprises many components: the random deviations

(errors) and the unknown (and therefore uncorrected) systematic deviations (errors) of all quantities that contribute to the measuring result.

According to ISO 14 253-1 for geometrical deviations (Geometrical Product Specifications (GPS)), for the measurement uncertainty a statistical confidence level of 95% applies, if not otherwise specified.

For measuring instruments, the maximum permissible error **MPE** is standardized, that is, the maximum uncertainty contributor from the measuring instrument. It is valid for specified conditions (e.g. temperature range).

13.11.2 Application

ISO 9001:1994 clause 4.11.1 specifies:

> Inspection, measuring and test equipment shall be used in a manner which ensures that the measurement uncertainty is known and is consistent with the required measuring capability.

The standard ISO 14 253-1 defines uncertainty zones at the specification (tolerance) limits. The width of the uncertainty zone is ± the expanded measurement uncertainty U; see Fig. 13.118. For tolerances of form, orientation,

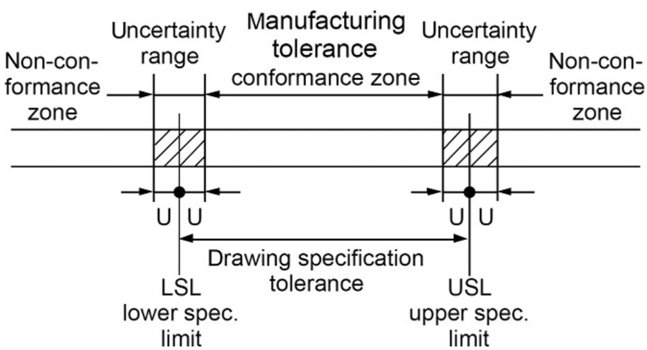


FIG. 13.118 Conformance zones, nonconformance zones and uncertainty zones (ranges) with specification zone limits, both ≠ 0

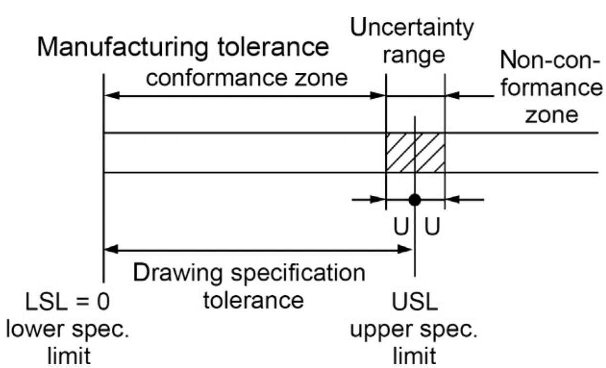


FIG. 13.119 Conformance zone, nonconformance zone and uncertainty zone (range) with one specification limit = 0

coaxiality/symmetry and run-out, with one tolerance limit of zero, there is only one uncertainty zone at the other tolerance limit; see Fig. 13.119.

The standard specifies the following decision rules for the case when the result of measurement falls within one of the uncertainty zones:

– Repeat the measurement using a measurement process with a smaller measurement uncertainty or repeat the estimate of the measurement uncertainty more precisely; if the measurement result still falls within an uncertainty zone, the following rule applies:
– the supplier cannot prove conformance with the specification (tolerance) in the case of an outgoing inspection; the customer cannot prove nonconformance with the specification (tolerance) in the case of an incoming inspection. Resellers should use the proof provided to them by their supplier in order to avoid a situation in which they cannot reject the delivery but they also cannot submit the delivery.

13.11.3 Assessment

The GUM Guide to the expression of uncertainty in measurement (European Standard EN 13 005) applies. The GUM has been elaborated by international organizations and has been adopted by most of the national laboratories for measurement and standards (e.g. NIST in the United States, NPL in the United Kingdom, PTB in Germany).

GUM gives the rules on how to assess the measurement uncertainty in a most correct way. It distinguishes between:

Evaluation type A: using statistical means (repeated measurements).
Evaluation type B: using uncertainty components (investigations).

Bases for the evaluation type B are:

– data from former measurements,
– experimental results, investigations,
– indications of the measuring device manufacturer,
– dates of calibration.

There is no longer a distinction between random deviations and uncorrected systematical deviations.

GUM is (because of its generality) very theoretical, very voluminous and therefore not easy to read and difficult to understand and to implement into industrial practice.

Therefore, for geometrical product specifications GPS (specifications for dimensions, geometry, roughness and waviness), ISO 14 253-2 has been developed. This standard contains a simplified procedure for the assessment of the measurement uncertainty and a simplified procedure for optimizing the cost of manufacturing and inspection (**PUMA: p**rocedure for **u**ncertainty

management). For manufacturing, the drawing specification is to be diminished by the measurement uncertainty U or 2U; see Figs 13.118 and 13.119.

The simplifications are conservative so they result in larger measurement uncertainty values (safe method).

The simplifications are:

- the sensitivity coefficients (see GUM) are equal to 1; the correlation coefficients ρ are either $\rho = 1, -1, 0$; if the uncertainty components are not known to be uncorrelated, full correlation is assumed either $\rho = 1$ or $\rho = -1$
- restriction to three distribution types (normal, rectangular, U distribution)
- conservative assumption of the distribution type; if not known to be normal, use rectangular or U distribution
- start the iteration procedure with evaluation type B.

The procedure of uncertainty budgeting is based on the error propagation law.

For the **black box method** (direct measurement), the combined standard uncertainty (standard deviation) u_c is

$$u_c = \sqrt{\left(\sum u_{xr}\right)^2 + \sum u_{xp}{}^2}$$

where

u_{xr} = standard deviation of correlated uncertainty contributor
u_{xp} = standard deviation of uncorrelated uncertainty contributor.

For the **transparent box method** (indirect measurement), when the geometrical deviation is assessed by indirect measurements, i.e. the deviation δ to be measured is calculated from the measured quantities $(x_1, x_2, \ldots x_i, \ldots x_{p+r})$ according to the function.

$$\delta = G\left(x_1, x_2, \ldots x_i, \ldots x_{p+r}\right)$$

the combined standard uncertainty of measurement u_c is

$$uc = \sqrt{ur^2 + \sum_{i=1}^{p}\left(\frac{\partial\delta}{\partial x_i}\,uxi\right)^2} \qquad ur = \sum_{i=1}^{r}\frac{\partial\delta}{\partial x_i}\,uxi$$

where

u_r = the contribution (“sum”) of the correlated components of the measurement uncertainty;
u_{xi} = the combined uncertainty of measurement of the number i measured value (function) that is part of the transparent box method of uncertainty estimation for the measurement of δ; u_{xi} can be the result (u_c) of either a black box or another transparent box method of uncertainty estimation;

r = the number of correlated components of the measurement uncertainty;
p = the number of uncorrelated components of the measurement uncertainty.

The uncertainty contributors u_x are due to the influences on the measuring process and are to be assessed as standard deviations in length. When only the limit values are known (range a), the standard deviation of the uncertainty contributor is

$$u_x = b\,a$$

where b the transformation coefficient given in Fig. 13.120.

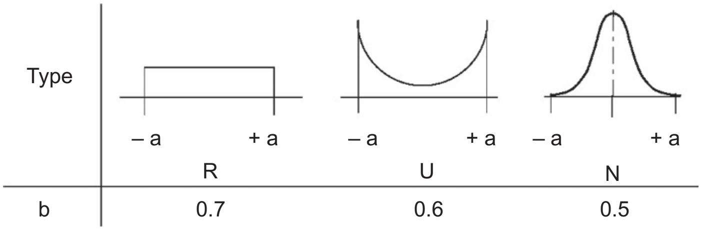


FIG. 13.120 Transformation coefficient b

The rectangular distribution is to be used when it is not certain that the normal distribution applies, and the U distribution is unlikely. The U distribution occurs with phenomena varying similar to a sine wave, e.g. many time-dependent phenomena.

The expanded measurement uncertainty U is

$$U = k\,u_c$$

If not otherwise specified, according to ISO 14 253-1 the coverage factor k = 2 applies corresponding to a confidence level of 95%.

ISO 14 253-2 contains a list of possible measurement uncertainty contributors (u_x) to the combined measurement uncertainty uc, e.g.:

– deviation of the calibration standard;
– deviation of the measuring equipment (e.g. maximum permissible error (MPE) of the measuring instrument);
– deviation caused by support and alignment of the workpiece (set up);
– deviation caused by incomplete tracing of the workpiece (form deviation);
– deviation caused by temperature influences (deviations from 20°C);
– deviation caused by measuring force (deformations);
– deviation caused by gravity influences on the workpiece (deformations);
– deviation caused by the metrologist.

When the influences are known in units other than length units, e.g. temperature units, they must be transformed into length units.

The PUMA method is an iterative procedure in which step by step, due to improved knowledge or due to improved measuring processes, the assessed measurement uncertainty is reduced and the manufacturing tolerance is increased.

In the first iteration, the evaluation type B is used and thereby time- and cost-consuming experiments are avoided.

Due to the squares of the measurement uncertainty components u_x, components with small values of ux contribute very little to the combined measurement uncertainty u_c. Therefore, if the measurement uncertainty shall be reduced, such contributors should be dealt with that have large values. ISO 14 253-2 describes a scheme for procedure and documentation of the assessment of the measurement uncertainty.

In the field of geometrical deviations, the assessment of the measurement uncertainty is often very difficult. It is difficult to identify which contributors are effective and to what magnitude. Therefore at present in the inspection of workpieces the measurement uncertainty is more or less roughly estimated according to experience. Means for a more precise assessment of measurement uncertainty are in preparation, as described in the following sections.

a) Uncertainty budgeting according to ISO 14 253-2

It is intended to develop a collection of examples of the assessment of measurement uncertainty for various methods of measurement of geometrical deviations, e.g. methods according to ISO TR 5460. The examples may be used as guidelines. (ISO 14 253-2 shows in Annex A one example of the estimation of the measurement uncertainty of a roundness measurement.)

Plans are underway to develop similar PC software that asks for the values of the contributors (e.g. temperatures, maximum permissible errors (MPEs) of the measuring instruments, deviations of auxiliary equipment) and then calculates the measurement uncertainty.

b) Virtual coordinate measuring system ISO 15 530-4

For coordinate measuring systems (CMMs), a computer program has been developed that estimates the measurement uncertainty by simulation of repeated measurements. For this program, the uncorrected deviations of the CMM are assessed with the aid of a calibrated sphere and a calibrated sphere plate. The program simulates measurements at the points originally measured, taking into account the errors of the CMM. From these simulated measurements, the program calculates the measurement uncertainty. After a single measurement, the instrument gives the measurement result together with the measurement uncertainty.

c) Comparison (substitution) method ISO 15 530-3

With this method, the workpiece and a similar calibrated object (e.g. a calibrated workpiece) are measured alternately. The measuring instrument serves as a comparator. The essential contributors to the measurement uncertainty are:

– due to the measuring procedure,
– due to the calibration,
– due to the workpiece (inhomogeneities of form deviations, roughness and material properties).

This procedure leads to relatively small measurement uncertainties, but the practical use is very limited because of the effort needed for calibration and logistics of the calibrated objects.

Another method is the task-related calibration of the measuring process. The calibrated object is measured repeatedly and the measurement uncertainty of the measuring process is assessed. This value is a measurement uncertainty contributor for the measurement of similar workpieces. This method also has limited practical use, because of the effort needed for calibration and logistics of the calibrated objects.

d) Measurement of long dimensions and of other geometrical deviations

With the measurement of long dimensions, e.g. diameters, widths or position toleranced dimensions, the influence of the temperature is often the largest, or even dominant, on the measurement uncertainty. Neumann [2] has found, for measurements of workpieces of steel with CMMs the measurement uncertainty contributor due to measurements deviating ±2 K from 20°C:

when temperature has not been corrected: $\pm\, U_{95} \approx \pm 40\,\mu m/m\,K$,
when temperature has been corrected: $\pm U_{95} \approx \pm(5\ldots20)\,\mu m/m\,K$.[7)]

When measuring deviations of form, orientation, coaxiality, symmetry and run-out, the measured value is close to zero. Therefore the influence of the temperature is much smaller and other influences (larger) are more important (e.g. MPE or calibration uncertainty of the measuring device and the repeatability of the measuring process).

13.11.4 Calibration of measuring instruments

Calibration is (according to GUM) a set of operations that establish under specified conditions the relationship between the value indicated by the measuring instrument and the corresponding value realized by standards.

A (measurement) standard is (according to GUM) a material measure, measuring instrument, reference material or measuring system intended to define, realize, conserve or reproduce a unit or one or more values of a quantity, e.g. a standard for a length.

In order to assess the measurement uncertainty, the measuring instrument must be calibrated and the calibration uncertainty must be taken into account. However, the calibration of metrological characteristics of measuring

7) Depending on the accuracy of the assessment of the temperature

equipment that have no influence on the intended measurement is superfluous and uneconomical. For example, a length measuring instrument used only for measurements of a short range of lengths need not be calibrated over the whole measuring range of the instrument.

13.11.5 Examples for uncertainty budgets

ISO 14 253-2:2011 gives examples for uncertainty budgets:

in Annex A for the calibration of a setting ring
in Annex B for the design of calibration hierarchy
in Annex C for the measurement of roundness.

This page intentionally left blank

Chapter 14

Differences Between ASME Y14.5 and ISO

14.1 Application

If not otherwise specified, e.g. in the drawing title box, for GPS the ISO standards apply worldwide; see ISO 8015. When the rules according to ASME Y14.5 apply, the drawing title box must have the indication ASME Y14.5; see ISO 8015, ASME Y14.5-2009 clause 1.1.3 and ASME Y14.5-2018 clause 1.3.

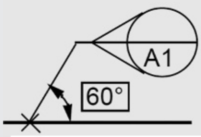
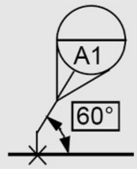
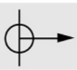
The following paragraphs describe the differences between ASME Y14.5 and ISO.

14.2 Symbols

The National Standard of the United States ASME Y14.5-2018 specifies, in addition to or deviating from ISO 1101, the symbols and drawing indications shown in Table 14.1.

Geometrical Dimensioning and Tolerancing for Design, Manufacturing and Inspection
https://doi.org/10.1016/B978-0-12-824061-8.00001-3

TABLE 14.1 Symbols according to ASME Y14.5

ASME		ISO
▷	Translation between datums (dimension variable)	[DV]
⟨CF⟩	United (continuous) feature	UF
Ⓤ	Unequally distributed tolerance	UZ
	Movable datum target moving direction given by:  Default: normal to true profile 	
- A -	Datum (former practice)	
AVG	Arithmetical mean	(SA)
[BSC]	Fixed datum, TED-position	
Ⓢ	Regardless of feature size	
Ⓘ	Independency principle	Default
⟨ST⟩	Statistical tolerance	
CR	Controlled radius	
	Dimension origin = datum	
△	Dynamic profile tolerance	OZ

14.3 Coplanarity

According to ASME Y14.5, when two or more flat surfaces are drawn in the same plane and profile toleranced without a datum, they are to be regarded as an interrupted or non-continuous surface, i.e. a common tolerance zone applies. According to ISO, the independency principle applies, i.e. the symbol CZ must be indicated after the tolerance value when they shall be regarded as an interrupted or non-continuous surface. See Fig. 14.1.

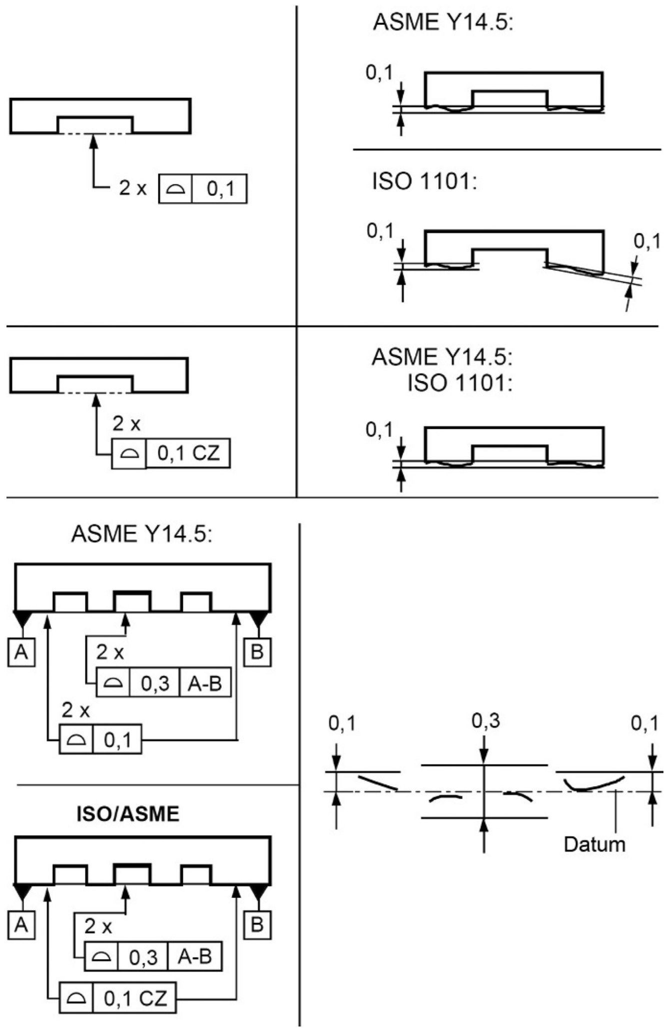


FIG. 14.1 Flat surfaces in a plane

14.4 Envelope requirement, Rule # 1

Rule # 1: for features of linear size (cylinder, sphere, plane-pair), the envelope requirement applies as default (without further indication in the drawing).

Exceptions:

– stock material
– flexible part in the free state
– features of size with form tolerances for axes or median surfaces

When the independency principle shall apply, Ⓘ shall be indicated after the size tolerance.

According to ISO 8015, the independency principle applies as default. According to ISO 14 405-1, two-point sizes apply as default for sizes. When the envelope requirement shall apply, this is to be indicated according to ISO 14 405-1, e.g. by the symbol Ⓔ after the size tolerance.

14.5 Radii

R in front of the dimension (e.g. R 2,4 ± 0,3) means a crescent-shaped tolerance zone tangential to the adjacent surfaces.

CR (controlled radius) in front of the dimension (e.g. CR 2,4 ± 0,3) means, in addition to the tolerance zone, a fair curve without reversals. Radii taken at all points on the part contour shall neither be smaller than the specified minimum limit nor larger than the maximum limit (see Fig. 14.2).

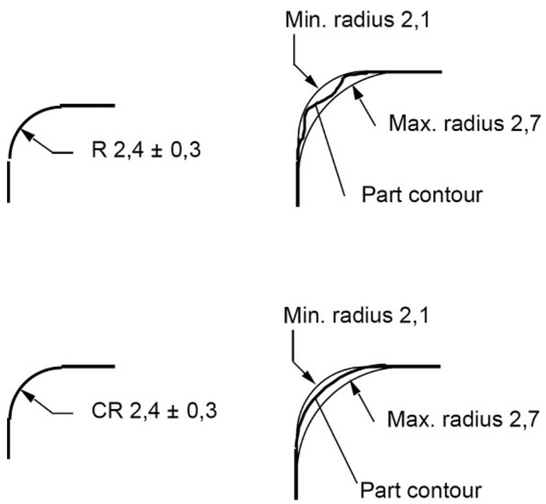


FIG. 14.2 Tolerancing of radii

Currently, there is no precise definition of the radius tolerance in the ISO standards.

14.6 Dimensioning origin

The symbol ⌖→ has a meaning similar to that of a datum; see Fig. 14.3.

Currently, there is no precise definition of the dimensioning origin in the ISO standards.

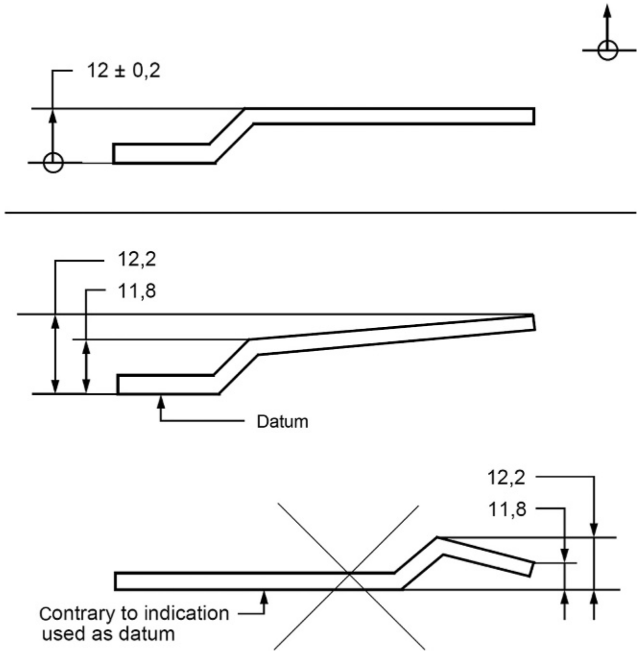


FIG. 14.3 Dimensioning origin; interpretation in the lower image is wrong

14.7 Angular tolerances

The meaning of angular tolerances is explained in Fig. 14.4. The variant in the figure below is omitted in ASME Y14.5-2009 (was in ASME Y14.5-1994).

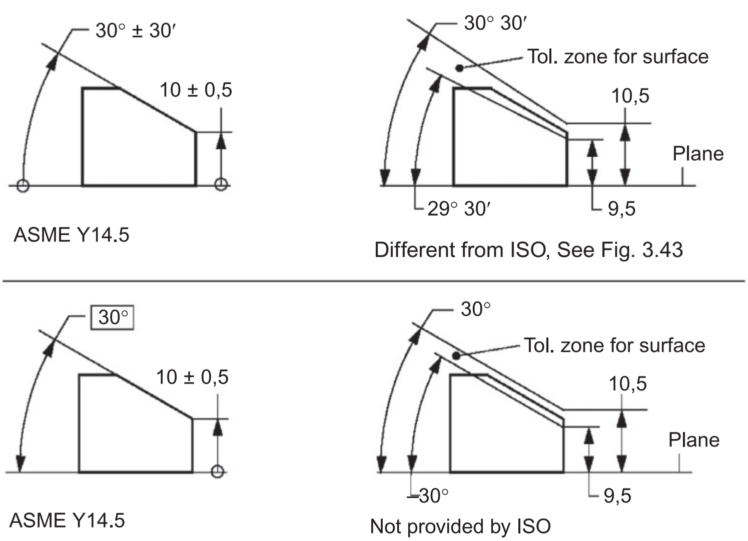


FIG. 14.4 Angular tolerances

According to ISO 14 405-2, the angular size tolerance applies to associated straight lines in intersection planes. Form deviations are not limited.

14.8 Cones

Tolerancing of cones by axial cone tolerances is explained in Fig. 14.5. This figure also shows the relationship to a radial tolerance and to a profile tolerance. According to ISO 3040, profile tolerances are used.

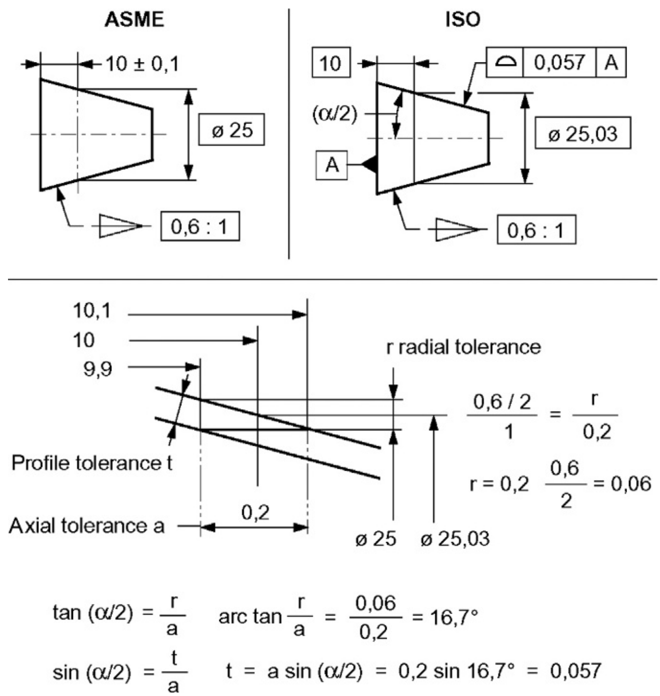


FIG. 14.5 Tolerancing of cones

14.9 Regardless of feature size RFS Ⓢ

According to a former rule (USASI Y14.5-1966 and ANSI Y14.5-1973), position tolerances were applied with the maximum material requirement at the toleranced feature and at the datum. This was the default without any further indication, such as Ⓜ. This is in contrast to the international practice, where Ⓢ (regardless of feature size) applies as the default (without any further indication) (ISO 8015, ISO 2692). This rule has been adopted by ASME Y14.5-2009. In the time of transition, it was always to be indicated whether Ⓢ or Ⓜ applied. With ASME Y14.5-2018, the indication of Ⓢ is no longer necessary.

14.10 Tolerances of position, coaxiality, symmetry

According to ASME Y14.5, position tolerances with the symbol ⌖ and without the symbol Ⓜ or Ⓛ apply to the (straight) axis or centre plane of the actual mating envelope. The mating envelope is the largest inscribed or the smallest circumscribed perfect feature of size (cylinder or plane-pair).

According to ASME Y14.5-2009, coaxiality or symmetry tolerances with the symbols ◎ and ⌯ apply to the deviated (imperfect) actual centre lines or median surfaces; see Fig. 14.6. The symbols ◎ and ⌯ must not be used together with Ⓜ and Ⓛ. According to ASME Y14.5-2018 the symbols shall no longer be used.

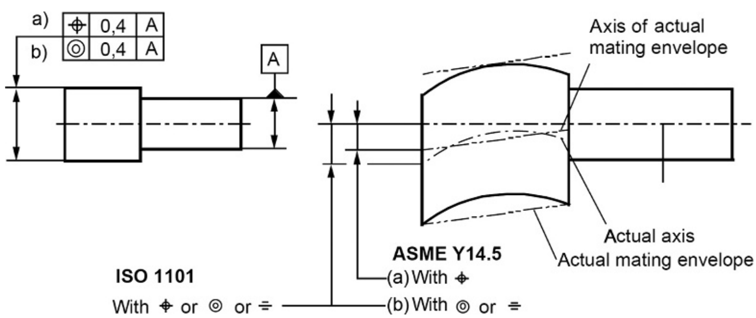


FIG. 14.6 Position tolerance according to ASME Y14.5-2009 and according to ISO 1101

According to ISO 1101, position tolerances as well as coaxiality and symmetry tolerances apply to the deviated (imperfect) median lines or median surfaces and may be used with or without Ⓜ and Ⓛ; see Fig. 14.6.

The difference between position tolerance and coaxiality tolerance (concentricity tolerance) according to ASME Y14.5-2009 is shown in Fig. 14.7.

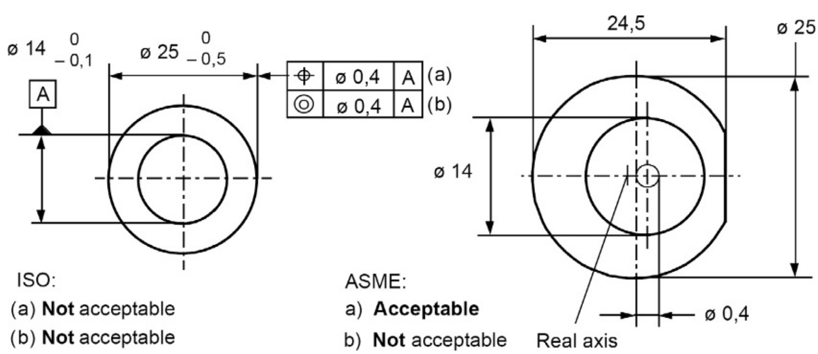


FIG. 14.7 Difference between position tolerance and coaxiality tolerance according to ASME Y14.5-2009 (workpiece within position tolerance but outside concentricity tolerance)

14.11 Orientation and location tolerances for axes or median planes

Orientation and location tolerances according to ASME Y14.5 apply to the (perfect) axes or median planes of the mating envelope (true geometric counterpart, largest inscribed or the smallest circumscribed perfect feature of size (cylinder or plane-pair)). Orientation and location tolerances according to ISO 1101 apply to the (imperfect) actual axes or median surfaces; see Fig. 14.8.

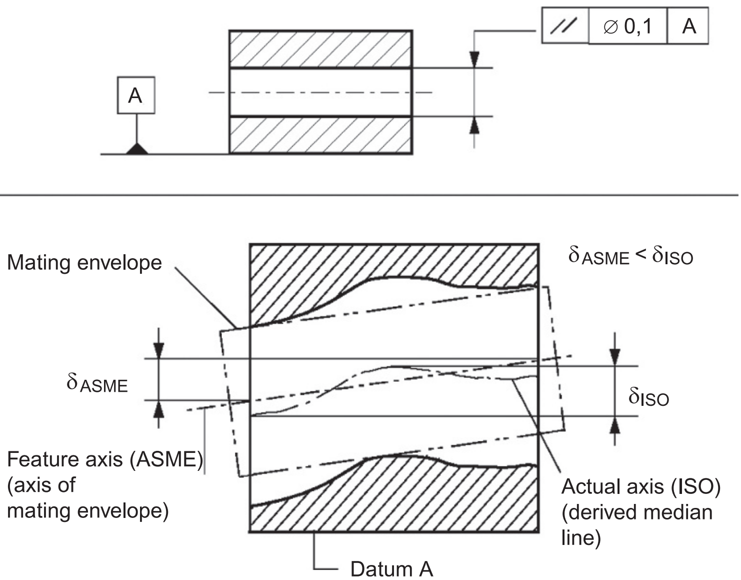


FIG. 14.8 Orientation (parallelism) tolerance according to ASME Y14.5 applied to the (perfect) axis of the mating envelope and according to ISO 1101 applied to the (imperfect) actual axis; similar applies to location tolerances

According to ASME Y14.5-2018 the symbols for coaxiality and for symmetry shall no longer be used.

14.12 Composite tolerancing and single tolerancing

ASME distinguishes between composite tolerancing and single tolerancing. Single tolerancing is identified by the tolerance symbol (position or profile) indicated with each tolerance (separately); see Fig. 14.9 c). Composite tolerancing is identified by one single tolerance symbol (position or profile) for two or three tolerances; see Fig. 14.9 a) and b).

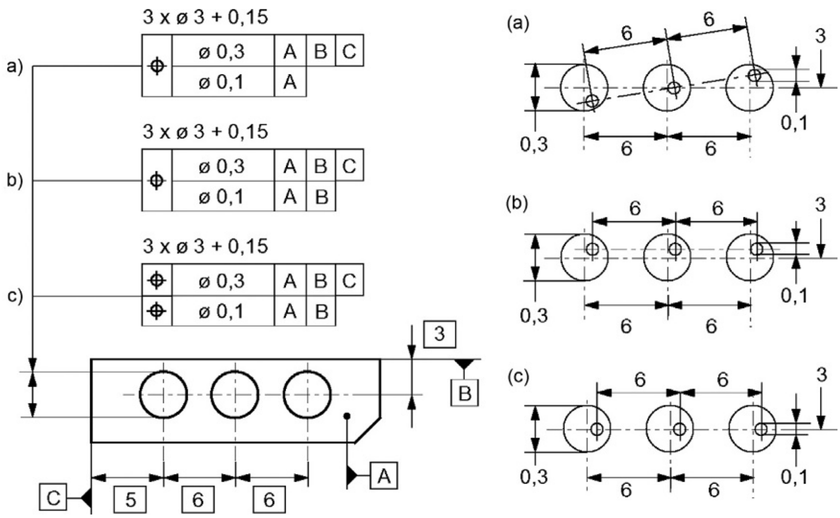


FIG. 14.9 Composite tolerancing a), b) and single tolerancing c)

In the case of single tolerancing, each tolerance applies as if no other tolerance were indicated, i.e. the theoretically exact dimensions from the datums specified in the tolerance indicator apply; see Fig. 14.9 c). For tolerance ⌀0,1 the TED 3 (distance to datum B) applies.

In the case of composite tolerancing, the tolerance symbol is only indicated once and applies for all (two or three) position or profile tolerances; see Fig. 14.9 a) and b). In the upper segment, the largest tolerance is to be indicated and applies to define the location of the group (pattern) relative to the datum system. The theoretically exact **location** of the tolerance zones is to be indicated by TEDs (pattern locating tolerance zone indicator framework (PLTZF)).

In the lower segment(s), the (smaller) tolerances apply regardless of the distances to the datums for the location of the tolerance zones relative to each other but for the **orientation** relative to the datum(s) (feature relating tolerance zone framework (FRTZF)).

In Fig. 14.9 the tolerance zones ⌀0,1 are

a) (composite tolerancing) perpendicular to datum A
b) (composite tolerancing) perpendicular to datum A and parallel to datum B
c) (single tolerancing) perpendicular to datum A, parallel and in theoretically exact distance to datum B

ISO 1101 and ISO 5458 do not provide this differentiation. The distance 3 always applies. When the distance to Datum B shall be disregarded, as according to ASME in the case of Fig. 14.9 b), this is to be indicated by the symbol >< after the datum letter in the tolerance indicator; see Fig. 14.10.

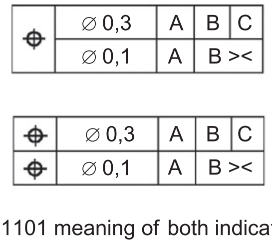


FIG. 14.10 Position tolerance for orientation only according to ISO

14.13 Multiple patterns of features

According to ASME Y14.5, when groups of features, e.g. holes, are position toleranced related to the same datum system (same datum letters, same order of datums, same modifiers Ⓜ, Ⓛ), they are regarded as patterns and the tolerances apply simultaneously. For example, in the case of Ⓜ, they are to be inspected with one gauge; see Fig. 14.11.

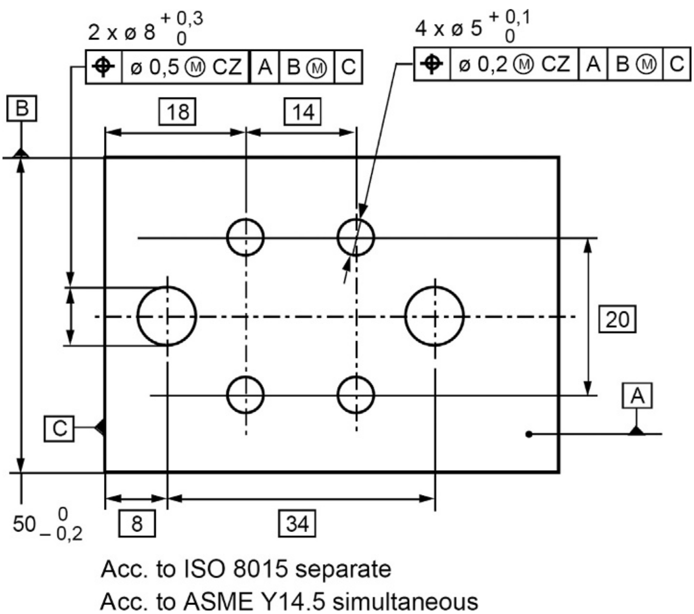


FIG. 14.11 Group of features with same datums

When the tolerances are not to be applied simultaneously but separately, with separate requirements, this must be indicated by "SEP REQT" at the tolerance indicator. ISO 1101 and ISO 5459 do not provide this rule. They apply the independency principle: if not otherwise indicated by SIM, the tolerances

for the patterns apply separately. This is the opposite of the ASME rule; see Fig. 14.11. Therefore it is recommended to indicate "SIM" at the tolerance indicator when the tolerances apply simultaneously; see Fig. 14.12.

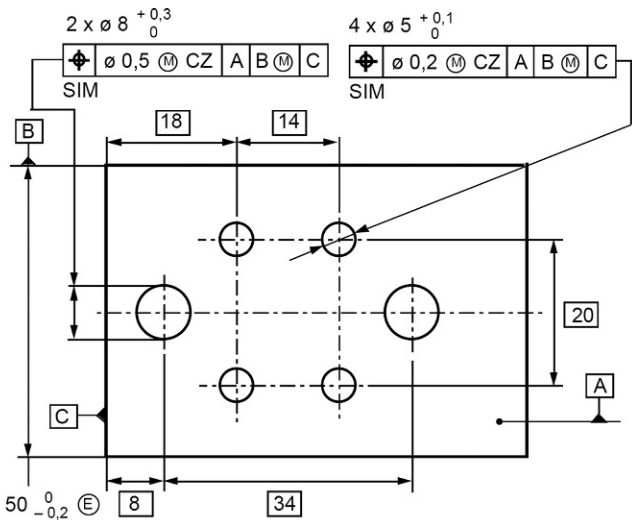


FIG. 14.12 Simultaneous requirements (simultaneous gauging)

When separate gauging is allowed, the indication is "separate" or SEP REQT instead of "SIM".

14.14 Unequally disposed profile tolerance zone

According to ASME Y14.5, the unequally disposed tolerance zone is to be indicated by the symbol Ⓤ after the tolerance value. The value after Ⓤ indicates the distance between the theoretically exact profile and the maximum material boundary. See Fig. 14.13.

According to ISO 1101 the unequally disposed tolerance zone is to be indicated by the symbol "UZ" after the tolerance value. The value after "UZ" indicates the distance between the theoretically exact profile and the centre line of the (shifted) tolerance zone. See Fig. 14.13. Positive values shift the tolerance zone outward of the material, and negative values shift the tolerance zone inward of the material.

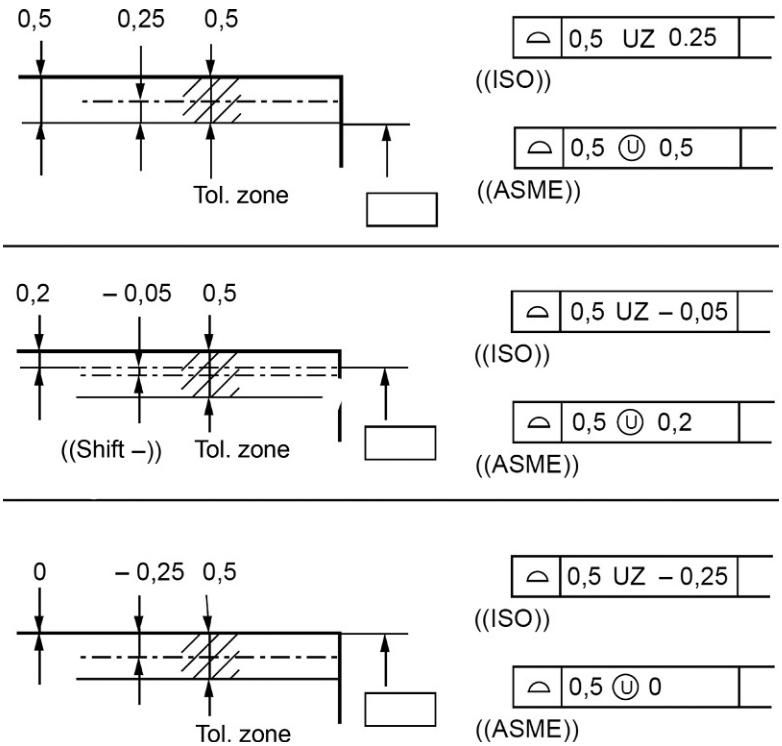


FIG. 14.13 Unequally disposed tolerance zone

14.15 Datums with indication of the locked degrees of freedom, customized datums

According to ASME Y14.5, the locked degrees of freedom may be indicated in [ ] in the tolerance indicator. See Fig. 14.14.

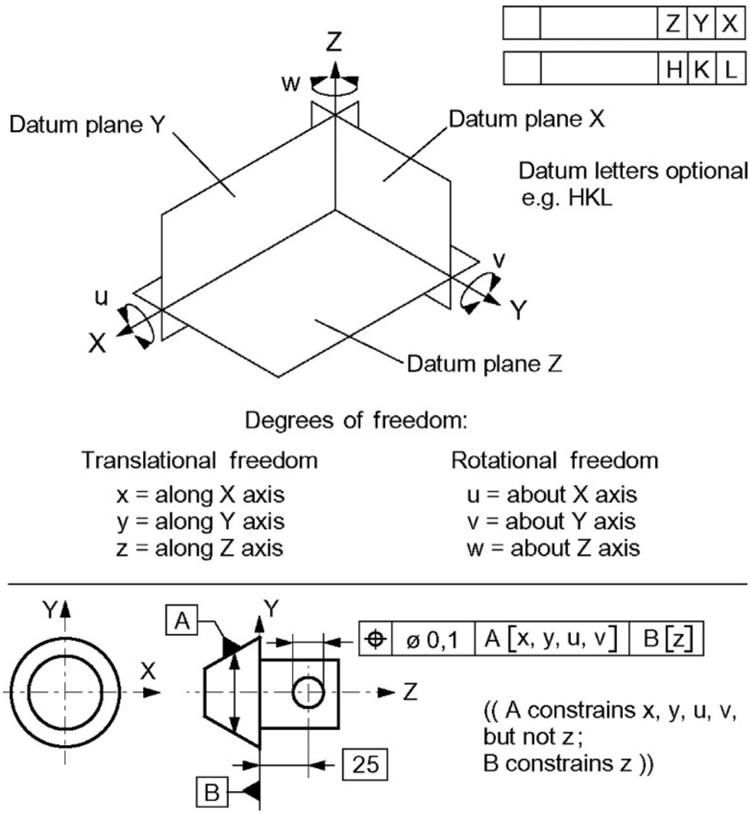


FIG. 14.14 Customized datums

Figure 14.15 shows an example according to ISO 5459 for the same purpose. Indicated are the applied situation features (point, straight line, plane) in [ ].

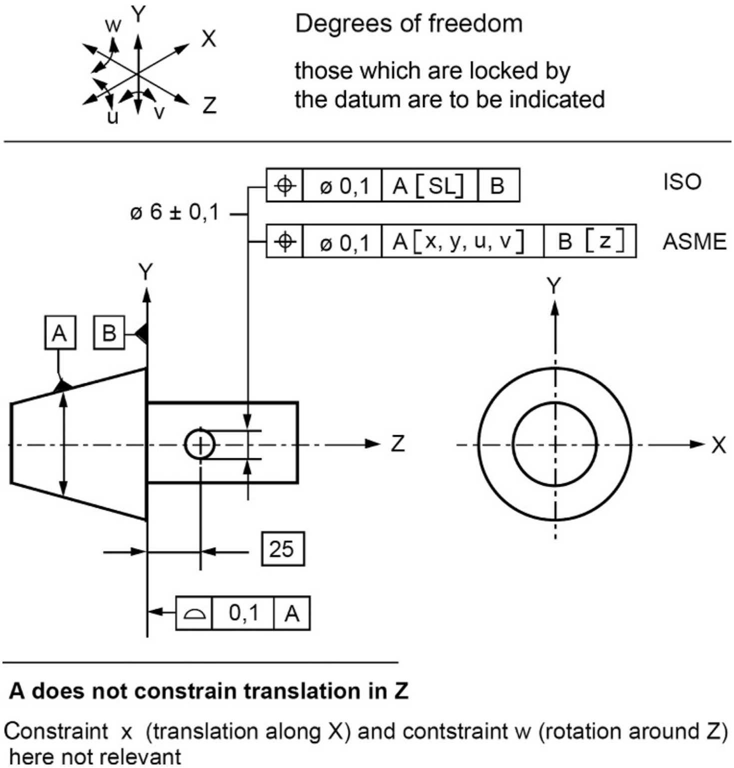


FIG. 14.15 Datum system with indication of the applied situation features; degree of freedom z is not locked by the cone but by the datum B

14.16 Datum systems

Figure 14.16 shows datum systems with datum B for locking the rotation around datum axis A according to ASME Y14.5.

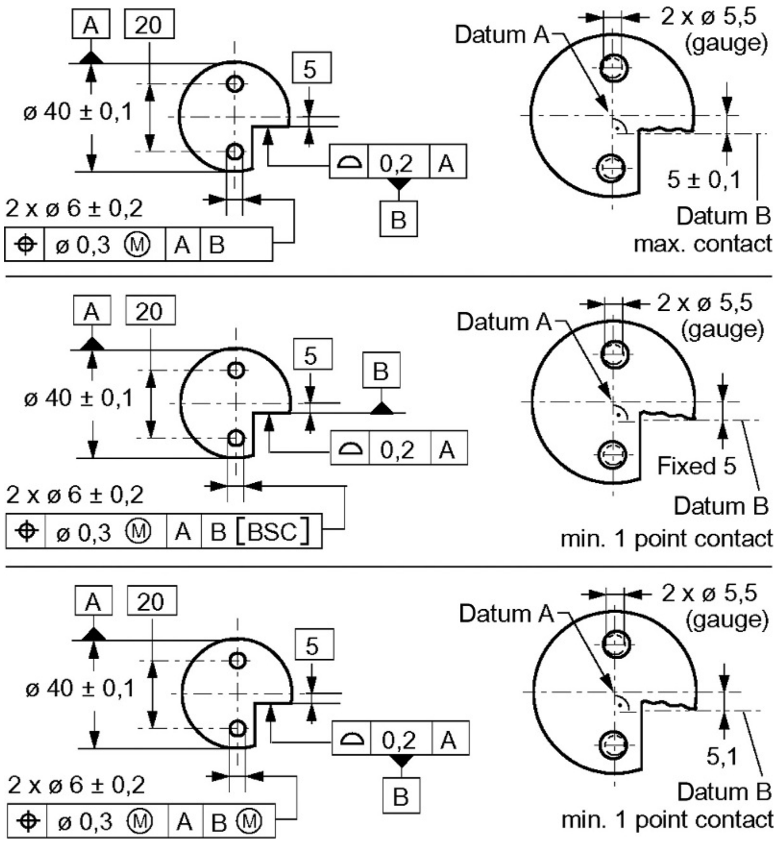


FIG. 14.16 Datums for locking the rotation around datum axis A

Above: The datum may vary within the profile tolerance 0,2 in order to have maximal contact.

Centre: The datum is fixed at the theoretical location. The workpiece must have (at least a point of) contact.

Below: The workpiece must have contact in the gauge at datum B (in order to limit the rotation to the left) ASME Y14.5-2009.

Figure 14.16 lower image requires contact at the gauge. This is in contradiction to ISO. ISO requires only that the workpiece fit into the gauge (Figs 14.17 and 14.18).

ASME Y14.5-2018 requires that at least one point is between 5 ± 0,1.

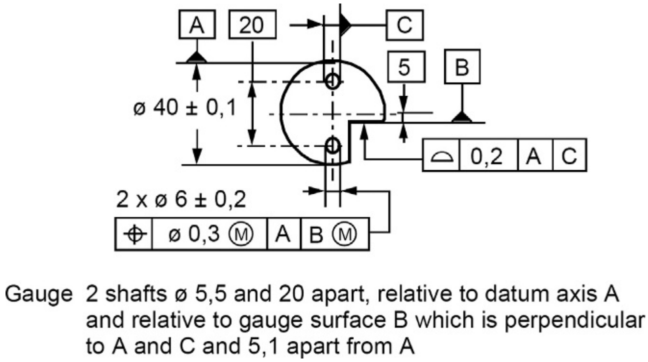


FIG. 14.17 ISO tolerancing similar to Fig. 14.16 lower image: contact at B not needed

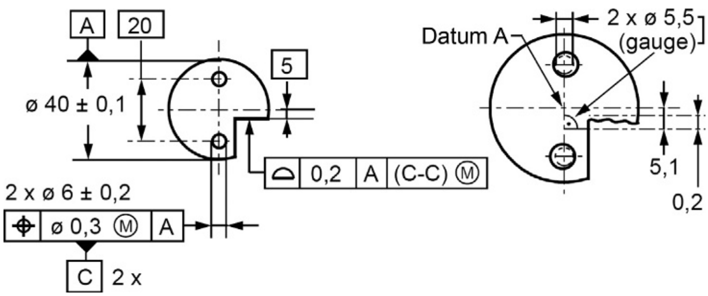


FIG. 14.18 ISO tolerancing similar to Fig. 14.16 lower image: contact at horizontal surface not needed

Figures 14.19 and 14.20 show tolerancings with the translation symbol and without.

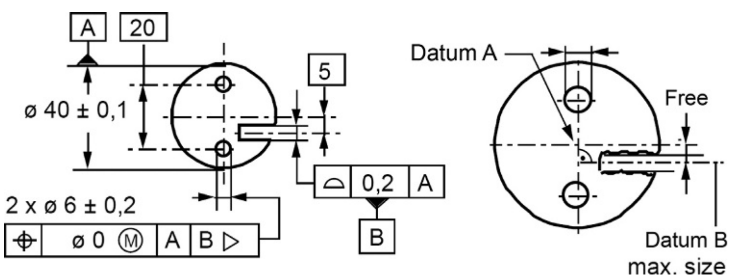


FIG. 14.19 ASME tolerancing with translation symbol

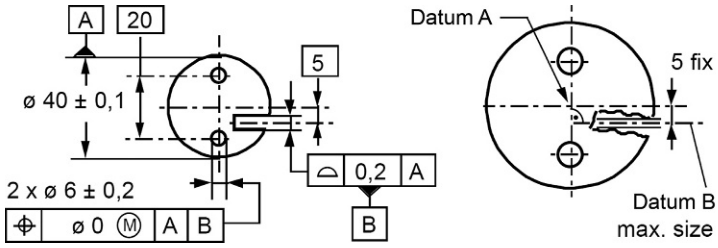


FIG. 14.20 ASME and ISO tolerancing without translation symbol

14.17 Roundness measurements

ASME B89.3.1 provides the possibility of specifying on drawings the measuring conditions for roundness tolerances, i.e.:

– reference method, to be selected from:

MRS: Minimum radial separation
LSC: Least-squares circle
MIC: Maximum inscribed circle
MCC: Minimum circumscribed circle

– filter, to be selected from:
0, 1,67, 5, 15, 50, 150, 1500 cycles per revolution
– stylus tip radius, to be selected from:
0,001, 0,003, 0,010, 0,030, 0,100 in.

See Fig. 14.21.

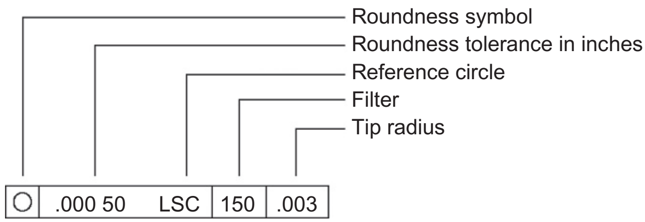


FIG. 14.21 Roundness tolerance with measuring conditions according to ASME B89.3.1

The filters are defined as electrical low-pass filters equivalent to two unloaded RC networks in series. Their nominal value (cut-off) corresponds to the sinusoidal frequency (cycles per revolution (cpr)) whose amplitude is 70,7% transmitted by the filter.

The former standard ISO 4291 specified a slightly different filter characteristic. The nominal value (cut-off) corresponds to the sine wavelength with 75% amplitude transmission. However, this difference is practically negligible. The new standard ISO 12 181 changes the nominal value (cut-off) corresponding to the sine wavelength with 50% amplitude transmission by the Gauss filter.

ASME B89.3.1 states further that, if not otherwise specified (default case), for roundness measurements (roundness tolerances), the following measuring conditions apply:

– Reference method	MRS
– Filter	50 cpr
– Tip radius	0,010 in (0,25 mm)

The ISO standards do not yet have standardized measuring conditions for the default case. If necessary, they must be agreed upon between the parties (e.g. by reference to ASME B89.3.1). See also 13.10.

14.18 Dynamic profile tolerance

The symbol △ stands for dynamic profile tolerance of features of size. This is, like the offset zone according to ISO 1101, a tolerance zone that controls the form but not the size of a feature of size. It uniformly progresses (expands or contracts) normal to the true profile.

Figure 14.22 shows a feature of size where the dynamic profile tolerance 0,04 is without reference to a datum. This zone may progress within the tolerance zone 0,2. It may translate and rotate. The lower section of the figure shows two possible zones.

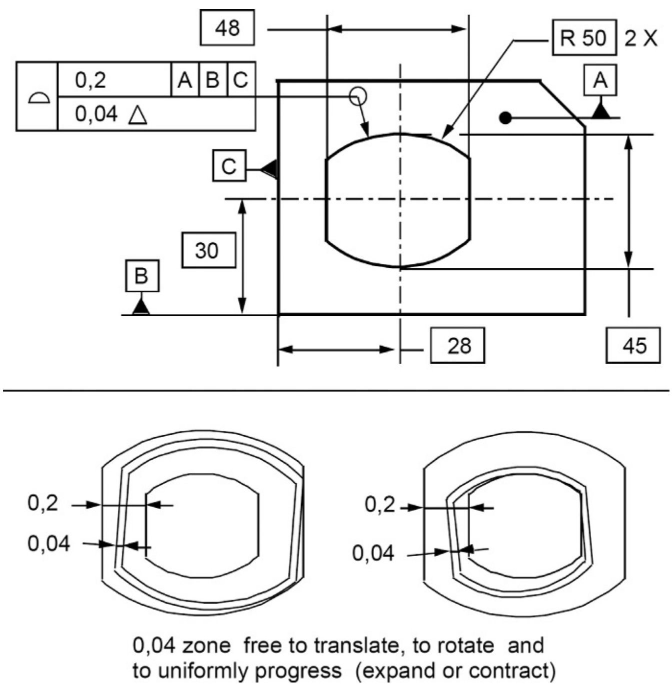


FIG. 14.22 Dynamic profile tolerance zone without reference to a datum

Figure 14.23 shows the dynamic profile tolerance as composite tolerancing with datums. In this composite tolerancing the datums control only the orientation (rotation) and not the translation. The 0,04 tolerance zones are fixed in rotation but not in translation. The lower section of this figure shows two possible zones.

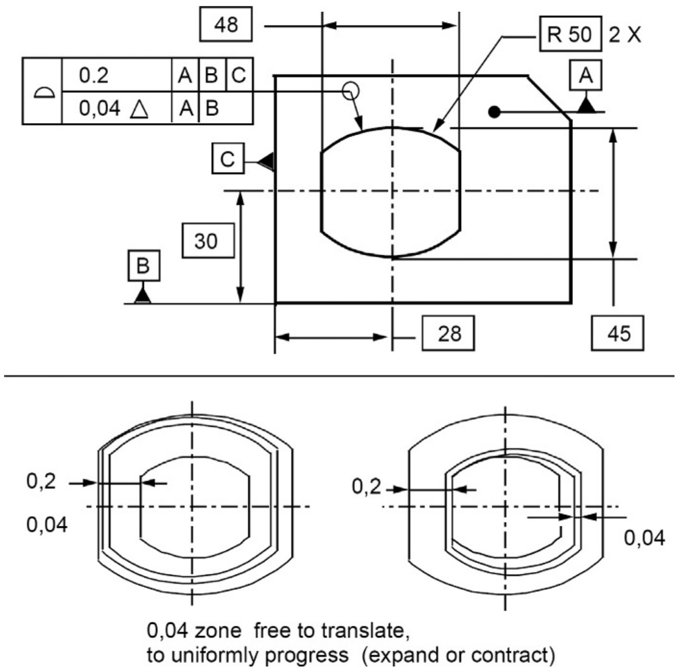


FIG. 14.23 Dynamic tolerance zone as composite tolerance zone with reference to datums

Figure 14.24 shows similar tolerancing for multiple features. It is recommended to consider multiple features as a single feature, although ASME Y14.5 does not mention it.

Figure 14.25 shows the dynamic profile tolerancing as single tolerancing with datums. The datums control the translations and the orientations (rotations). The figure shows two possible zones.

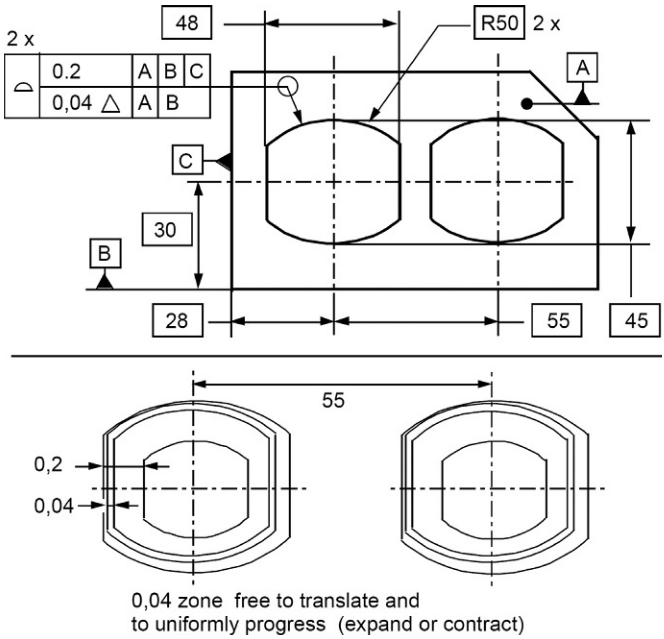


FIG. 14.24 Dynamic tolerance zone as composite tolerance zone with reference to datums and the constraint of both tolerance zones having the same size

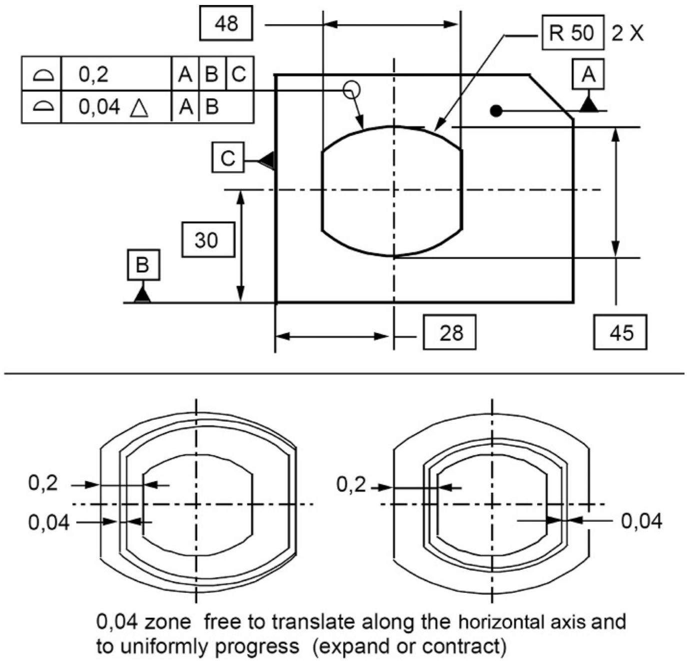


FIG. 14.25 Dynamic tolerance zone as single tolerance zone with reference to datums

With single tolerancing, multiple features are considered to be a single feature, unless INDIVIDUALLY follows the single feature control indicator. Figure 14.26 shows this case.

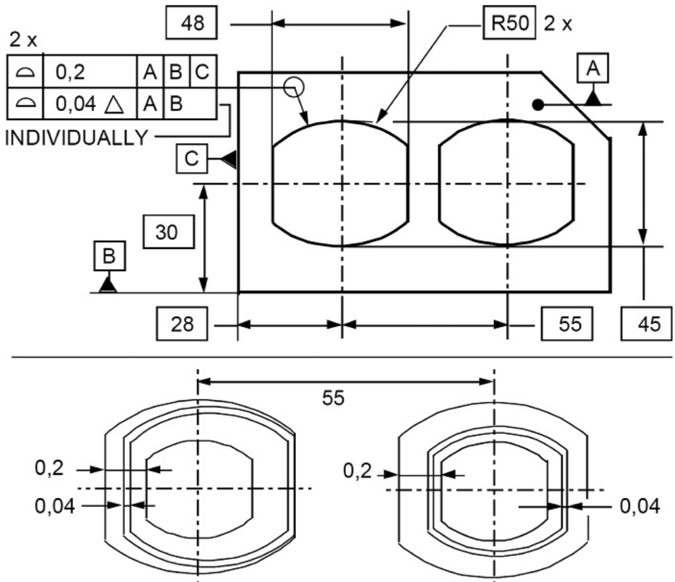


FIG. 14.26 Dynamic tolerance zone as single tolerance zone with reference to datums and the disclaimer INDIVIDUALLY

14.19 Terminology

Some terms used in ASME Y14.5 differ from those given by ISO (see Table 14.2).

TABLE 14.2 Comparison of ASME Y14.5 and ISO terminologies

ASME Y14.5M	ISO
Basic dimension	Theoretically exact dimension (TED)
Feature control indicator	Tolerance indicator
Variation	Deviation
True position (TP)	Theoretically exact position
Reference dimension	Auxiliary dimension

ASME Y14.5-2009 specifies FIM (Full Indicator Movement): the total movement of an indicator when appropriately applied to a surface to measure its variations. The former terms FIR (Full Indicator Reading) and TIR (Total Indicator Reading) have the same meaning as FIM.

ASME Y14.5-2009 defines the terms "resultant condition", "inner boundary" and "outer boundary" (which are not yet used in ISO standards).

Resultant condition: worst-case boundary, i.e.:

the inner boundary for holes with Ⓜ and for shafts with Ⓛ,
the outer boundary for holes with Ⓛ and for shafts with Ⓜ.

The boundary is the form that must not be exceeded by the feature. For a graphical definition of inner and outer boundary, see Fig. 14.27.

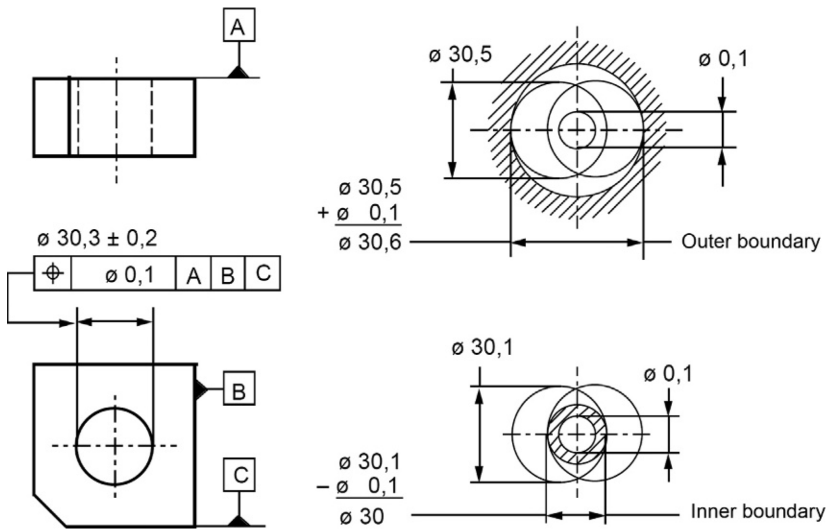


FIG. 14.27 Outer and inner boundary

The one boundary is given by the gauge or the cut-out contour and the position tolerance. The other boundary (resultant boundary) is given by the maximum material size or the least material size and the enlarged position tolerance. The enlarged position tolerance is the position tolerance plus or minus the size tolerance. See Figs 14.28 to 14.31.

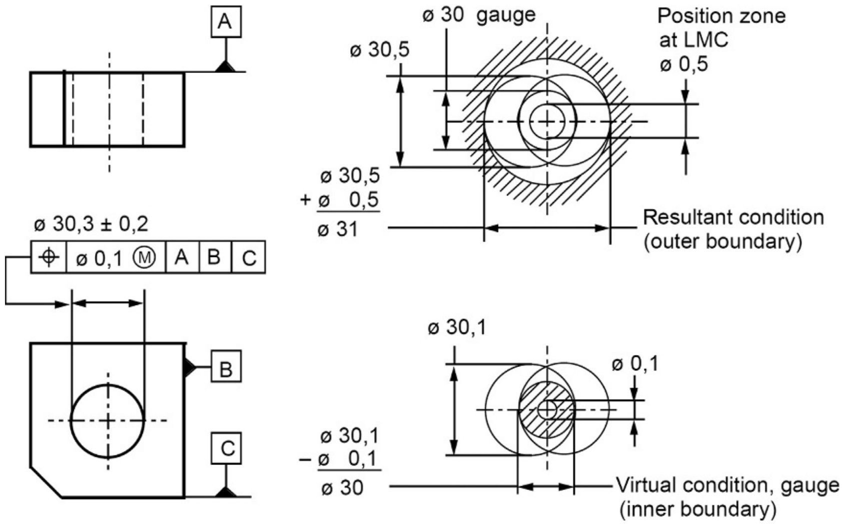


FIG. 14.28 Virtual and resultant condition boundaries, internal feature, MMC drawing indication

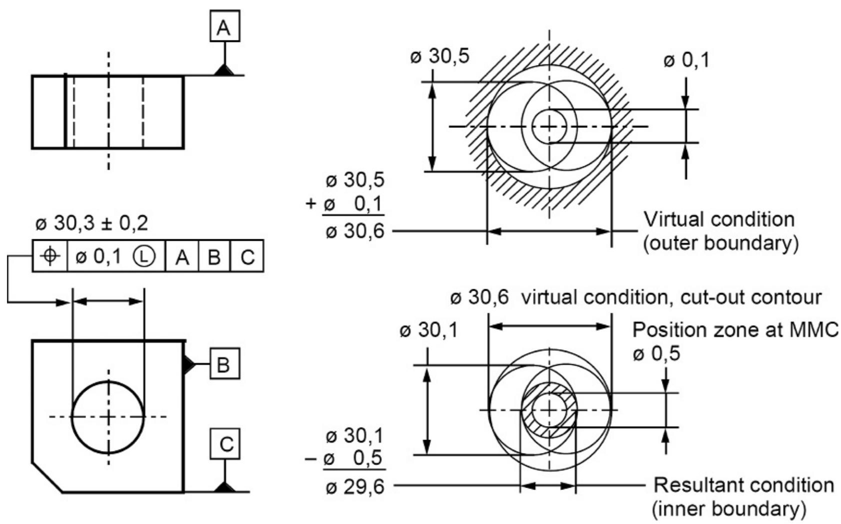


FIG. 14.29 Virtual and resultant condition boundaries, internal feature, LMC drawing indication

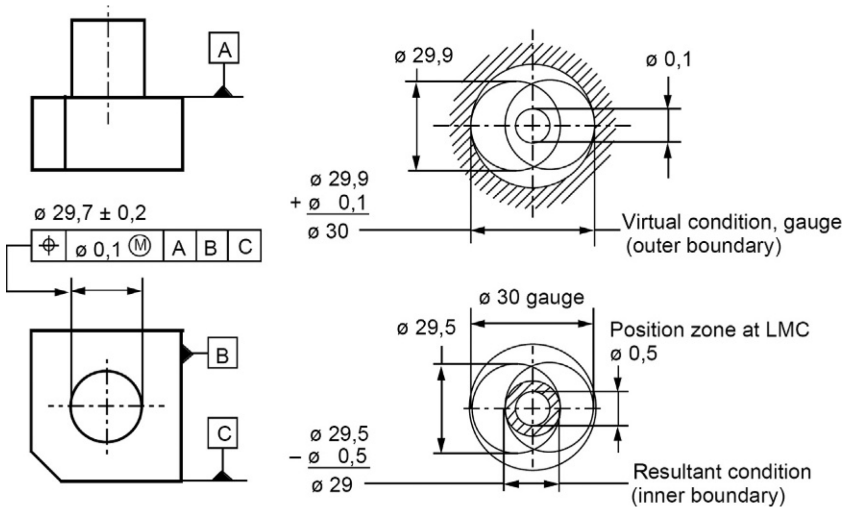


FIG. 14.30 Virtual and resultant condition boundaries, external feature, MMC drawing indication

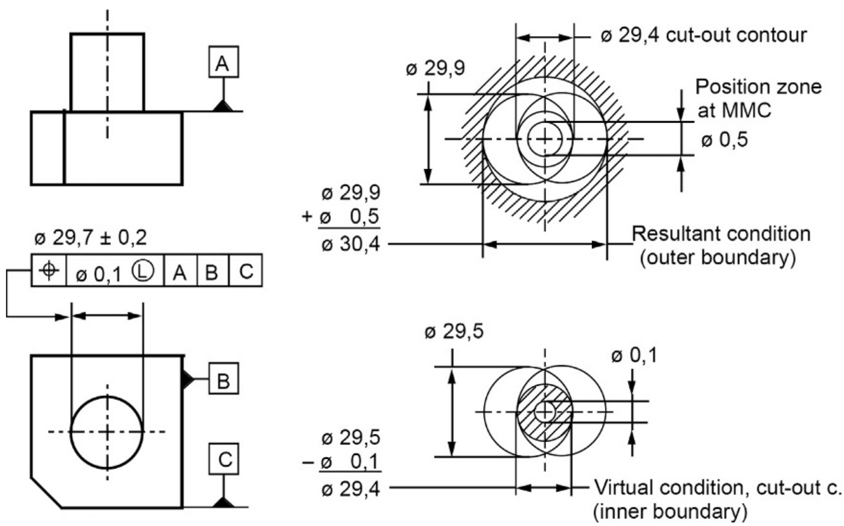


FIG. 14.31 Virtual and resultant condition boundaries, external feature, LMC drawing indication

14.20 Planar datum, definition

ASME Y14.5-2018 has changed the definition of the datum. It now applies that if irregularities on a datum feature are such that the part is unstable (i.e. it rocks) when it is brought into contact with the corresponding true geometric counterpart, the default requirement is that the part be adjusted to a single solution that minimizes the separation between the feature and the true geometric counterpart. If a different procedure is desired (e.g. candidate datum set, least squares), it shall be specified.

According to ISO 5459 the successor of the minimum rock requirement applies (Chebyshev extern, which is the same as according to ASME Y14.5-2018).

According to ASME Y14.5-2009 the "candidate method" applies: contacting planes are associated to the datum feature in all possible orientations. Those who fulfil certain requirements concerning the contacting points apply as candidates. When with one of the candidates all tolerances referred to this datum are respected, this candidate applies as the datum and the workpiece is to be accepted. See Fig. 14.32.

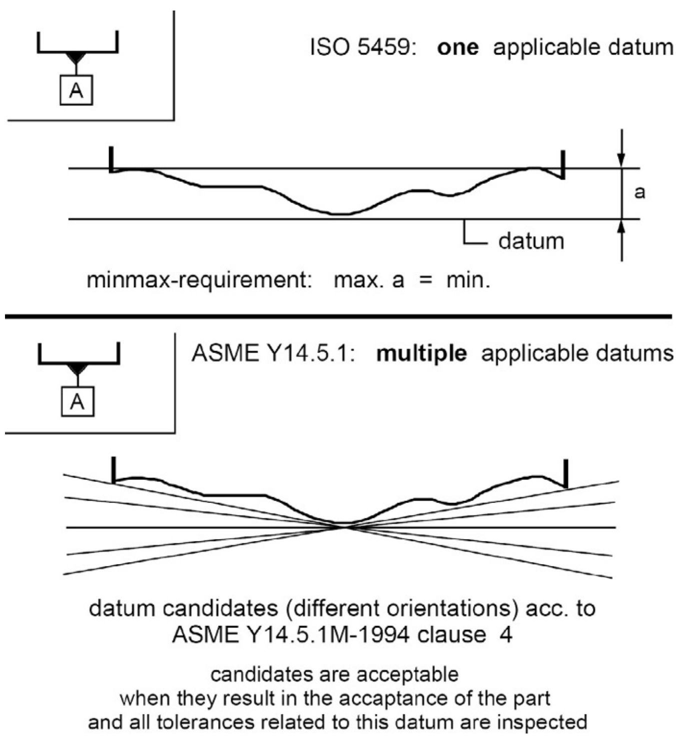


FIG. 14.32 Planar datum according to ISO 5459 and ASME Y14.5.1

14.21 Survey of differences between ASME and ISO

Table 14.3 together with Tables 14.1 and 14.2 and with Figs 14.1 and 14.2 give a survey of the differences between ASME and ISO standards.

TABLE 14.3 Differences between ASME and ISO standards

Designation	ASME Y14.5	ISO
Envelope requirement Ⓔ Rule# 1	Ⓔ applies without drawing indication see 14.4	Ⓔ or 0 Ⓜ to be indicated or reference to ISO 14 405-E (or to DIN 7167) see 14.4 and 4.3
Radius	Surface to be contained within 3-D tolerance zone see 14.5	No precise definition
Angle	Surface to be contained within 3-D tolerance zone see 14.7	Contacting lines in sections planes see 4.8
Position tolerance	Applies to the (straight) axis or (plane) median plane of the actual mating envelope see 14.11	Applies to the actual axis (not straight) or actual median surface (not planar) see 14.11
Symmetry tolerance concentricity tolerance (coaxiality tolerance)	Applies to actual axis or actual median surface (different from position tolerance) see 14.10	Applies to actual axis or actual median surface (same as position tolerance) see 14.10
Composite tolerance	Theor. exact dim. relative to the datum does not apply to the lower tolerance frame see 14.12	Theoretical exact dimensions apply always unless otherwise specified (by ><), see Figs 3.34 and 3.76
Multiple pattern	Default: SIM REQT see 14.13	To be indicated as applicable: SIM or separate, see 14.13
Roundness	Default: MRS, 50 cpr, r = 0.01 in; see 14.17	Default not yet standardized
Movable datum targets	Direction of movement: default: normal to true profile see Table 14.1	Direction of movement: indicated by symbol see Table 14.1
Dynamic profile tolerance	See Figs 14.22 to 14.26	See Figs 3.28, 3.36, 3.46, 3.49 and 6.11

14.22 Drawings for both systems, ISO and ASME

14.22.1 General

Drawings for both systems, ISO and ASME, may be identified by the indication in the drawing title block:

ISO 8015, ISO 1101, ASME Y14.5 without rule # 1

The symbols shown in Table 14.4 may be added to the drawings for the United States. This table provides the meanings of the symbols which are not in ASME Y14.5.

TABLE 14.4 Symbols which are not in ASME Y14.5

Ⓔ	Envelope requirement	ISO 14404-1
><	Orientation only	ISO 5459
CZ	Combined (common) zone	ISO 1101
CZR	Combined zone, only group internal orientation	ISO 5458
CT	Common tolerance	ISO 14405-1
UF	United feature (= ⟨CF⟩)	ISO 1101
UZ	Unequally distributed tol. zone (different Ⓤ)	ISO 1101
OZ	Offset zone, size variable (Chebyshev) (= △)	ISO 1101
VA	Variable angle	ISO 1101
ACS	Any cross section (datum section line or point)	ISO 1101
SIM	Simultaneous requirement	ISO 1101
[DV]	Dimension variable (= ▷)	ISO 5459[1)]
[DF]	Dimension fixed (default ASME)	ISO 5459[1)]
[Tx]	Translation x fixed (= [x])	ISO 5459[1)]
[Rx]	Rotation x fixed (= [u])	ISO 5459[1)]
[-]	Only for definition of sec. or tert. datum	ISO 5459[1)]
◁ // | A	Intersection plane orientation for ⌒	ISO 1101
◁ // | A ▷	Tolerance zone plane orientation	ISO 1101
◄ // | A	Tolerance (measuring) direction	ISO 1101

[1)]ISO / DIS 5459:2017

When the same symbology has different interpretations according to ISO and ASME, the recommendations in the following paragraphs should be followed.

14.22.2 Coplanarity

Whenever coplanarity is required, CZ should be indicated; see Fig. 3.10.

14.22.3 Envelope requirement

The use of the more economical principle of independency according to ISO 8015 is recommended. When the envelope requirement is necessary, the symbol Ⓔ (ISO 8015) or position zero Ⓜ (ASME Y14.5) is to be indicated; see example in Fig. 4.18.

14.22.4 Angular tolerances

Angular dimensional tolerances (see 4.8 and Fig. 3.43) should be avoided.

For tolerancing of angularity, profile or position according to ISO 1101 is recommended (Figs 3.35 and 3.76).

14.22.5 Radii tolerances

For radii, ± tolerances should be avoided. They are not precisely defined according to ISO. It is recommended to use profile tolerancing according to ISO 1101 (see Fig. 3.26) or to use transitions according to ISO 21 204 (see Figs 9.14 and 9.15).

14.22.6 Roundness tolerances

It is recommended to use the symbol ⌒ and to avoid the symbol ○. See Fig. 3.58 and 14.17.

14.22.7 Coaxiality and symmetry tolerances

It is recommended to use position; see Fig. 3.73 and 14.22.9.

14.22.8 Simultaneous requirements for groups with different position tolerances

It is recommended to always indicate “SIM” when it applies (see Fig. 14.12) and “separate” when it applies.

14.22.9 Orientation tolerances of axes or median surfaces

ISO 1101 tolerances the actual (imperfect) axes or median surfaces. ASME Y14.5 tolerances the (perfect) axes or median planes of the mating envelopes;

see Fig. 14.8. It is recommended to tolerance in such a way that both versions fit to the workpiece function and are acceptable. Alternative: indicate “ISO” or “ASME” after the tolerance indicator (tolerance indicator).

14.22.10 Position tolerancing

It is recommended to avoid position tolerancing of planar surfaces and to use profile tolerancing, as it is the rule according to ASME Y14.5; see 14.3.

ISO 1101 tolerances the actual (imperfect) axes or median surfaces with position tolerances. ASME Y14.5 tolerances the (perfect) axes or median planes of the mating envelopes with position tolerances; see Fig. 14.8. It is recommended to tolerance in such a way that both versions fit to the workpiece function and are acceptable. Alternative: indicate “ISO” or “ASME” after the tolerance indicator.

14.22.11 Composite tolerancing

It is recommended to always indicate “orientation only” (symbol ><) when it applies; see Fig. 14.10.

14.22.12 Maximum material requirement, least material requirement

For tolerances with MMR or LMR, it is recommended to use position and to avoid coaxiality and symmetry, as this is the rule according to ASME Y14.5. The same applies to perpendicularity, parallelism and angularity.

This page intentionally left blank

Glossary

These are easily understandable descriptions of terminology; for the standardized definitions, see the standards.

Actual In reality.
Association Operation to fit an ideal feature to a non-ideal feature according to a criterion (e.g. Chebyshev, Gauss, minimum circumscribed, maximum inscribed, etc.).
Characteristic See Geometrical characteristic.
Chebyshev criterion Maximum distance between considered (e.g. toleranced) feature and reference feature minimized (also referred to as minimax criterion) [ISO 1101].
Default Applies, if not otherwise indicated.
Deviations global deviation, local deviation.
Global deviation Only one value for the entire feature, e.g. Gauss size.
Local deviation Different values at different locations, e.g. two-point size.
Envelope element Smallest ideal cylinder or plane pair for a shaft, largest ideal cylinder or plane pair for a hole or slot, associated to the real feature.
Envelope requirement Requires feature within ideal gauge contour of maximum material size [ISO 2692].
Features Types; see 2.2.
Gauss criterion Sum of squares of distances between considered (e.g. toleranced) feature and reference feature minimized.
Geometrical characteristic Geometrical property of a workpiece [ISO 25 378].
L_1 norm Volume between considered feature and reference feature minimized [ISO 5459]; see Table 3.10.
L_2 norm Sum of squares of distances between considered feature and reference feature minimized (Gauss criterion) [ISO 5459]; see Table 3.10.
L_∞ norm Maximum distance between considered feature and reference feature minimized (Chebyshev criterion) [ISO 5459]; see Table 3.10.
Least material requirement Requires at the datum feature and/or at the toleranced feature the ideal "cut- out contour" completely within the material. [ISO 2692].
Maximum material requirement Requires datum feature and/or toleranced feature maximum material virtual size within ideal gauge contour [ISO 2692].
Minimax criterion See Chebyshev criterion.
Modifier Modifies or precisely defines a characteristic, e.g. Ⓔ, Ⓜ, .
Multivariate characteristic Characteristic with (+ or −) sign.
Non-size dimension Dimension which is not a size, e.g. centre distance, step dimension, radius.
Operations Types; see 2.2.
Parameter characteristic Defined by a parameter, e.g. size tolerance.
Plane-pair Two opposite parallel planes connected by a size.
Position tolerance Geometrical tolerance of a surface, a line or a point, preferable for derived features [ISO 1101].
Profile tolerance Geometrical tolerance of a surface or a line, preferable for integral features [ISO 1101].
Projected tolerance zone Tolerance zone for the projection of the derived feature, in order to enable the assembly [ISO 1101].

Reciprocity requirement Specifies with Ⓜ︎Ⓡ that the maximum material limit of size may be exceeded, provided the feature fits into the gauge contour; specifies with ⓁⓇ that the least material limit of size may be exceeded, provided the feature contains the cut-out contour completely [ISO 2692].

Size Linear: dimension of a cylindrical feature or a feature established by two opposite parallel planes (plane pair) [ISO 14 405-1]. Angular: dimension of a cone or wedge [ISO 14 405-3].

Univariate characteristic Characteristic without (+ or −) sign.

Variable size Size with ± tolerance.

Zone characteristic Defined by a zone, e.g. profile tolerance.

For further terms, see 2.2.

Standards

ISO 1:2016	Geometrical product specifications (GPS) – Standard reference temperature for the specification of geometrical and dimensional properties
ISO 286-1:2010	Geometrical product specifications (GPS) – ISO code system for tolerances on linear sizes – Part 1: Basis of tolerances, deviations and fits
ISO 286-1:2010/Cor 1:2013	
ISO 286-2:2010	Geometrical product specifications (GPS) – ISO code system for tolerances on linear sizes – Part 2: Tables of standard tolerance classes and limit deviations for holes and shafts
ISO 286-2:2010/Cor 1:2013	
ISO 463:2006	Geometrical product specifications (GPS) – Dimensional measuring equipment – Design and metrological characteristics of mechanical dial gauges
ISO 463:2006/Cor 1:2007	
ISO 463:2006/Cor 2:2009	
ISO 1101:2017	Geometrical product specifications (GPS) – Geometrical tolerancing – Tolerances of form, orientation, location and run-out
ISO 1119:2011	Geometrical product specifications (GPS) – Series of conical tapers and taper angles
ISO 1302:2002	Geometrical product specifications (GPS) – Indication of surface texture in technical product documentation
ISO 1502:1996	ISO general purpose metric screw threads – Gauges and gauging
ISO 1660:2017	Geometrical product specifications (GPS) – Geometrical tolerancing – Profile tolerancing
ISO 1938-1:2015	Geometrical product specifications (GPS) – Dimensional measuring equipment – Part 1: Plain limit gauges of linear size
ISO 1938-2:2017	Geometrical product specifications (GPS) – Dimensional measuring equipment – Part 2: Reference disk gauges
ISO 2538-1:2014	Geometrical product specifications (GPS) – Wedges – Part 1: Series of angles and slopes
ISO 2538-2:2014	Geometrical product specifications (GPS) – Wedges – Part 2: Dimensioning and tolerancing

ISO 2692:2014	Geometrical product specifications (GPS) – Geometrical tolerancing – Maximum material requirement (MMR), least material requirement (LMR) and reciprocity requirement (RPR)
ISO 2768-1:1989	General tolerances – Part 1: Tolerances for linear and angular dimensions without individual tolerance indications
ISO 2768-2:1989	General tolerances – Part 2: Geometrical tolerances for features without individual tolerance indications
ISO 3040:2016	Geometrical product specifications (GPS) – Dimensioning and tolerancing – Cones
ISO 3274:1996	Geometrical product specifications (GPS) – Surface texture: Profile method – Nominal characteristics of contact (stylus) instruments
ISO 3274:1996/Cor 1:1998	
ISO 3611:2010	Geometrical product specifications (GPS) – Dimensional measuring equipment: Micrometers for external measurements – Design and metrological characteristics
ISO 3650:1998	Geometrical product specifications (GPS) – Length standards – Gauge blocks
ISO 3650:1998/Cor 1:2008	
ISO 4291:1985	Methods for the assessment of departure from roundness – Measurement of variations in radius
ISO 4759-1:2000	Tolerances for fasteners – Part 1: Bolts, screws, studs and nuts – Product grades A, B, C
ISO 5458:2018	Geometrical product specifications (GPS) – Geometrical tolerancing – Positional tolerancing
ISO 5459:2011	Geometrical product specifications (GPS) – Geometrical tolerancing – Datums and datum systems
ISO 7863:1984	Height setting micrometers and riser blocks
ISO 8015:2011	Geometrical product specifications (GPS) – Fundamentals – Concepts, principles and rules
ISO 8062-1:2007	Geometrical product specifications (GPS) – Dimensional and geometrical tolerances for moulded parts – Part 1: Vocabulary
ISO/TS 8062-2:2013	Geometrical product specifications (GPS) – Dimensional and geometrical tolerances for moulded parts – Part 2: Rules
ISO 8062-3:2007	Geometrical product specifications (GPS) – Dimensional and geometrical tolerances for moulded parts – Part 3: General dimensional and geometrical tolerances and machining allowances for castings
ISO 8062-3:2007/Cor 1:2009	
ISO 8062-4	Geometrical product specifications (GPS) – Dimensional and geometrical tolerances for moulded parts – Part 4: General tolerances for castings using profile tolerancing in a general datum system
ISO 8512-1:1990	Surface plates – Part 1: Cast iron
ISO 8512-2:1990	Surface plates – Part 2: Granite
ISO 8785:1998	Geometrical product specification (GPS) – Surface imperfections – Terms, definitions and parameters

ISO 9493:2010	Geometrical product specifications (GPS) – Dimensional measuring equipment: Dial test indicators (lever type) – Design and metrological characteristics
ISO 10 135:2007	Geometrical product specifications (GPS) – Drawing indications for moulded parts in technical product documentation (TPD)
ISO 10 360-1:2000	Geometrical product specifications (GPS) – Acceptance and reverification tests for coordinate measuring machines (CMM) – Part 1: Vocabulary
ISO 10 360-1:2000/Cor 1:2002	
ISO 10 360-2:2009	Geometrical product specifications (GPS) – Acceptance and reverification tests for coordinate measuring machines (CMM) – Part 2: CMMs used for measuring linear dimensions
ISO 10 360-3:2000	Geometrical product specifications (GPS) – Acceptance and reverification tests for coordinate measuring machines (CMM) – Part 3: CMMs with the axis of a rotary table as the fourth axis
ISO 10 360-4:2000	Geometrical product specifications (GPS) – Acceptance and reverification tests for coordinate measuring machines (CMM) – Part 4: CMMs used in scanning measuring mode
ISO 10 360-4:2000/Cor 1:2002	
ISO 10 360-5:2010	Geometrical product specifications (GPS) – Acceptance and reverification tests for coordinate measuring machines (CMM) – Part 5: CMMs using single and multiple stylus contacting probing systems
ISO 10 360-6:2001	Geometrical product specifications (GPS) – Acceptance and reverification tests for coordinate measuring machines (CMM) – Part 6: Estimation of errors in computing Gaussian associated features
ISO 10 360-6:2001/Cor 1:2007	
ISO 10 360-7:2011	Geometrical product specifications (GPS) – Acceptance and reverification tests for coordinate measuring machines (CMM) – Part 7: CMMs equipped with imaging probing systems
ISO 10 360-8:2013	Geometrical product specifications (GPS) – Acceptance and reverification tests for coordinate measuring systems (CMS) – Part 8: CMMs with optical distance sensors
ISO 10 360-9:2013	Geometrical product specifications (GPS) – Acceptance and reverification tests for coordinate measuring systems (CMS) – Part 9: CMMs with multiple probing systems
ISO 10 360-10:2016	Geometrical product specifications (GPS) – Acceptance and reverification tests for coordinate measuring systems (CMS) – Part 10: Laser trackers for measuring two-point distances
ISO 10 360-12:2016	Geometrical product specifications (GPS) – Acceptance and reverification tests for coordinate measuring systems (CMS) – Part 12: Articulated arm coordinate measurement machines (CMM)
ISO 10 579:2010	Geometrical product specifications (GPS) – Dimensioning and tolerancing – Non-rigid parts
ISO 10 579:2010/Cor 1:2011	
ISO 12 085:1996	Geometrical product specifications (GPS) – Surface texture: Profile method – Motif parameters

ISO 12 085:1996/Cor 1:1998	
ISO 12 179:2000	Geometrical product specifications (GPS) – Surface texture: Profile method – Calibration of contact (stylus) instruments
ISO 12 179:2000/Cor 1:2003	
ISO 12 180-1:2011	Geometrical product specifications (GPS) – Cylindricity – Part 1: Vocabulary and parameters of cylindrical form
ISO 12 180-2:2011	Geometrical product specifications (GPS) – Cylindricity – Part 2: Specification operators
ISO 12 181-1:2011	Geometrical product specifications (GPS) – Roundness – Part 1: Vocabulary and parameters of roundness
ISO 12 181-2:2011	Geometrical product specifications (GPS) – Roundness – Part 2: Specification operators
ISO 12 780-1:2011	Geometrical product specifications (GPS) – Straightness – Part 1: Vocabulary and parameters of straightness
ISO 12 780-2:2011	Geometrical product specifications (GPS) – Straightness – Part 2: Specification operators
ISO 12 781-1:2011	Geometrical product specifications (GPS) – Flatness – Part 1: Vocabulary and parameters of flatness
ISO 12 781-2:2011	Geometrical product specifications (GPS) – Flatness – Part 2: Specification operators
ISO 13 102:2012	Geometrical product specifications (GPS) – Dimensional measuring equipment: Electronic digital-indicator gauge – Design and metrological characteristics
ISO 13 225:2012	Geometrical product specifications (GPS) – Dimensional measuring equipment; Height gauges – Design and metrological characteristics
ISO 13 385-1:2011	Geometrical product specifications (GPS) – Dimensional measuring equipment – Part 1: Callipers; Design and metrological characteristics
ISO 13 385-2:2011	Geometrical product specifications (GPS) – Dimensional measuring equipment – Part 2: Calliper depth gauges; Design and metrological characteristics
ISO 13 565-1:1996	Geometrical product specifications (GPS) – Surface texture: Profile method; Surfaces having stratified functional properties – Part 1: Filtering and general measurement conditions
ISO 13 565-1:1996/Cor 1:1998	
ISO 13 565-2:1996	Geometrical product specifications (GPS) – Surface texture: Profile method; Surfaces having stratified functional properties – Part 2: Height characterization using the linear material ratio curve
ISO 13 565-2:1996/Cor 1:1998	
ISO 14 253-1:2017	Geometrical product specifications (GPS) – Inspection by measurement of workpieces and measuring equipment – Part 1: Decision rules for proving conformity or nonconformity with specifications
ISO 14 253-2:2011	Geometrical product specifications (GPS) – Inspection by measurement of workpieces and measuring equipment – Part 2: Guidance for the estimation of uncertainty in GPS measurement, in calibration of measuring equipment and in product verification

ISO 14 253-2:2011/Cor 1:2013	
ISO 14 253-3:2011	Geometrical product specifications (GPS) – Inspection by measurement of workpieces and measuring equipment – Part 3: Guidelines for achieving agreements on measurement uncertainty statements
ISO/TS 14 253-4:2010	Geometrical product specifications (GPS) – Inspection by measurement of workpieces and measuring equipment – Part 4: Background on functional limits and specification limits in decision rules
ISO 14 253-5:2015	Geometrical product specifications (GPS) – Inspection by measurement of workpieces and measuring equipment – Part 5: Uncertainty in verification testing of indicating measuring instruments
ISO/TR 14 253-6:2012	Geometrical product specifications (GPS) – Inspection by measurement of workpieces and measuring equipment – Part 6: Generalized decision rules for the acceptance and rejection of instruments and workpieces
ISO 14 405-1:2016	Geometrical product specifications (GPS) – Dimensional tolerancing – Part 1: Linear sizes
ISO 14 405-2:2011	Geometrical product specifications (GPS) – Dimensional tolerancing – Part 2: Dimensions other than linear sizes
ISO 14 405-3:2016	Geometrical product specifications (GPS) – Dimensional tolerancing – Part 3: Angular sizes
ISO 14 406:2010	Geometrical product specifications (GPS) – Extraction
ISO 14 638:2015	Geometrical product specifications (GPS) – Matrix model
ISO 14 978:2018	Geometrical product specifications (GPS) – General concepts and requirements for GPS measuring equipment
ISO/TS 15 530-1:2013	Geometrical product specifications (GPS) – Coordinate measuring machines (CMM): Technique for determining the uncertainty of measurement – Part 1: Overview and metrological characteristics
ISO 15 530-3:2011	Geometrical product specifications (GPS) – Coordinate measuring machines (CMM): Technique for determining the uncertainty of measurement – Part 3: Use of calibrated workpieces or measurement standards
ISO/TS 15 530-4:2008	Geometrical product specifications (GPS) – Coordinate measuring machines (CMM): Technique for determining the uncertainty of measurement – Part 4: Evaluating task-specific measurement uncertainty using simulation
ISO/TR 16 015:2003	Geometrical product specifications (GPS) – Systematic errors and contributions to measurement uncertainty of length measurement due to thermal influences
ISO 16 610-1:2015	Geometrical product specifications (GPS) – Filtration – Part 1: Overview and basic concepts
ISO 16 610-20:2015	Geometrical product specifications (GPS) – Filtration – Part 20: Linear profile filters: Basic concepts
ISO 16 610-21:2011	Geometrical product specifications (GPS) – Filtration – Part 21: Linear profile filters: Gaussian filters
ISO 16 610-22:2015	Geometrical product specifications (GPS) – Filtration – Part 22: Linear profile filters: Spline filters
ISO 16 610-28:2016	Geometrical product specifications (GPS) – Filtration – Part 28: Profile filters: End effects

ISO 16 610-29:2015	Geometrical product specifications (GPS) – Filtration – Part 29: Linear profile filters: Spline wavelets
ISO 16 610-30:2015	Geometrical product specifications (GPS) – Filtration – Part 30: Robust profile filters: Basic concepts
ISO 16 610-31:2016	Geometrical product specifications (GPS) – Filtration – Part 31: Robust profile filters: Gaussian regression filters
ISO/TS 16 610-32:2009	Geometrical product specifications (GPS) – Filtration – Part 32: Robust profile filters: Spline filters
ISO 16 610-40:2015	Geometrical product specifications (GPS) – Filtration – Part 40: Morphological profile filters: Basic concepts
ISO 16 610-41:2015	Geometrical product specifications (GPS) – Filtration – Part 41: Morphological profile filters: Disk and horizontal line-segment filters
ISO 16 610-49:2015	Geometrical product specifications (GPS) – Filtration – Part 49: Morphological profile filters: Scale space techniques
ISO 16 610-60:2015	Geometrical product specification (GPS) – Filtration – Part 60: Linear areal filters – Basic concepts
ISO 16 610-61:2015	Geometrical product specification (GPS) – Filtration – Part 61: Linear areal filters – Gaussian filters
ISO 16 610-71:2014	Geometrical product specifications (GPS) – Filtration – Part 71: Robust areal filters: Gaussian regression filters
ISO 16 610-85:2013	Geometrical product specifications (GPS) – Filtration – Part 85: Morphological areal filters: Segmentation
ISO 17 450-1:2011	Geometrical product specifications (GPS) – General concepts – Part 1: Model for geometrical specification and verification
ISO 17 450-2:2012	Geometrical product specifications (GPS) – General concepts – Part 2: Basic tenets, specifications, operators, uncertainties and ambiguities
ISO 17 450-3:2016	Geometrical product specifications (GPS) – General concepts – Part 3: Toleranced features
ISO 17 450-4:2017	Geometrical product specifications (GPS) – Basic concepts – Part 4: Geometrical characteristics for quantifying GPS deviations
ISO/TS 17 863:2013	Geometrical product specification (GPS) – Tolerancing of moveable assemblies
ISO/TS 17 865:2016	Geometrical product specifications (GPS) – Guidelines for the evaluation of coordinate measuring machine (CMM) test uncertainty for CMMs using single and multiple stylus contacting probing systems
ISO 18 391:2016	Geometrical product specifications (GPS) – Population specification
ISO 20 170:2019	Geometrical product specifications (GPS) – Decomposition of geometrical characteristics for manufacturing control
ISO/TS 21 619:2018	Geometrical product specifications (GPS) – Types of documents with GPS
ISO 22 432:2011	Geometrical product specifications (GPS) – Features utilized in specification and verification
ISO 22 514-2:2017	Statistical methods in process management – Capability and performance – Part 2: Process capability and performance of time-dependent process models

ISO/TS 23 165:2006	Geometrical product specifications (GPS) – Guidelines for the evaluation of coordinate measuring machine (CMM) test uncertainty
ISO/TR 23 605:2018	Technical product specification (TPS) – Application guidance – International model for national implementation
ISO 25 378:2011	Geometrical product specifications (GPS) – Characteristics and conditions – Definitions
GUM	(JCGM 100:2008 Evaluation of measurement data – Guide to the expression of uncertainty in measurement)
VIM	(JCGM 200:2012 International vocabulary of metrology – Basic and general concepts and associated terms)
ASME Y14.5-2009	Dimensioning and Tolerancing
ASME Y14.5-2018	Dimensioning and Tolerancing
ASME Y14.5.1M-10994	Mathematical Definition of Dimensioning and Tolerancing Principles
ASME B89.3.1-1988	Measurement of Out-of-Roundness
BS 8888-2017	Technical product documentation and specifications

The following standards are only available in German (the figures give an impression of the intention):

TGL 39 092	Methods of measuring geometrical deviations – General principles
TGL 39 093	Methods of measuring deviations from straightness
TGL 39 094	Methods of measuring deviations from flatness
TGL 39 095	Methods of measuring deviations from parallelism
TGL 39 096	Methods of measuring deviations from roundness
TGL 39 097	Methods of measuring deviations from cylindricity
TGL 39 098	Methods of measuring deviations of the longitudinal section profile
TGL 43 041	Methods of measuring straightness deviations of axes
TGL 43 042	Methods of measuring deviations from coaxiality

Publications

[1]	Kirschling	*Quality Assurance and Tolerances* (New York: Springer-Verlag 1991)
[2]	Neumann	*Industrial Coordinate Metrology* (Landsberg/Lech: verlag moderne industrie 2000)
[3]	Barth	Kombinierte Zweipunkt-Dreipunkt-Messung zum gleich-zeitigen Bestimmen von Maß- und Kreisformabweichungen. *Feingerätetechnik, Berlin* **33** (1984) 1 S. 21–23
[4]	Wirtz, Maduda	Fertigungsgerechte Rundheitsmessung. *Fertigung* **5**/72, S. 127–137
[5]	Shakarji Srinivasan	Toward a New Mathematical Definition of Datums in Standards to Support Advanced Manufacturing. MSEC2018-6305, *Proceedings of the ASME 2018 Manufacturing Science and Engineering Conference (MSEC2018)*, June 18–22, 2018, College Station, Texas, USA

This page intentionally left blank

Index

Note: Page numbers followed by *f* indicate figures, and *t* indicate tables.

D

E

F

K

L

M

R

S

U

THIRD EDITION

Geometrical Dimensioning and Tolerancing for Design, Manufacturing and Inspection

A Handbook for Geometrical Product Specification Using ISO and ASME standards

Geometrical Dimensioning and Tolerancing for Design, Manufacturing and Inspection, Third Edition, presents the state of the art in geometrical dimensioning and tolerancing. It describes the international standardization in this field laid down in ISO Standards and also indicates how this differs from the American Standard ASME Y14.5M, in concise and clear language.

The general principles of geometrical dimensioning and tolerancing are described, helping to define precision-related specifications unambiguously and consistently with the constraints of the manufacturing and inspection processes. Principles for the inspection of geometrical deviations are given, together with a basis for tolerancing suitable for inspection, as well as practical examples that show correct application and offer advice beyond the scope of the standards.

Since publication of the second edition of this book in 2006, more than 10 ISO GPS standards have been revised, involving the introduction of new symbols and concepts, and in many cases default interpretation of the tolerance indicators has changed; in addition, two new versions of American standard ASME Y14.5 (2009 and 2018) have appeared. All of these developments have been addressed in this new edition in full, together with changes to measuring equipment and technical information related to inspection methods.

Key Features

- Reflects the latest ISO standards up to 2019 and ASME Y14.5 –2018
- The rules and cases of geometrical tolerances are explained clearly with a wealth of examples and application cases presented with excellent technical drawings
- Manufacturing processes for specific tolerances are addressed
- A detailed chapter covers everything a practitioner needs to know about the inspection of geometrical tolerances

About the Author

Georg Henzold spent many years as manager of the department for standardization of a manufacturer for power plant machinery. He was also a long-standing chairman of the committee dealing with the standardization in the field of geometrical dimensioning and tolerancing in the German Standardization Institute DIN and in the European Committee for Standardization CEN. He is a long-time delegate in the pertinent committees of the International Standardization Organization (ISO).

ISBN 978-0-12-824061-8

9 780128 240618

ELSEVIER

Butterworth-Heinemann
An imprint of Elsevier
elsevier.com/books-and-journals