BS 8888:2020

BSI Standards Publication

Technical product documentation and specification

bsi.

...making excellence a habit.™

Publishing and copyright information

The BSI copyright notice displayed in this document indicates when the document was last issued.

Published by BSI Standards Limited 2019

ISBN 978 0 539 02682 5

ICS 01.100.01, 01.110

The following BSI references relate to the work on this document:
Committee reference TPR/1/8
Draft for comment 19/30384745 DC

Publication history

First published as BS 308–1, BS 308–2 and BS 308–3, September 1927

Second editions, December 1943

Third editions, December 1953

Fourth editions, November 1964

Fifth editions, October 1972

Sixth editions, August 1984 (BS 308–1), October 1985 (BS 308–2) and August 1990 (BS 308–3)

Seventh edition of BS 308 1, December 1993

First published as BS 8888 August 2000

Second edition, October 2002

Third edition, October 2004

Fourth edition, October 2006

Fifth edition, October 2008

Sixth edition, December 2011

Seventh edition, December 2013

Eighth edition, January 2017

Ninth (present) edition, January 2020

Amendments issued since publication

Date	Text affected

Contents

Summary of pages

This document comprises a front cover, an inside front cover, pages i to xiv, pages 1 to 197, and a back cover.

Foreword

Publishing information

This British Standard is published by BSI Standards Limited, under licence from The British Standards Institution, and came into effect on 31 January 2020. It was prepared by Technical Committee TDW/4/8, *BS 8888 Technical product specification*, under the authority of Technical Committee TDW/4, *Technical product realization*. A list of organizations represented on these committees can be obtained on request to their secretary.

Supersession

This British Standard supersedes BS 8888:2017, which is withdrawn.

Information about this document

This edition of the standard is a full revision. It introduces relevant international standards published since the 2017 edition. It also incorporates more fully than previous editions some of the fundamental requirements of the key international standards relevant to the preparation of technical product specifications, such as BS EN ISO 1101 and BS EN ISO 5459. It is hoped that UK industry will find this edition of BS 8888 more user-friendly than previous editions. It aims to help organizations better understand and implement the full complement of international standards developed by ISO/TC 213, *Geometrical product specifications and verification*, and ISO/TC 10, *Technical product documentation*.

This publication can be withdrawn, revised, partially superseded or superseded. Information regarding the status of this publication can be found in the Standards Catalogue on the BSI website at bsigroup.com/standards, or by contacting the Customer Services team.

Where websites and webpages have been cited, they are provided for ease of reference and are correct at the time of publication. The location of a webpage or website, or its contents, cannot be guaranteed.

Several figures are used with kind permission of Iain Macleod Associates Ltd.

Relationship with other publications

The function of BS 8888 is to draw together, in an easily accessible manner, the full complement of international standards relevant to the preparation of technical product specifications. However, it is not the intention for BS 8888 to be a “stand-alone” standard. TDW/4 is responsible for a suite of related national standards, including the various parts of BS 8887, BS 8889 and PD 68888, a new training document.

BS 8888 was taken up by the Ministry of Defence in 2006 as part of its DEF-STAN for defence project specification.

Presentational conventions

The provisions of this standard are presented in roman (i.e. upright) type. Its requirements are expressed in sentences in which the principal auxiliary verb is “shall”.

Commentary, explanation and general informative material is presented in smaller italic type, and does not constitute a normative element.

In addition, information boxes provide additional informative material that might otherwise have appeared in informative annexes but was felt to be best placed in the main body of text.

All dimensions shown in the figures in this British Standard are in millimetres.

Contractual and legal considerations

This publication does not purport to include all the necessary provisions of a contract. Users are responsible for its correct application.

Compliance with a British Standard cannot confer immunity from legal obligations.

Section 1 Scope

1.1 Scope

This British Standard specifies requirements for technical product documentation and specification, principally for manufacturing industries and organizations associated with engineering disciplines, such as mechanical, electrical, nuclear, automotive, aerospace and medical devices.

The needs of building, architectural, civil, and structural engineering and construction services industries continue to be covered by the BS 1192 series of standards.

Most of this British Standard consists of an implementation of the ISO system for technical product documentation and specification. The ISO system is defined in a large number of interlinked and related international standards which are referenced in this British Standard.

The purpose of this British Standard is to facilitate the use of the ISO system by providing:

- an index to the international standards which make up the ISO system, referencing them according to their area of application;
- key elements of the ISO standards to facilitate their application;
- references to additional British and European standards where they provide information or guidance over and above that provided by ISO standards; and
- commentary and recommendations on the application of the standards where this is deemed useful.

The requirements refer to international and European standards which have been implemented as British Standards, either in the BS EN, BS EN ISO, BS ISO series or as international standards renumbered as British Standards.

Annex A (normative) contains a list of normative references, indispensable for the application of this British Standard.

Annex B (informative) contains a list of informative references.

Annex C (informative) gives information about association.

Annex D (normative) contains requirements for enhanced document security.

Annex E (informative) gives information about the implementation of datums and datum features in CAD.

Annex F (informative) demonstrates former practice for tolerancing.

Annex G (informative) gives selected ISO fits for holes and shafts.

Annex H (informative) explains how data representing a manufactured surface are filtered by the measurement process.

Annex I (informative) gives further information about the dimensioning of slots.

1.2 Normative references

The documents listed in Annex A are referred to in the text in such a way that some or all of their content constitutes provisions of this document[1]. For dated references, only the edition cited applies.

---

[1] Documents that are referred to solely in an informative manner are listed in the Bibliography (Annex B).

For undated references, the latest edition of the referenced document (including any amendments) applies.

1.3 Terms and definitions

For the purposes of this British Standard, the terms and definitions given in BS EN ISO 10209 and BS EN ISO 14660-1 apply, together with the following.

1.3.1 date of issue

point in time at which the technical product specification is officially released for its intended use

NOTE 1 The date of issue is important for legal reasons, e.g. patent rights, traceability.

*NOTE 2 For the implications of the date of issue, see **3.1.3**.*

1.3.2 ISO GPS system

geometrical product specification and verification system developed in ISO by ISO/TC 213

1.3.3 technical product documentation (TPD)

means of conveying all or part of a design definition or specification of a product

1.3.4 technical product specification (TPS)

technical product documentation comprising the complete design definition and specification of a product for manufacturing and verification purposes

NOTE A TPS, which might contain drawings, 3D models, parts lists or other documents forming an integral part of the specification, in whatever format they might be presented, might consist of one or more TPDs.

Section 2 Standards underpinning BS 8888

2.1 General

The following documents shall be applied as “global” standards in support of BS 8888.

ISO/IEC Guide 98-3	*Guide to the expression of uncertainty in measurement (GUM)*
ISO/IEC Guide 99	*International vocabulary of metrology – Basic and general concepts and associated terms (VIM)*

The principles in Section 3 shall always be applied where conformity with BS 8888 is claimed.

Section 3 Technical product specification (TPS): Principles and concepts

3.1 Principles of specification

3.1.1 Types of technical product specification

3.1.1.1 2D TPS (2D drawing only)

COMMENTARY ON ***3.1.1.1***

This subclause relates to a 2D drawing in hardcopy or electronic format. Although 3D CAD can be used to derive the 2D drawing, the 3D model does not form any part of the TPS.

A 2D TPS shall consist of 2D drawing information only. The drawing(s) shall provide all the necessary views, sections, datums, dimensions, tolerances and general notation to provide suitable and sufficient information to manufacture and verify the workpiece.

NOTE A workpiece can consist of a component or an assembly.

3.1.1.2 Combined 3D and 2D TPS

COMMENTARY ON ***3.1.1.2***

This subclause relates to a 2D drawing (specification) with 3D CAD geometry (definition) which in combination fully specifies a component or assembly. A 3D/2D TPS uses both 3D models and 2D drawings in combination to provide suitable and sufficient information to manufacture and verify the workpiece.

A combined 2D and 3D TPS shall conform to BS ISO 16792.

In the absence of any dimension on the drawing, values shall be obtained from an interrogation of the 3D model geometry.

Unless otherwise stated, all tolerance requirements shall appear on the 2D drawing.

Unless otherwise stated, any dimension explicitly defined in any 2D TPS shall take precedence over any dimension obtained by interrogating the 3D model.

3.1.1.3 3D TPS (3D CAD model only)

COMMENTARY ON ***3.1.1.3***

This subclause relates to 3D CAD data in electronic format. There is no 2D drawing.

A 3D TPS shall conform to BS ISO 16792.

The 3D TPS shall fully specify all geometry, datums, dimensions and tolerances in order to provide suitable and sufficient information to manufacture and verify the workpiece. A 3D TPS shall consist of 3D model information only. The 3D model(s) shall provide the CAD nominal geometry and sufficient datums, dimensions, tolerances and general notation necessary to provide suitable and sufficient information to manufacture and verify the workpiece.

3.1.1.4 Working with 3D CAD data

COMMENTARY ON ***3.1.1.4***

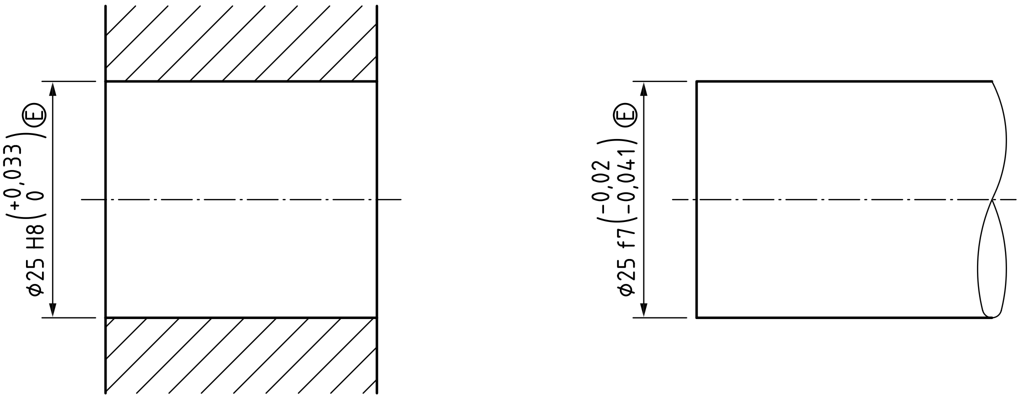
See BS ISO 16792.

Where the 3D CAD geometry includes features of size, two different practices are widely used. These are to model the feature at "nominal" size or to model the feature at "mid-limit" size (see Figure 1).

"Nominal" size is the dimension size to which the tolerance is applied. "Mid-limit" size is the size half-way through the tolerance range.

Where symmetrical size tolerances are used, "nominal" and "mid-limit" are the same. When non-symmetrical tolerances are used, such as limit and fit codes, they are different.

Figure 1 – *Hole and shaft sizes*



Hole

Nominal size: Ø25

Maximum material size: Ø25

Mid-limit size: Ø25,0165

Least material size: Ø25,033

Shaft

Nominal size: Ø25

Maximum material size: Ø24,98

Mid-limit size: Ø24,9695

Least material size: Ø24,959

When a nominal size is provided by means of a specification or dimension, all features of size can be modelled to nominal size or to mid-limit value of the tolerance limits. Where a nominal size is not provided, all features of size shall be modelled to the mid-limit value of the tolerance limits, unless otherwise stated (see Table 1).

Within a 3D CAD model, wherever non-symmetrical size tolerances are used, one practice shall be consistently followed.

Table 1 – *Mean and nominal values for 3D CAD model*

Dimension	Nominal value	Value to be modelled when working to nominal	Mid-limit value	Value to be modelled when working to "mid-limit"
Ø15 Ø13	Not stated	Ø14	Ø14	Ø14
20,4 19,6	Not stated	20	20	20
10 ±1	10	10	10	10
Ø25 H8Ⓔ	25	25	25,0165	Ø25,0165
Ø 25 f7Ⓔ	25	25	24,9695	Ø24,9695
$\varnothing 13{,}5^{+1{,}5}_{-0{,}5}$	13,5	13,5	14	Ø14
$\varnothing 14^{+1}_{0}$	14	14	14,5	Ø14,5
R0,5 max.	R0,5	R0,5	R0,25A)	R0,25

A) Min. value = 0.

3.1.2 General principles of specification

The specifier shall determine what level of detail is required in a specification and define the workpiece in an appropriate manner (see Table 2).

*COMMENTARY ON **3.1.2***

When a specification leaves certain requirements undefined, or open to more than one interpretation, it is ambiguous.

The more ambiguity which exists in a specification, the greater the risk that a manufactured product conforms to that specification but is still unfit for purpose. Conversely, there is also greater risk of rejecting a workpiece which is fit for purpose.

The less ambiguity which exists in a specification, the greater the likelihood that the conforming manufactured product is fit for purpose.

It is not always necessary to minimize or eliminate the ambiguity in a specification. A high level of ambiguity can be acceptable when the specifier is willing to rely on the skill, craftsmanship and knowledge of the manufacturer or fitter to achieve a satisfactory result. The benefit is a very simple specification, but the consequences could include a high degree of variation from one part to the next, a lack of interchangeability and inconsistent quality.

Higher levels of ambiguity also mean much greater probability of disputes over whether a manufactured product is acceptable or not.

The use of datums, geometrical tolerances, surface texture requirements and edge tolerances is not obligatory. However, not using these specification elements results in greater ambiguity.

Table 2 – *Level of detail in a TPS Technical product documentation – Digital product definition data*

Specification elements included	**Level of detail**			
	Sketch	**Technical model/ drawing**	**Technical model/drawing with geometrical tolerancing**	**ISO GPS specification**
3D/2D geometry	✓	✓	✓	✓
Dimensions		✓	✓	✓
+/− Tolerances		✓	✓	✓
Datums			✓	✓
Geometrical tolerances			✓	✓
Surface texture requirements				✓
Edge tolerances				✓
Level of ambiguity in the specification	**Very high**	**High**	**Low**	**Very low**

3.1.3 Date of issue principle

A TPS shall always be interpreted according to those versions of the standards which governed its interpretation on its date of issue, unless otherwise indicated.

3.1.4 Reference conditions

Unless otherwise stated, all geometrical properties and tolerances for a workpiece given in a TPS shall be considered to apply at 20 °C.

NOTE 1 See BS EN ISO 1.

By default, all GPS specifications shall apply at reference conditions, including the workpiece’s freedom from contaminants.

NOTE 2 See BS EN ISO 8015.

Any additional or other conditions that apply, e.g. humidity conditions, shall be defined on the drawing.

3.1.5 Interpretation

A TPS shall indicate which standards, or systems of standards, govern its interpretation.

When the format of data in a TPS conforms to the British and ISO standards identified in BS 8888, a reference to BS 8888 itself shall be sufficient to indicate that these standards apply to the specification. Such a reference shall take the following form.

CONFORMS TO BS 8888

NOTE 1 This note may be placed in the title block, in a drawing note, or elsewhere within the drawing frame.

NOTE 2 The phrase "format of data" refers to 2D and 3D TPS, and includes aspects such as format of attributes, layout of drawing borders and title blocks, layout of views and projections, format of letters, numbers, dimensions and tolerances, and use of different line types and thicknesses.

When the geometry of a workpiece is defined according to the requirements of ISO GPS (see **3.3**), which are documented in standards such as BS EN ISO 5459 and BE EN ISO 1101, a reference to BS EN ISO 8015 is sufficient to indicate that all ISO GPS standards apply to the interpretation of that specification. Such a reference shall take the following form.

TOLERANCING ISO 8015

NOTE 3 This note may be placed in the title block, in a drawing note, or elsewhere within the drawing frame. This note is required in addition to any reference to BS 8888.

This marking is a BS 8888 requirement, and not a requirement of BS EN ISO 8015. This is necessary to avoid possible misinterpretation of which system of standards govern the interpretation of a TPS.

3.1.6 Decimal principle

Non-indicated decimals shall be taken as zeroes.

NOTE 1 0,2 is the same as 0,200 000 000 000 000 000 000 000 000 000 ... etc.

NOTE 2 10 is the same as 10,000 000 000 000 000 000 000 000 000 000 ... etc.

3.1.7 Rigid workpiece principle

Unless otherwise stated, a workpiece shall be considered as having infinite stiffness, and all geometrical properties and tolerances given in a TPS shall be considered to apply in the absence of deformation from any external forces (including gravity). Any additional or other conditions that apply shall be defined in the drawing.

3.1.8 Non-rigid workpieces

A non-rigid workpiece is one that deforms to an extent that in the free state it is beyond the dimensional and/or geometrical tolerances on the drawing. A non-rigid workpiece shall therefore have the free-state deformation removed for verification by applying pressure or forces that replicate, but do not exceed, those expected under normal assembly conditions.

NOTE 1 This is known as the "restrained condition". The restrained condition normally corresponds to the assembly condition but might also correspond to certain operational conditions. Dimensions, tolerances and geometrical tolerances may be applied to either the free state or to the restrained condition, or to both.

NOTE 2 Free state is the condition of a part subjected only to the force of gravity.

NOTE 3 This is particularly important when aligning and/or measuring large and/or flexible constructions.

A TPS for a non-rigid workpiece shall include:

a) the indication ISO10579-NR near the title block or in the notes;

b) a definition of restraint, e.g. use of tooling, application of forces or pressure, and if necessary an indication of the direction of gravity;

c) if required, a definition of the condition under which free-state tolerances apply, e.g. orientation and direction of gravity.

When ISO 10579-NR is indicated on the drawing, all dimensions, tolerances and geometrical tolerances shall apply to the restrained condition unless otherwise stated. Any dimensions, tolerances and geometrical tolerances which apply to the free state shall be identified with the Ⓕ symbol (see *7.10.2*).

3.2 Fundamental concepts

3.2.1 Properties

A TPS defines various properties and requirements. Some properties or requirements may be defined for an entire workpiece (e.g. a material specification), some for part of a workpiece (e.g. a particular surface treatment), and some for individual features on a workpiece (e.g. a size requirement).

Properties or requirements may also be defined for a process or procedure.

Properties which may be specified include:

- size;
- location;
- orientation;
- form;
- surface texture;
- surface imperfections;
- properties of edges;
- material;
- coatings and finishes;
- hardness;
- chemical composition;
- treatments which are to be applied to the workpiece;
- processing requirements;
- recycling requirements;
- packaging requirements; and
- handling requirements.

Properties of location and orientation may be defined solely with toleranced dimensions (see Section 5), but this results in ambiguous specifications which are open to a wide range of interpretations.

For an unambiguous definition of the size, location, orientation and form of features on the workpiece, the ISO system of Geometrical Product Specification shall be used.

NOTE This utilizes datums and geometrical tolerances to minimize ambiguity (see Section 6 and Section 7).

3.2.2 Feature principle

A workpiece shall be considered to be made up of a number of features limited by natural boundaries. By default, every specification requirement for a feature or relation between features shall apply to the entire feature and each specification requirement shall apply only to one feature or one relation between features.

NOTE This default can only be overridden by explicit indications on the drawing. The system for classifying features is outlined in the following information box.

Classification of features

ISO GPS defines several different types of feature. The main types of features are:

a) integral features: features which consist of, or represent, surfaces on the workpiece;

b) features of size: integral features which have a size characteristic and include (but are not limited to) the following types of feature:

   1) two parallel opposed planes;

   2) a cylinder;

   3) a sphere;

   4) two non-parallel planes (a wedge);

   5) a cone;

   6) the two rounded ends of a slot or protrusion (see Annex I); and

   7) a torus;

c) derived features: theoretical features such as axes or median lines, which are derived from a feature of size (axis or median line, median plane or median surface and centre points).

ISO GPS also defines a number of different "worlds" or "states" in which features can exist. In some "worlds" the features exist in an ideal form, such as in the specification, while in other states they exist in a non-ideal form, such as surfaces on the manufactured workpiece (see Figure 2).

Nominal features

In the "world" of the specification, features are represented in an ideal state. These features have ideal form, and ideal relationships with each other. Features in this world are known as nominal features.

Real features

In the "world" of the manufactured workpiece (the physical world), features are non-ideal and are known as real features.

Extracted features

In the "world" of verification, a feature is represented by a set of data obtained by sampling the workpiece with measuring instruments. This set of data represents the non-ideal real feature and is known as an extracted feature.

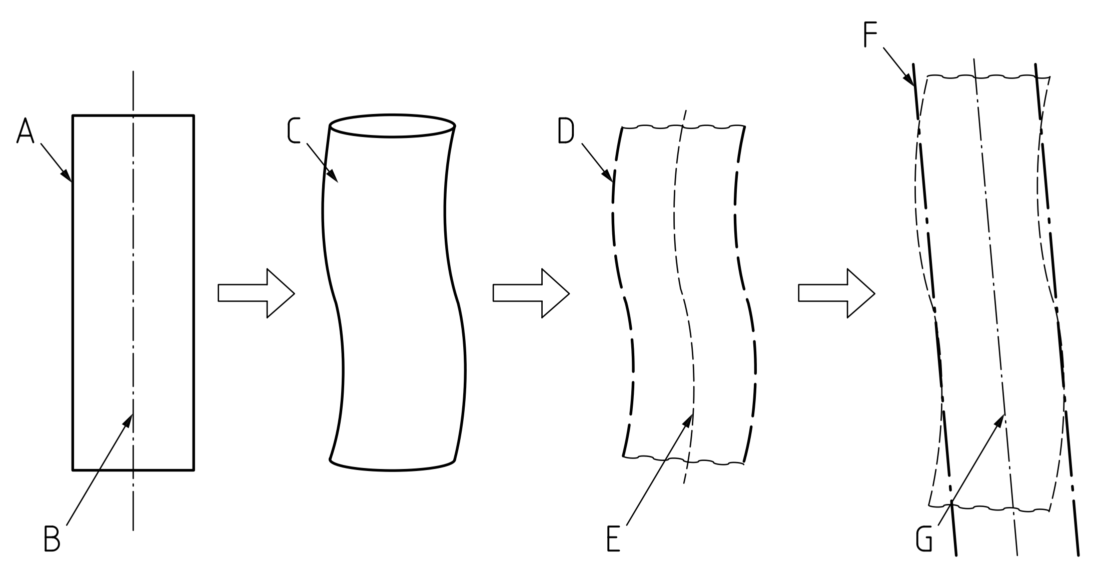
NOTE 1 The extracted feature is used when determining whether a tolerance requirement has been met.

Associated features

In the "world" of verification, an ideal feature may be "fitted" to, or associated with, either the real feature or the extracted feature. This ideal feature is known as an associated feature.

NOTE 2 The associated feature is used when determining the orientation of size measurements, or when defining a datum. A range of different association methods may be used.

Figure 2 – *Interrelationship of the geometrical feature definitions*



Key

A Nominal integral feature

B Nominal derived feature

C Real feature

D Extracted integral feature

E Extracted derived feature

F Associated integral feature

G Associated derived feature

3.2.3 Interpretations of limits of size for a feature of size

3.2.3.1 Relevant standards

Limits of size for a feature of size shall be interpreted according to the principles and rules defined in the following standards.

BS EN ISO 8015	*Geometrical product specifications (GPS) – Fundamentals – Concepts, principles and rules*
BS EN ISO 14405-1	*Geometrical product specifications (GPS) – Dimensional tolerancing – Part 1: Linear sizes*
BS EN ISO 14405-2	*Geometrical product specifications (GPS) – Dimensional tolerancing – Part 2: Dimensions other than linear or angular sizes.*
BS EN ISO 14405-3	*Geometrical product specifications (GPS) – Dimensional tolerancing – Part 3: Angular sizes*
BS EN ISO 17450-1	*Geometrical product specifications (GPS) — General concepts — Part 1: Model for geometrical specification and verification*
BS EN ISO 17450-2	*Geometrical product specifications (GPS) — General concepts — Part 2: Basic tenets, specifications, operators, uncertainties and ambiguities*
BS EN ISO 17450-3	*Geometrical product specifications (GPS) — General concepts — Part 3: Toleranced features*
BS EN ISO 22432	*Geometrical product specifications (GPS) — Features used in specification and verification*

3.2.3.2 Interpretations of limits of size for a linear feature of size

A linear size specification shall be interpreted as a two-point size requirement unless otherwise indicated.

When a different specification operator is required, it shall be invoked using the appropriate modifier applied to the individual size specification, or by defining a different default linear size specification operator for the entire TPS (see BS EN ISO 14405-1).

If the default size specification operator is to be changed for an entire TPS, the following indication shall appear in or near the title block of each drawing:

LINEAR SIZE ISO 14405 X

where X is the modifier representing the required operator.

For example, if the envelope requirement is to be made the default interpretation of linear size specifications for the entire TPS, the indication shall be:

LINEAR SIZE ISO 14405 Ⓔ

NOTE 1 It is recommended that a note on the drawing be used in addition to the ISO 14405 reference, for clarity.

The consequence of using the default "two-point" definition of a linear size tolerance is that size limits only apply to any "two-point" size measurement of a feature. Therefore, the size limits do not control the form deviations of the feature. For example:

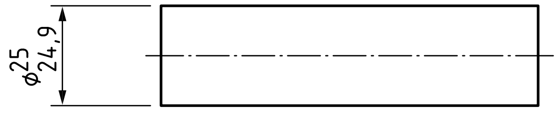
- the size limits on a cylindrical feature do not control its roundness or straightness; and
- the size limits on a feature consisting of two parallel, opposed planes do not control the flatness of the two planes.

Form deviations can be controlled by:

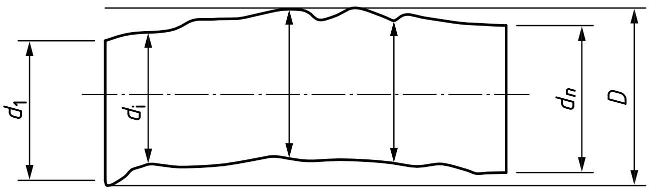
- individually specified geometrical tolerances;
- general geometrical tolerances;
- the use of the envelope requirement, Ⓔ [where the maximum material limit of size defines an envelope of perfect form for the relevant surfaces (see BS EN ISO 14405-1)]; or
- the use of another global size characteristic (see BS EN ISO 14405-1).

NOTE 2 Figure 3 gives examples of how size limits could be interpreted where no form control is defined and the specification is incomplete.

Figure 3 – *Possible interpretations of size limits where no form control is defined and the specification is incomplete*



a) Drawing presentation



Local sizes d_i lie within the range Ø25 to Ø24,9.

No global size requirement is specified, so D can be greater than Ø25.

No form control is specified, so properties such as roundness and straightness can vary greatly.

b) Size limit being met, but no control over the form of the feature

Some national standards apply (or have applied in the past) the envelope requirement to all linear features of size by default. As the envelope requirement has been the default, they have not used a symbol to indicate this requirement; rather, they use (or used) a note to indicate when this is not required. This system of tolerancing is sometimes described as the principle of dependency, or the application of the Taylor Principle.

Standards which apply, or have applied, the envelope requirement by default include:

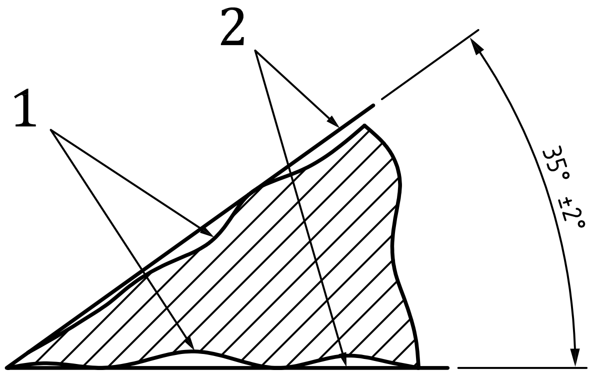
- BS 308[2];
- ASME Y14.5 [1][3]; and
- DIN 7167.

3.2.3.3 Interpretations of limits of size for an angular feature of size

An angular size specification shall be interpreted as a two-line angular size requirement, with minimax association, unless otherwise indicated.

NOTE 1 For angular size, the angle applies to an ideal line or plane fitted (associated) to the manufactured surface. "Minimax" association (also known as a "Chebyshev" association) is a method used for fitting an ideal feature to a manufactured surface (see Figure 4). The minimax association works by minimizing the maximum deviations between the ideal feature and the manufactured feature. It can be further modified by applying a material constraint such as "outside the material" or "inside the material", but as this makes no difference to the orientation of the ideal feature, no material constraint is specified when evaluating angular sizes.

Figure 4 – *Two-line angular size: Second association with minimax criterion*



Key

1 Extracted lines

2 Associated lines

When a different specification operator is required, it shall be invoked using the appropriate modifier applied to the individual angular size specification, or by defining a different default angular size specification operator for the entire TPS (see BS EN ISO 14405-3).

If the default size specification operator is to be changed for an entire TPS, the following indication shall appear in or near the title block of each drawing:

ANGULAR SIZE ISO 14405 X

where X is the modifier representing the required operator.

---

[2] Withdrawn

[3] The requirement that there is an envelope of perfect form corresponding to the maximum material size of the feature is defined as Rule #1 in ASME Y14.5 [1].

For example, if the global angular size with least squares association is to be made the default interpretation of angular size specifications for the entire TPS, the indication shall be:

ANGULAR SIZE ISO 14405 H

NOTE 2 It is recommended that a note on the drawing be used in addition to the ISO 14405 reference, for clarity.

3.2.3.4 Implied dimensions

NOTE Implied dimensions are dimensional requirements which are not explicitly indicated on a 2D engineering drawing, but which have the same meaning as a dimensional requirement which is explicitly stated.

The following rules shall govern the use and interpretation of implied annotation on engineering drawings.

These rules do not apply to 3D specifications; where all dimensional requirements are defined in the model, implied dimensions shall not be used where there is a significant risk of misinterpretation.

a) Where two features are aligned, there is no requirement to indicate a linear dimension of 0 or an angular dimension of 0°.

b) Where two features are parallel to each other, there is no requirement to indicate an angular dimension of 0° or 180°.

c) Where two features are at 90° to each other, there is no requirement to indicate an angular dimension of 90°.

d) Where several features are equispaced around a pitch circle (see BS ISO 129-1), there is no requirement to indicate an angular dimension, although it might be advisable to do so. Terms such as “equispaced” and “equally spaced” shall not be used.

e) If the features concerned have their locations and/or orientations controlled through the use of geometrical tolerances, the implied dimensions shall be taken as theoretically exact dimensions (TEDs; see **7.8**).

f) If the features concerned have their locations and/or orientations controlled through the use of +/− or limit tolerances, the implied dimensions shall also be toleranced. In the absence of other indications, they shall be subject to a general tolerance, or else the TPS would remain incomplete (see **5.4**).

Tolerances can never be implied and shall always be indicated.

Datums can never be implied and shall always be indicated.

3.3 Geometrical product specification

3.3.1 Interpretation and invocation principle

Once a portion of the ISO GPS system is invoked on a mechanical engineering product specification, the entire ISO GPS system shall be invoked, unless otherwise indicated on the specification e.g. by reference to a relevant document.

“Unless otherwise indicated on the specification” means that it is indicated on the specification that this has been prepared in accordance with a certain standard. That standard, and not the ISO GPS system, shall be used to interpret those elements of the specification which are covered by that standard.

NOTE 1 The ISO GPS system is defined in a hierarchy of standards that includes the following types of standards in the given order:

- *fundamental GPS standards;*
- *general GPS standards; and*
- *complementary GPS standards.*

See BS EN ISO 14638 for further details.

The "entire ISO GPS system shall be invoked" means that fundamental and general GPS standards shall apply and, consequently, that the reference temperature given in BS EN ISO 1 and the decision rules given in BS EN ISO 14253-1 shall apply, unless otherwise indicated.

NOTE 2 The purpose of the invocation principle is to provide the formal traceability for these GPS standards and rules.

The rules given in standards at a higher level in the hierarchy shall apply in all cases, unless rules in standards at lower levels in the hierarchy specifically give other rules.

All rules given in fundamental GPS standards shall apply, in addition to those specifically given in general GPS standards, e.g. BS EN ISO 1101, except where the rules in the general GPS standard are explicitly different from the rules given in fundamental GPS standards and unless the rules in a specific complementary GPS standard give other rules that apply within its scope.

All rules given in fundamental and general GPS standards shall apply in addition to the rules specifically given in complementary GPS standards, e.g. BS EN 22768-1, except in the cases where the rules in the complementary GPS standard are explicitly different from the rules given in fundamental and general GPS standards.

3.3.2 Independency principle

By default, every GPS requirement for a feature or relation between features shall be fulfilled independently of other requirements, except when it is stated in a standard or by special indication (e.g. modifiers Ⓜ or Ⓛ according to BS EN ISO 2692 or CZ or CZR according to BS EN ISO 5458) as part of the actual specification.

Section 4 Technical product documentation (TPD)

Various types of documents are used to provide appropriate technical information to define the features and requirements for a component or assembly (see Section 3). A parts list can be included on the drawing. The most commonly used document types are as follows (see BS ISO 29845 for further document types).

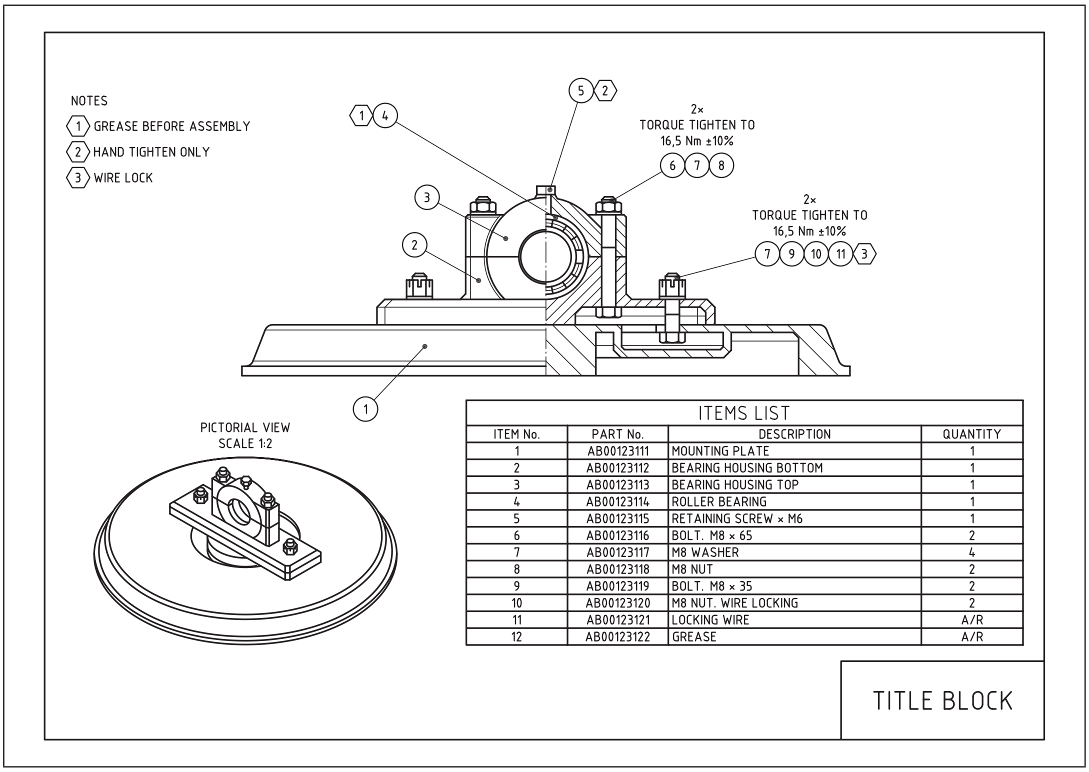
Assembly drawing

An assembly drawing (see Figure 5) represents the relative position and/or shape of a group of assembled parts. It depicts the constituents of a parts list.

The list of parts is normally provided in a separate document, i.e. a parts list, but can be included within the drawing, with the individual parts annotated with ballooned numbers matching the item numbers in the parts list.

Relevant information can also be added, torque settings can either be shown adjacent to the item balloon or in a torque table, and other information, e.g. lubricating grease, thread adhesive, can be shown alongside corresponding flag notes. Overall and key dimensions can also be included, along with mass and centre of gravity.

Figure 5 – *Example of assembly drawing*



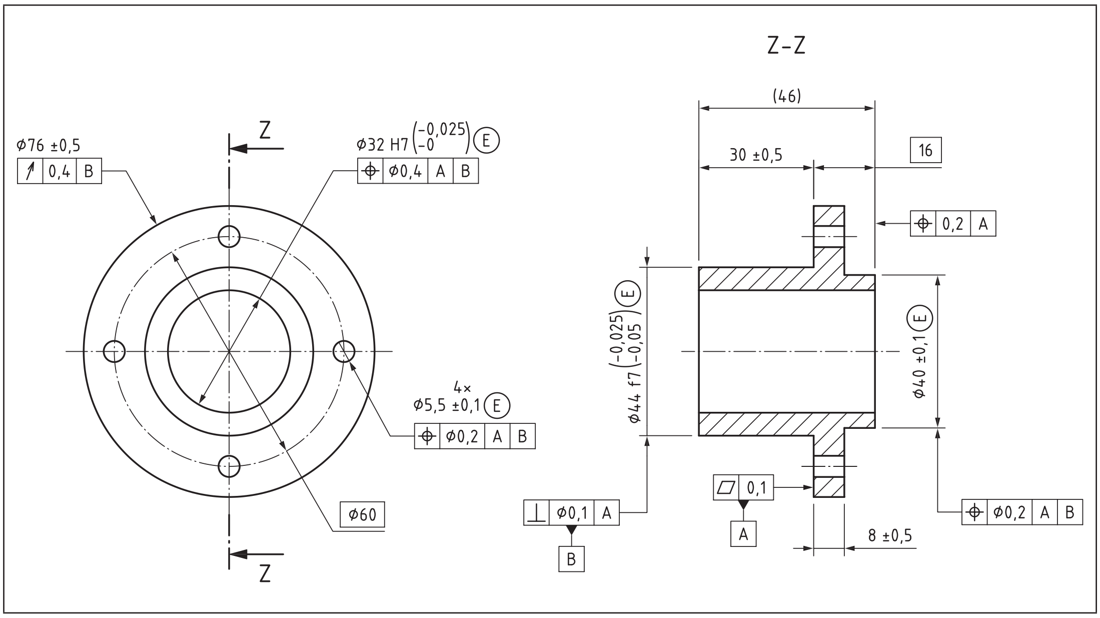
ITEMS LIST

ITEM No.	PART No.	DESCRIPTION	QUANTITY
1	AB00123111	MOUNTING PLATE	1
2	AB00123112	BEARING HOUSING BOTTOM	1
3	AB00123113	BEARING HOUSING TOP	1
4	AB00123114	ROLLER BEARING	1
5	AB00123115	RETAINING SCREW × M6	1
6	AB00123116	BOLT. M8 × 65	2
7	AB00123117	M8 WASHER	4
8	AB00123118	M8 NUT	2
9	AB00123119	BOLT. M8 × 35	2
10	AB00123120	M8 NUT. WIRE LOCKING	2
11	AB00123121	LOCKING WIRE	A/R
12	AB00123122	GREASE	A/R

Part drawing

A part drawing includes all the necessary information required for the definition of the part, e.g. material properties, dimensions, tolerances, surface texture (see Figure 6 over page).

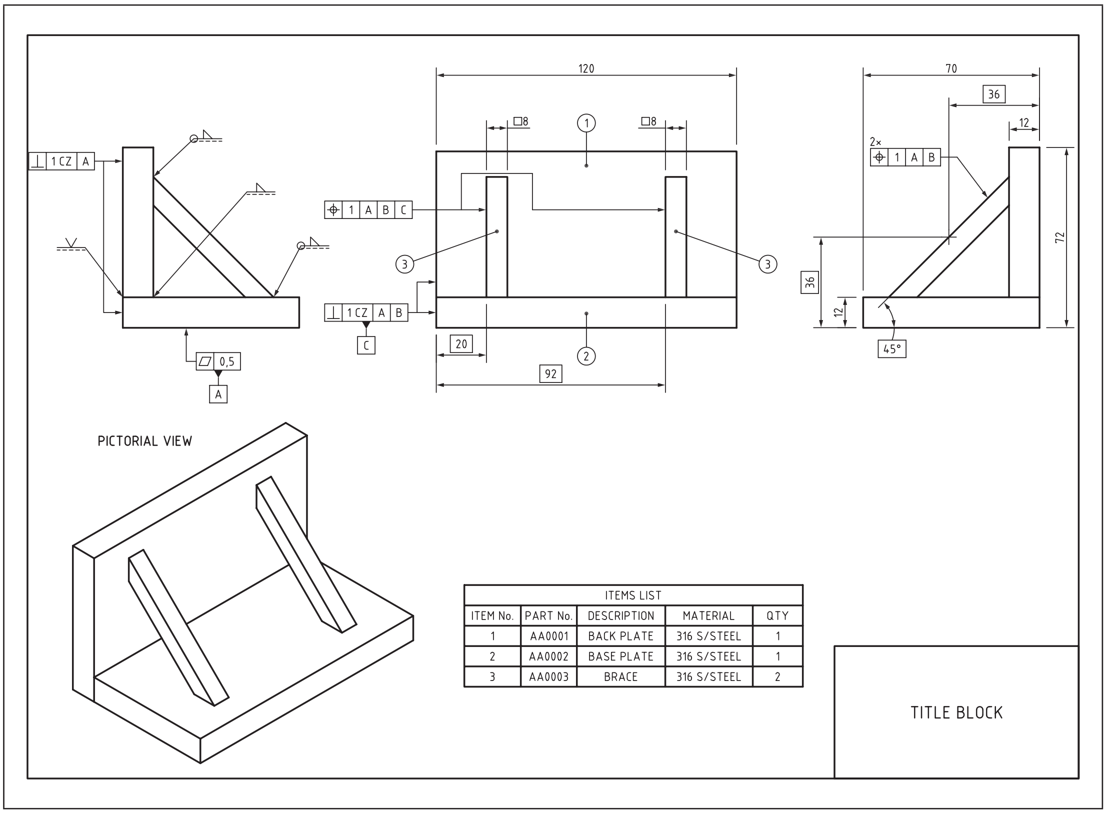
Figure 6 – *Example of part drawing*



Fabrication drawing

A fabrication drawing depicts a workpiece which is permanently joined together by means of welding, brazing, adhesion or other method. The constituents need to be fully specified (see Figure 7).

Figure 7 – *Example of fabrication drawing*



ITEMS LIST				
ITEM No.	PART No.	DESCRIPTION	MATERIAL	QTY
1	AA0001	BACK PLATE	316 S/STEEL	1
2	AA0002	BASE PLATE	316 S/STEEL	1
3	AA0003	BRACE	316 S/STEEL	2

4.1 Graphical representation and annotation of 3D data (3D modelling output)

Graphical representation and annotation of 3D models shall conform to the following standard.

BS ISO 16792, *Technical product documentation – Digital product definition data practices*

NOTE See Section 3.

4.2 Drawing sheets

*COMMENTARY ON **4.2***

The complete drawing sheet comprises the drawing space, title block, border and frame.

4.2.1 General

The drawing sheet shall conform to BS EN ISO 5457 and BS EN ISO 7200.

4.2.2 Sizes

As required by BS EN ISO 5457, the original drawing shall be made on the smallest sheet permitting the necessary clarity and resolution.

NOTE 1 The preferred sizes of the trimmed and untrimmed sheets, as well as the drawing space, are given in Table 3.

*NOTE 2 The range of sheet sizes chosen could be rationalized through the use of scales (see **4.4**).*

Table 3 – *Sizes of trimmed and untrimmed sheets and the drawing space*[A)]

Designation	Trimmed sheet (T)		Drawing space		Untrimmed sheet (U)	
	a_1[B)]	b_1[B)]	a_2 ±0,5	b_2 ±0,5	a_3 ±2	b_3 ±2
A0[C)]	841	1 189	821	1 159	880	1 230
A1	594	841	574	811	625	880
A2	420	594	400	564	450	625
A3	297	420	277	390	330	450
A4 landscape	210	297	190	267	240	330
A4 portrait	297	210	277	180	330	240

[A)] Dimensions in millimetres.

[B)] For tolerances see BS EN ISO 216.

[C)] For sizes >A0 see BS EN ISO 216.

4.2.3 Graphical features

4.2.3.1 Title block

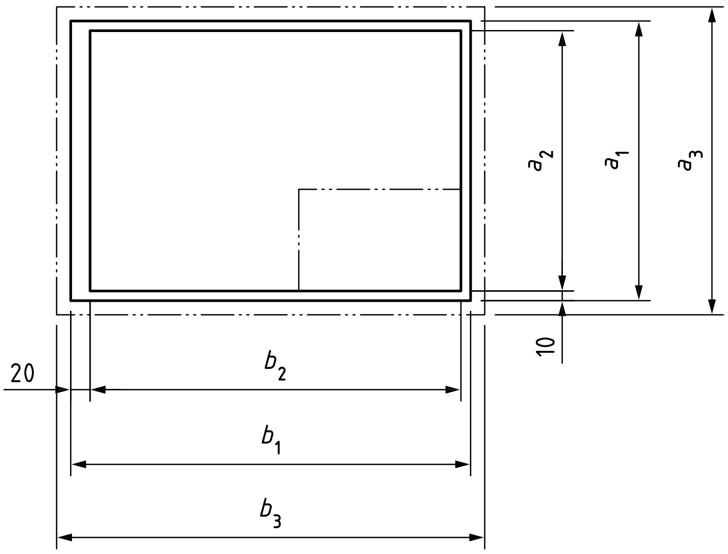
NOTE 1 For the dimensions and layout of title blocks, see BS EN ISO 7200.

Sheet sizes A0 to A3 shall be used in landscape orientation only (see Figure 8) and the title block shall be located in the bottom right-hand corner of the drawing space.

NOTE 2 A4 sheets can be used in landscape or portrait orientation.

For sheet size A4, the title block shall be situated in the bottom right-hand corner when used in landscape orientation, or the shorter (bottom) part of the drawing space when used in portrait orientation.

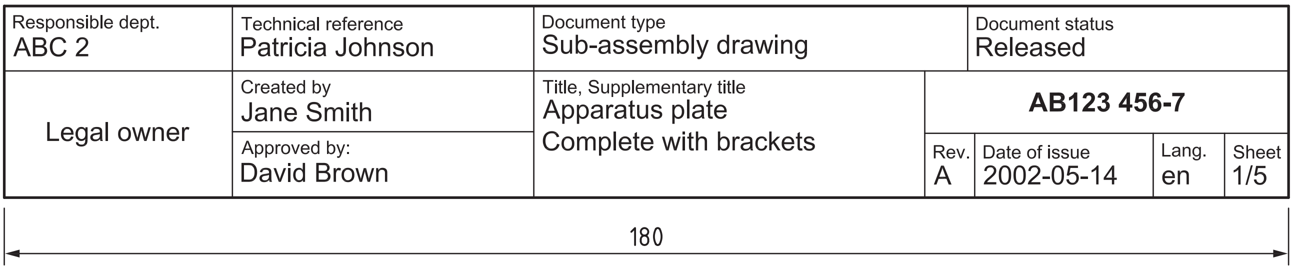
Figure 8 – ***Size A4 to A0***



The total width of the title block shall be 180 mm to fit an A4 portrait sheet where the left margin is 20 mm and the right margin 10 mm. The same title block shall be used for all paper sizes (see Figure 9).

NOTE 3 In a TPS there is a minimum of eight mandatory data fields which can appear in the title block or elsewhere in the TPS or data management system (see Table 4).

Figure 9 – ***Title block in compact form***



Responsible dept.
ABC 2

Technical reference
Patricia Johnson

Document type
Sub-assembly drawing

Document status
Released

Legal owner

Created by
Jane Smith

Approved by:
David Brown

Title, Supplementary title
Apparatus plate
Complete with brackets

AB123 456-7

Rev.
A

Date of issue
2002-05-14

Lang.
en

Sheet
1/5

Table 4 – ***Identifying, descriptive and administrative data fields in the title block***

Field name	Language-dependent	Recommended number of characters	Field type
Legal owner	–	Unspecified	Identifying
Identification number	No	16	Identifying
Date of issue	No	10	Identifying
Segment/sheet number	No	4	Identifying
Title	Yes	25/30[A)]	Descriptive
Approval person	No/Yes[B)]	20	Administrative
Creator	No/Yes[B)]	20	Administrative
Document type	Yes	20	Administrative

[A)] 30 to support two-byte-character languages such as Japanese or Chinese.

[B)] "Yes" to support presentation in different types of alphabet.

4.2.3.2 Borders and frame

Borders enclosed by the edges of the trimmed sheet and the frame limiting the drawing space shall be provided with all sizes. The border shall be 20 mm wide on the left edge, including the frame, and it can be used as a filing margin. All other borders shall be 10 mm wide.

The frame for limiting the drawing space shall be executed with continuous wide lines.

4.2.3.3 Grid reference system

The sheets shall be divided into fields in order to permit easy location of details, additions, revisions, etc., on the drawing. The individual fields shall be referenced using capital letters (I and O shall not be used) in the left and right borders, and numerals in the top and bottom borders. For the size A4, they shall be located only at the top and the right side. The start of the ascending sequences of letters and numerals shall begin at the sheet origin, which can either be determined manually or defined by the CAD tool.

The size of letters and characters shall be 3.5 mm. The length of the fields shall be 50 mm, starting either at the sheet origin or from the centring mark.

NOTE The number of fields depends on sheet size (see Table 5).

The letters and numerals shall be placed in the grid reference border and shall be written in vertical characters in accordance with BS EN ISO 3098-2.

The number of fields shall be extended as required for non-standard sheet sizes.

The grid reference system lines shall be executed with continuous narrow lines.

Table 5 – *Number of fields*

Designation	**A0**	**A1**	**A2**	**A3**	**A4**
Long side	24	16	12	8	6
Short side	16	12	8	6	4

4.3 Line types and line widths

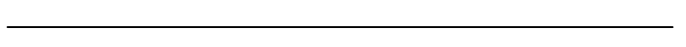
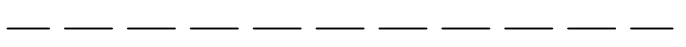
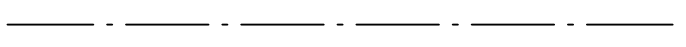
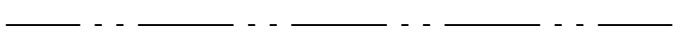
4.3.1 Types of line

Lines shall conform to BS EN ISO 128-20.

NOTE 1 Examples of the most common line type and width combinations for engineering drawings are given in Table 6.

NOTE 2 See BS EN ISO 128-20 and BS ISO 128-24 for more information on lines.

Table 6 – *Basic line types*

No.	Representation	Description	Application
01.1		Continuous narrow line	Dimension lines, extension lines, leader lines, hatching, roots of screw threads, termination of interrupted views
01.2		Continuous wide line	Visible edges, visible outlines, crests of threads, limit of thread length, section arrows
02.1		Dashed narrow line	Hidden edges, hidden outlines
02.2		Dashed wide line	Permissible areas of surface treatment
04.1		Long dashed-dotted narrow line	Centre lines, lines and planes of symmetry, pitch circle of gears or holes, spread of surface-hardened areas
04.2		Long dashed-dotted wide line	Position of cutting planes, restricted area for surface treatment or application of tolerance requirement
05.1		Long dashed-double-dotted narrow line	Outline of adjacent parts, extreme positions of moveable parts, initial outline prior to forming, projected tolerance zones and outline of datum target areas

NOTE Example applications in the use of line types can be found in Figure 5, Figure 6, Figure 8 and Figure 15 to Figure 18.

4.3.2 Line width

In accordance with BS EN ISO 128-20, the width, *d*, of all types of line shall be one of the following, depending on the type and size of drawing.

0,13 mm, 0,18 mm, 0,25 mm, 0,35 mm, 0,5 mm, 0,7 mm, 1 mm, 1,4 mm, 2 mm.

NOTE 1 The widths of extra wide, wide and narrow lines are in the ratio 4:2:1.

NOTE 2 For mechanical engineering drawings, two line thicknesses (typically 0,7 and 0,35 or 0,5 with 0,25) are sufficient for most purposes.

The line width of any one line shall be constant throughout the whole line (see also **4.7.2**, Table 8).

4.3.3 Colours

According to the requirements of BS EN ISO 128-20, lines shall be drawn black or white depending on the background. Other standardized colours can also be used for drawing standardized lines and, in this case, the meaning of the colours shall be explained.

4.4 Scales

*COMMENTARY ON **4.4***

With the advent of CAD systems and the ability to view drawings electronically at any size, the importance of using a standard range of scales has greatly diminished.

Traditionally, the recommended scales for use on technical drawings were as set out in Table 7.

Table 7 – *Scales*

Category	**Recommended scales**		
Enlargement scales	50:1	20:1	10:1
	5:1	2:1	
Full size	1:1	1:1	1:1
Reduction scales	1:2	1:5	1:10
	1:20	1:50	1:100
	1:200	1:500	1:1 000
	1:2 000	1:5 000	1:10 000

The scale for a drawing shall be chosen according to the complexity of the object to be depicted and the purpose of the representation. In all cases, the selected scale shall be large enough to permit easy and clear interpretation of the information depicted. The scale and the size of the object, in turn, shall determine the size of the drawing. Where no scaleable views are presented, for example a chart drawing or item list, the term N/A ("Not applicable") shall be indicated in the title block scale field.

Details that are too small for complete dimensioning in the main representation shall be shown adjacent to the main representation in a separate detail view (or section) which is drawn to a larger scale.

3D models produced on CAD systems shall always be produced at 1:1.

NOTE For more information on scales, see BS EN ISO 5455.

4.5 Lines

4.5.1 Lines and terminators

Lines shall conform to the following standards, as appropriate.

BS EN ISO 128-20	*Technical drawings – General principles of presentation – Part 20: Basic conventions for lines*
BS EN ISO 128-21	*Technical drawings – General principles of presentation – Part 21: Preparation of lines by CAD systems*
BS ISO 128-22	*Technical drawings – General principles of presentation – Part 22: Basic conventions and applications for leader lines and reference lines*
BS ISO 128-23	*Technical drawings – General principles of presentation – Part 23: Lines on construction drawings*
BS ISO 128-24	*Technical drawings – General principles of presentation – Part 24: Lines on mechanical engineering drawings*
BS ISO 128-25	*Technical drawings – General principles of presentation – Part 25: Lines on shipbuilding drawings*

4.5.2 Lines, terminators and origin indicators

Arrows and terminators composed of lines shall conform to the following standard.

BS ISO 129-1	*TPD – Presentation of dimensions and tolerances – Part 1: General principles*

4.6 Number formats

4.6.1 General

Numbers used in dimensions and tolerances shall conform to 4.6.2. Where a “thousands” separator is required, a space shall be used; this can be used with metric and imperial numbers, and both before and after the decimal marker.

4.6.2 Metric values

The decimal marker shall be a comma.

Numbers smaller than 1 shall have a zero in front of the decimal marker:

e.g. 0,25 not ,25.

Non-indicated decimals shall be taken as zeros; i.e. 0,25 is the same as 0,250 000 000 000 000 000 .... and not rounded up to 0,25.

Trailing zeros shall be omitted:

e.g. 16 not 16,0.

A note on imperial numbers:

ISO standards make no provision for the use of imperial units, so when they are used, it is recommended that the American ASME Y14.5 [1] standard conventions are used. ASME Y14.5 [1] specifies the following requirements for imperial (inch) dimension and tolerance values:

- *the decimal marker shall be a full stop;*
- *values smaller than 1 shall not have a zero in front of the decimal marker, e.g.:*

  Ø.25 not Ø0.25;
- *trailing zeros shall be added where necessary, to ensure that the dimension and tolerance have the same number of decimal places, e.g.:*

  .500±.005.

4.7 Lettering

4.7.1 General

Lettering shall conform to the following standards, as appropriate.

BS EN ISO 3098-0	*Technical Product Documentation – Lettering – Part 0: General requirements*[4]
BS EN ISO 3098-2	*Technical product documentation – Lettering – Part 2: Latin alphabet, numerals and marks*
BS EN ISO 3098-3	*Technical product documentation – Lettering – Part 3: Greek alphabet*
BS EN ISO 3098-4	*Technical product documentation – Lettering – Part 4: Diacritical and particular marks for the Latin alphabet*
BS EN ISO 3098-5	*Technical product documentation – Lettering – Part 5: CAD lettering of the Latin alphabet, numerals and marks*
BS EN ISO 3098-6	*Technical product documentation – Lettering – Part 6: Cyrillic alphabet*

---

[4] Withdrawn.

4.7.2 Lettering size

Lettering heights and line widths shall be appropriate for the sheet size and method of reproduction.

NOTE Two types of lettering are described within BS EN ISO 3098-0[5]*, type A and type B. Upper case characters of type B have a height of 10 times the line width. Type A take the same character width as type B but with the height increased by a factor of* $\sqrt{2}$*. BS ISO 129-1 recommends the use of type B lettering. Preferred sizes are summarized in Table 8.*

Table 8 – ***Preferred line widths and lettering heights for drawing sheet sizes***

Sheet size designation	**A0**	**A1**	**A2**	**A3**	**A4**
Line group (wide line)	0,7	0,7	0,7	0,5	0,5
Narrow line	0,35	0,35	0,35	0,25	0,25
Type B lettering height	3,5	3,5	3,5	2,5	2,5
Type B lower case height	2,5	2,5	2,5	1,75	1,75
Type B alternative height (line width)	5,0 (0,5)	5,0 (0,5)	5,0 (0,5)	3,5 (0,35)	3,5 (0,35)

4.7.3 Notes

Notes of a general nature shall, wherever practicable, be grouped together and not distributed over the drawing. Notes relating to specific details shall appear near the relevant feature, but not so near as to crowd the view.

Where emphasis is required, larger characters shall be used.

NOTE Underlining of notes is not recommended.

4.8 Projections

4.8.1 General

Projections shall conform to one of the following standards.

BS EN ISO 5456-2	*Technical drawings – Projection methods – Part 2: Orthographic representations*
BS EN ISO 5456-3	*Technical drawings – Projection methods – Part 3: Axonometric representations*
BS ISO 5456-4	*Technical drawings – Projection methods – Part 4: Central projection*
BS EN ISO 10209	*Technical product documentation – Vocabulary – Terms relating to technical drawings, product definition and related documentation*

NOTE BS EN ISO 5456-1 contains a survey of the various projection methods.

4.8.2 Conventions for arrangement of views on a TPD

4.8.2.1 General

Three main conventions shall be used for arranging the views on a TPD:

a) labelled views (see **4.8.2.3**);

b) first angle orthographic projection (see **4.8.2.4**);

c) third angle orthographic projection (see **4.8.2.5**).

NOTE 1 The order of this list is not meant to indicate a preference.

NOTE 2 Other projection methods exist. See BS EN ISO 5456 (all parts).

---

[5] Withdrawn.

4.8.2.2 Choice of views

When views (including sections and sectional views) are needed, these shall be selected according to the following principles.

a) The number of views (and sections and sectional views) shall be limited to the minimum necessary but shall be sufficient to fully delineate the object without ambiguity.

b) The need for hidden outlines and edges shall be avoided.

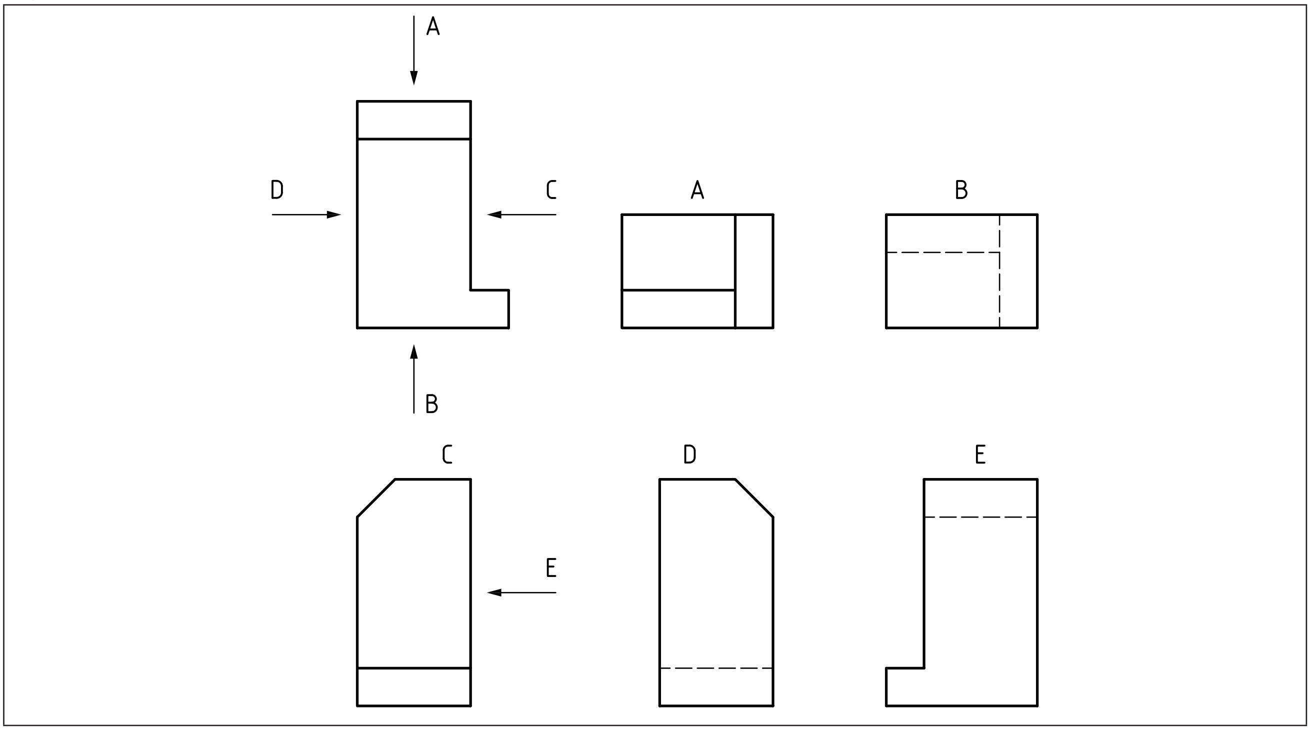
c) The unnecessary repetition of a detail shall be avoided.

4.8.2.3 Labelled view method

As required by BS ISO 128-30, the most informative view of an object shall be used as the front or principal figure, taking into consideration, for example, its functioning position, position of manufacturing or mounting. Each view, with the exception of the front or principal figure (view, plan, principal figure), shall be given clear identification with a capital letter, repeated near the reference arrow needed to indicate the direction of the viewing for the relevant view. Whatever the direction of viewing, the capital letter shall always be positioned in normal relation to the direction of reading and be indicated either above or on the right side of the reference arrow.

The capital letters identifying the referenced views shall be placed immediately above the relevant views (see Figure 10).

Figure 10 – ***Labelled view method***



4.8.2.4 First angle projection method

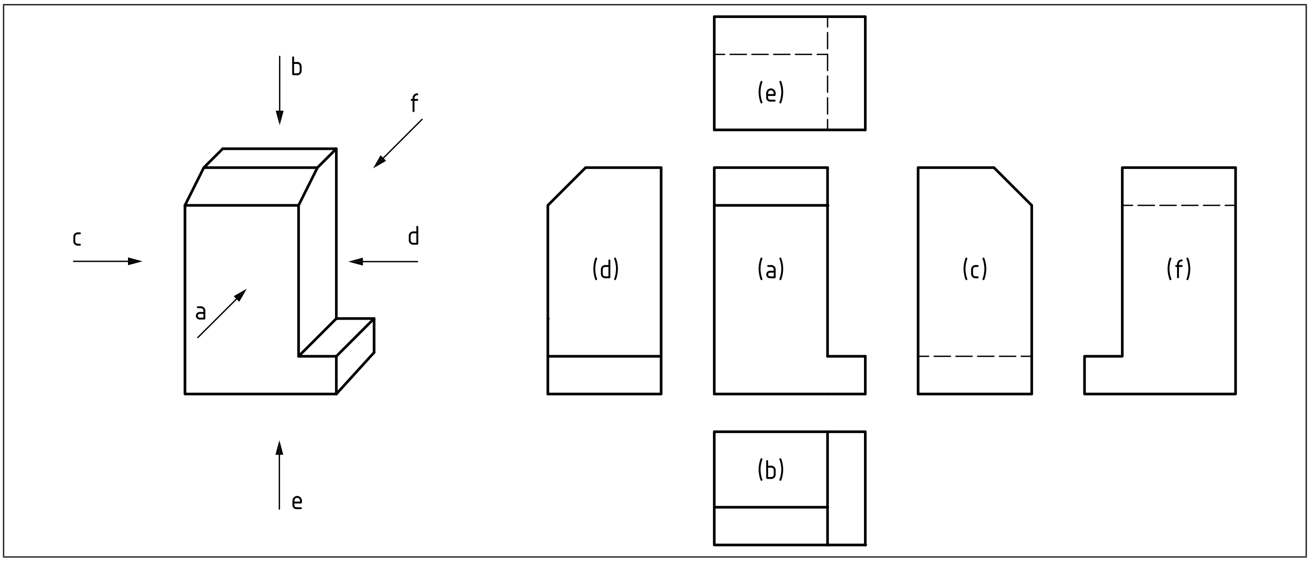
4.8.2.4.1 General

NOTE A more detailed description of the first angle projection method is to be found in BS ISO 128-30 and BS EN ISO 5456-2.

With reference to the front view (a), the other views shall be arranged as follows (see Figure 11).

- The view from above (b) shall be placed underneath.
- The view from below (e) shall be placed above.
- The view from the left (c) shall be placed on the right.
- The view from the right (d) shall be placed on the left.
- The view from the rear (f) shall be placed on the left or right, as convenient.

Figure 11 – *First angle projection method*

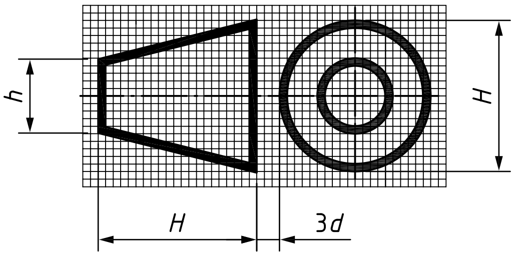


4.8.2.4.2 First angle projection method – Graphical symbol

As specified by BS ISO 128-30, the graphical symbol for the first angle projection method shall be as shown in Figure 12.

The proportions and dimensions of this graphical symbol shall be as specified in BS ISO 128-30.

Figure 12 – *First angle projection method: Graphical symbol*



4.8.2.5 Third angle projection method

*COMMENTARY ON **4.8.2.5***

A more detailed description of the third angle projection method is given in BS ISO 128-30 and BS EN ISO 5456-2.

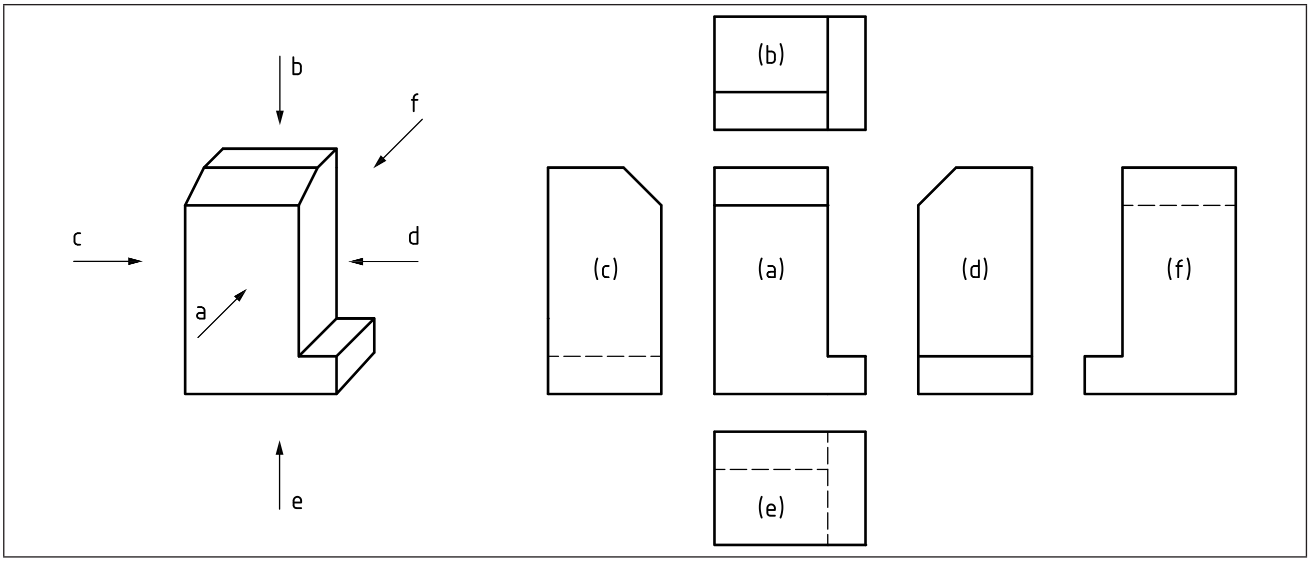
4.8.2.5.1 General

With reference to the front view (a), the other views shall be arranged as follows (see Figure 13).

- The view from above (b) shall be placed above.
- The view from below (e) shall be placed underneath.
- The view from the left (c) shall be placed on the left.

- The view from the right (d) shall be placed on the right.
- The view from the rear (f) shall be placed on the left or right, as convenient.

Figure 13 – *Third angle projection method*

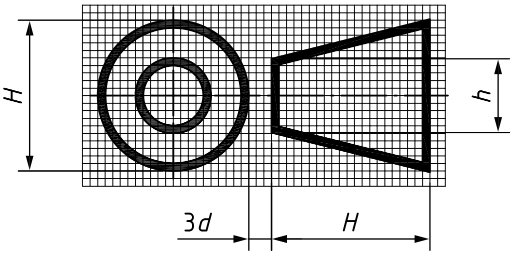


4.8.2.5.2 Third angle projection method – Graphical symbol

As specified by BS ISO 128-30, the graphical symbol for the third angle projection method shall be as shown in Figure 14.

The proportions and dimensions of this graphical symbol shall be as specified in BS ISO 128-30.

Figure 14 – *Third angle projection method: Graphical symbol*



4.9 Views

4.9.1 General

Views shall conform to the following standards.

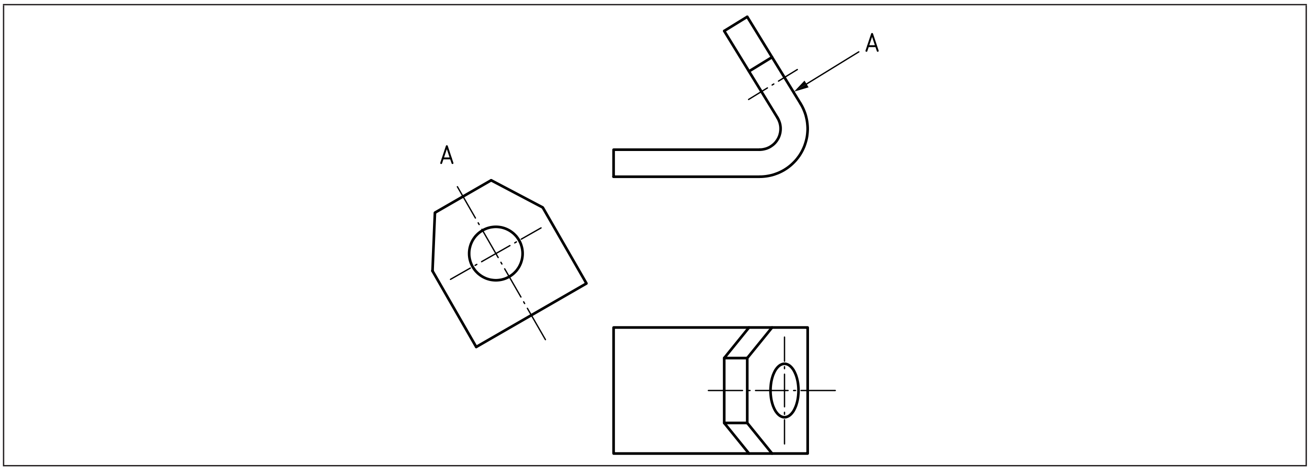
BS ISO 128-30	*Technical drawings – General principles of presentation – Part 30: Basic conventions for views*
BS ISO 128-34	*Technical drawings – General principles of presentation – Part 34: Views on mechanical engineering drawings*

4.9.2 Auxiliary views

Where true representation of features is necessary, but cannot be achieved on the orthographic views, the features shall be shown in projected auxiliary views.

NOTE An example is shown in Figure 15.

Figure 15 – ***Auxiliary view showing true shape of inclined surface***



4.10 Sections

Sections shall conform to the following standards.

BS ISO 128-40	*Technical drawings – General principles of presentation – Part 40: Basic conventions for cuts and sections*
BS ISO 128-44	*Technical drawings – General principles of presentation – Part 44: Sections on mechanical engineering drawings*
BS ISO 128-50	*Technical drawings – General principles of presentation – Part 50: Basic conventions for representing areas on cuts and sections*

NOTE 1 BS ISO 128-44 and BS ISO 128-50 contain presentational defects in some figures (e.g. line types, line thickness, terminators and letter heights). It is stressed that the text of these standards is technically correct and users should, therefore, regard the figures as illustrations only.

As specified by BS ISO 128-44, ribs, fasteners, shafts, spokes of wheels, etc. shall not be cut in longitudinal sectional views and shall therefore not be represented as sections.

NOTE 2 In practice, it is not normally possible to avoid cutting ribs and spokes of wheels.

NOTE 3 Like views, sections might be shown in a position other than that indicated by the arrows for the direction of their viewing.

NOTE 4 A section in one plane is shown in Figure 16.

NOTE 5 A section in two parallel planes is shown in Figure 17.

NOTE 6 A section in three contiguous planes is shown in Figure 18.

Figure 16 – *Section view in one plane*

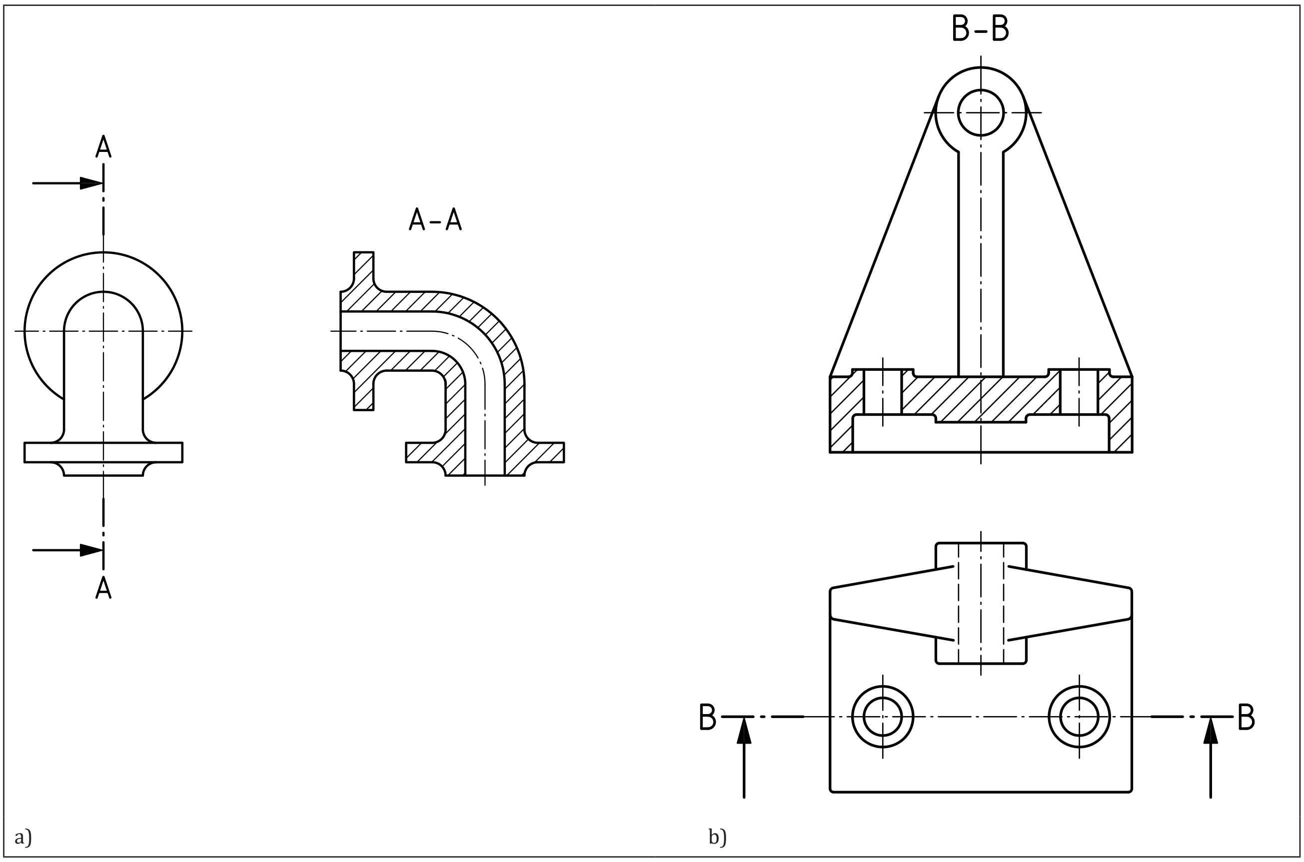


Figure 17 – *Section view in two parallel planes*

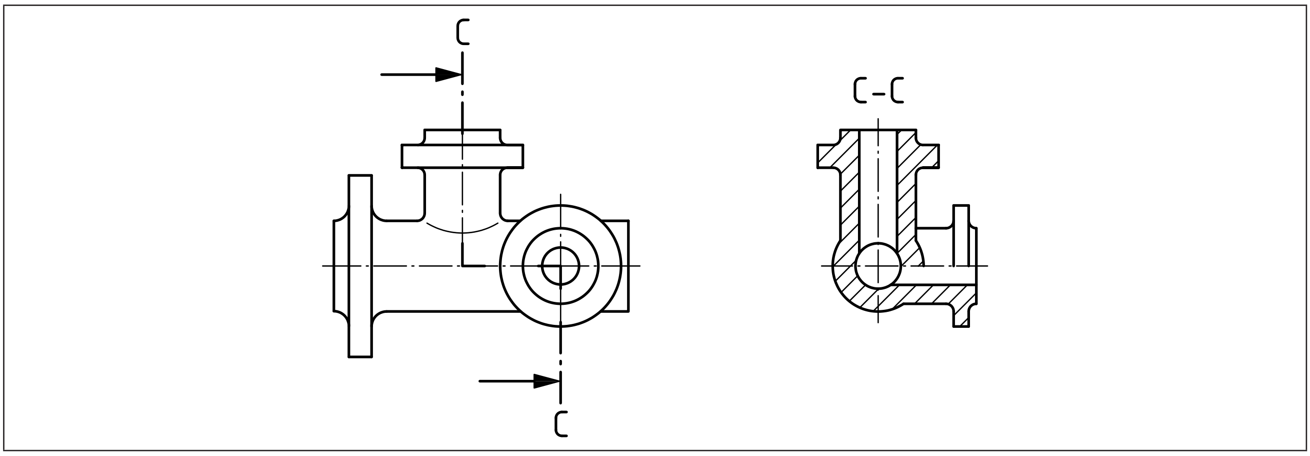
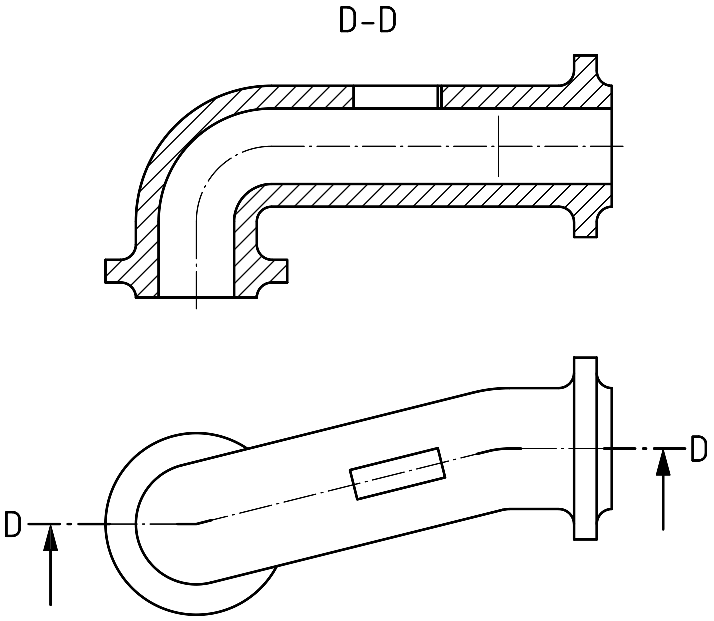


Figure 18 – ***Section view in three contiguous planes showing true shape of inclined surface***



4.11 Representation of components

4.11.1 General

Conventions used for the representation of components shall conform to the following standards, as appropriate.

BS EN ISO 2162-1	*Technical product documentation – Springs – Part 1: Simplified representation*
BS EN ISO 2162-2	*Technical product documentation – Springs – Part 2: Presentation of data for cylindrical helical compression springs*
BS EN ISO 26909	*Springs – Vocabulary*
BS EN ISO 2203	*Technical drawings – Conventional representation of gears*
BS ISO 1219-1	*Fluid power systems and components – Graphical symbols and circuit diagrams – Part 1: Graphical symbols for conventional use and data-processing applications*
BS 3238-1	*Graphical symbols for components of servo-mechanisms – Part 1: Transductors and Magnetic Amplifiers*
BS 3238-2	*Graphical symbols for components of servo-mechanisms – Part 2: General Servo-mechanisms*
BS EN ISO 13715	*TPD – Edges of undefined shape – Indication and dimensioning*
BS EN ISO 2553	*Welded and allied processes – Symbolic representation on drawings – Welded joints*
BS EN ISO 4063	*Welding and allied processes – Nomenclature of processes and reference numbers*

BS EN ISO 5261	*Technical drawings – Simplified representation of bars and profile sections*
BS EN ISO 5845-1	*Technical drawings – Simplified representation of the assembly of parts with fasteners – Part 1: General principles*
BS EN ISO 6410-1	*Technical drawings – Screw threads and threaded parts – Part 1: General conventions*
BS EN ISO 6410-2	*Technical drawings – Screw threads and threaded parts – Part 2: Screw thread inserts*
BS EN ISO 6410-3	*Technical drawings – Screw threads and threaded parts – Part 3: Simplified representation*
BS EN ISO 6411	*Technical drawings – Simplified representation of centre holes*
BS EN ISO 6412-1	*TPD – Simplified representation of pipelines – Part 1: General rules and orthogonal representation*
BS EN ISO 6412-2	*TPD – Simplified representation of pipelines – Part 2: Isometric projection*
BS EN ISO 6412-3	*TPD – Simplified representation of pipelines – Part 3: Terminal features of ventilation and drainage systems*
BS EN ISO 6413	*Technical drawings – Representation of splines and serrations*
BS EN ISO 8826-1	*Technical drawings – Roller bearings – Part 1: General simplified representation*
BS EN ISO 8826-2	*Technical drawings – Roller bearings – Part 2: Detailed simplified representation*
BS EN ISO 9222-1	*Technical drawings – Seals for dynamic application – Part 1: General simplified representation*
BS EN ISO 9222-2	*Technical drawings – Seals for dynamic application – Part 2: Detailed simplified representation*
BS EN ISO 15785	*Technical drawings – Symbolic presentation and indication of adhesive, fold and pressed joints*

NOTE The BS ISO 128 series of standards covers the general subject of component representation.

4.11.2 Representation of moulded, cast and forged components

Dimensional tolerancing for metal and metal alloy castings shall conform to the following standards, as appropriate.

BS EN ISO 8062-1	*Geometrical product specifications (GPS) – Dimensional and geometrical tolerances for moulded parts – Part 1: Vocabulary*
PD CEN ISO/TS 8062-2	*Geometrical Product Specifications (GPS) – Dimensional and geometrical tolerances for moulded parts – Part 2: Rules*
BS EN ISO 8062-3	*Geometrical product specifications (GPS) – Dimensional and geometrical tolerances for moulded parts – Part 3: General dimensional and geometrical tolerances and machine allowances for casting*

4.11.3 Representation of moulded components

Indications for parting lines, draft angles and other elements of moulded components shall conform to the following standard.

BS EN ISO 10135	*Geometrical product specifications (GPS) – Drawing indications for moulded parts in technical product documentation (TPD)*

Section 5 Dimensioning

5.1 General

Dimensioning and tolerancing shall conform to the following standards, as appropriate.

BS ISO 129-1	*TPD – Presentation of dimensions and tolerances – Part 1: General principles*
BS EN ISO 14405-2	*Geometrical product specifications (GPS) – Dimensional tolerancing – Part 2: Dimensions other than linear sizes*
BS EN ISO 1119	*Geometrical product specifications (GPS) – Series of conical tapers and taper angles*
BS EN ISO 1660	*Geometrical product specifications (GPS) – Geometrical tolerancing – Profile tolerancing*
BS 1916-1	*Limits and fits for engineering – Part 1: Guide to limits and tolerances*
BS 1916-2	*Limits and fits for engineering – Part 2: Guide to the selection of fits in BS 1916-1*
BS 1916-3	*Limits and fits for engineering – Part 3: Guide to tolerances, limits and fits for large diameters*
BS EN ISO 3040	*Geometrical product specifications (GPS) – Dimensioning and tolerancing – Cones*
BS 3734-1	*Rubber – Tolerances for products – Part 1: Dimensional tolerances*[6]
BS 4500	*Limits and fits – Guidance for system of cone (taper) fits and tolerances for cones from C = 1:3 to 1:500, lengths from 6 mm to 630 mm and diameters up to 500 mm*
BS EN ISO 5458	*Geometrical Product Specifications (GPS) – Geometrical tolerancing – Pattern and combined geometrical specification*
BS EN ISO 6410-1	*Technical drawings – Screw threads and threaded parts – Part 1: General conventions*
BS 7010	*Code of practice for a system of tolerances for the dimensions of plastic mouldings*
BS EN ISO 7083	*Technical drawings – Symbols for geometrical tolerancing – Proportions and dimensions*
BS EN ISO 8015	*Geometrical product specifications (GPS) – Fundamentals – Concepts, principles and rules*
BS ISO 10579	*Geometrical product specifications (GPS) – Dimensioning and tolerancing – Non-rigid parts*
BS EN ISO 13920	*Welding – General tolerances for welded constructions – Dimensions for lengths and angles – Shape and position*
BS EN ISO 8062-1	*Geometrical product specifications (GPS) – Dimensional and geometrical tolerances for moulded parts – Part 1: Vocabulary*

---

[6] Withdrawn.

PD CEN ISO/TS 8062-2	*Geometrical Product Specifications (GPS) – Dimensional and geometrical tolerances for moulded parts – Part 2: Rules*
BS EN ISO 8062-3	*Geometrical product specifications (GPS) – Dimensional and geometrical tolerances for moulded parts – Part 3: General dimensional and geometrical tolerances and machining allowances for castings*
BS EN ISO 286-1	*Geometrical product specifications (GPS) – ISO code system for tolerances on linear sizes – Part 1: Basis of tolerances, deviations and fits*
BS EN ISO 286-2	*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*

5.2 Dimensioning methods

5.2.1 Presentation rules

5.2.1.1 Dimensioning shall conform to BS ISO 129-1.

NOTE The dimensioning of shipbuilding drawings is specified in BS ISO 129-4.

5.2.1.2 Only the dimensions which are necessary to unambiguously define the nominal geometry of the product shall be specified.

NOTE The dimensions shown should be for the purposes of describing the function, production or verification of the product. Auxiliary dimensions may also be included, but do not form part of the specification.

5.2.1.3 The size of a feature or relationship between features shall be dimensioned only once.

5.2.1.4 Unless otherwise specified, dimensions shall be indicated for the finished state of the dimensioned feature.

NOTE However, it might be necessary to give additional dimensions at intermediate stages of production if they are shown on the same drawing (e.g. the size of a feature prior to carburizing and finishing).

5.2.1.6 All dimensions, graphical symbols and annotations shall be indicated such that they can be read from the bottom or right-hand side (main reading directions) of the drawing. Dimensioning and annotations of 3D models shall be in accordance with BS ISO 16792.

NOTE Dimensions alone are not sufficient to define the requirements of a product.

5.2.1.7 Dimension shall be used with other specification techniques as applicable, e.g. geometrical tolerancing or surface texture requirements.

5.2.1.8 Lettering on drawings shall be in accordance with BS EN ISO 3098 (all parts).

5.2.1.9 There shall be only one lettering height for dimension and tolerance indication for a TPS.

5.2.1.10 All dimensions shall be toleranced, either via a general tolerance or by direct indication of tolerance or limit indications, except for the following cases:

a) min. (see **5.4.3**);

b) max. (see **5.4.3**);

c) auxiliary dimension (see **5.2.8**); and

d) theoretically exact dimension (TED) (see **7.8**).

5.2.2 Presentation of decimals

Dimensional values indicated in decimal notation shall use a comma as the decimal marker.

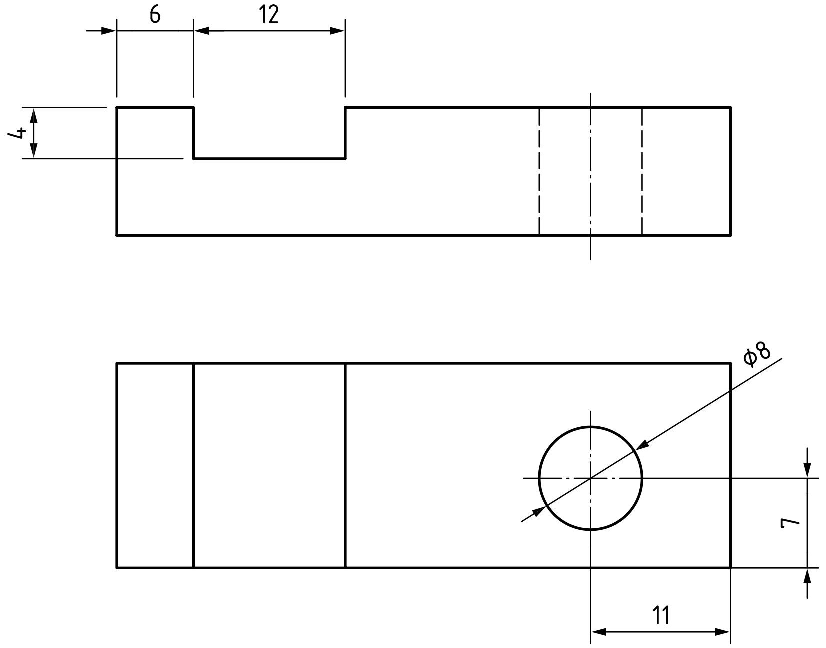
Each group of three digits, counting from the decimal marker to the left and to the right, shall be separated from other digits by a small space (e.g. 12 345,067 8).

NOTE The use of a comma or a point for this purpose is deprecated. It is further recommended that items in lists are separated by semicolons.

5.2.3 Positioning of dimensions

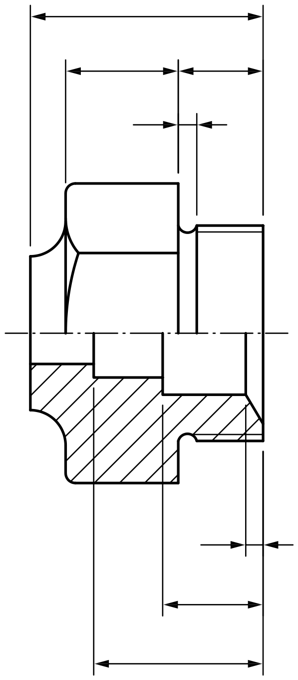
Dimensions shall be placed on that view or section which shows the relevant feature(s) most clearly (see Figure 19).

Figure 19 – ***Positioning of dimensions***



Dimensions for internal features and dimensions for external features shall, where possible, be arranged and indicated in separate groups of dimensions to improve readability (see Figure 20).

Figure 20 – ***Arrangement and indication of dimensions internal and external features***



Whenever possible, dimensions shall not be placed within the contour of the depicted item.

5.2.4 Units of dimensions

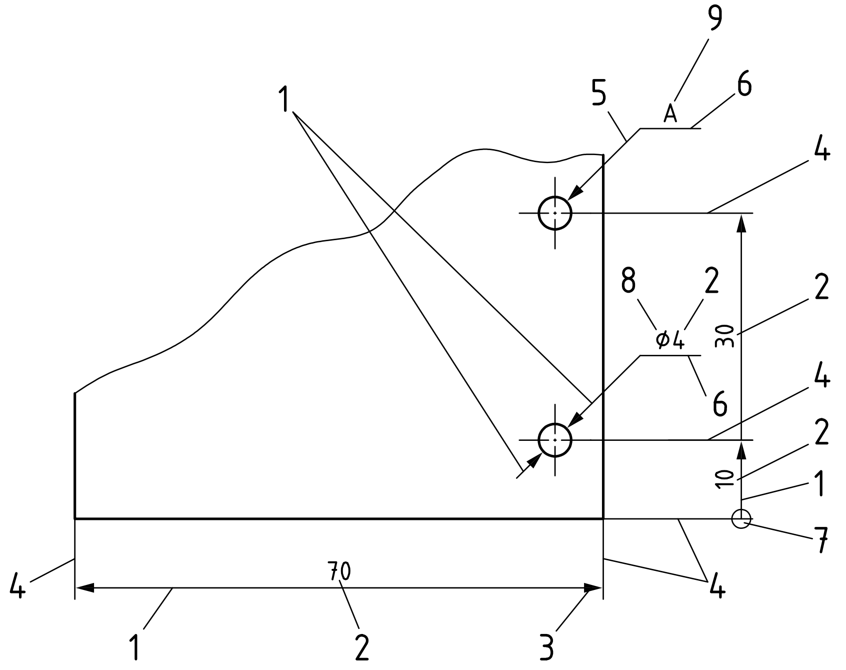
The units of a dimension shall be specified with the dimension. Typically, the predominant unit of measure on a drawing is specified in the drawing title block and the unit omitted from the individual dimensions. Any dimensions expressed in a different unit of measure shall indicate that unit of measure.

5.2.5 Elements of dimensioning: usage

5.2.5.1 General

The various elements of dimensioning shall be indicated as illustrated in Figure 21.

Figure 21 – ***Elements of dimensioning***



Key

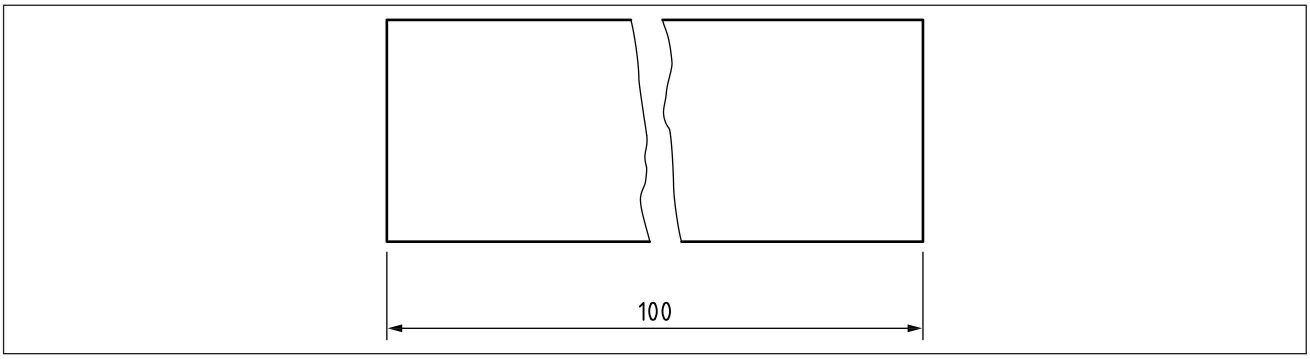
1	Dimension line	6	Reference line
2	Nominal dimensional value	7	Origin symbol
3	Terminator (in this case an arrowhead)	8	Property indicator
4	Extension line	9	Reference letter
5	Leader line		

5.2.5.2 Dimension line

Dimension lines shall be indicated as continuous narrow lines in accordance with BS EN ISO 128-20.

Where the feature is shown broken, the corresponding dimension line shall be shown unbroken (see Figure 22).

Figure 22 – ***Dimension line of feature that is broken***

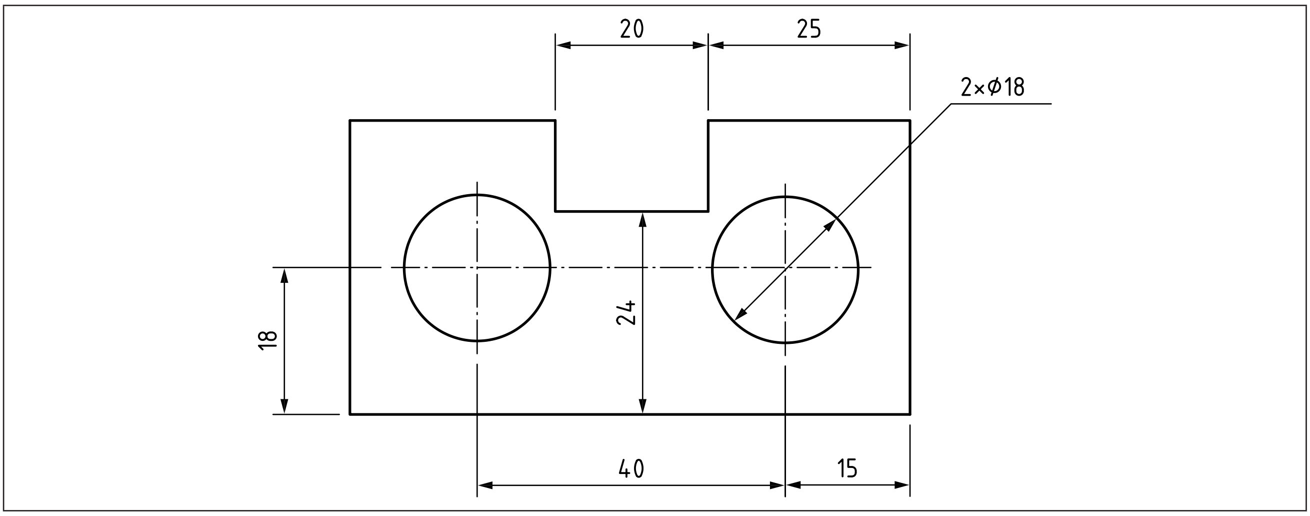


Dimension lines of holes can be indicated oblique through the centre of the hole (see Figure 23).

Intersection of dimension lines with any other line should be avoided, but where intersection is unavoidable they shall be shown unbroken (see Figure 22).

The centre line or outline of a feature or their extensions can be used in place of an extension line (see Figure 23). However, a centre line or the outline of a feature shall not be used as a dimension line.

Figure 23 – ***Dimension lines of holes***



5.2.5.3 Extension lines

Extension lines shall be drawn as continuous narrow lines in accordance with BS ISO 128-20.

Extension lines shall not be drawn between views and shall not be drawn parallel to the direction of hatching.

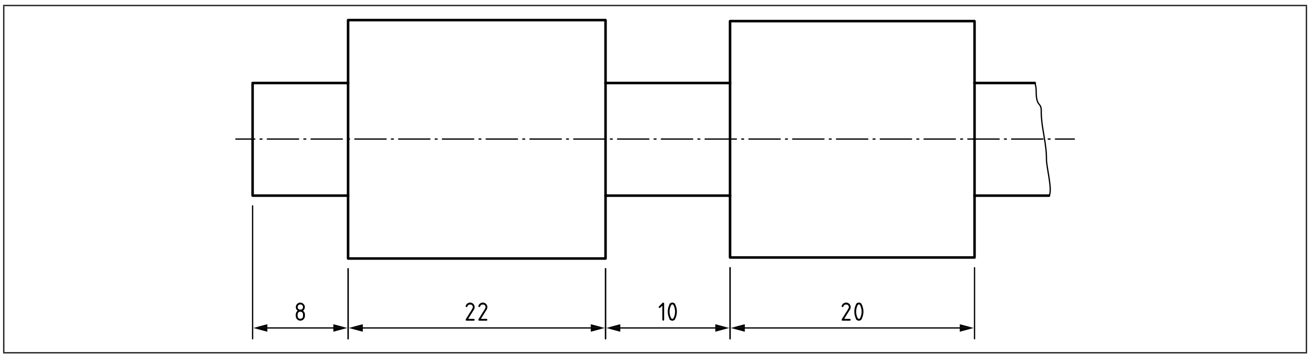
Extension lines shall extend approximately eight times the line width beyond their associated dimension line.

Extension lines shall be drawn perpendicular to the corresponding physical length (see Figure 24, Figure 25 and Figure 26).

For circular features the extension line shall be drawn as a continuation of the feature shape (see Figure 62).

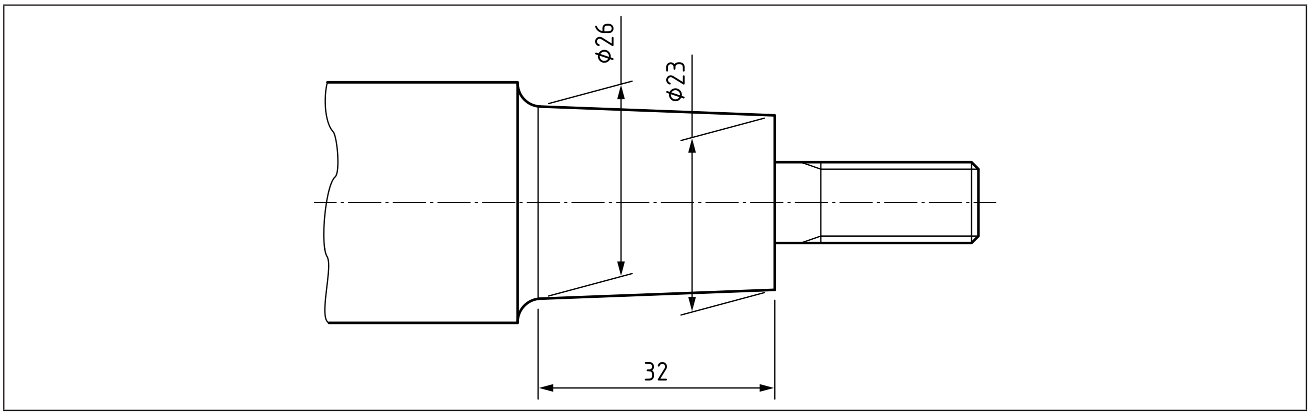
NOTE It is advisable to have a gap between the feature and the beginning of the extension line.

Figure 24 – *Extension lines*



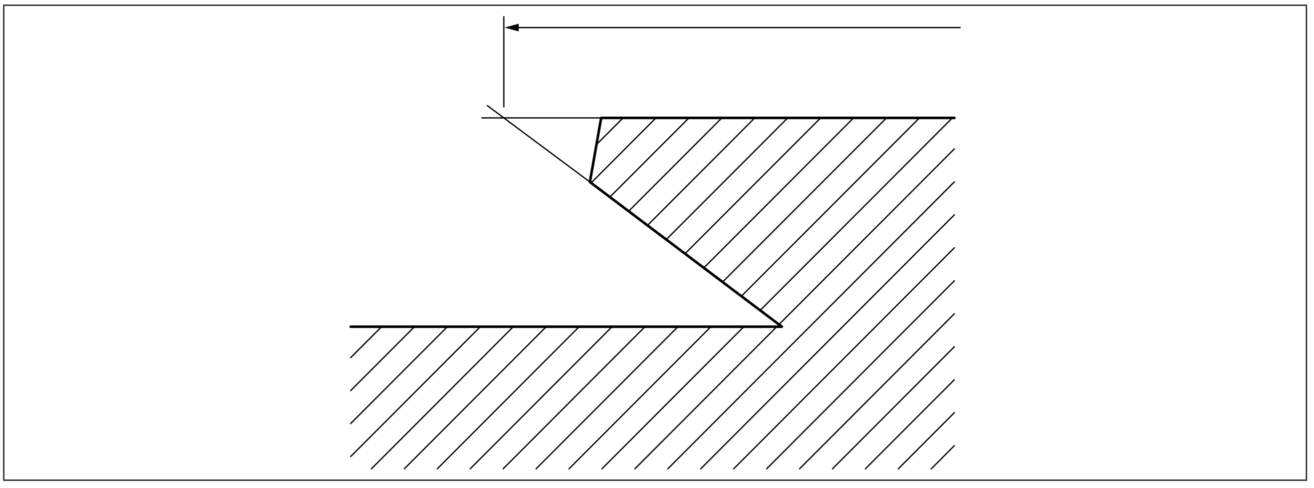
The extension lines may be drawn oblique to the feature but shall be parallel to each other (see Figure 25).

Figure 25 – *Oblique extension lines*



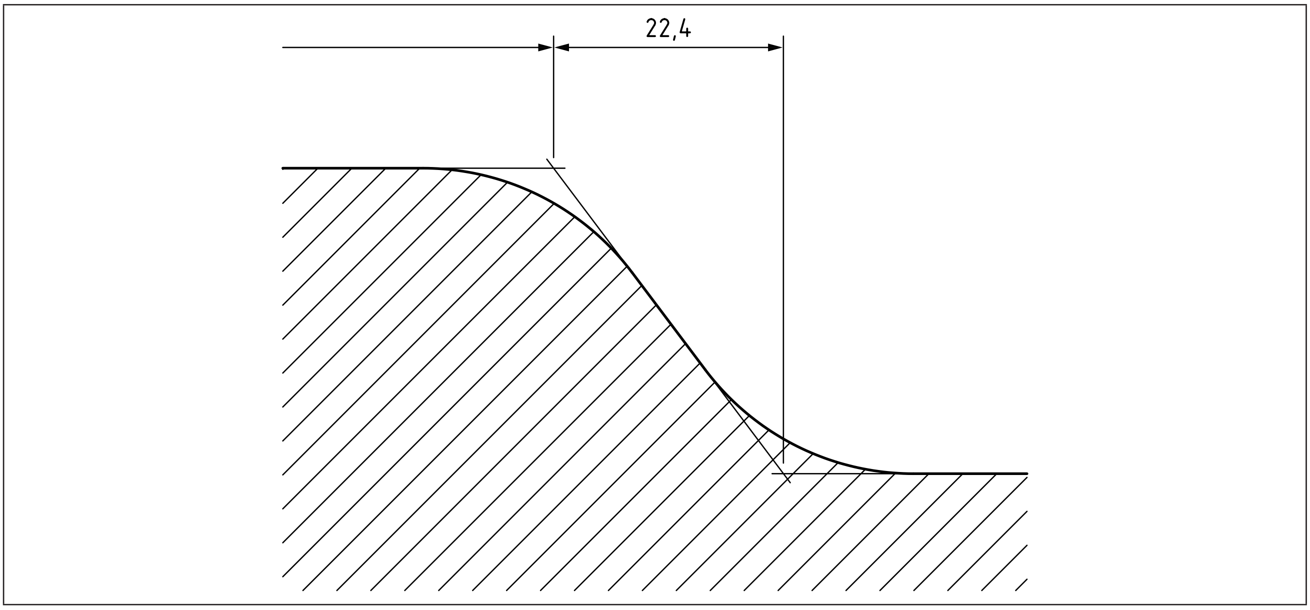
Intersecting projected contours of outlines shall extend approximately eight times the line width beyond the point of intersection (see Figure 26).

Figure 26 – *Intersection of projected contours of outlines*



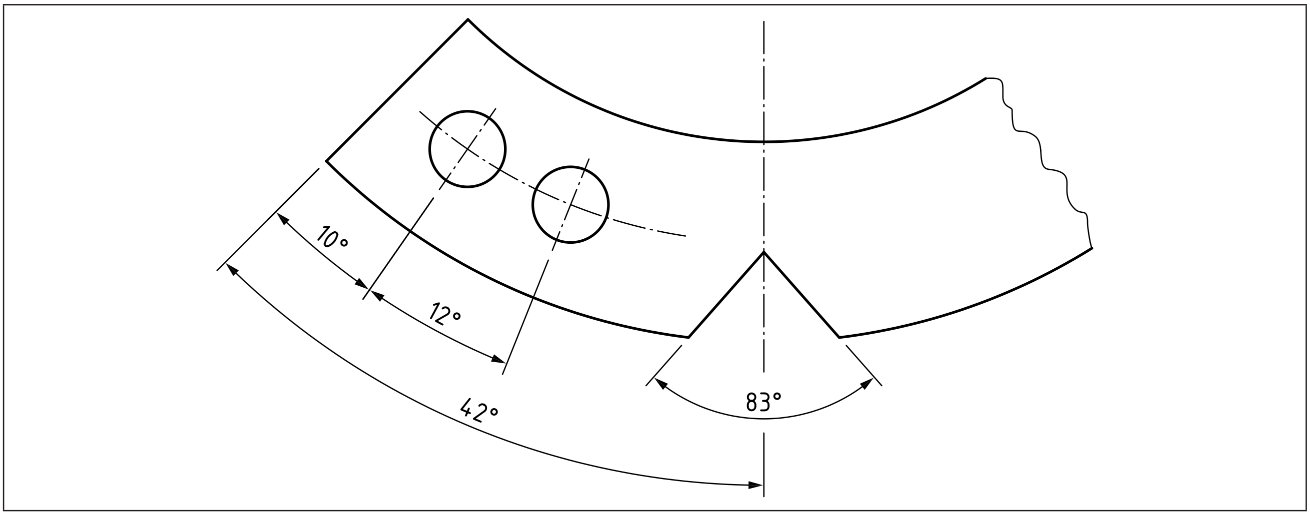
In the case of projected contours of transitions and similar features, the extension lines shall apply at the point of intersection of the projection lines (see Figure 27).

Figure 27 – ***Intersection of projected contours of transitions and similar features***



In the case of angular dimensions, the extension lines shall be the extensions of the angle legs (see Figure 28).

Figure 28 – ***Extension lines of angular dimensions***



5.2.5.4 Leader line

Leader lines shall be drawn in accordance with BS ISO 128-22.

5.2.6 Dimensional values

5.2.6.1 Indication

Dimensional values shall be indicated on drawings in characters of sufficient size to ensure complete legibility on the original drawing, as well as on reproductions made from microfilms (see BS EN ISO 6428).

NOTE Lettering ISO 3098-BVL (Type B, Vertical, Latin) is recommended.

5.2.6.2 Positions of dimensional values

Dimensional values shall be placed parallel to their dimension line and near the middle of and slightly above that line (see Figure 29 and Figure 30).

Dimensional values shall be placed in such a way that they are not crossed or separated by any line.

Figure 29 – ***Position of dimensional values***

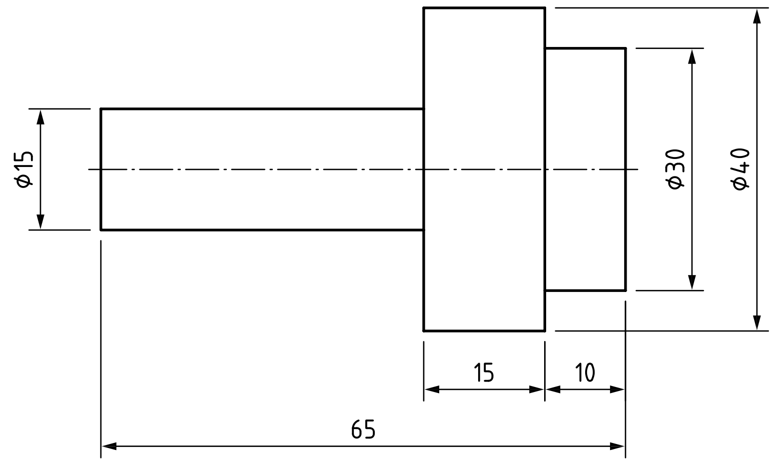
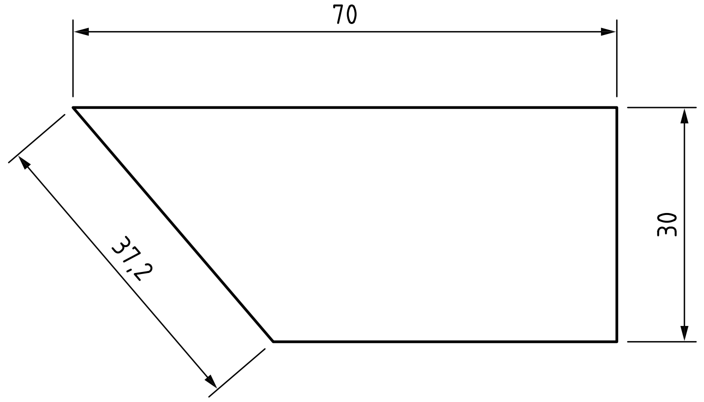


Figure 30 – ***Position of dimensional values***



Values of linear dimensions on oblique dimension lines shall be oriented as shown in Figure 31. The values shall be indicated so that they can be read from the bottom or right-hand side of the drawing.

Values of angular dimensions shall be oriented as shown in Figure 32. Angular dimensions shall be placed on top of the dimension line and follow the same rule as linear dimensions (see Figure 31).

Figure 31 – ***Orientation of linear dimensions***

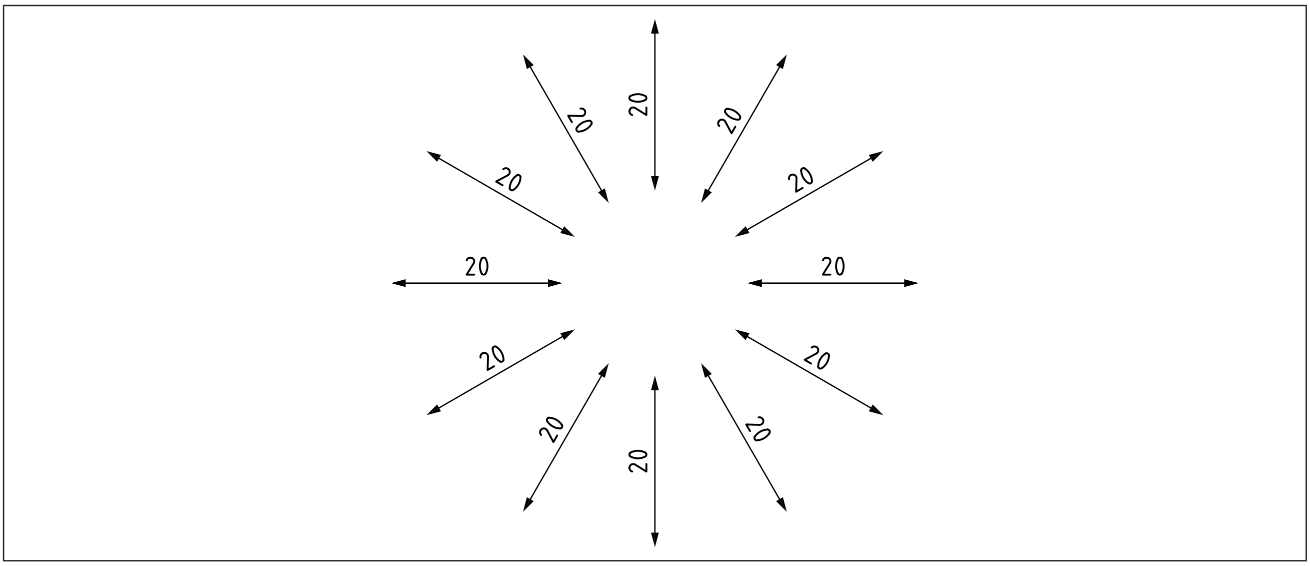
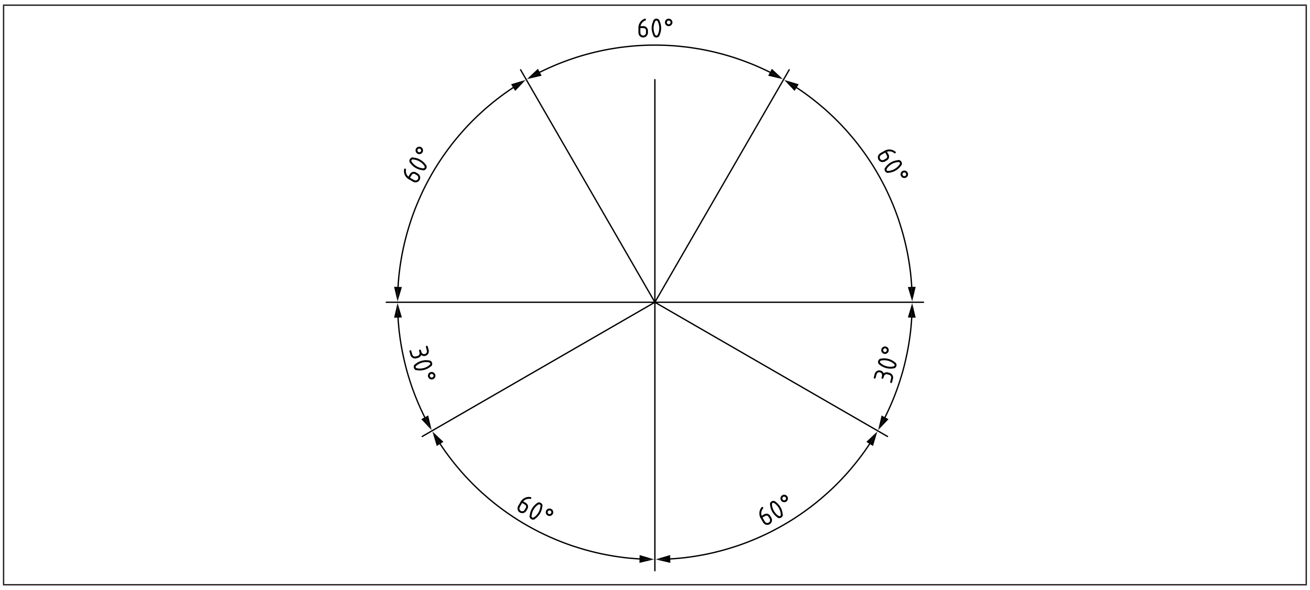


Figure 32 – ***Orientation of angular dimensions***



5.2.7 Indications of special dimensions

The symbols shown in Table 9 shall be used with dimensions to identify the feature characteristics. The following symbols shall directly precede the dimensional value without space (see Figure 33 to Figure 39).

Table 9 – *Special dimensions*

Property indicator symbol	Description	Associated property	Example of presentation
⌀	Diameter	Cylindrical feature or circular feature described by its diameter	Figure 38
R	Radius	Cylindrical feature or circular feature described by its radius	Figure 33
□	Square	Square feature with four equal angles and four equal sides described by its side dimension	Figure 34
S⌀	Spherical diameter	Spherical feature described by its diameter	Figure 35
SR	Spherical radius	Spherical feature described by its radius	Figure 36
⌒	Arc length	Curvilinear dimension of non-flat feature (e.g. arc length)	Figure 142 and Figure 146
t=	Thickness (of thin objects)	Two offset features defined by its thickness	Figure 37
↧	Depth	Depth of a hole or internal feature	Figure 38b) and d)
⌴	Cylindrical counterbore	Cylindrical hole with a flat bottom described by its diameter and depth	Figure 38b) and d)
⌵	Countersink	Circular chamfer described by a diameter and angle	Figure 39b)
○→	Developed length	Length of feature prior to bending or forming	–
↔	Between	Indication of the extent of a restricted area, used in conjunction with reference letters	Figure 144 and Figure 145

Figure 33 – *Indications of special dimensions: Radius*

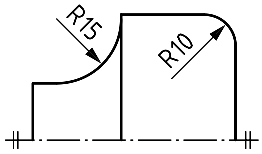


Figure 34 – *Indications of special dimensions: Square*

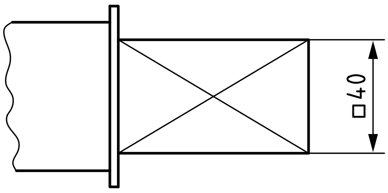


Figure 35 – *Indications of special dimensions: Spherical diameter*

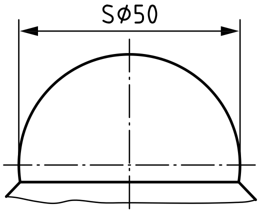


Figure 36 – *Indications of special dimensions: Spherical radius*

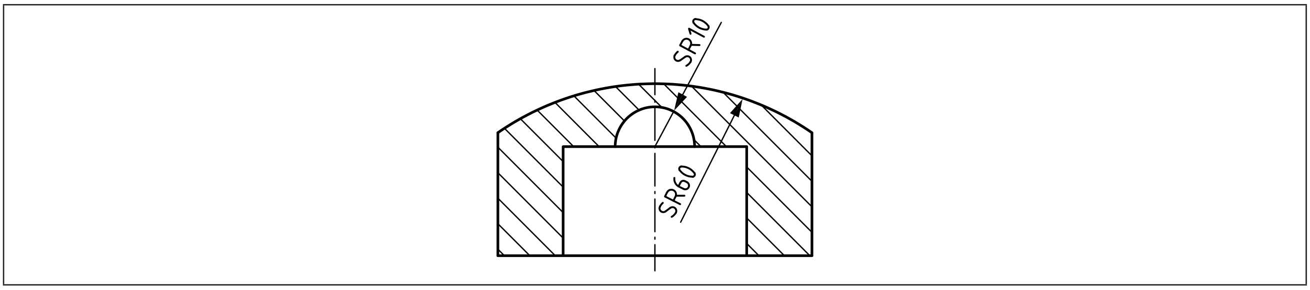
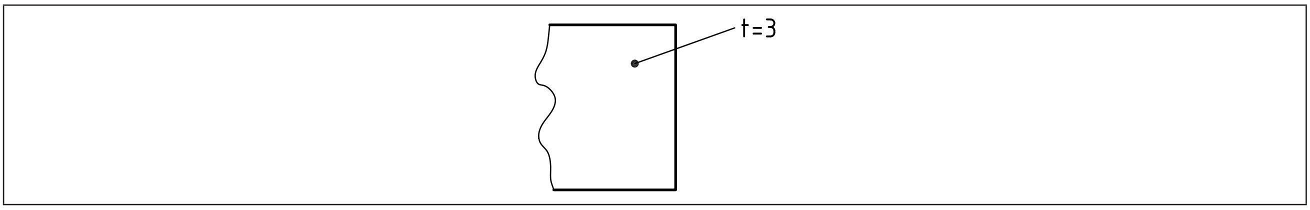
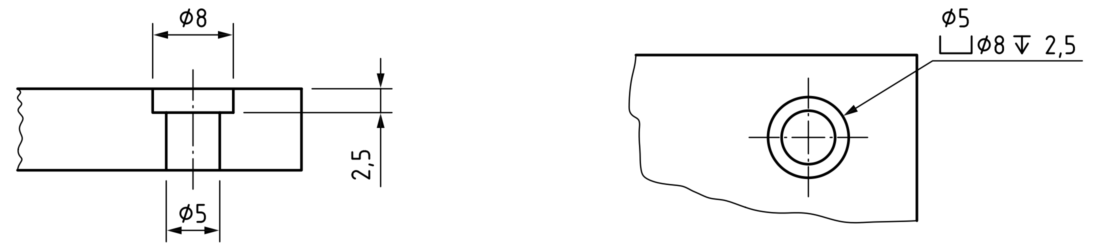


Figure 37 – *Indications of special dimensions: Thickness*



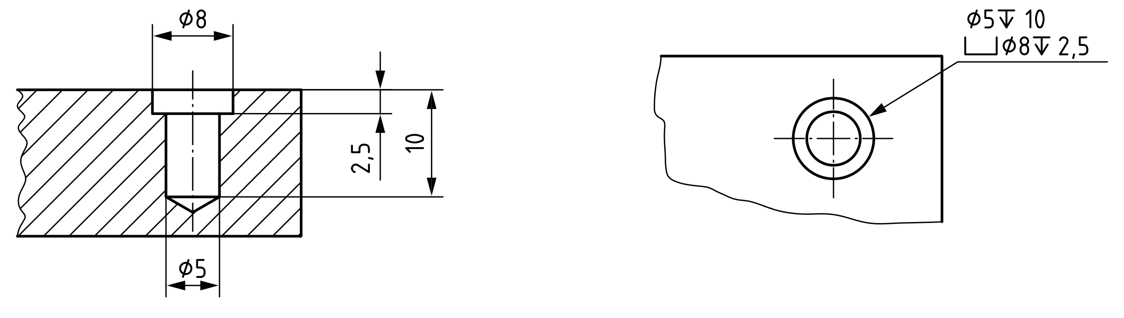
When counterbore is indicated, the diameter, and depth if needed, of the counterbore shall be specified below the dimension value of the hole (see Figure 38). When countersink is indicated, the size and inclusive angle of the countersink shall be specified below the dimensional value of the hole (see Figure 39).

Figure 38 – *Diameter and depth of counterbore*



a) Possible indication

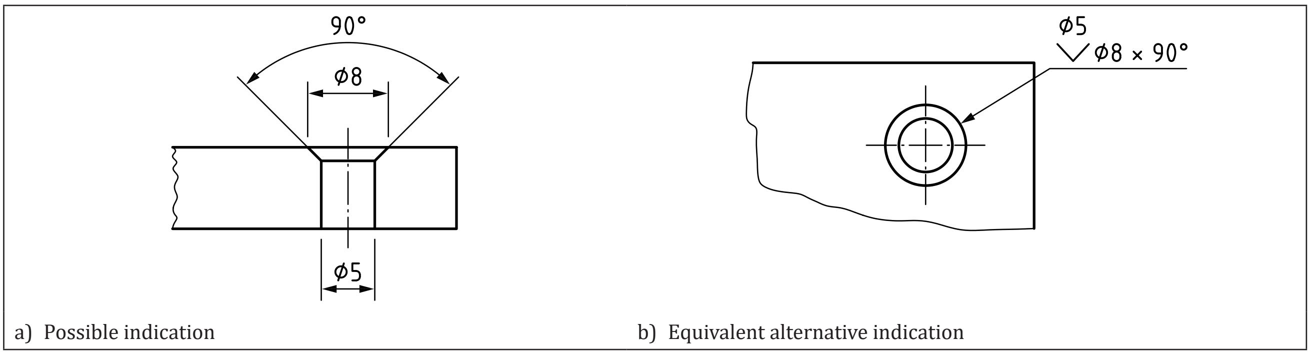
b) Equivalent alternative indication



c)

d)

Figure 39 – ***Size and angle of countersink***



a) Possible indication b) Equivalent alternative indication

5.2.8 Auxiliary dimensions

Auxiliary dimensions shall be indicated within parentheses and do not constitute an integral part of the specification or requirement (see Figure 40 and Figure 41).

NOTE Auxiliary dimensions in drawings are for information only and not for manufacturing and verification purposes. Auxiliary dimensions should have limited use.

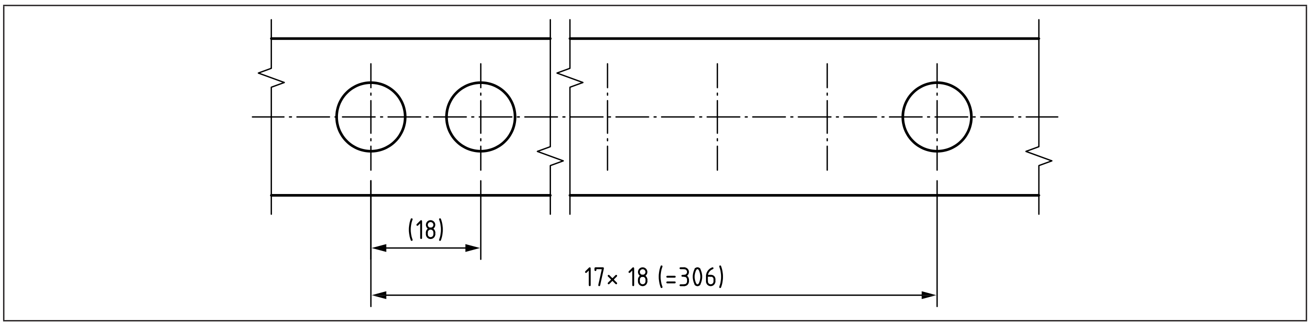
5.2.9 Equally-spaced and repeated features

5.2.9.1 Equally-spaced features

Where features have the same spacing and are uniformly arranged, their dimensioning shall, where appropriate, be simplified as follows.

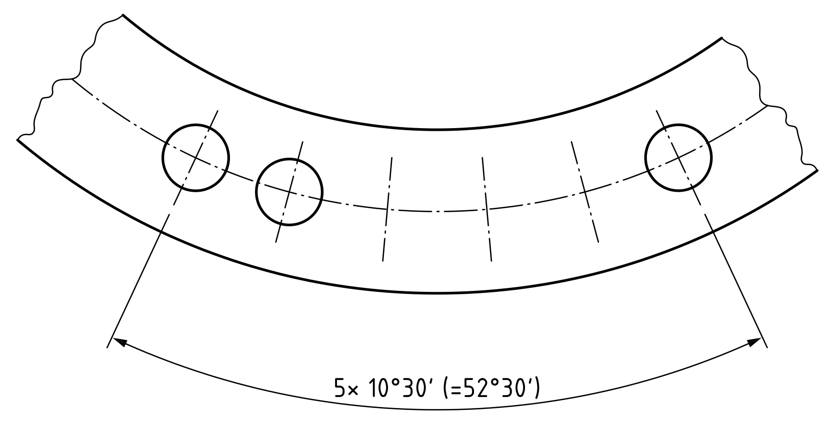
- Repeated linear and angular spacing can be indicated with the number of spacings and their dimensional value separated by the symbol "×". The number of features shall directly precede the symbol "×" without a space and the dimensional value shall be preceded by a space, e.g. 17× 18. If there is any risk of confusion between the length of the space and the number of spacings, one space may additionally be dimensioned (see Figure 40).

Figure 40 – ***Number of equally-spaced features and the dimensional value***



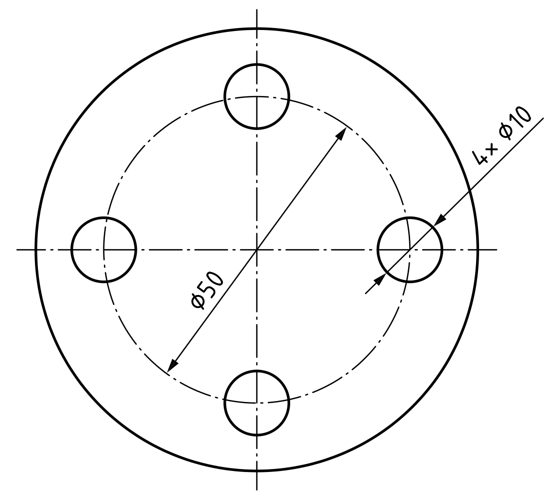
- The sum of the linear or angular spacing of the indicated features is an auxiliary dimension (see **5.2.8** and Figure 40 and Figure 41). The total representation shall be indicated as the number of spacings times the dimensional value of the spacings, and the sum given in parenthesis preceded by the equal sign.
- Angular spacing can be dimensioned as shown in Figure 41.

Figure 41 – *Dimensioning of angular spacing*



- The angles of the spacings can be omitted where spacings are self-evident and the indication does not lead to confusion (see Figure 42).

Figure 42 – *Angles of spacings*



5.2.9.2 Repeated features

Where clarity is not impaired, features having the same dimensional value shall be indicated by that value preceded by the number of features and the symbol “×” (see Figure 43 and Figure 44).

Figure 43 – *Indication of features having the same dimensional value*

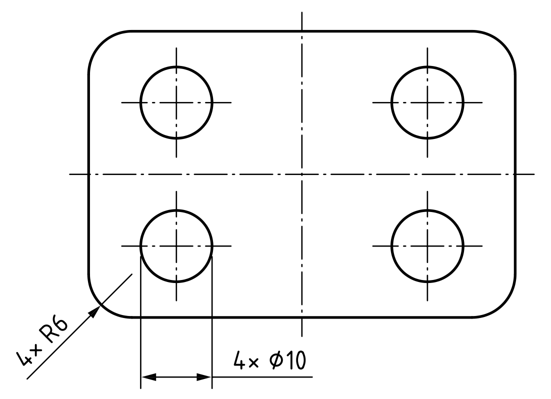
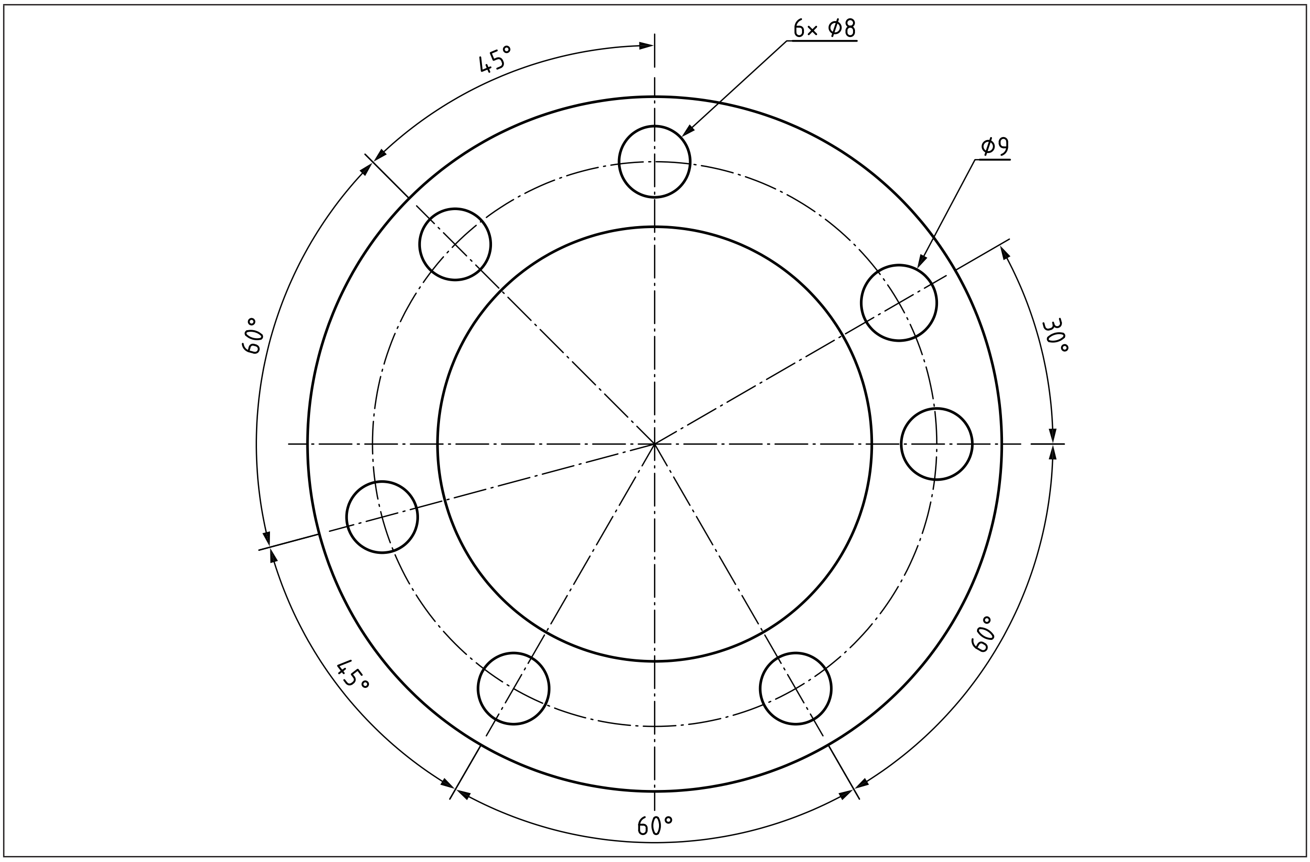


Figure 44 – *Indication of features having the same dimensional value*

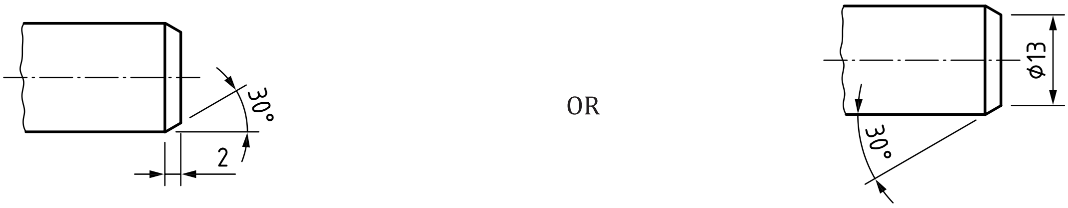


5.2.10 Chamfers

5.2.10.1 External chamfer

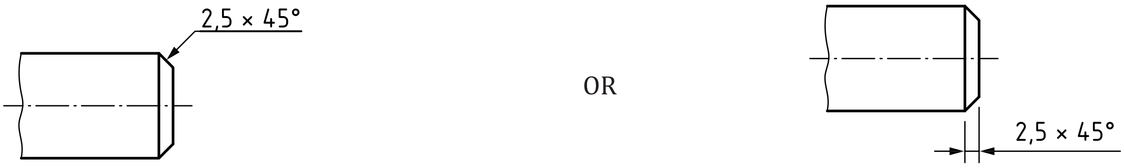
An external chamfer shall be dimensioned conventionally if the angle of the chamfer is not equal to 45° (see Figure 45).

Figure 45 – *Dimension of external chamfer not equal to 45°*



If the chamfer angle is 45°, the indication can be simplified (see Figure 46).

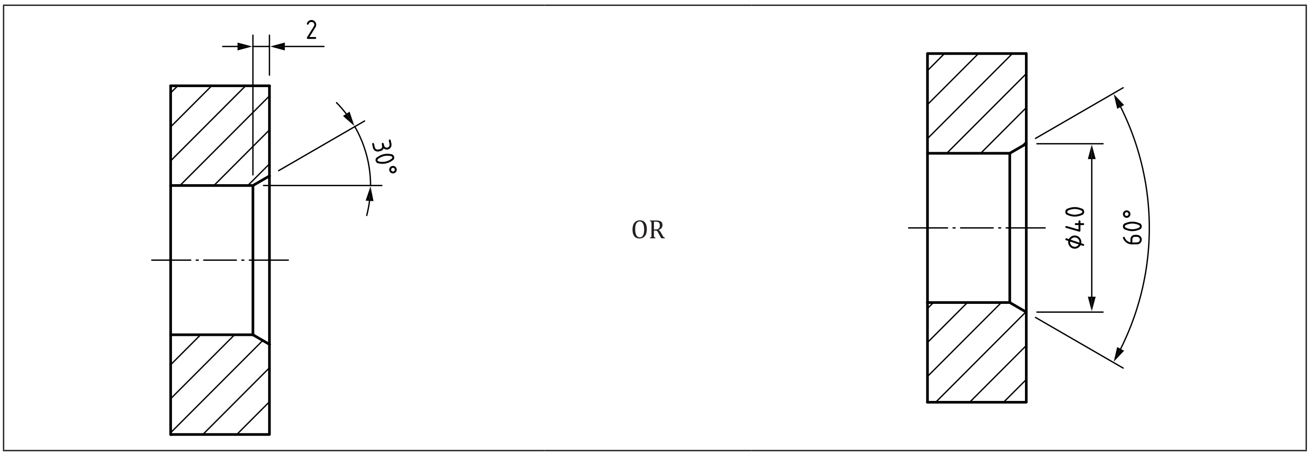
Figure 46 – *Dimension of external chamfer equal to 45°*



5.2.10.2 Internal chamfer

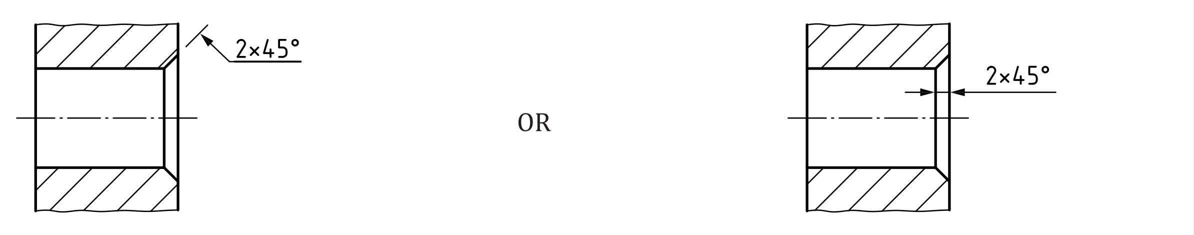
An internal chamfer shall be dimensioned conventionally if the angle of the chamfer is not equal to 45° (see Figure 47).

Figure 47 – *Dimension of internal chamfer not equal to 45°*



If the chamfer angle is 45° the indication can be simplified (see Figure 48).

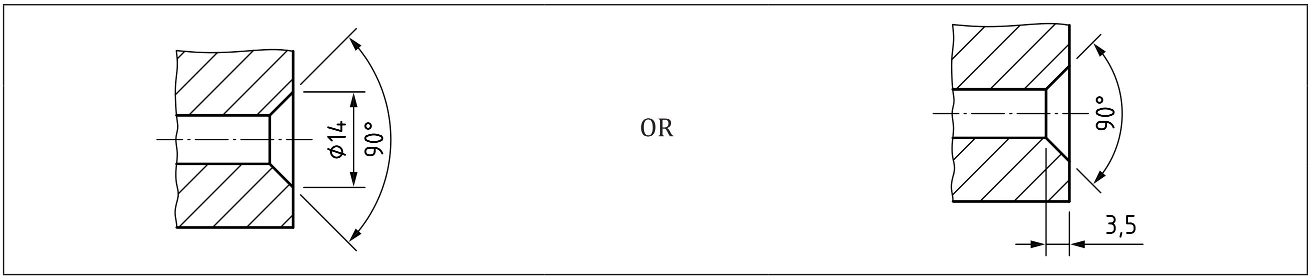
Figure 48 – *Dimension of internal chamfer equal to 45°*



5.2.11 Countersink

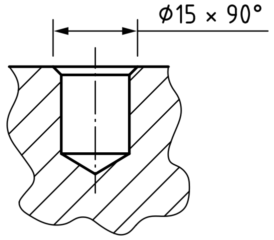
A countersink shall be dimensioned by showing either the required diametrical dimension at the surface and the included angle, or the depth and the included angle (see Figure 49).

Figure 49 – *Dimension of countersink*



If the included angle is 90°, the indication can be simplified in accordance with BS ISO 15786 (see Figure 50).

Figure 50 – ***Dimension of countersink: simplification***



5.2.12 Symmetrical parts

The dimensions of symmetrical arranged features shall be indicated once only (see Figure 51 to Figure 53). The dimension shall indicate the total number of features which occur in the part.

Usually, the line of symmetry of features shall not be dimensioned (see Figure 51 to Figure 53).

In the case of half or quarter representations, and if also required in the case of full representations, a symmetry symbol (see BS ISO 128-30) shall be indicated at both ends of the line of symmetry (see Figure 51 to Figure 53).

In the case of half or quarter representations, the dimension lines that need to cross the line of symmetry shall extend past the axis of symmetry; the second termination shall then be omitted (see Figure 51 to Figure 53).

Figure 51 – ***Symmetrical parts***

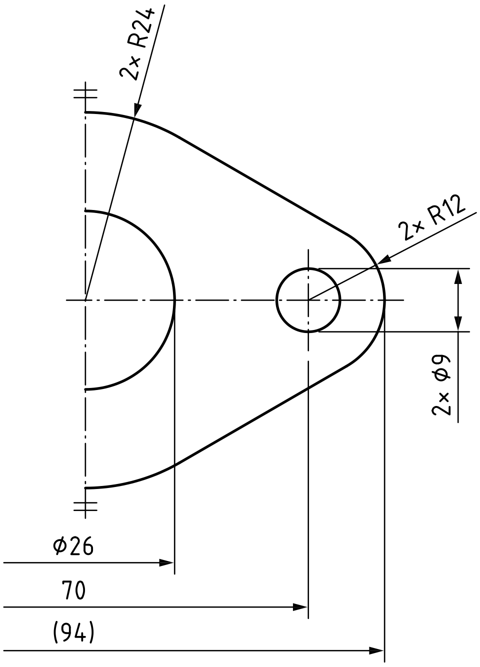


Figure 52 – *Symmetrical parts*

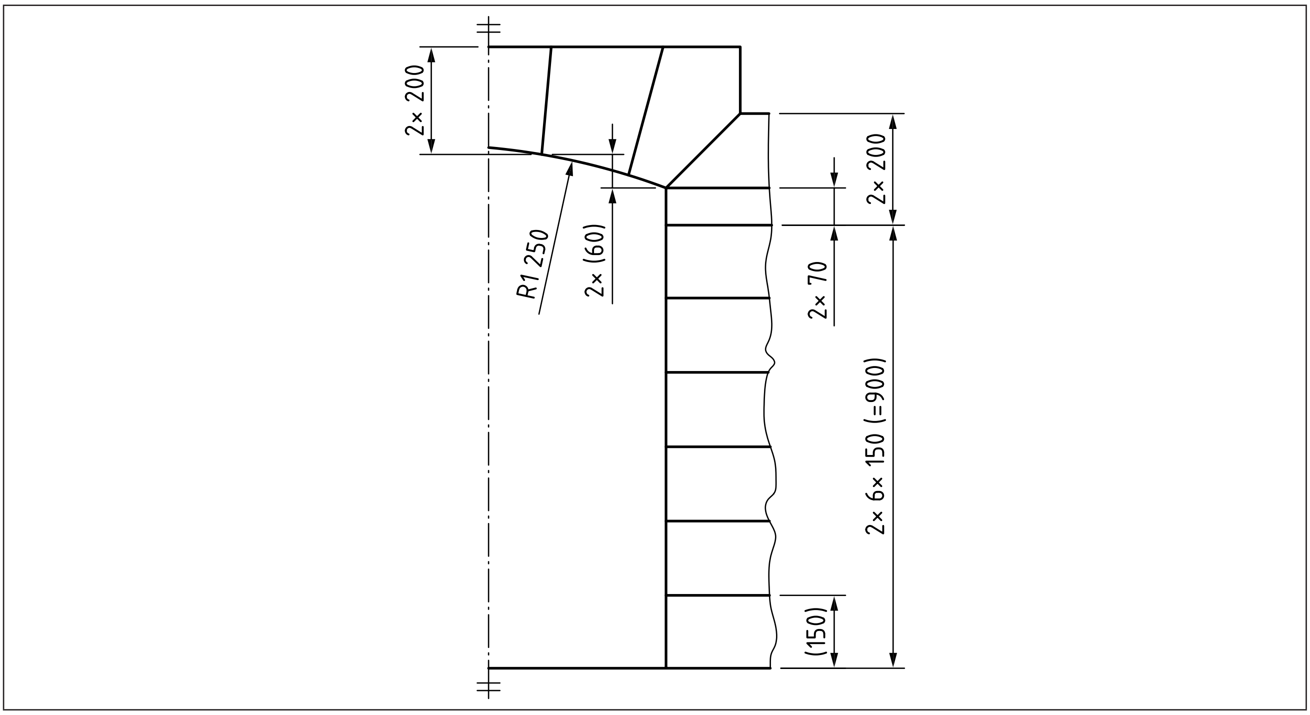
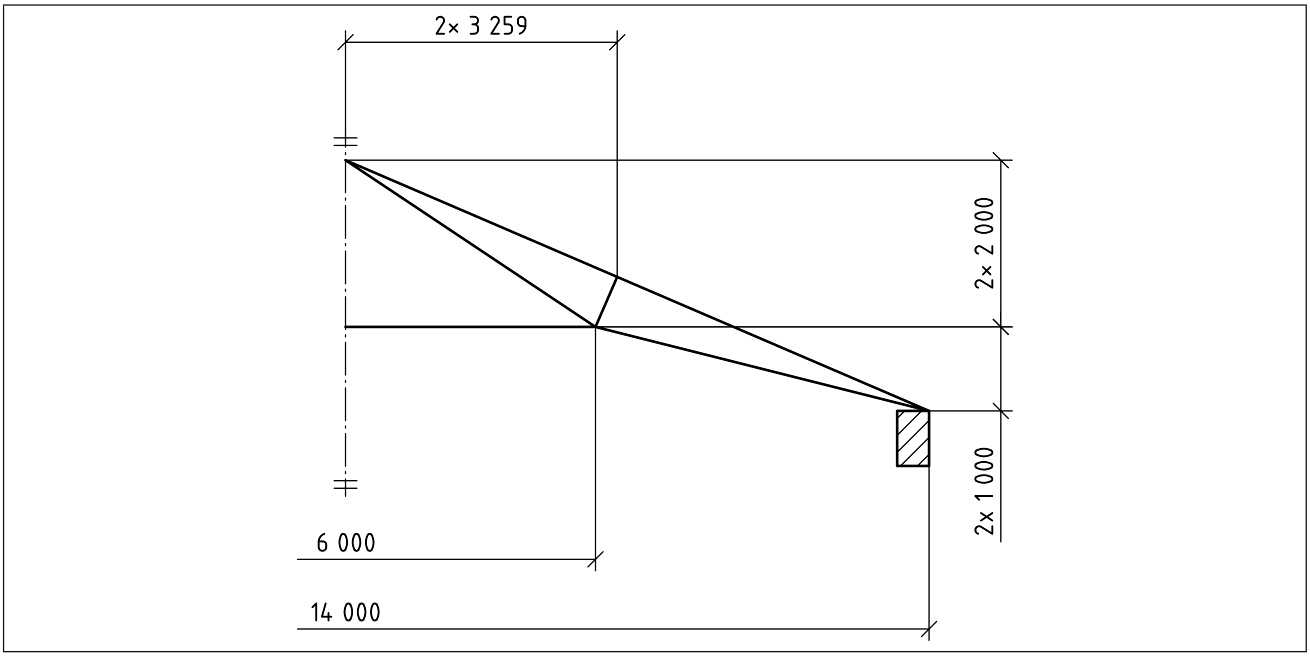


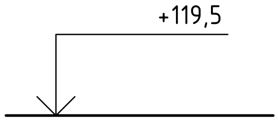
Figure 53 – *Symmetrical parts*



5.2.13 Indication of levels

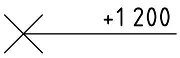
Levels on vertical views, sections and cuts shall be indicated by an open 90° arrowhead connected with a vertical line and horizontal line above which the numerical value of level is placed (see Figure 54).

Figure 54 – ***Indication of level on vertical view***



Levels for specified points on horizontal (planes) views and sections shall be indicated by a numerical value of the level placed above a line connected to the point indicated by "✕" (see Figure 55).

Figure 55 – ***Indication of level on horizontal view or section***



5.2.14 Dimensioning of curved features

5.2.14.1 Curved feature defined by radii

Where appropriate, a curved feature shall be defined by radii, as shown in Figure 56 and Figure 57.

Figure 56 – ***Example of curved feature defined by radii***

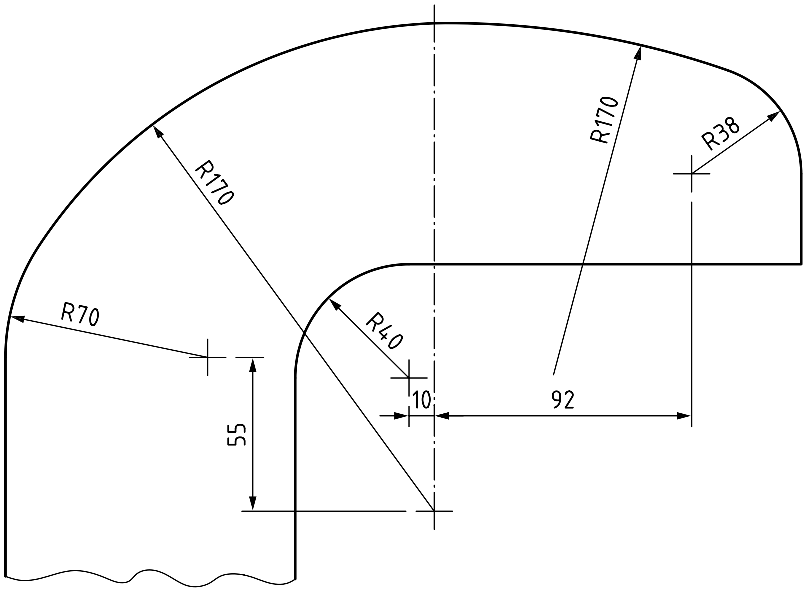
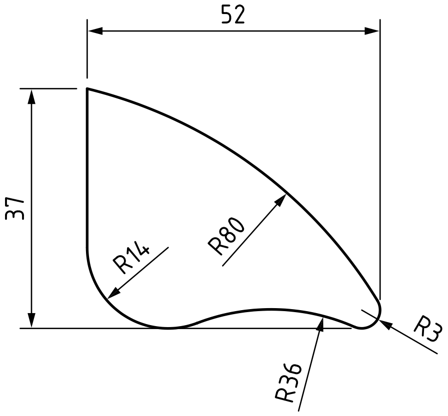


Figure 57 – ***Example of curved feature defined by radii***



5.2.14.2 Curved feature defined by coordinate dimensions

Dimensioning of curved features using linear or polar coordinates shall be indicated by dimensions to points on the profile (see Figure 58 and Figure 59).

Figure 58 – ***Curved feature defined by coordinate dimensions***

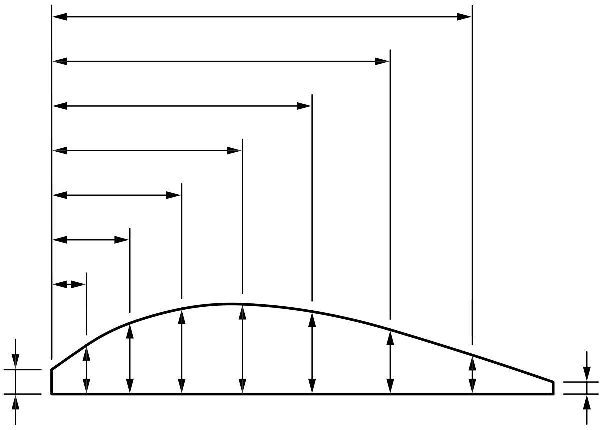
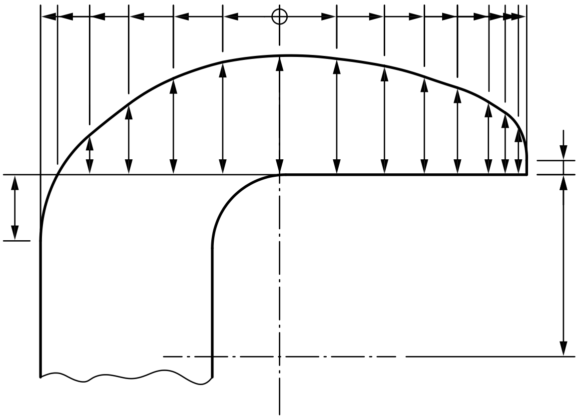


Figure 59 – ***Curved features defined by coordinate dimensions***



5.2.15 Arrangements of dimensions

5.2.15.1 General

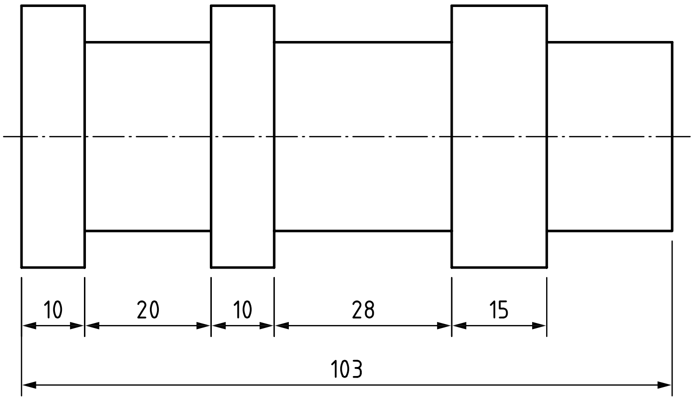
Dimensional values indicated in decimal notation shall use a comma as the decimal marker.

5.2.15.2 Chain dimensioning

When using chain dimensioning, chains of single dimensions shall be arranged in a row (see Figure 60).

Chains of single dimensions shall be used only where the possible accumulation of tolerances does not impinge on the functional requirements of the workpiece.

Figure 60 – *Example of chain dimensioning*



5.2.15.3 Parallel dimensioning

The dimension lines shall be drawn parallel in one, two or three orthogonal directions, or concentrically (see Figure 61 and Figure 62).

Figure 61 – *Example of parallel dimensioning*

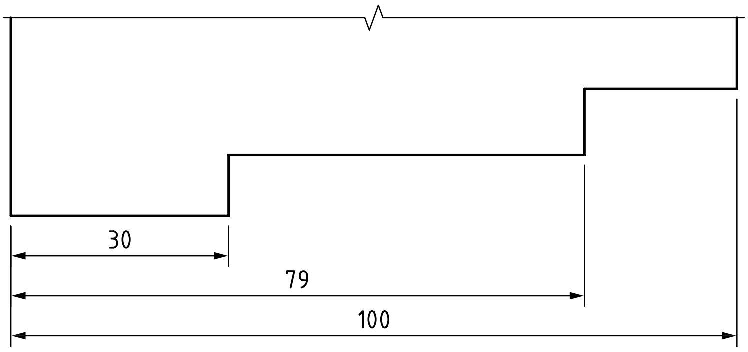
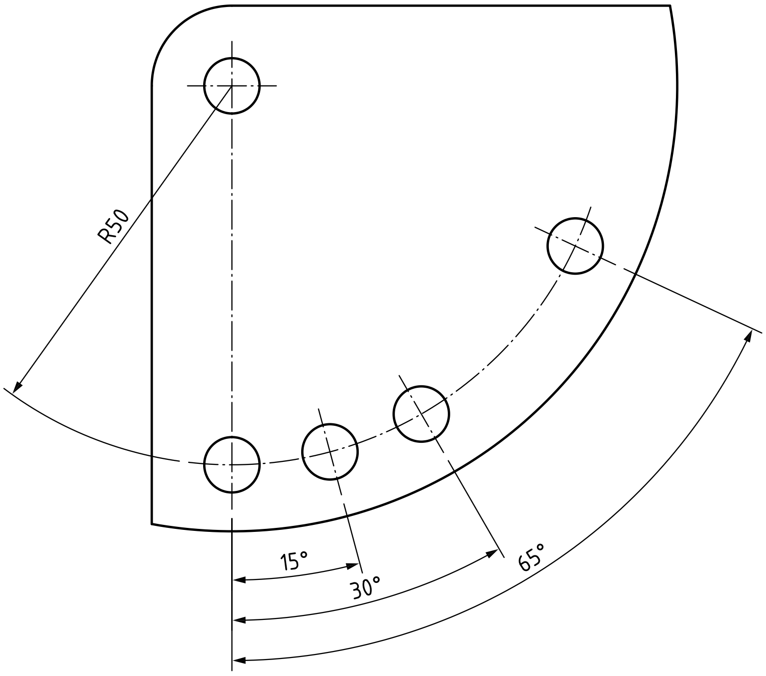


Figure 62 – *Example of parallel dimensioning*



5.2.15.4 Running dimensioning

5.2.15.4.1 General

Running dimensioning is simplified parallel dimensioning. The origin(s) of the dimension line(s) shall be indicated in accordance with **5.2.5** (see Figure 21).

Dimensional values shall be placed near the terminator, remote from the origin, and can be either:

- in line with the corresponding extension line (see Figure 63); or
- above and clear of the dimension line (see Figure 64).

NOTE An alternate representation of running dimensions can be used where:

- *the origin of the dimension(s) is shown in an appropriate location to indicate where the dimensions are measured from;*
- *the dimension values are shown on abbreviated dimension lines, where only one arrow is used, directed to the feature to which the dimension value applies (see Figure 63).*

5.2.15.4.2 Running dimensioning in two directions

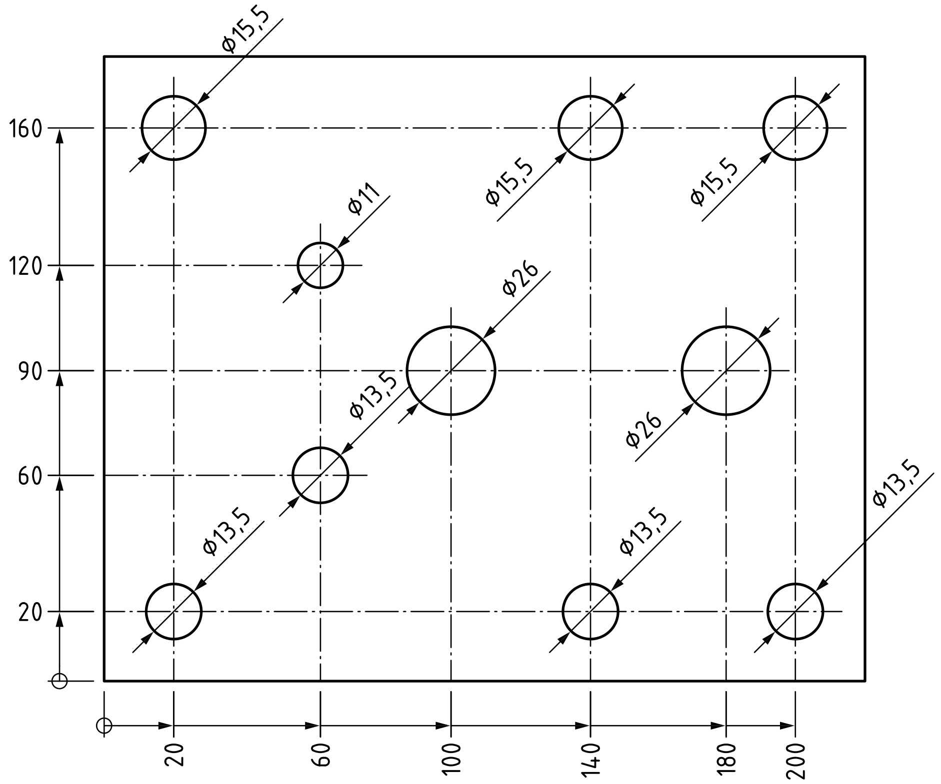
Running dimensioning in two directions shall utilize either:

- one continuous dimension line and one origin; or
- two dimension lines, inclined to each other, and two origins (see Figure 63).

In the case of two directions that are rectangular to each other, the running dimension shall be indicated by two origin symbols.

NOTE Figure 63 shows the running dimension values on the same line as the corresponding origin symbol.

Figure 63 – *Two dimension lines, inclined to each other, and two origins*

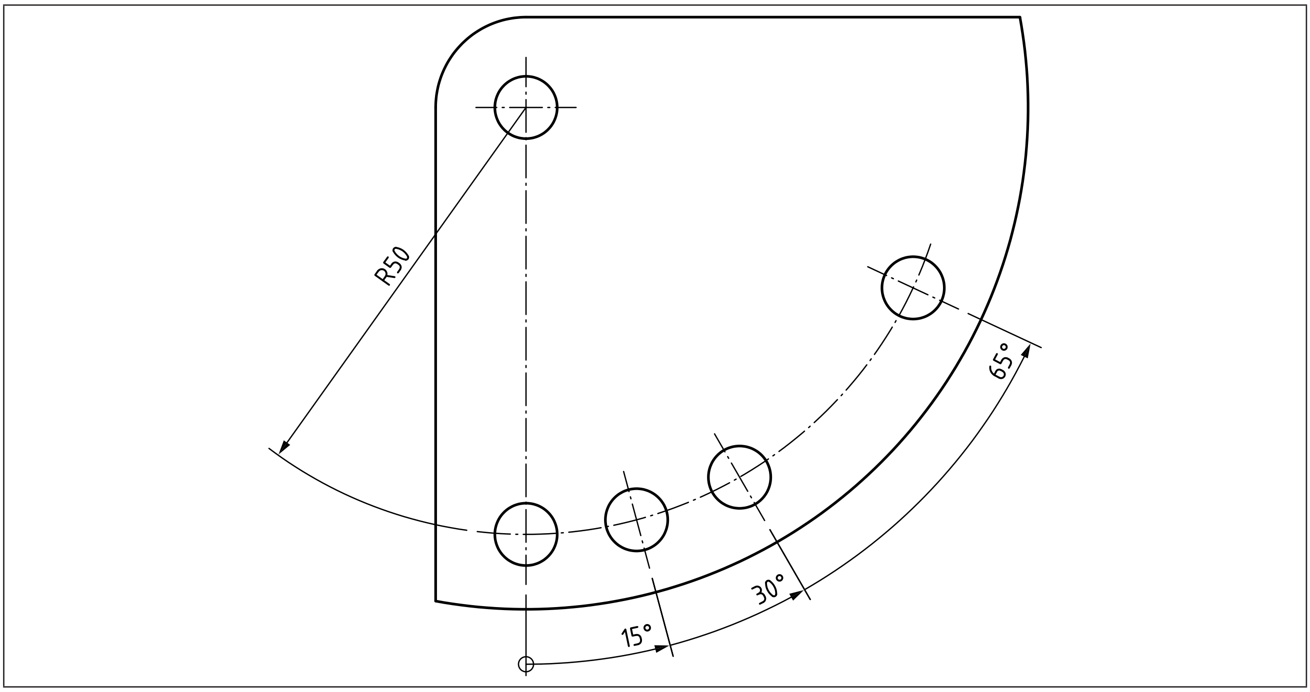


NOTE It is permissible to indicate only one dimension in the case of running dimensions in two directions.

5.2.15.4.3 Running dimensioning of angles

Running dimensioning of angles shall be indicated by one origin symbol and arrowheads to the extension lines of the feature (see Figure 64).

Figure 64 – *Running dimensioning of angles*



5.2.16 Cartesian coordinate dimensioning

*COMMENTARY ON **5.2.16***

Cartesian coordinates are defined starting from the origin by linear dimensions in orthogonal directions (see Figure 65 and Figure 66). Neither dimension lines nor extension lines are drawn.

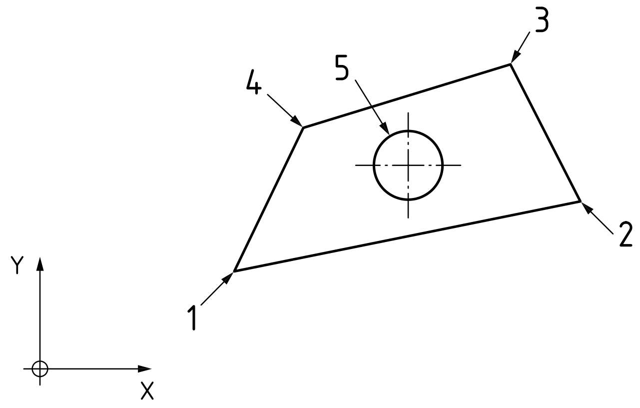
The dimensional values indicated in the negative directions shall have negative signs.

The origin used for coordinate dimensioning shall be indicated (see Figure 65).

The coordinates can be indicated by a reference letter(s) or number which appears in a table together with the value of the coordinate, or by the direct indication of the coordinates.

NOTE The reference letter(s) or number or the coordinate value can be placed adjacent to the coordinate location or indicated using a leader line.

Figure 65 – *Cartesian coordinate dimensioning*

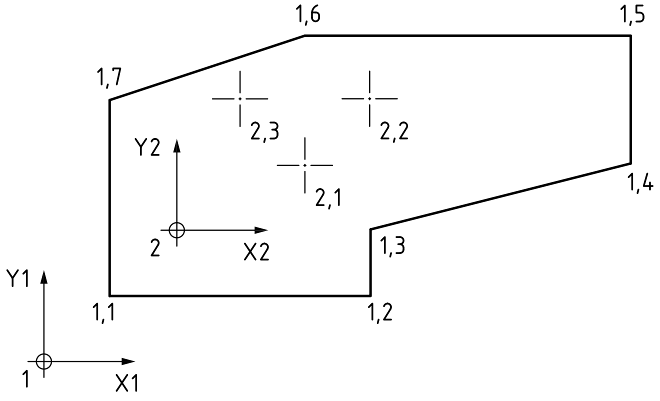


Position	X	Y	*d*
1	10	10	–
2	60	20	–
3	50	40	–
4	20	30	–
5	35	25	Ø10

NOTE Designations for position can be numerical or alphanumerical.

Where the main coordinate system has subsystems, the origin of the coordinate systems and the specific positions within the coordinate systems shall be numbered continuously by Arabic numerals. A point shall be used as a separation symbol (see Figure 66).

Figure 66 – *Cartesian coordinate dimensioning*



Coordinate origin	Position	X1, X2	Y1, Y2	*d*
1	1	0	0	–
1	1,1	10	10	–
1	1,2	50	10	–
1	1,3	50	20	–
1	1,4	80	30	–
1	1,5	80	50	–
1	1,6	30	50	–
1	1,7	10	40	–
1	2	20	20	Ø10
2	2,1	20	10	Ø5
2	2,2	30	20	Ø10
2	2,3	10	20	Ø5

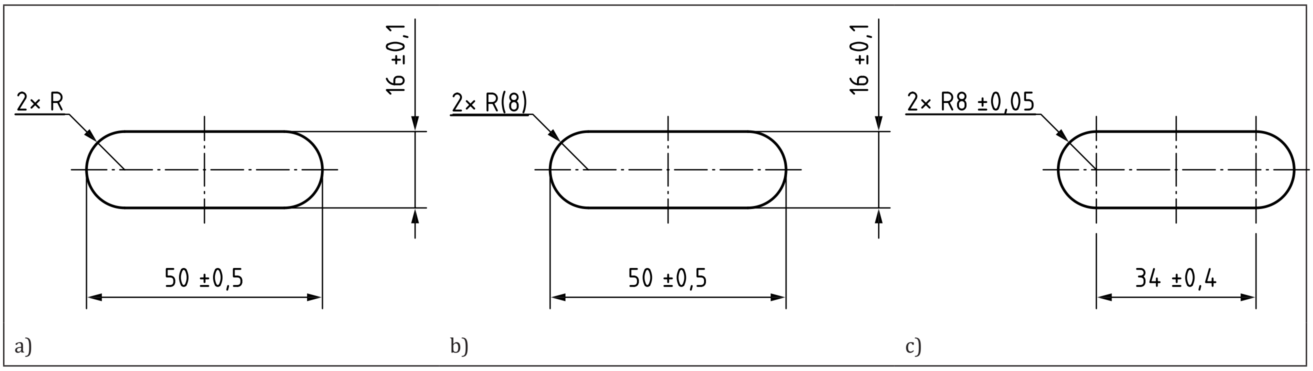
5.3 Dimensioning common features

5.3.1 Dimensioning of slots

Slots shall be specified in accordance with BS ISO 129-1 (see Figure 67).

NOTE For further information, see Annex I.

Figure 67 – *Dimensioning slots with rounded ends*



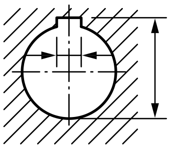
5.3.2 Keyways

Keyways in hubs or shafts shall be dimensioned by one of the methods shown in Figure 68.

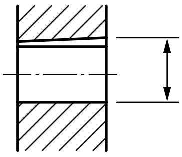
NOTE 1 Limit tolerances and/or geometrical tolerances are also required.

NOTE 2 Further information on keys and keyways is given in BS 4235-1 and BS 4235-2.

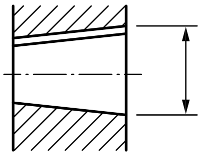
Figure 68 – *Dimensioning of keyways*



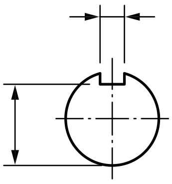
a) Parallel hub



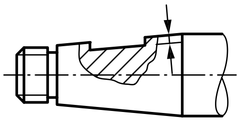
b) Tapered keyway



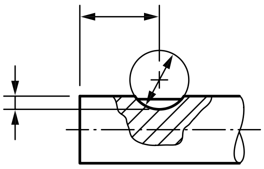
c) Parallel keyway



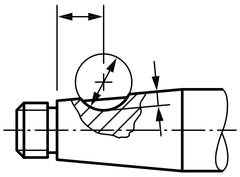
d) Parallel shaft



e) Parallel keyway in tapered shaft



f) Parallel shaft



g) Tapered shaft

5.3.3 Screw threads

The following standards provide the definition for metric ISO screw threads.

BS 3643-1	*ISO metric screw threads – Part 1: Principles and basic data*
BS 3643-2	*ISO metric screw threads – Part 2: Specification for selected limits of size*
BS 4827	*Specification for ISO miniature screw threads – Metric series*
BS ISO 261	*ISO general purpose metric screw threads – General plan*
BS ISO 262	*ISO general purpose metric screw threads – Selected sizes for screws, bolts and nuts*
BS ISO 965-1	*ISO general purpose metric screw threads – Tolerances – Part 1: Principles and basic data*

Screw threads shall be specified according to functional requirement.

5.4 Dimensional tolerancing

5.4.1 General

*COMMENTARY ON **5.4.1***

All features on component parts always have a size and a geometrical shape. For the deviation of size and for the deviations of the geometrical characteristics (form, orientation and location) the function of the part requires limitations which, when exceeded, impair this function.

The tolerancing on the drawing shall be complete to ensure that the elements of size and geometry of all features are controlled, i.e. nothing shall be implied or left to judgement in the workshop or in the inspection department.

NOTE 1 The use of general tolerances for size and geometry simplifies the task of ensuring that the prerequisites are met.

NOTE 2 See BS EN 22768-1 for more information on general tolerances.

5.4.2 Methods of specifying tolerances

The necessary tolerances shall be specified in one or more of the following ways:

a) separate indication on the drawing;

b) reference to general tolerances noted on the drawing;

c) reference to a standard containing general tolerances;

d) reference to other documents;

5.4.3 Indication of tolerances

5.4.3.1 General

Dimensional tolerancing for mechanical engineering shall be in accordance with BS EN ISO 14405 (all parts).

NOTE 1 BS EN ISO 14405 (all parts) can also be applied to fields other than mechanical engineering. Some of these rules are summarized here for information.

NOTE 2 Depending on the field of application, the tolerances of dimensions can be indicated by:

- *limit deviations;*
- *limits of dimension;*
- *general tolerances.*

All tolerances shall apply to the represented state of the feature in the technical drawing.

When tolerance limits are indicated in a vertical orientation (e.g. limit deviations, dimension limit values) the decimal marker of the upper and lower shall be aligned. When a tolerance limit is not shown with a decimal marker, the remaining digits shall be aligned as if the decimal marker had been displayed (see Figure 69, Figure 70, Figure 72, Figure 73 and Figure 75).

If two deviations relating to the same dimension are to be shown, both shall be expressed to the same number of decimal places (see Figure 69), except if one of the deviations is zero. This shall also be applied if the limits of size are indicated (see Figure 70).

Figure 69 – ***Example: expression of two deviations to the same number of decimal places***

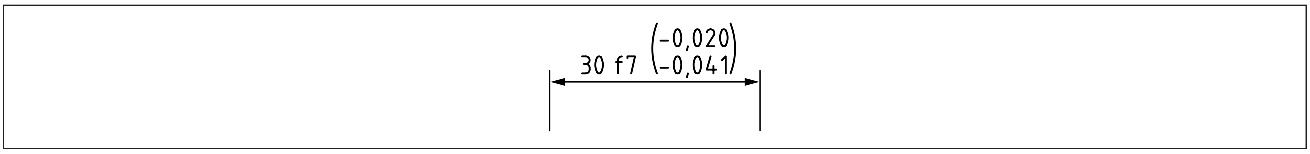
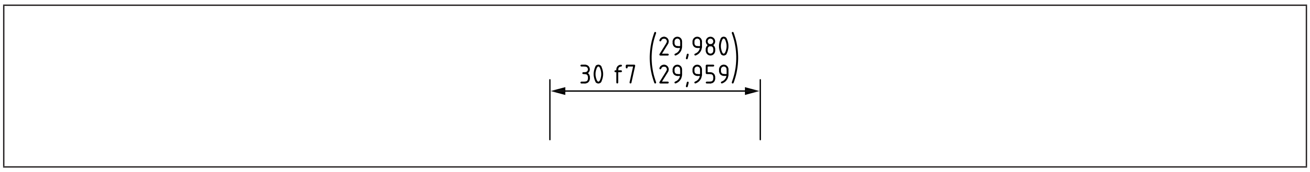
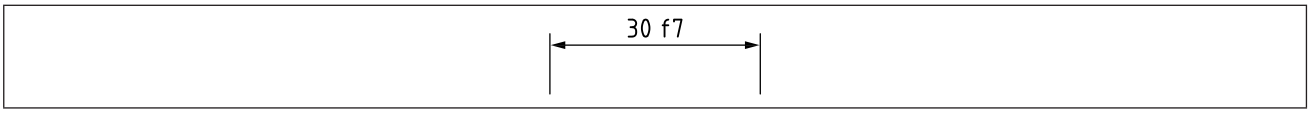


Figure 70 – ***Example: expression of two limits of size to the same number of decimal places***



NOTE 3 For dimensions displayed in accordance with BS EN ISO 2861, it is not necessary to express the values of the deviations unless they are needed (see Figure 71).

Figure 71 – ***Example: expression of deviations from dimensions displayed in accordance with BS EN ISO 286-1***



5.4.3.2 Limit deviations

The components of a toleranced dimension shall be indicated in the following order (see Figure 72 to Figure 76):

a) the dimensional value;

b) the limit deviations.

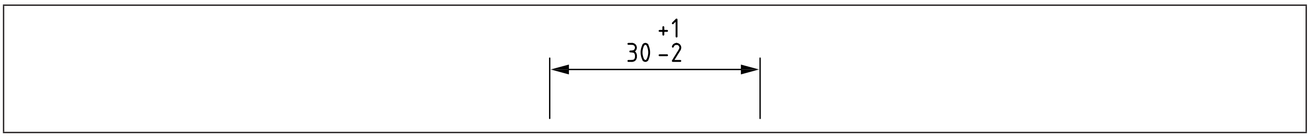
A space shall separate the dimensional values and the tolerance indication, e.g.:

- 55 ±0,2;
- 20 min;
- ø10 h7.

Limit deviations shall be expressed in the same unit as the dimensional value.

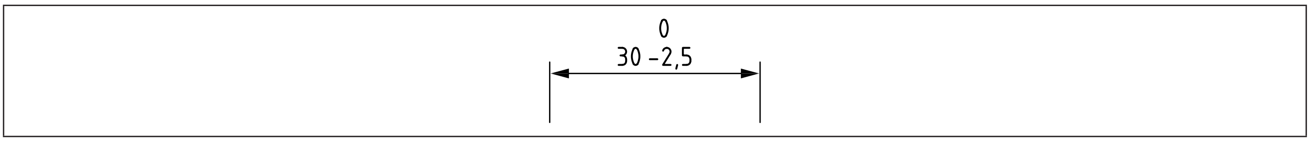
Limit deviations shall be indicated by indicating the upper deviation above the lower deviation (see Figure 72 and Figure 74).

Figure 72 – ***Components of a toleranced dimension***



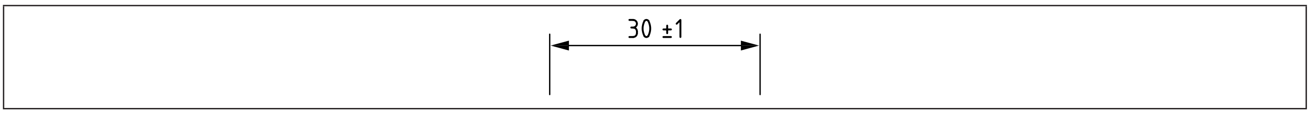
If one of the two limit deviations is zero, this shall be expressed explicitly by the digit zero shown without sign (see Figure 73).

Figure 73 – ***Components of a toleranced dimension***



If the tolerance is symmetrical in relation to the dimensional value, the limit deviation shall be indicated only once, preceded by the plus-minus sign (±) (see Figure 74).

Figure 74 – ***Components of a toleranced dimension***

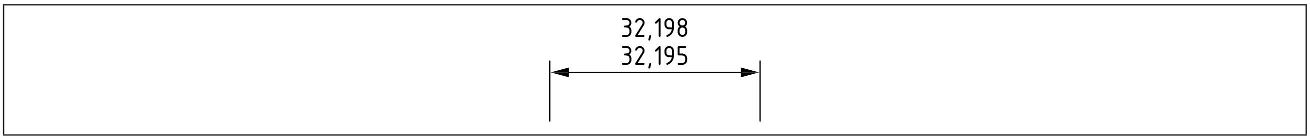


5.4.3.3 Limits of dimension

5.4.3.3.1 Maximum and minimum limit dimensions

The limits of dimensions shall be indicated by a maximum and a minimum dimension (see Figure 75). The larger dimension shall be placed above the smaller dimension.

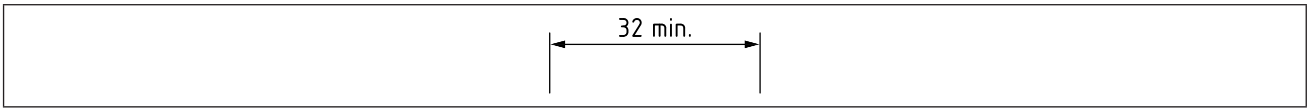
Figure 75 – ***Limits of dimensions***



5.4.3.3.2 Single limit dimensions

To limit the dimension in one direction only, the word “min.” or “max.” shall be added after the dimensional value (see Figure 76).

Figure 76 – ***Single limit dimension***

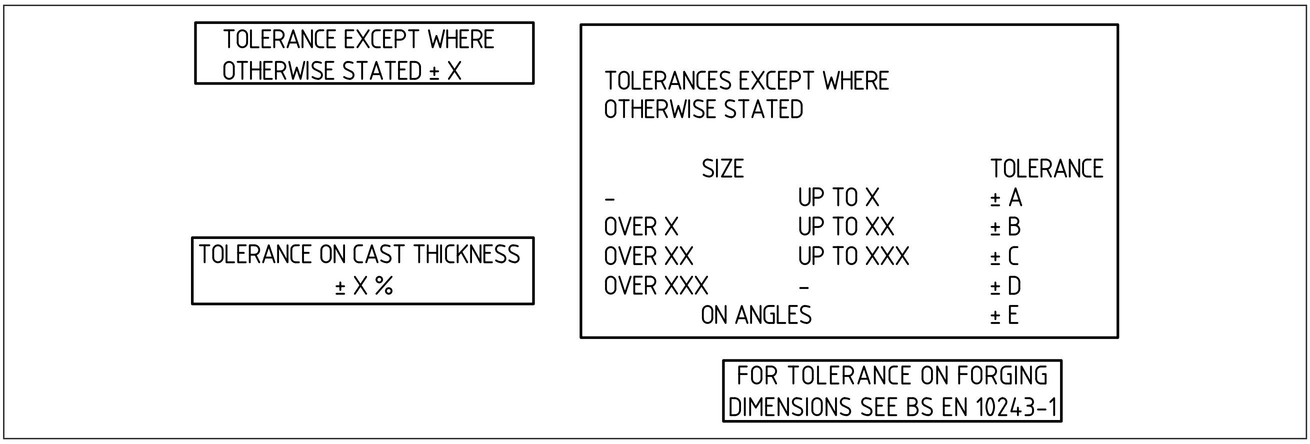


5.4.3.4 General tolerances

5.4.3.4.1 General tolerance notes

Where general tolerances are used these shall be noted on the drawing (see Figure 77).

Figure 77 – ***Use of general tolerance notes***



5.4.3.4.2 General tolerance standards

When use is made of BS EN 22768-1 for general tolerances, tolerances for dimensions shall be as given in Table 10, Table 11 and Table 12.

Table 10 – ***Permissible deviations for linear dimensions except for broken edges***

Tolerance class		**Permissible deviations for basic size range**							
Designation	**Description**	**0,5$^{A)}$ up to 3**	**over 3 up to 6**	**over 6 up to 30**	**over 30 up to 120**	**over 12 up to 400**	**over 400 up to 1 000**	**over 1 000 up to 2 000**	**over 2 000 up to 4 000**
f	fine	±0,05	±0,05	±0,1	±0,15	±0,2	±0,3	±0,5	–
m	medium	±0,1	±0,1	±0,2	±0,3	±0,5	±0,8	±1,2	±2
c	coarse	±0,2	±0,3	±0,5	±0,8	±1,2	±2	±3	±4
v	very coarse	–	±0,5	±1	±1,5	±2,5	±4	±6	±8

A) For nominal sizes below 0,5 mm, the deviations shall be indicated adjacent to the relevant nominal size(s).

NOTE Values in millimetres.

Table 11 – ***Permissible deviations for broken edges (external radii and chamfer heights)***

Tolerance class		**Permissible deviations for basic size range**		
Designation	**Description**	**0,5$^{A)}$ up to 3**	**over 3 up to 6**	**over 6**
f m	fine medium	±0,2	±0,5	±1
c v	coarse very coarse	±0,4	±1	±2

A) For nominal sizes below 0,5 mm, the deviations shall be indicated adjacent to the relevant nominal size(s).

NOTE Values in millimetres.

Table 12 – *Permissible deviations of angular dimensions*

Tolerance class		**Permissible deviations for ranges of lengths, in millimetres, of the shorter side of the angle concerned**				
Designation	**Description**	**up to 10**	**over 10 up to 50**	**over 50 up to 120**	**over 120 up to 400**	**over 400**
f	fine	±1°	±0°30′	±0°20′	±0°10′	±0°5′
m	medium					
c	coarse	±1°30′	±1°	±0°30′	±0°15′	±0°10′
v	very coarse	±3°	±2°	±1°	±0°30′	±0°20′

NOTE Due to the inherent risk of unintentionally over-specifying form and orientation controls that can result from the use of general geometrical tolerances, reference to BS EN 22768-2 is inadvisable. BS EN 22768-2 is in the process of being withdrawn. In such cases, general tolerance notes could be used to apply a common tolerance to many of the features on a drawing. The example shown in Figure 77 illustrates the wide field of application of this system.

5.4.3.5 Tolerances in other documents

When tolerances are controlled by another document or method, the document reference shall be indicated on the drawing.

5.4.4 Functional dimensions

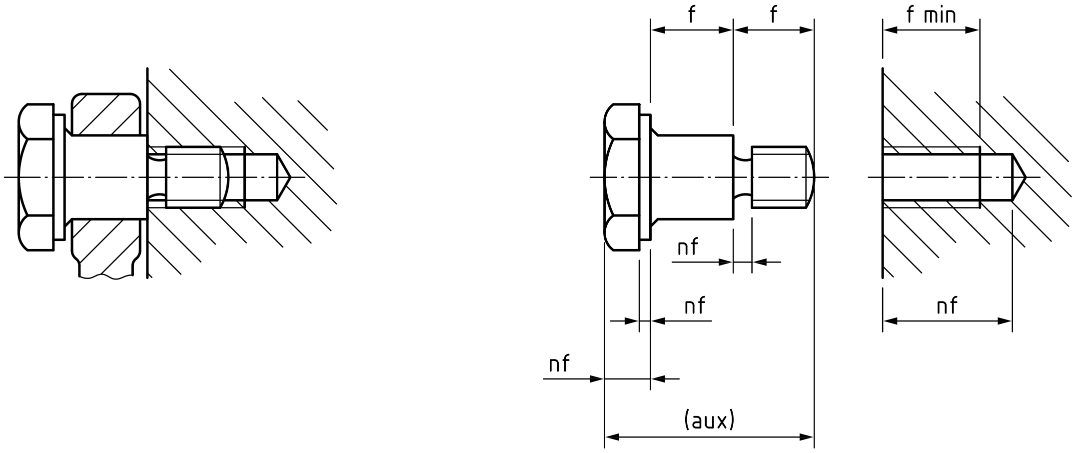
Functional dimensions shall be expressed directly (see Figure 78).

NOTE The application of this principle results in the selection of origin or datum features on the basis of the function of the product and the method of locating it in any assembly of which it could be a part.

If any datum feature other than one based on the function of the product is used, finer tolerances might be necessary and products which would satisfy functional requirements can be rejected because they exceed these finer tolerances (Figure 79).

*Expressing functional dimensions does not preclude the use of any dimensioning arrangement (see **5.2.15**) providing functional requirements are satisfied.*

Figure 78 – *Application of dimensions to suite functional requirements*

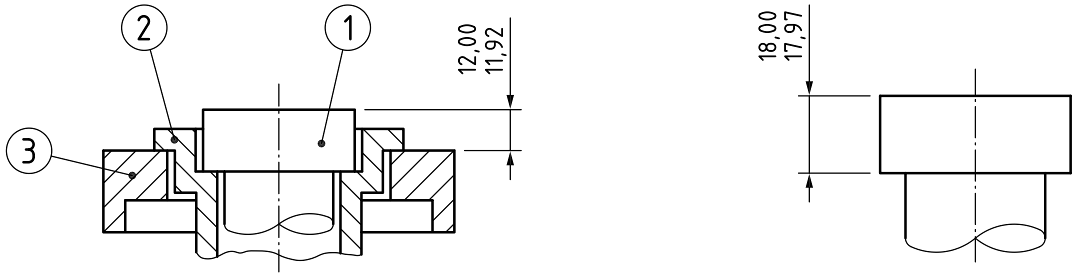


Key

f	Functional dimension
aux	Auxiliary dimension without tolerance for information only
nf	Non-functional dimension

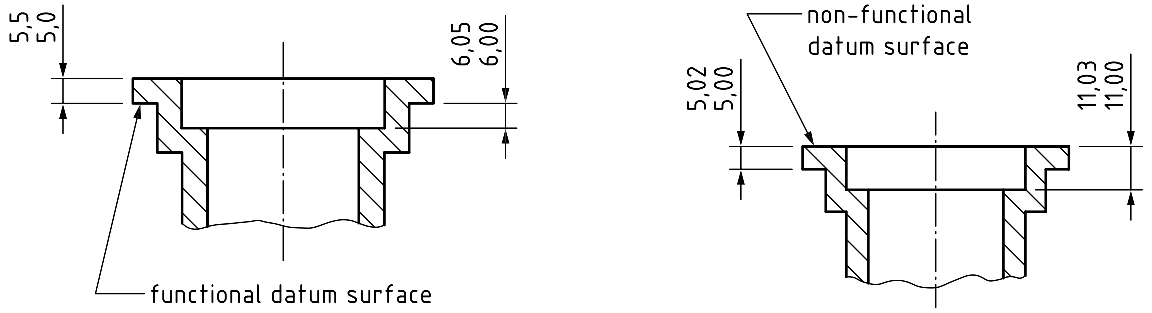
NOTE Geometric requirements have been omitted for simplicity.

Figure 79 – *Effect of functional dimensioning on tolerance values*



a) Assembly drawing showing a given functional requirement, namely the limits of height of the top face of item 1 above the top face of item 3, with a tolerance of 0,08 mm

b) Detail of head of item 1 showing given limits of size, with a tolerance of 0,03 mm



c) Item 2 dimensioned from a functional datum surface

d) Item 2 dimensioned from a non-functional datum surface

NOTE 1 One direct dimension with a tolerance of 0,05 mm is needed to satisfy the condition shown in a). A nominal flange thickness of 5 mm has been assumed. This value is non-functional in this case.

NOTE 2 Tolerances have had to be reduced; two dimensions with tolerances of, say, 0,02 mm for the flange and 0,03 mm are now needed to satisfy the condition shown in a).

Section 6 Datums and datum systems

COMMENTARY ON CLAUSE 6

This section describes some of the main definitions, principles and rules of working with datums and datum features. A more extensive description of datums and datum systems is given in in BS EN ISO 5459. This section is not intended to replace BS EN ISO 5459; it is only intended to make some of the key content more accessible.

Some additional options for working with datums are also described here which are not yet published in ISO standards (e.g. the use of non-default association methods). As some of these options are already in use in industry, they are included in this section with the caveat that whenever they are used, a clear explanation also appears in the specification, and the additional caveat that, when these options do become available through published ISO standards, their symbology and format might differ from those set out here.

6.1 General

Datums and datum systems shall be used where there is a requirement to define the geometry of a workpiece using geometrical tolerances.

A datum system shall be defined in the datum section of a tolerance indicator or other specification element (see **6.2.4**).

Where the datum system is composed of more than one datum, the datum system shall define the sequence in which the datums are used.

6.2 Definitions and explanations

6.2.1 Datum

A datum is a geometrically ideal feature which can be used to define the location or orientation of a tolerance zone.

A datum can be:

a) a point;

b) a straight line;

a) a flat plane; or

b) a combination of a), b) and c).

A datum consists of the situation feature(s) (see definition in **6.2.10**) of a datum feature (see definition in **6.2.2**).

The concept of six degrees of freedom (see definition in **6.2.9**) is very useful in understanding the way in which datums work.

6.2.2 Datum feature

The datum feature is a real integral feature or part of an integral feature.

NOTE The datum feature is often a feature on the workpiece itself but can also be a feature on a tool or fixture to which the workpiece is in contact.

6.2.3 Common datum

A common datum is a datum based on more than one datum feature.

6.2.4 Datum system

A datum system is a set of one or more datums used to locate or orientate a tolerance zone for a geometrical tolerance, or to define planes and directions for other specification elements such as intersection plane indicators (see **8.4.1**).

The datum system is used to constrain some or all of the six degrees of freedom for a tolerance zone (see **6.2.9**).

NOTE Not all geometrical tolerances require datum systems.

6.2.5 Datum section

A datum section is the section in a specification element containing one or more datum indicators (see Figure 80).

NOTE A datum section can be used as a part of a tolerance indicator, an intersection plane indicator, an orientation plane indicator, a collection plane indicator or a direction feature indicator (see BS EN ISO 1101).

Figure 80 – ***Datum section of a tolerance indicator defining a datum system***

a) Tolerance with a datum system consisting of two datums

b) Tolerance with a datum system consisting of three datums

6.2.6 Primary datum

The primary datum is the datum listed first in the datum section of a geometrical specification.

This datum is not influenced by constraints from other datums.

6.2.7 Secondary datum

The secondary datum is the datum listed second in the datum section of a geometrical specification in a datum system.

In a datum system, the secondary datum has its location and orientation constrained to the primary datum, unless otherwise indicated.

NOTE In BS EN ISO 5459:2011, the secondary datum is described as only having orientation constraints to the primary datum. This is contrary to how datum systems are set up in most cases. The next revision of BS EN ISO 5459 will change this to location and orientation constraints.

6.2.8 Tertiary datum

The tertiary datum is the datum listed third in the datum section of a geometrical specification.

In a datum system, the tertiary datum has its location and orientation constrained to the primary and secondary datums, unless otherwise indicated.

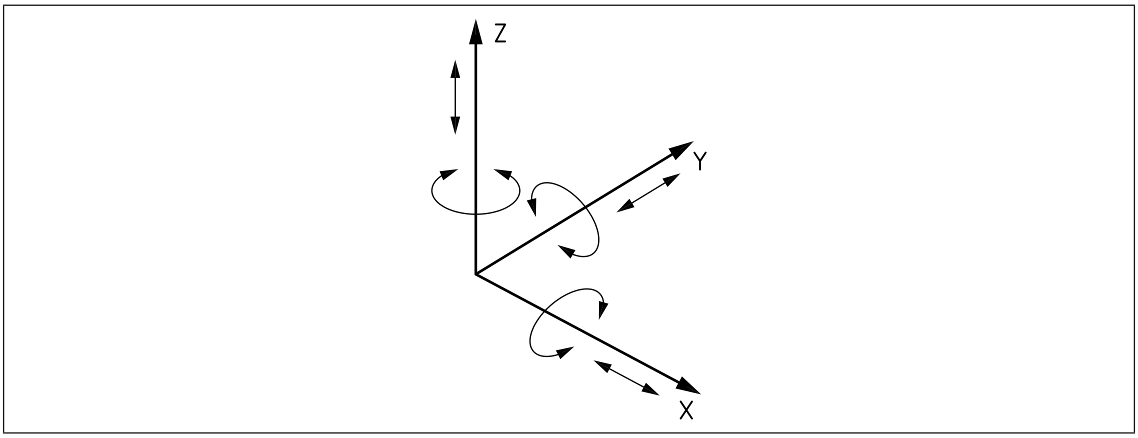
NOTE In BS EN ISO 5459:2011, the tertiary datum is described as only having orientation constraints to the primary and secondary datums. This is contrary to how datum systems are set up in most cases. The next revision of BS EN ISO 5459 will change this to location and orientation constraints.

6.2.9 Six degrees of freedom

A rigid body is said to have six degrees of freedom (see Figure 81). These can be considered as the freedom to translate along each of the three axes of a Cartesian coordinate system, and to rotate around each of these axes. Thus, a rigid body has three translational degrees of freedom and three rotational degrees of freedom.

Some or all of these degrees of freedom can be locked by constraining the rigid body in various ways.

Figure 81 – ***Degrees of freedom***



6.2.10 Situation feature

A situation feature is a point, line, plane or combination of these elements, used to define the location or orientation of a feature.

For example, the situation feature for a planar surface is a flat plane, the situation feature for a cylinder is a straight line (the axis of the cylinder), and the situation feature for a sphere is a point (the centre point of the sphere) (see Table 13).

Locking the location or orientation of a situation feature causes some of the degrees of freedom of the surface to be locked.

Some types of surface have more than one situation feature. A cone has two situation features, a straight line (axis) and a point.

The situation features for any real (non-ideal) manufactured surface are based on the associated feature. The associated feature is an ideal feature which is fitted to the real feature, or to data extracted from the real feature (see BS EN ISO 17450-1:2011, **6.3**, and BS EN ISO 5459).

Table 13 – *Situation features*

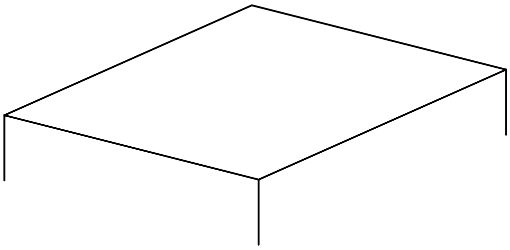
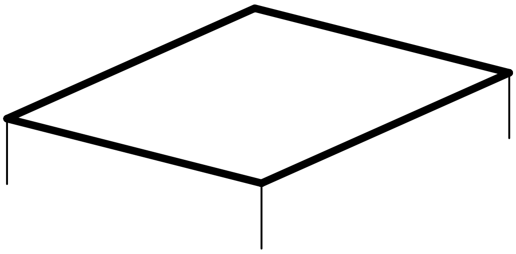
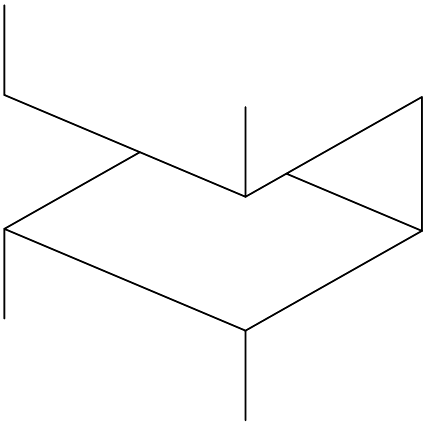
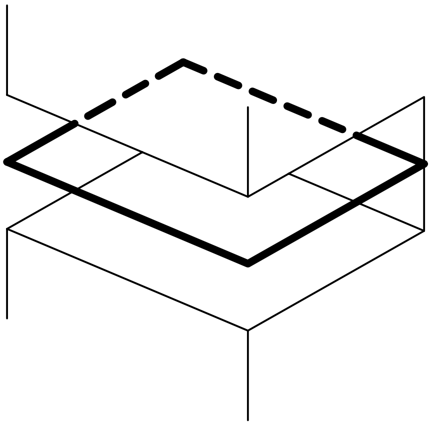
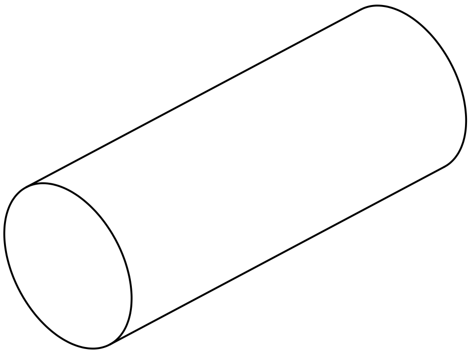
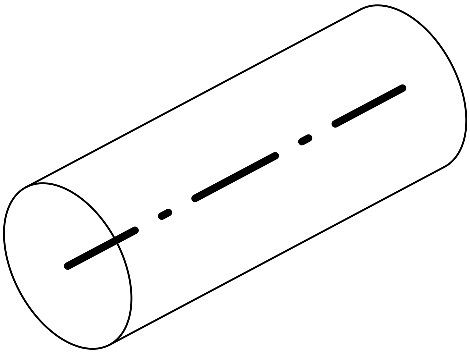
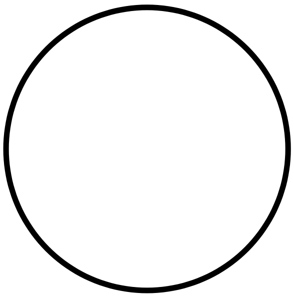
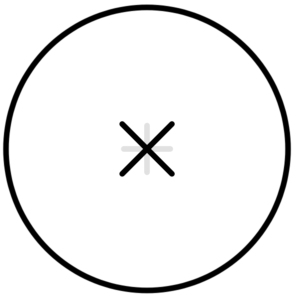
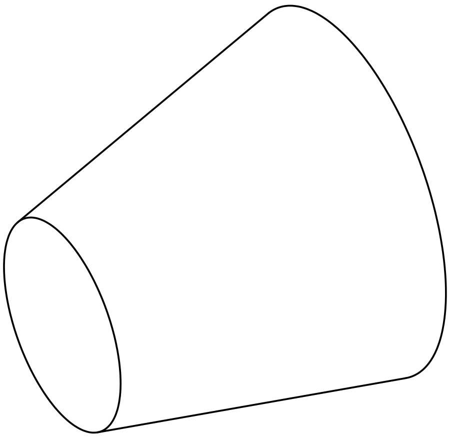
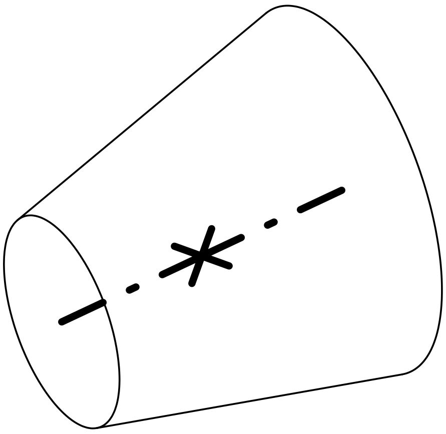
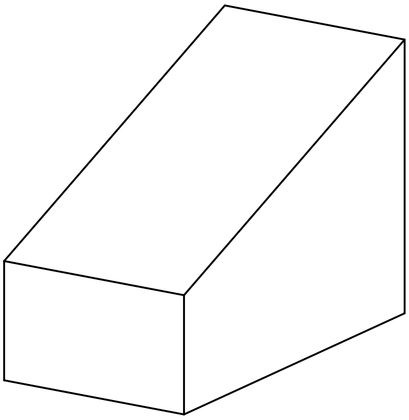
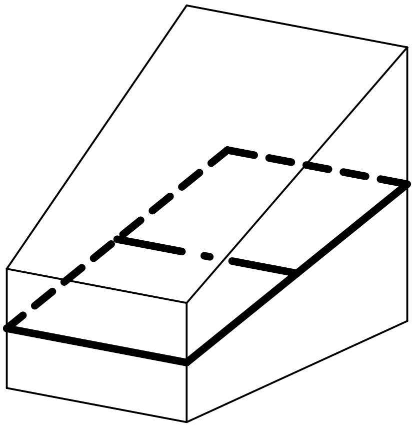
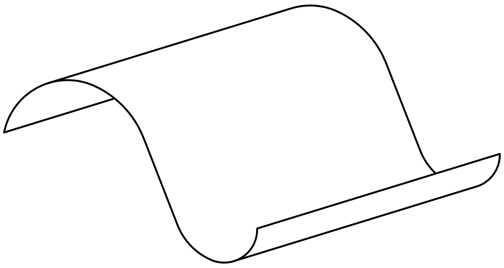
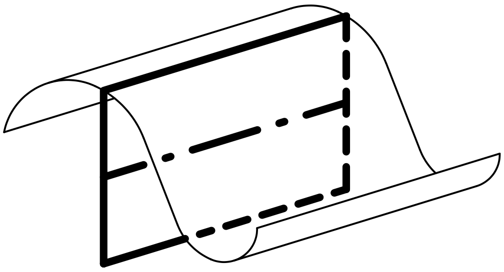
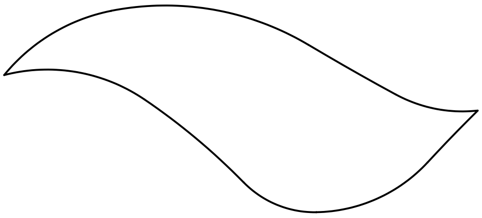
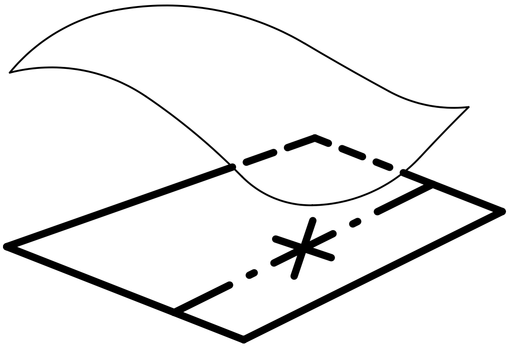
Feature type	**Situation feature(s)**
Plane: 	Plane: 
Two parallel opposed planes: 	Plane: 
Cylinder: 	Straight line: 
Sphere: 	Point: 

Table 13 – *Situation features (Cont.)*

Feature type	**Situation feature(s)**
Cone: 	Straight line point and point: 
Wedge (prismatic): 	Plane and straight line: 
Extruded profile: 	Plane and straight line: 
Complex: 	Plane straight line and point: 

6.2.11 Associated feature

An associated feature is an ideal feature which is fitted to a non-ideal (real or extracted) feature using an association criterion.

NOTE 1 The associated features used to establish datums are normally intended to simulate the contact or interface with other components.

NOTE 2 See Annex C for further information about association.

6.2.12 Association criterion

An association criterion is an objective function combined with association constraints.

Example 1 (see Figure C.3):

The default association for a datum plane based on a nominally flat datum feature is for the associated plane to be in contact with the datum feature, but outside the material, arranged such as to minimize the maximum distances from the datum to any point on the datum feature. This is known as a “minimax” association.

Example 2 (see Figure C.5):

The default association for a datum axis based on a cylindrical datum feature is to use the minimum circumscribed feature for an external cylinder, or the maximum inscribed feature for an internal feature.

NOTE 1 See Annex C for a full list of the different association criteria.

NOTE 2 The next planned revision of BS EN ISO 5459 will change the default association criterion to the Gaussian (least squares) objective function with the “outside the material” constraint. This will result in datum definitions which more closely correspond to the “real life” behaviour of mechanical parts and have improved reproducibility.

6.2.13 Objective function (for association)

An objective function is the method or algorithm which defines how the associated feature is fitted to a real feature.

Objective functions include:

- minimax;
- Gaussian (least squares);
- minimum circumscribed; and
- maximum inscribed.

NOTE See Annex C for further details.

6.2.14 Association constraints

Association constraints are additional constraints which may be combined with the objective function when constructing an associated feature.

Association constraints include:

- location constraints (e.g. when constructing a secondary or tertiary datum);
- orientation constraints (e.g. when constructing a secondary or tertiary datum);
- material constraints (e.g. outside the material, inside the material, etc); and
- size constraints (e.g. size fixed or size variable).

NOTE See Annex C for further details.

6.3 Identifying datum features: Placement of the datum feature indicator

6.3.1 Datums based on single feature

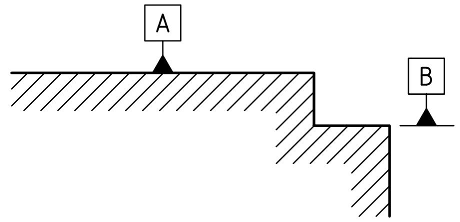
Where a datum is based on a single datum feature, the datum feature shall be identified with a datum feature indicator.

*NOTE The datum is based on the entire surface of the datum feature, unless otherwise indicated. If the datum is based on part of the datum feature, datum targets are used (see **6.8.1**).*

If the feature is not a feature of size, the datum feature indicator shall be placed in one of the following positions.

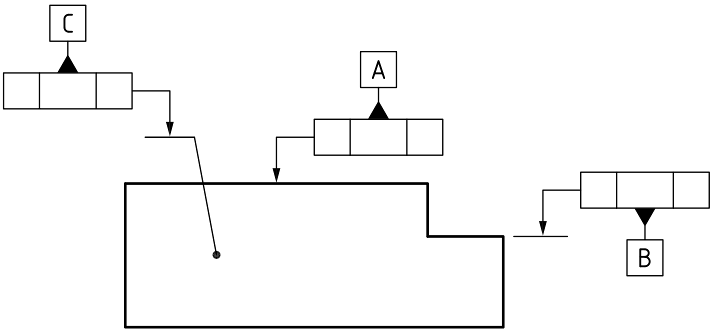
a) Attached to an outline, or an extension line from an outline, representing the feature in profile (as shown in Figure 82).

Figure 82 – ***Datum feature indicators attached to an outline and an extension line from an outline***



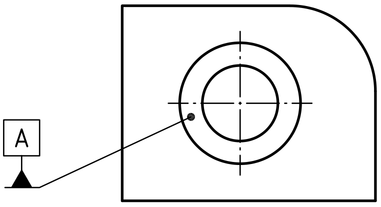
b) Attached to a tolerance frame which is attached to the feature (as shown in Figure 83).

Figure 83 – ***Datum feature indicators attached to a tolerance frame attached to the feature***



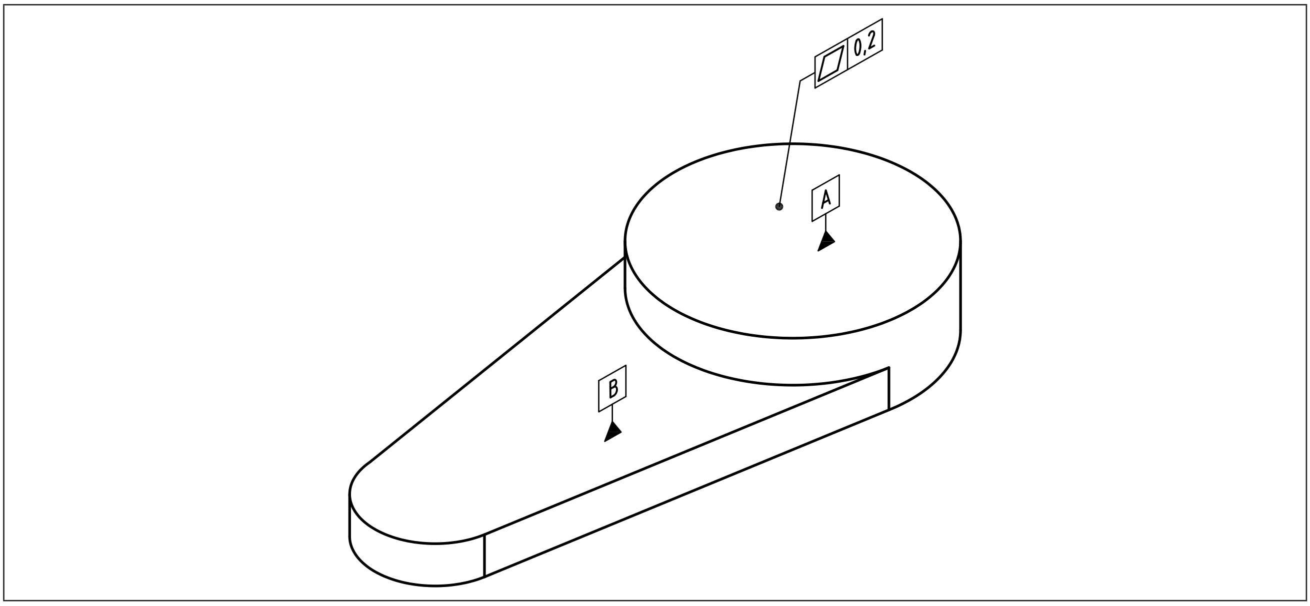
c) Attached to a visible surface with a leader line ending in a filled dot on the surface (as shown in Figure 84).

Figure 84 – ***Datum feature indicator attached with leader line ending in a filled dot on surface***



d) Attached to a visible surface on a 3D TPS (as shown in Figure 85).

Figure 85 – *Datum feature indicators attached to surface on 3D TPS*

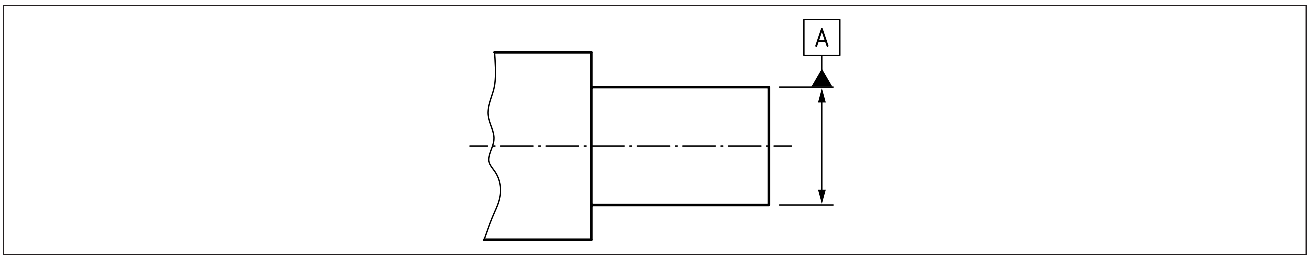


If the datum feature is a feature of size, the datum feature indicator shall be placed in one of the following positions.

1) In line with the dimension line for the size dimension of the feature (as shown in Figure 86).

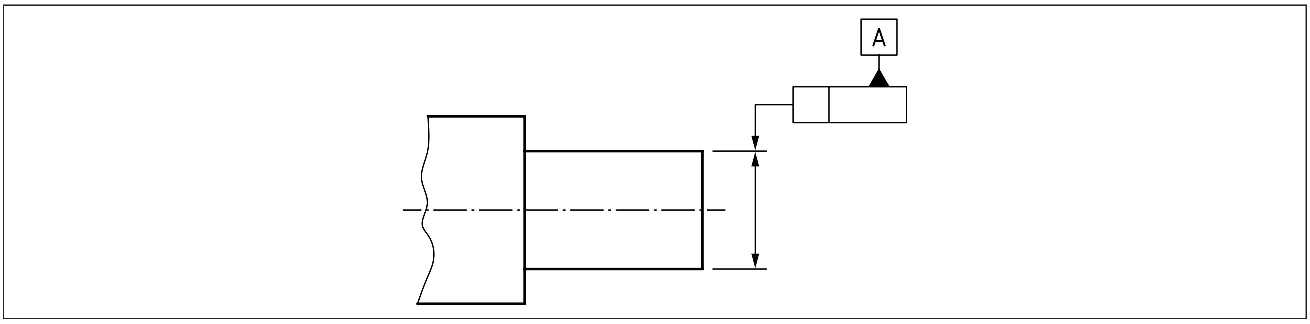
NOTE The datum feature indicator is placed in line with the size dimension to unambiguously identify the feature that the datum is to be derived from.

Figure 86 – *Datum feature indicator placed in line with dimension line for size dimension*



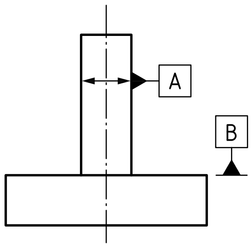
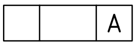
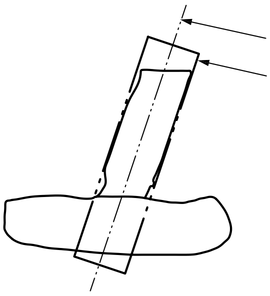
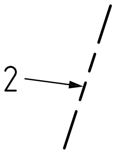
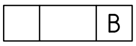
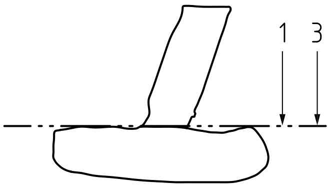
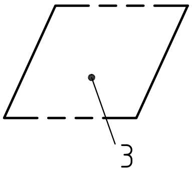
2) Attached to a tolerance frame which has its leader line in line with the dimension line for the size dimension of the feature (as shown in Figure 87).

Figure 87 – *Datum feature indicators attached to tolerance frame*



NOTE 2 An example of a datum based on a single datum feature is given in Table 14.

Table 14 – *Example: single datum*

Indication of datum feature	**Indication of datum in tolerance frame**	**Illustration of the meaning**	**Invariance class and situation feature (see 7.7)**	**Datum**
			Cylindrical Axis of associated cylinder	
			Planar Associated plane	

Key

1 Associated feature (without orientation constraint)

2 Straight line which is the situation feature of the associated cylinder (its axis)

3 Plane which is the situation feature of the associated plane (the associated plane itself)

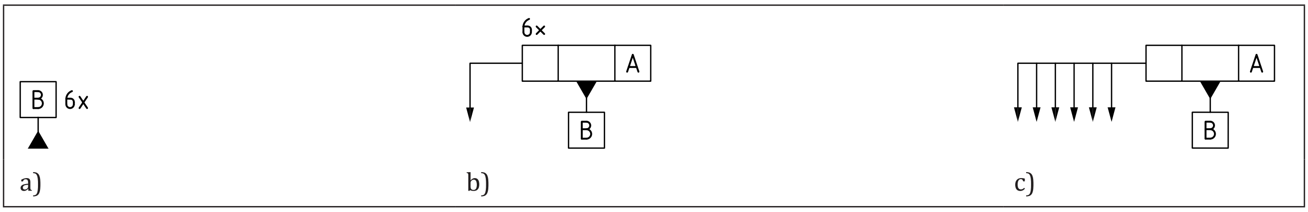
NOTE Association for single datums is described in BS EN ISO 5459:2011, Annex A.

6.3.2 Datums based on multiple datum features

Where multiple datum features are to be used to define a specific situation feature of a datum, this shall be shown in accordance with Figure 88.

NOTE Where multiple individual datum features are used to define a single datum, simplification can be used (see Figure 88).

Figure 88 – *Examples of datum indications based on multiple datum features*

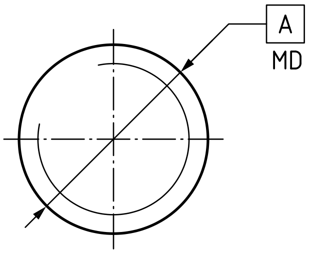


a) b) c)

6.3.3 Datums based on threaded features, gears or splines

If the datum feature is a threaded feature, it shall be treated like a cylindrical feature of size. The datum feature indicator shall be placed in line with the thread dimension or attached to a tolerance frame which is attached in line with the thread dimension (see Figure 89). By default, the datum shall be a datum axis based on the pitch cylinder of the thread. The datum definition can be altered by placing the MD (major diameter) or LD (minor diameter) modifier adjacent to the datum feature indicator (see BS EN ISO 1101).

Figure 89 – ***Datum feature indicator attached to tolerance frame in line with thread dimension***



If the datum feature is a cylindrical gear form or a splined cylindrical feature, it shall be treated like a cylindrical feature of size.

The datum feature indicator shall be placed in line with the size dimension or attached to a tolerance frame which is attached in line with the size dimension for the feature.

There is no default definition of the datum axis based on a gear form or spline. The datum definition shall be completed by placing the PD (pitch diameter), MD or LD modifier adjacent to the datum feature indicator (see BS EN ISO 1101).

6.4 Deriving datums from datum features

The datum feature shall be identified in a TPD with a datum feature identifier (see **6.3**).

The datum section of a specification element shall determine whether the datum is primary, secondary or tertiary.

An associated feature shall be constructed for the datum feature using an association criterion.

NOTE 1 The associated feature may be constructed either for the real datum feature or for the extracted datum feature. Constructing the associated feature for the real datum feature involves using physical devices such as surface tables, angle plates, v-blocks, collets. Constructing the associated feature for the extracted datum feature involves a mathematical operation to build the associated feature based on the data which have been sampled from the datum feature.

For a primary datum feature, the associated feature shall be based on the datum feature alone, with no additional location or orientation constraints.

For a secondary datum feature, the associated feature shall be based on the datum feature, but with additional location and orientation constraints relating it to the primary datum, unless otherwise indicated.

For a tertiary datum feature, the associated feature shall be based on the datum feature, but with additional location and orientation constraints relating it to the primary and secondary datums, unless otherwise indicated.

The datum is the situation feature(s) of the associated feature.

NOTE 2 See Annex C for examples.

6.5 Indications of situation features in a technical product specification

*COMMENTARY ON **6.5***

When a datum feature has a single situation feature, the situation feature has a unique location.

For example, if the datum feature is a cylinder, the situation feature is a straight line, and it can only be centred in the associated cylinder; if the datum feature is a flat surface, the situation feature can only be a tangent plane.

When the datum feature has more than one situation feature, at least one of the situation features does not have a unique location.

For example, if the datum feature is a cone, the situation features are a line and a point. The line (the axis of the cone) has only one location, but the point can be taken to be anywhere along that line.

The location of the point is determined by the specifier; it can be taken as the point where a "gauge diameter" for the cone is defined or as the point at the apex of the cone, or at any other defined location on the axis of the cone.

Where necessary, the situation features shall be identified on a drawing or in a CAD model.

The situation features shall have a theoretically exact relationship with the nominal datum feature.

When these are shown on the specification, the situation features shall be represented as follows:

a) point: by a cross;

b) line: by a long-dashed double-dotted narrow line (line type 05.1 from BS ISO 128-24:1999); when shown end on, the line is represented by a point [see a)];

c) plane: by a rectangular plane outlined with a long dashed double-dotted narrow line; when shown edge on, the plane is represented by a line [see b)].

The situation features shall be labelled with an identifier as follows:

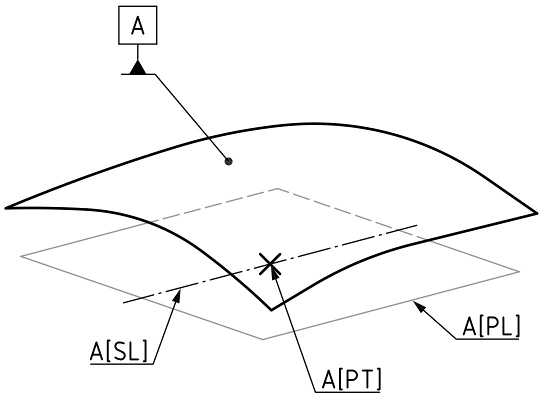
1) point: by an identifier consisting of the name of the datum, followed by PT in square brackets;

2) line: by an identifier consisting of the name of the datum, followed by SL in square brackets (see Figure 90);

3) plane: by an identifier consisting of the name of the datum, followed by PL in square brackets (see Figure 90).

Where necessary, these identifiers shall be shown in more than one drawing view.

Figure 90 – ***Example identifiers for situation features***



NOTE 2 See Annex E for further information regarding implementation in CAD.

6.6 Common datums

6.6.1 General

COMMENTARY ON 6.6.1

A common datum is a single datum based on more than one datum feature.

A common datum shall be identified by listing the individual datum feature identifiers in a single datum compartment of the datum section. The individual datum feature identifiers shall be separated by a hyphen, e.g.:

◎	⌀0,2	A–B

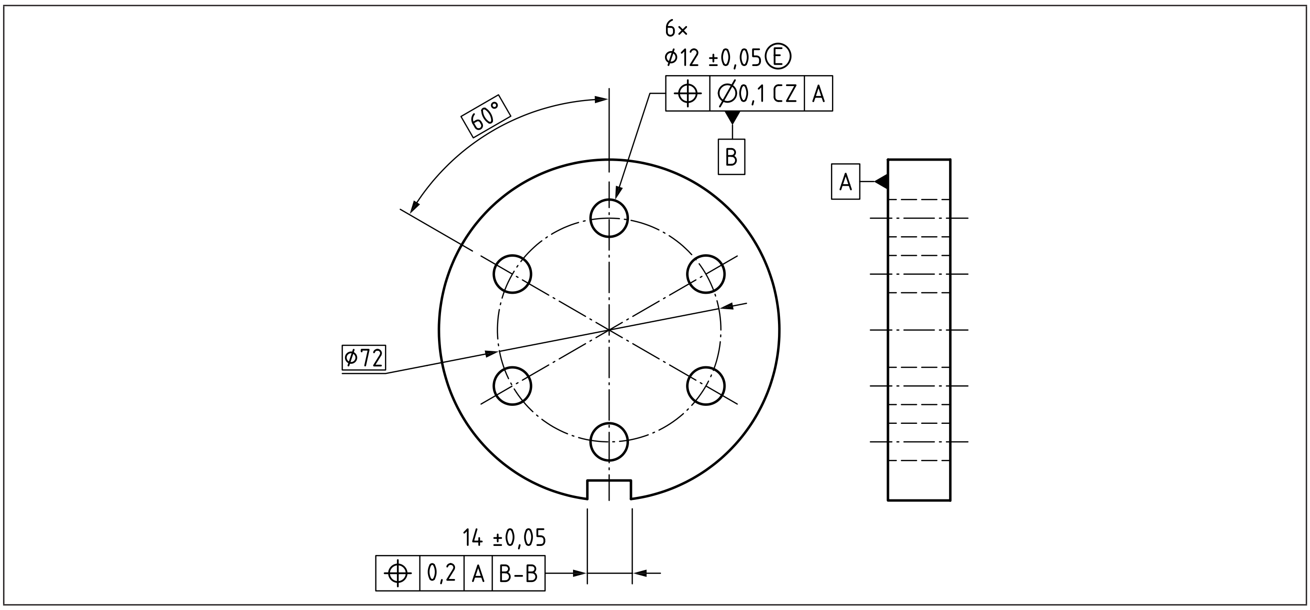
Where a common datum is based on a group of identical features, which are identified with a single datum feature indicator, as in Figure 91, the datum feature identifier shall be shown twice, separated by a hyphen, e.g.:

⌖ | 0,2 | A | B-B

When a common datum is used, the relationship between the individual datum features shall be toleranced.

When a common datum is constructed, the theoretically exact relationships between the individual datum features shall also apply to the associated features which are used to construct the datum.

Figure 91 – ***Common datum established by a group of identical features***



6.6.2 Common datums based on two coaxial cylinders

Where a single datum axis defined by two coaxial, cylindrical features used simultaneously (as shown in Figure 92) is a primary datum, it shall consist of:

a) a single axis constructed from the minimum circumscribing cylinder which can be constructed around datum feature A; and

b) the minimum circumscribing cylinder which can be constructed around datum feature B simultaneously,

where the two minimum circumscribing cylinders are additionally constrained to be coaxial with each other (see Figure 93).

Figure 92 – ***Common datum established from two coaxial cylinders***

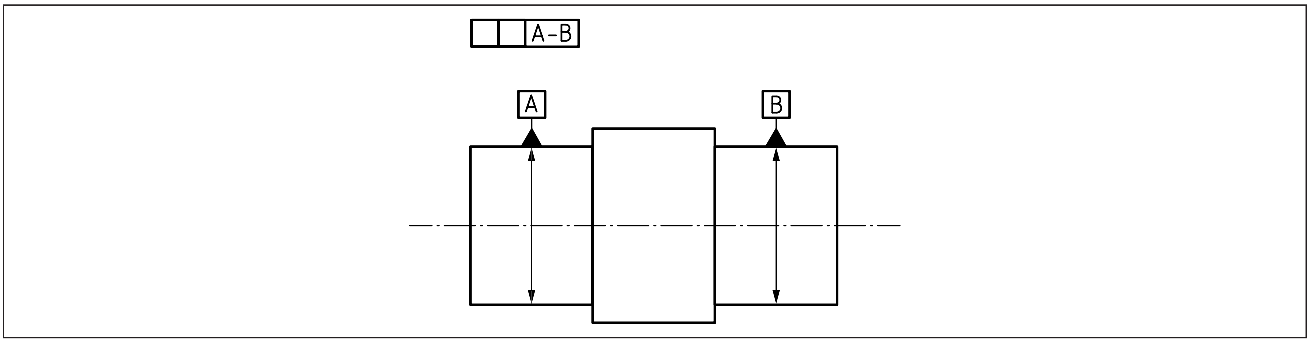
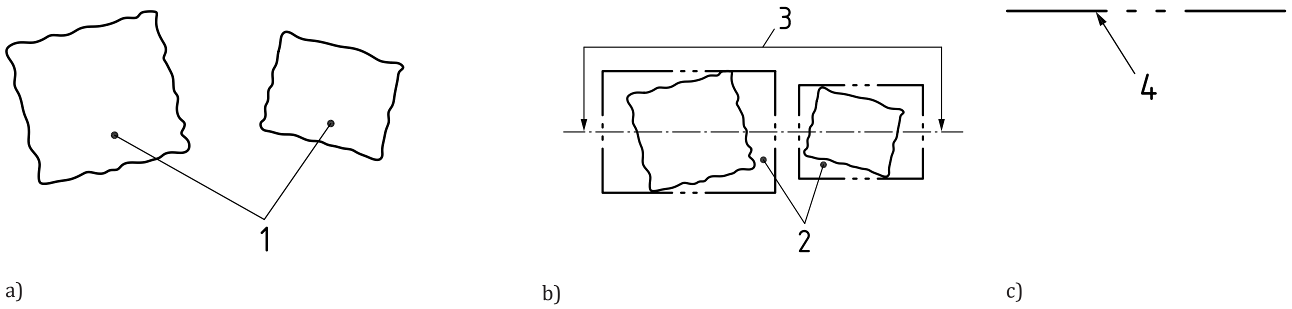


Figure 93 – ***Establishing a common datum from two coaxial cylinders***



a) b) c)

Key

1 Datum feature: real integral feature

2 Associated features with variable diameter

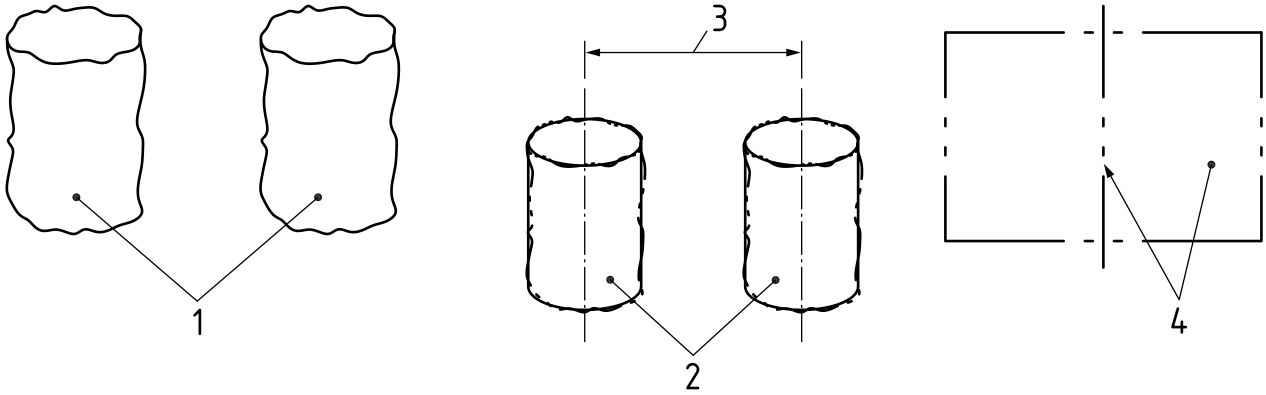
3 Orientation constraint (parallelism) and location constraint (coaxiality)

4 Single datum – situation feature of the associated feature: straight line (common axis)

6.6.3 Common datums based on two parallel cylinders

Where a single datum defined by two parallel, cylindrical holes is a primary datum, a maximum inscribing cylinder shall be constructed for each of the two holes simultaneously, with the additional constraint that the two inscribing cylinders shall remain parallel with each other, and a fixed distance (*l*) apart. The datum shall consist of two situation features, an axis and a plane. The axis shall be a mean between the axes of the two inscribing cylinders, and the plane shall be a plane through that axis, and the axis of either one of the inscribing cylinders (see Figure 94).

Figure 94 – ***Establishing a common datum from two parallel cylinders***



a) b) c)

Key

1 Datum features: real integral features

2 Associated features with variable diameters

3 Orientation constraint (parallelism) and location constraint (*l* distance)

4 Common datum – situation feature of the collection surface: plain and straight line

6.6.4 Common datums based on a group of cylinders

Where a single datum defined by a group of five, nominally identical cylindrical holes (as shown in Figure 95) is a primary datum, a maximum inscribing cylinder shall be constructed for each of the five holes simultaneously, with the additional constraint that the five inscribing cylinders shall remain parallel with each other and centred on a pitch circle of diameter Ød at 72° increments. The datum shall consist of two situation features, a line and a plane. The line shall be a mean between the axes of the five inscribing cylinders, and the plane shall be a plane through that axis, and the axis of one of the inscribing cylinders (see Figure 96).

NOTE These principles can be equally applied to non-parallel cylinders.

Figure 95 – ***Pattern of five cylinders: input for reading (drawing indication) and for writing (design intent)***

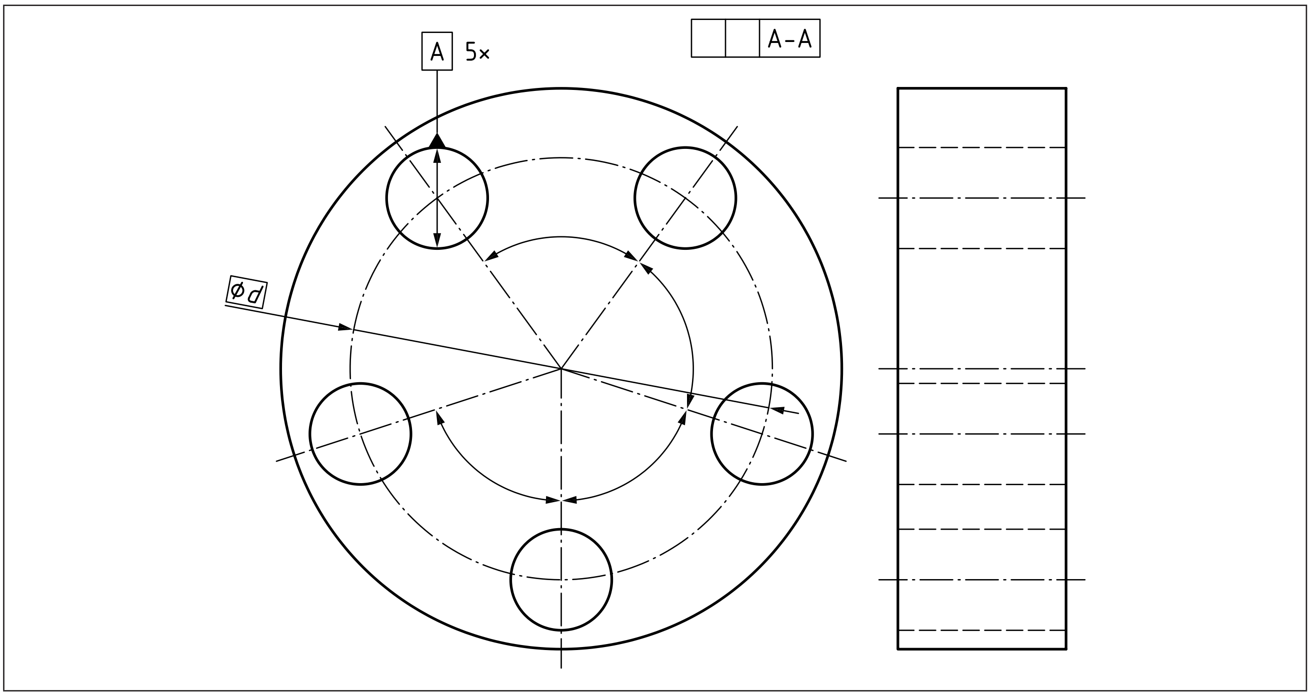
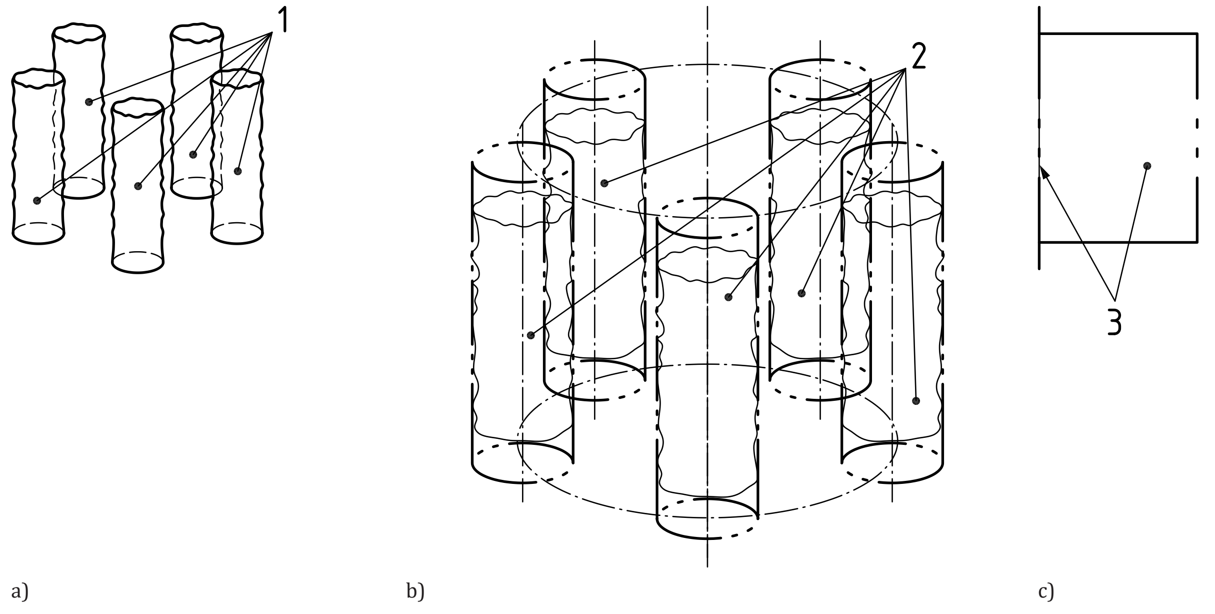


Figure 96 – ***Establishing a common datum from a pattern of five cylinders***



Key

1	Datum features: real integral features	3	Common datum – situation features of the collection surface: plain and straight line
2	Associated features with variable diameters, orientation constraint (parallelism) and location constraint (given by the TEDs and angles of 72°)		

6.6.5 Association for common datums

COMMENTARY ON 6.6.5

The association method for common datums requires that a collection of ideal single surfaces be fitted simultaneously (in one step) to several non-ideal surfaces.

The process of association for common datums shall include location and orientation constraints between the different associated features. These constraints shall be:

a) the new intrinsic characteristics defined by the collection of the features;

b) defined either explicitly or implicitly by TEDs (implicit orientation constraint: 0°, 90°, 180°, 270° and implicit location constraint: 0 mm).

The internal constraints for association described for single datums are also applicable for common datums, but complementary constraints (e.g. coplanarity, coaxiality) between the associated features shall be added.

6.6.6 Default association criteria

The following constraints for establishing a common datum shall be applied to each associated feature included in the collection defined by the common datum indication:

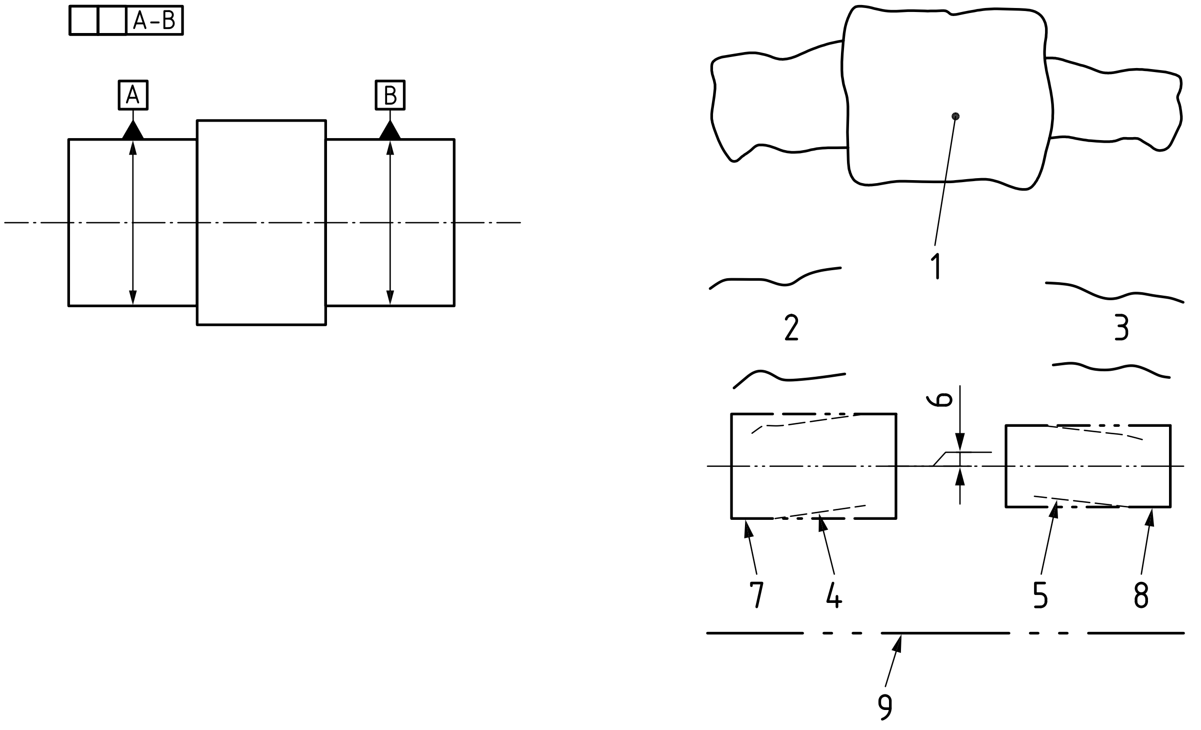
a) be outside the material of its corresponding filtered feature;

b) respect the orientation and location constraints defining the relationship between the nominal features in the collection (indicated by an explicit or implicit TED), while taking into account any modifiers (see **6.9.2**, Figure 113).

NOTE 1 The objective function is to simultaneously minimize the maximum distance normal to the associated feature between each associated feature and its filtered feature.

NOTE 2 Figure 97 illustrates the process used to establish a common datum from two surfaces nominally cylindrical and coaxial.

Figure 97 – ***Association for a common datum based on two coaxial cylinders***



a) Indication

b) Meaning of common datum A-B

Key

1	Real workpiece
2	Datum feature, arbitrarily numbered No 1 of the collection surface for the common datum
3	Datum feature numbered No. 2 of the collection surface for the common datum
4	Filtered feature of datum feature No. 1
5	Filtered feature of datum feature No. 2
6	Orientation constraint and location constraint between the associated features of the collection features (coaxiality)
7	Associated feature of datum feature No. 1 (see BS EN ISO 5459)
8	Associated feature of datum feature No. 2 (see BS EN ISO 5459)
9	Common datum (in this case, the axis of the collection surface – the two coaxial associated cylinders)

6.7 Datum systems and the six degrees of freedom

COMMENTARY ON 6.7

A datum system consists of one or more datums (see 6.2.4).

In a datum system, the datums shall be used to lock some, or all, of the six degrees of freedom available to the tolerance zone.

NOTE 1 The effect of datums can be modified with the "orientation only" symbol (see 6.9.1), specifying situation features or controlling specific degrees of freedom:

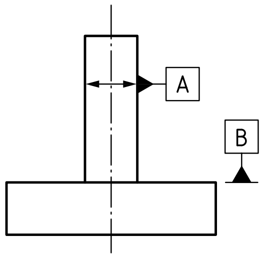
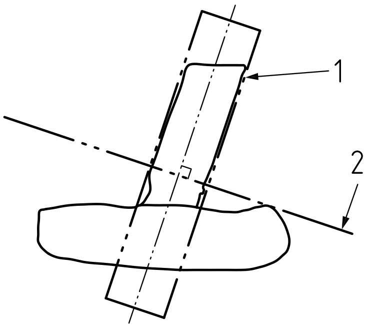
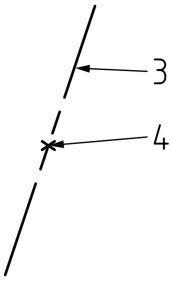
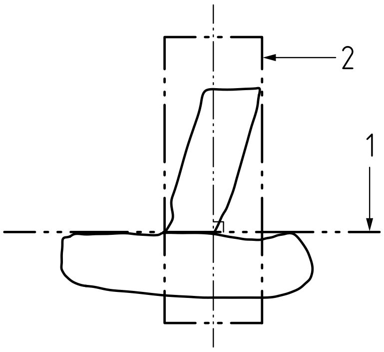
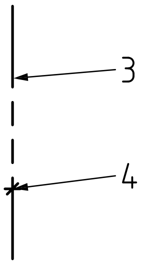
a) *the primary datum locks whichever degrees of freedom it can lock;*

b) *the secondary datum locks whichever degrees of freedom it can lock, and which have not already been locked by the primary datum;*

c) *the tertiary datum locks whichever degrees of freedom it can lock, and which have not already been locked by the primary and secondary datum.*

NOTE 2 Table 15 and Table 16 give examples of datum systems.

Table 15 – *Examples: datum systems*

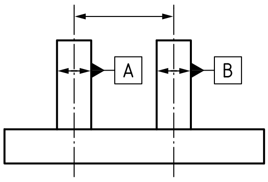
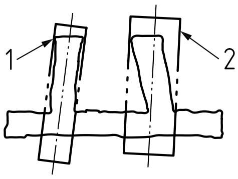
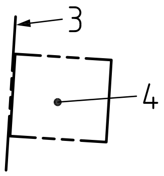
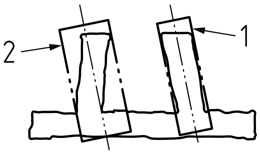
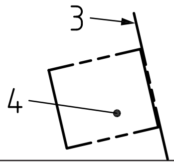
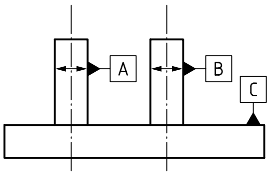
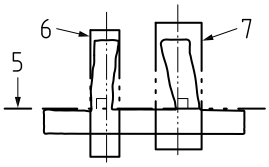
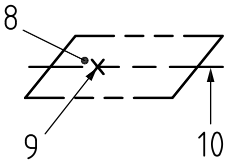
Indication of datum feature	Indication of datum in tolerance frame	Meaning on workpiece	Resulting common datum or datum system	Degrees of freedom controlled
 	A | B	 	 	Five degrees of freedom locked. Rotation about the axis (item 3) not locked.
	B | A	 	 	

Key

1	First associated feature without orientation constraint
2	Second associated feature with orientation constraint from the first associated feature
3	Straight line which is the situation feature of the associated cylinder (its axis)
4	Point of intersection between the straight line and the plane

NOTE Association for single datums is described in BS EN ISO 5459:2011, Annex A.

Table 16 – *Examples: datum systems*

Indication of datum feature	Indication of datum in tolerance frame	Meaning on workpiece	Resulting common datum or datum system	Degrees of freedom that can be locked
	A B			Five degrees of freedom locked. Translation along the axis (item 3) not locked.
	B A			
	C A B			All six degrees of freedom locked.

Key

1	First associated cylinder without constraint
2	Second associated cylinder with parallelism constraint from the first associated feature
3	Straight line which is the axis of the first associated cylinder
4	Plane including the axes of the two associated cylinders
5	First associated feature without a constraint
6	Second associated feature with a perpendicularlity constraint from the first associated feature
7	Third associated feature with a perpendicularlity constraint from the first associated feature (and parallelism constraint from the second feature)
8	Plane which is the first associated feature
9	Point of intersection between the plane and the axis of the second associated feature
10	Straight line which is the intersection between the associated plane and the plane containing the two axes

NOTE 1 The orientation and location of datums are different depending on the datum indication in the tolerance frame. Not all possibilities for establishing the datums are covered.

NOTE 2 Association for single datums is described in BS EN ISO 5459:2011, Annex A.

6.8 Datum targets

6.8.1 General

A datum target shall be used in the following instances:

a) when a datum is based on part of a datum feature, to identify the portion of the datum feature to be used; and

b) to represent a contacting feature with the same nominal geometry.

Where applicable, a datum target shall as necessary take the form of:

1) a point;

2) a line; or

3) an area.

A datum target shall be indicated by a datum target indicator constructed from:

i) a datum target frame;

ii) a datum target symbol; and

iii) a leader line linking the two symbols (directly or through a reference line).

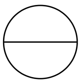
Where necessary, the same datum target shall be indicated on several appropriate views in order to provide an unambiguous definition (see Figure 108, Figure 109, Figure 110 and Figure 111).

NOTE A geometrical specification with a datum reference which is defined by datum targets may be applied to the surface or feature on which those datum targets are identified.

6.8.2 Datum target frame

The datum target frame shall be a circle, divided into two compartments by a horizontal line (see Figure 98). The lower compartment shall be reserved for the datum feature identifier, followed by a digit (from 1 to *n*), corresponding to the datum target number. The upper compartment shall be reserved for additional information, such as dimensions of the target area.

Figure 98 – *Single datum target frame*



6.8.3 Datum target symbol

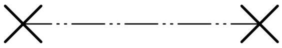
The datum target symbol indicates the type of datum target.

A datum target point shall be identified with a cross (see Figure 99).

Figure 99 – *Datum target point*

A datum target line shall be indicated by a long-dashed double-dotted narrow line (type 05.1 of BS ISO 128-24:1999). If the line is open, each end shall be terminated by a cross (see Figure 100). Where applicable, this line shall be straight, circular or a line of any shape.

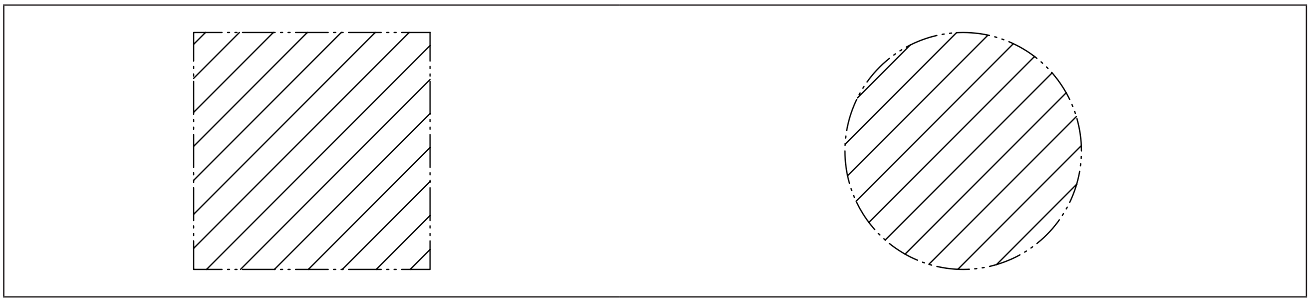
Figure 100 – *Non-closed datum target line*



A datum target area shall be indicated with an outline using a long-dashed double-dotted narrow line (type 05.1 of BS ISO 128-24:1999), and cross-hatched (see Figure 101).

*NOTE For 3D TPS a shaded surface can be used (see BS ISO 16792:2015, **10.3.4**).*

Figure 101 – *Datum target area*



6.8.4 Leader line

The datum target frame shall be connected directly, or through a reference line, to the datum target symbol by a leader line terminated with or without an arrow or a dot (see Figure 102, Figure 103 and Figure 104).

Figure 102 – *Indicator for single datum target point*

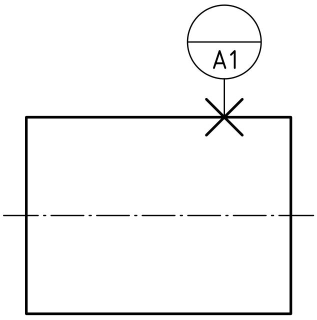


Figure 103 – *Indicator for single datum target line*

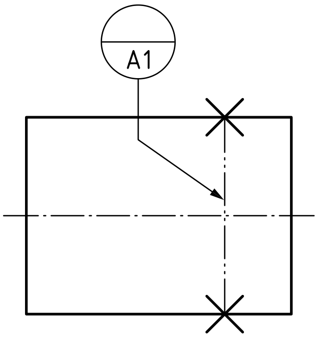
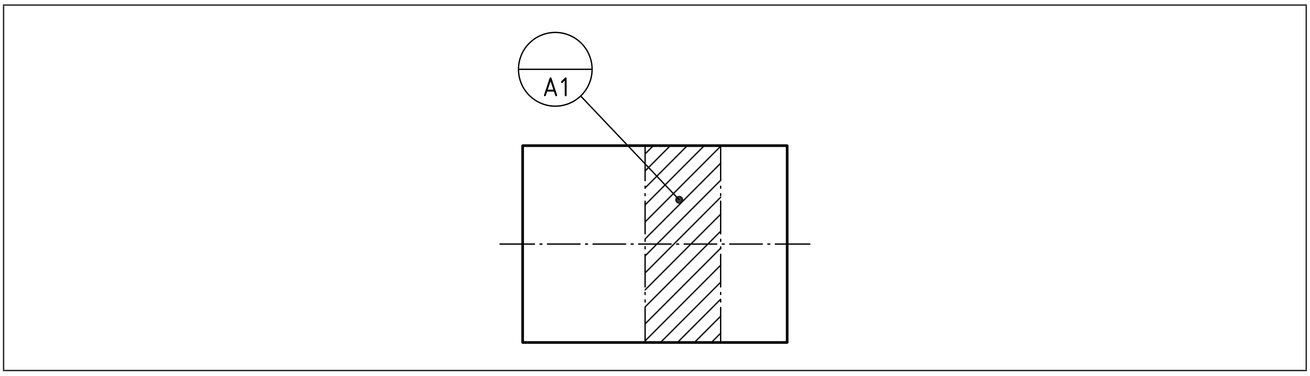


Figure 104 – ***Indicator for single datum target surface***

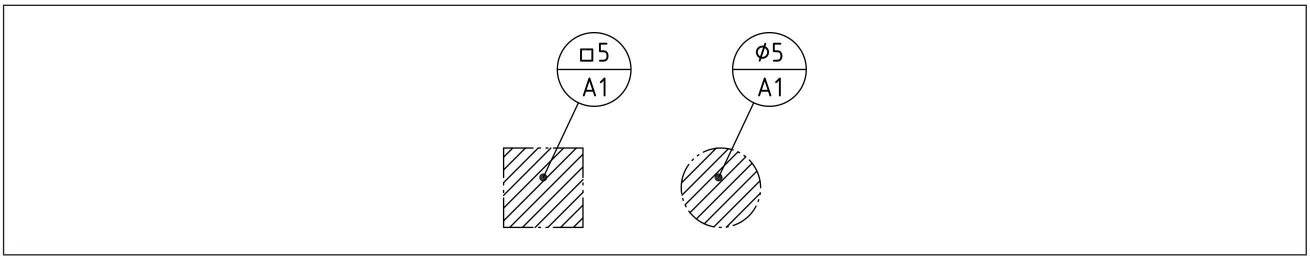


NOTE The orientation of the leader line connecting the frame with the datum target symbol is unimportant.

6.8.5 Datum target areas

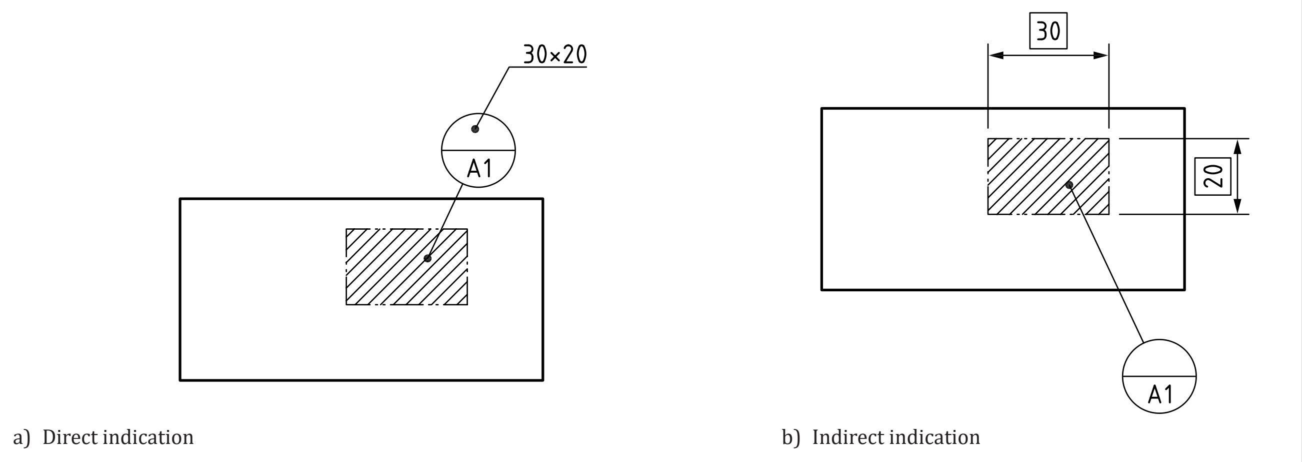
The upper compartment of the datum target frame shall only be used when defining a datum target area (see Figure 105).

Figure 105 – ***Indicator for single datum target area***



Where applicable, the size of the area shall, as appropriate, be indicated in the upper compartment or using dimensions elsewhere on the specification (in which case the upper compartment remains blank) (see Figure 106).

Figure 106 – ***Direct and indirect indication for single datum target area***



a) Direct indication

b) Indirect indication

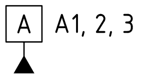
NOTE The dimensions defining the size of the datum target area are theoretically exact.

6.8.6 Datum feature indicators used with datum targets

If a single datum feature is established from one or more datum targets belonging to only one surface, the datum feature identifier identifying the surface shall be repeated close to the datum indicator, followed by the list of numbers (separated by commas) identifying the targets (see Figure 107). Each individual datum target shall be identified by a datum target indicator indicating the datum feature identifier, the number of the datum target and, if applicable, the dimensions of the datum target.

NOTE 1 The list of datum targets can be placed anywhere adjacent to the datum feature identifier.

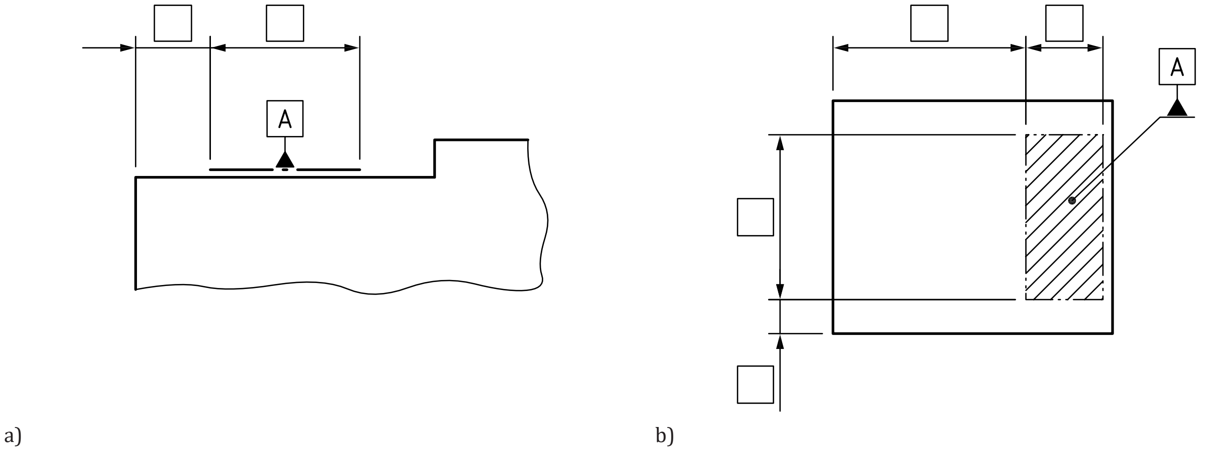
Figure 107 – ***Indication of datums established from datum targets***



NOTE 2 In the case of a large number of datum targets (e.g. when defining a datum for a large, flexible panel), an abbreviated indication, such as "A1-20", is acceptable.

NOTE 3 If there is only one datum target, the drawing indication may be simplified by placing the datum indicator as shown in Figure 108 and Figure 109.

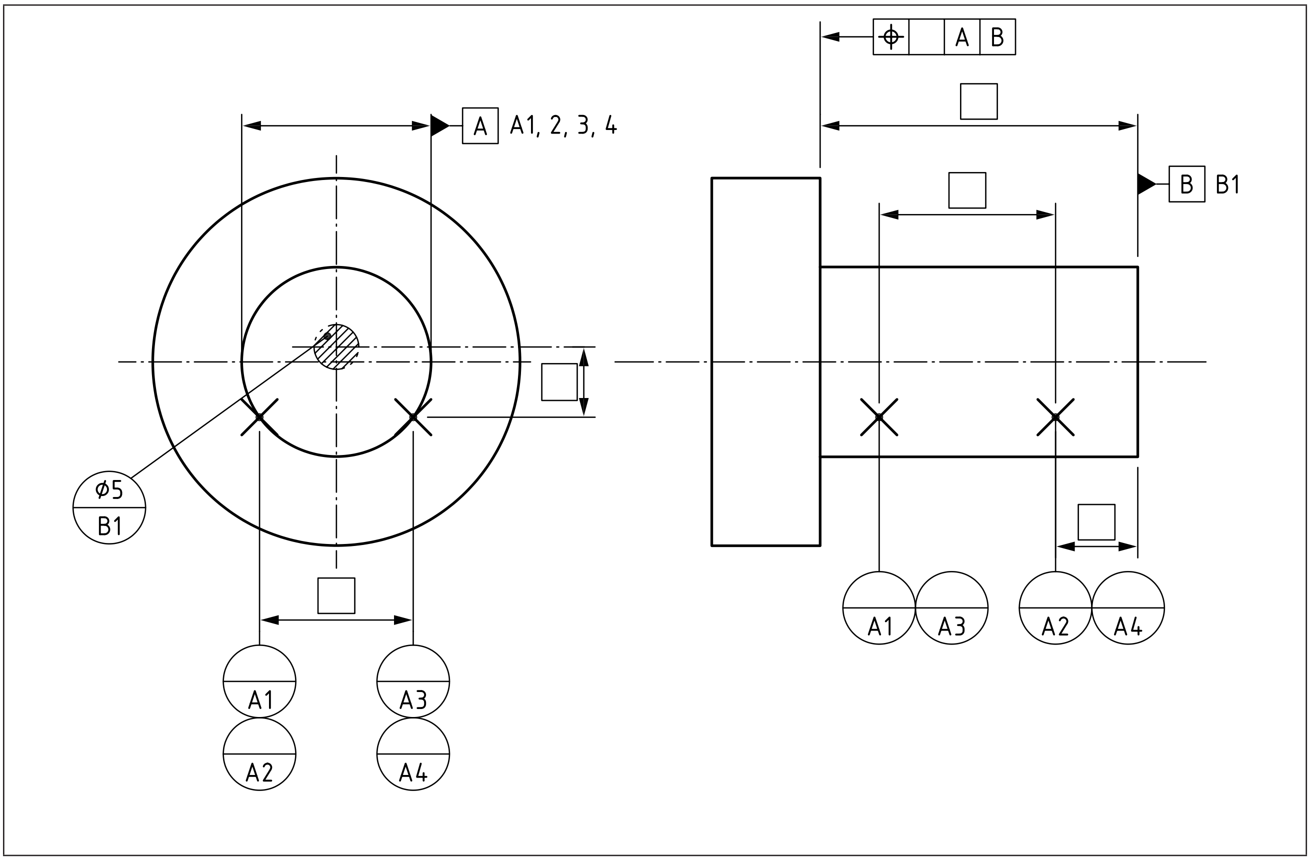
Figure 108 – ***Simplification of drawing indication when there is only one datum target area***



6.8.7 Datums based on more than one datum target

If a single datum is based on more than one datum target, the relationships between the datum targets shall as necessary be defined with theoretically exact dimensions (see Figure 109).

Figure 109 – ***Datums based on more than one datum target***



NOTE 1 In this case the TEDs can be used without any corresponding geometrical tolerance. The tolerance on the location of the datum targets is not part of the definition of the workpiece geometry, so does not need to be defined on the design specification.

When datum targets are used to define datums on a rigid workpiece (see Figure 110 and Figure 111), the primary datum shall normally be defined with three datum targets, the secondary datum with two datum targets, and the tertiary datum with one datum target, except where the geometry of the datum feature necessitates the use of more datum targets in order to ensure a unique solution (see Figure 109).

If a primary datum plane is defined with more than three datum targets, ISO standards do not currently define how this would be interpreted. In such a case, a statement shall be included in the TPS to explain how the datum is to be constructed, e.g.:

DATUM A IS A LEAST SQUARES PLANE CONSTRUCTED FROM DATUM TARGETS A1-8.

NOTE 2 This requirement also applies when a secondary datum plane is based on more than two datum targets, and a tertiary datum plane is based on more than one datum target.

Figure 110 – ***Application of datum targets***

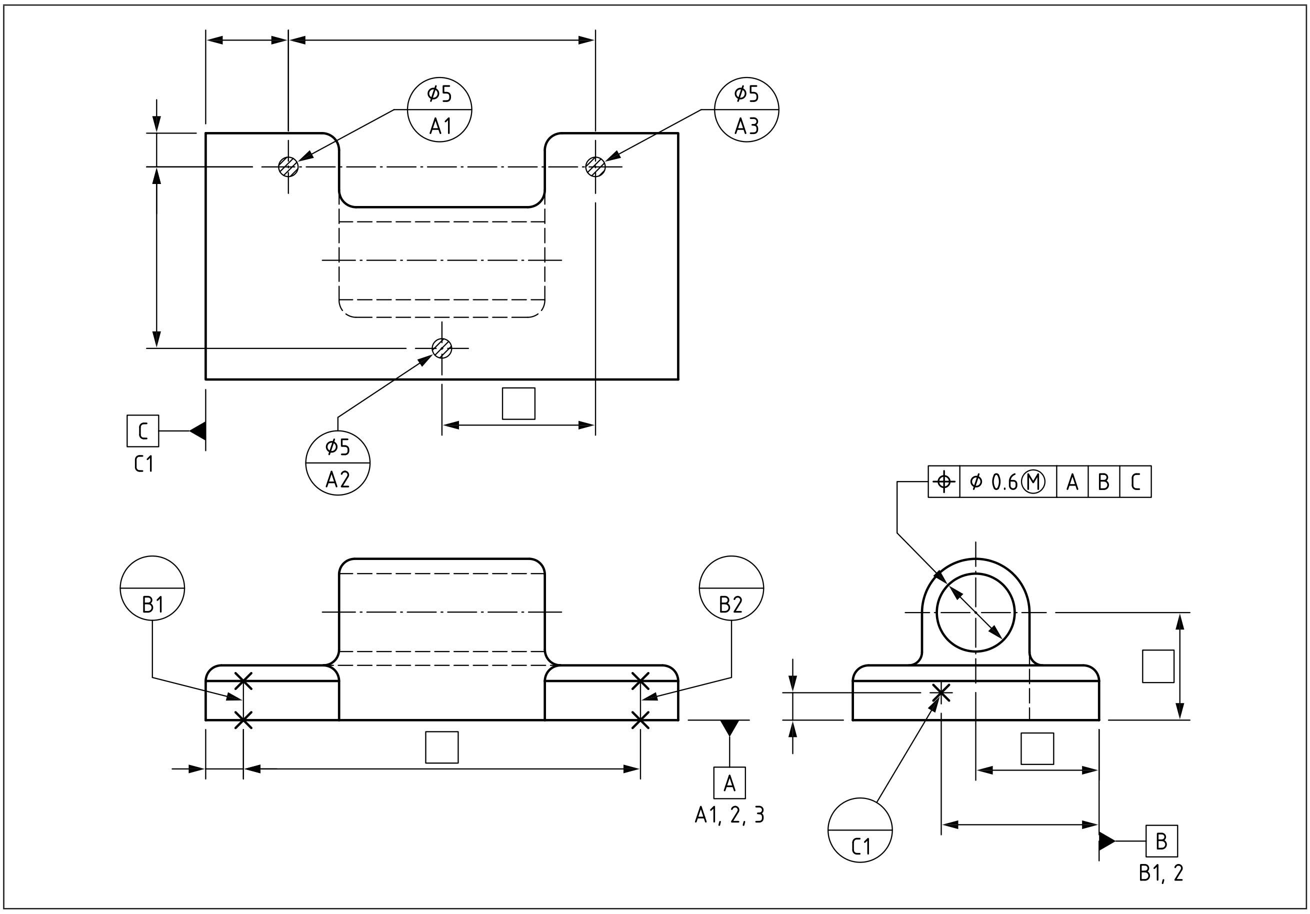
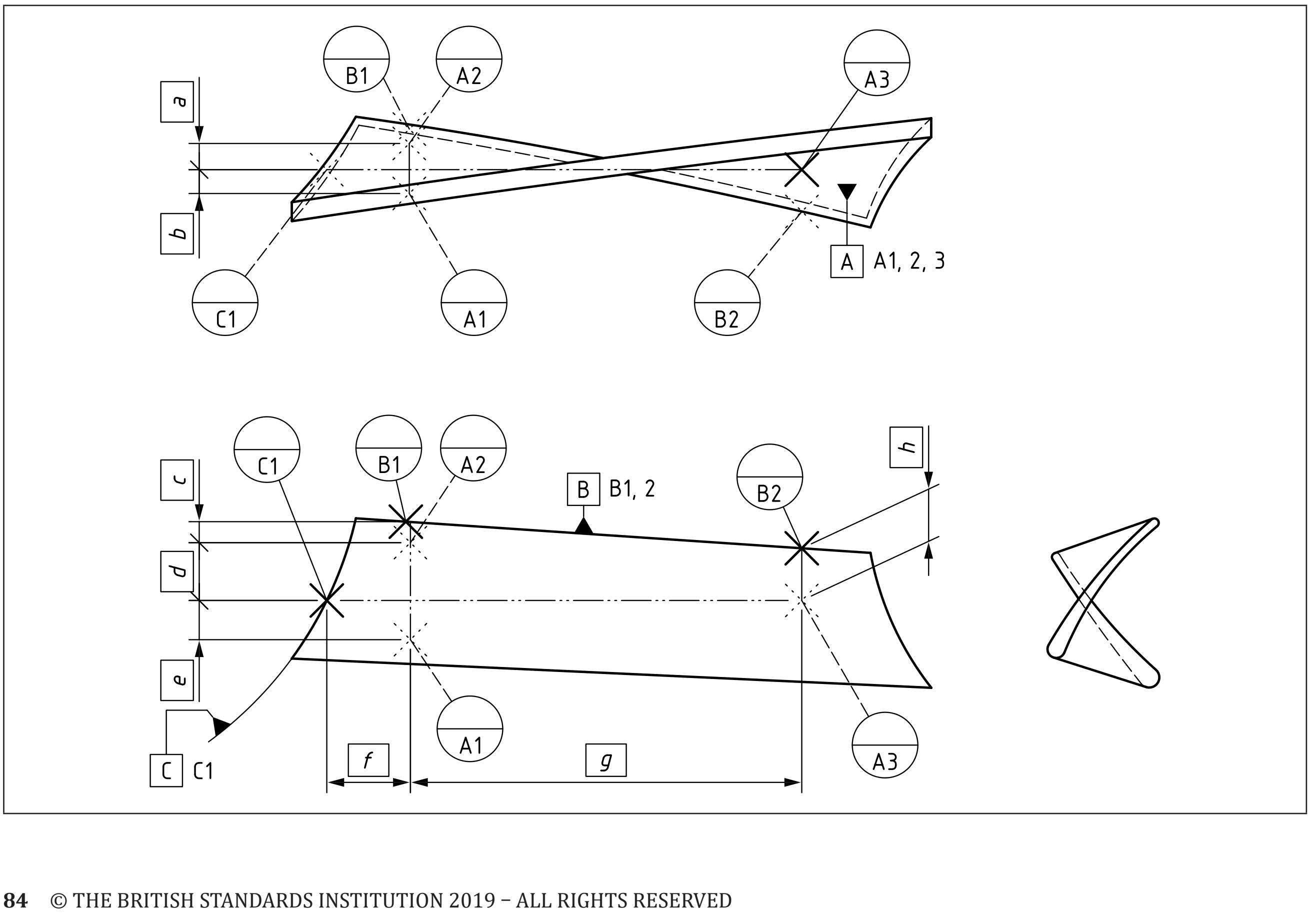


Figure 111 – ***Application of datum targets***



6.9 Modifying the effects of datums

COMMENTARY ON 6.9

The effect of a datum can be modified in several ways.

6.9.1 "Orientation only" modifier

COMMENTARY ON 6.9.1

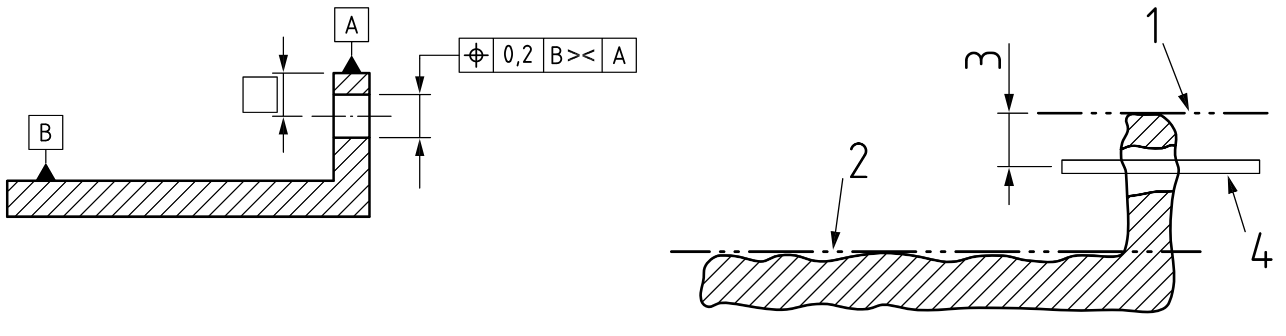
In some instances, a datum might be required only to control the orientation of a tolerance zone, and not its location.

To prevent the datum controlling the location of the tolerance zone, the "orientation only" modifier shall be used in the datum reference compartment of the tolerance frame (see Figure 112).

The "orientation only" modifier shall consist of two angle brackets used in the following arrangement, "><".

If the datum reference can only control the orientation of the tolerance zone in any case, the "><" modifier is redundant and shall be omitted.

Figure 112 – *Application of "orientation-only" modifier*



a) Indication

b) Meaning

NOTE The modifier >< allows only the orientation degrees of freedom of the tolerance zone, and not its translation degree of freedom (managed by the datum A in this case), to be locked from the datum B.

Key

1 Associated plane with outside material constraint and with orientation constraint from datum B

2 Associated plane with outside material constraint (datum B)

3 Distance relative to the location

4 Tolerance zone with orientation constraint from datum B and location constraint from datum A

6.9.2 Situation feature modifiers

COMMENTARY ON 6.9.2

A datum can be based on a set of more than one situation feature. In some instances, the full effect of all the situation features defining the datum can prevent the tolerance requirement from being correctly defined. The tolerance requirement might need to utilize the effect of only a subset of the situation features.

Individual situation features shall be invoked, as appropriate, by using the following modifiers (see Figure 113).

a) If the situation feature to be utilized is a plane, the [PL] modifier shall be used.

b) If the situation feature to be utilized is a straight line, the [SL] modifier shall be used.

c) If the situation feature to be utilized is a point, the [PT] modifier shall be used.

NOTE More than one of these modifiers can be used with a single datum reference.

Figure 113 – ***Indication of which modifier is needed in the set of situation features***

B[PL] a) The plane	B[SL] b) The straight line
B[PT] c) The point	B[PL][SL] d) The plane and the straight line

6.9.3 Specific control of degrees of freedom

NOTE 1 By default, a datum locks all available degrees of freedom not already locked by any preceding datum.

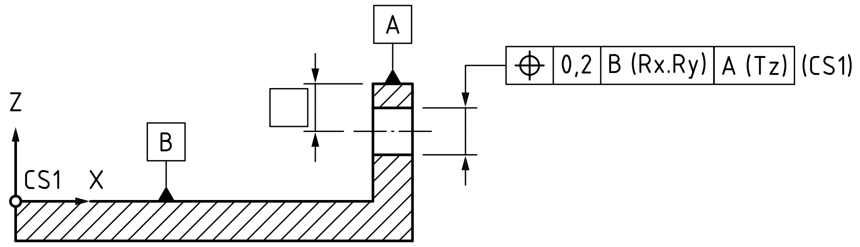
To specify that a datum is to control individual degrees of freedom and not the complete set, those degrees of freedom shall be indicated directly in the datum section of the geometrical specification.

One or more complimentary indications, enclosed in square brackets, shall be placed after the datum feature reference to specify the degrees of freedom to be controlled.

NOTE 2 There are six possible complimentary indications, namely Tx, Ty, Tz, Rx, Ry and Rz, representing the three translational and three rotational degrees of freedom, based on a Cartesian coordinate system.

To be unambiguous about the translation or rotation to be controlled, a coordinate reference system to which the complimentary indications are related shall be specified and identified in the technical product documentation. This identification shall then be placed after any tolerance indicator using the complimentary datum indications and enclosed in square brackets (see Figure 114).

Figure 114 – ***Complimentary datum indication***



NOTE 3 The reference coordinate system does not have a mandatory relationship with any nominal coordinate systems of the technical product documentation. Several unrelated coordinate systems can exist.

In some application the orientation only modifier (see 6.9.1) may be used as an alternative.

Section 7 Geometrical specifications

7.1 Geometric tolerances

Geometrical tolerancing shall conform to the following standards, as appropriate.

BS EN ISO 1101	*Geometrical product specifications (GPS) – Geometrical tolerancing – Tolerances of form, orientation, location and run-out*
BS EN ISO 1660	*Geometrical product specifications (GPS) – Geometrical tolerancing – Profile tolerancing*
BS EN ISO 2692	*Geometrical product specifications (GPS) – Geometrical tolerancing – Maximum material requirement (MMR), least material requirement (LMR) and reciprocity requirement (RPR)*
BS EN ISO 5458	*Geometrical Product Specifications (GPS) – Geometrical tolerancing – Pattern and combined geometrical specification*
BS EN ISO 5459	*Geometrical product specifications (GPS) – Geometrical tolerancing – Datums and datum systems*
BS EN ISO 7083	*Technical drawings – Symbols for geometrical tolerancing – Proportions and dimensions*
BS EN ISO 8015	*Geometrical product specifications (GPS) – Fundamentals – Concepts, principles and rules*
BS EN ISO 10579	*Geometrical product specifications (GPS) – Dimensioning and tolerancing – Non-rigid parts*
BS EN ISO 12180-1	*Geometrical product specifications (GPS) – Cylindricity – Part 1: Vocabulary and parameters of cylindrical form*
BS EN ISO 12180-2	*Geometrical product specifications (GPS) – Cylindricity – Part 2: Specification operators*
BS EN ISO 12181-1	*Geometrical product specifications (GPS) – Roundness – Part 1: Vocabulary and parameters of roundness*
BS EN ISO 12181-2	*Geometrical product specifications (GPS) – Roundness – Part 2: Specification operators*
BS EN ISO 12780-1	*Geometrical product specifications (GPS) – Straightness – Part 1: Vocabulary and parameters of straightness*
BS EN ISO 12780-2	*Geometrical product specifications (GPS) – Straightness – Part 2: Specification operators*
BS EN ISO 12781-1	*Geometrical product specifications (GPS) – Flatness – Part 1: Vocabulary and parameters of flatness*
BS EN ISO 12781-2	*Geometrical product specifications (GPS) – Flatness – Part 2: Specification operators*
BS EN ISO 16610 (all parts)	*Geometrical product specifications (GPS) – Filtration*
BS EN ISO 17450-1	*Geometrical product specifications (GPS) – Basic concepts – Part 1: Model for geometrical specification and verification*
BS EN ISO 17450-2	*Geometrical product specifications (GPS) – Basic concepts – Part 2: Basic tenets, specifications, operators, uncertainties and ambiguities*

BS EN ISO 17450-3	*Geometrical product specifications (GPS) – Basic concepts – Part 3: Toleranced features*
BS EN ISO 17450-4	*Geometrical product specifications (GPS) – Basic concepts – Part 4: Geometrical characteristics for quantifying GPS deviations*
BS EN ISO 22432	*Geometrical product specifications (GPS) – Features utilized in specification and verification*
BS EN ISO 25378	*Geometrical product specifications (GPS) – Characteristics and conditions – Definitions*

7.2 Basic concepts

A geometrical specification applied to a feature defines the tolerance zone within which that feature shall be contained (see 7.6).

The tolerance shall apply to the whole extent of the considered feature, unless otherwise specified as in 7.12 and 7.10.

Unless a more restrictive indication is required, for example by an additional tolerance or explanatory note, the toleranced feature shall be of any form within this tolerance zone.

NOTE 1 A feature is a specific portion of the workpiece, such as a point, a line or a surface; these features can be integral features (e.g. the external surface of a cylinder) or derived (e.g. a median line or median surface) (see BS EN ISO 14660-1).

NOTE 2 Indicating geometrical tolerances on a drawing does not necessarily imply the use of any particular method of production, measurement or gauging.

7.3 Symbols

Symbols for geometrical characteristics shall be as indicated in Table 17 and Table 18, and as indicated in BS EN ISO 1101.

Table 17 – *Symbols for geometrical characteristics*

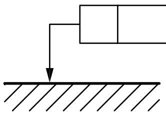
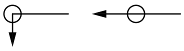
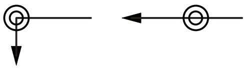
Tolerances	**Characteristics**	**Symbol**	**Datum needed**
Form	Straightness	⏤	no
	Flatness	⏥	no
	Roundness	○	no
	Cylindricity	⌭	no
	Line profile	⌒	no
	Surface profile	⌓	no
Orientation	Parallelism	//	yes
	Perpendicularity	⊥	yes
	Angularity	∠	yes
	Line profile	⌒	yes
	Surface profile	⌓	yes

(Continues)

Table 17 – *Symbols for geometrical characteristics (Cont.)*

Tolerances	**Tolerances**	**Symbol**	**Datum needed**
Location	Position	⌖	yes or no
	Coaxiality/concentricity	◎	yes
	Symmetry	⌯	yes
	Line profile	⌒	yes
	Surface profile	⌓	yes
Run-out	Circular run-out	↗	yes
	Total run-out	⌰	yes

Table 18 – *Additional symbols*

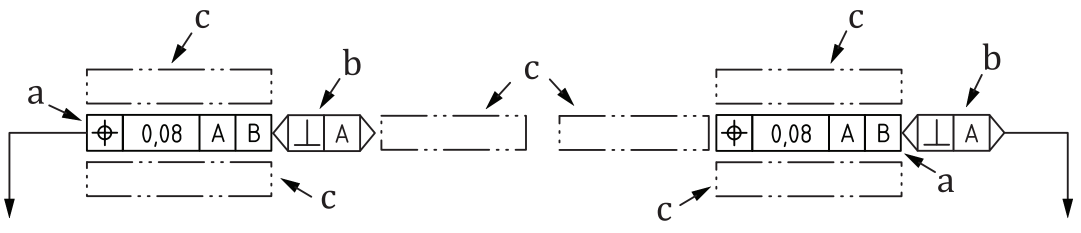
Description	**Symbol**
Geometrical specification	
Theoretically exact dimensions	$\boxed{50}$
Projected tolerance zone	Ⓟ
Maximum material requirement	Ⓜ
Least material requirement	Ⓛ
Free-state condition (non-rigid parts)	Ⓕ
All-around	
All over	
Between	◄►
Combined zone	CZ
Combined zone (orientation only)	CZR
Separate zone	SZ
Unequal zone	UZ
United feature	UF
Minor diameter	LD
Major diameter	MD
Pitch diameter	PD
Any cross-section	ACS
Simultaneous requirement	SIM*n*

NOTE Further symbols and modifiers are standardized in BS EN ISO 1101 and other standards.

7.4 Geometrical specification indication

A geometrical specification indication shall consist of a tolerance indicator, optional plane and feature indications and optional adjacent indications (see Figure 115).

Figure 115 – ***Elements of a geometrical specification indication***



Key

a	Tolerance indicator
b	Plane and feature indicator
c	Adjacent indications

The geometrical specification indication shall be attached to the relevant feature(s) with a leader line which can be attached to either end of the indication.

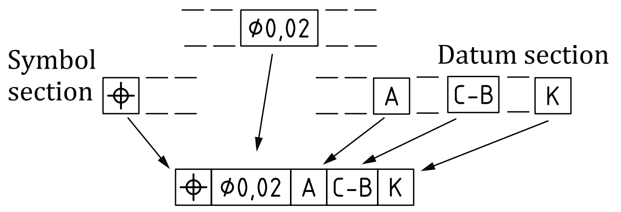
7.5 Tolerance indicator

7.5.1 General

The requirements shall be shown in a rectangular frame which is divided into two or more compartments, in accordance with BS EN ISO 1101 (see Figure 116). The frame shall contain the following sections in the following order:

- the symbol section identifying the geometrical characteristic;
- the tolerance zone section defining the shape and size of the tolerance zone;
- if applicable, a datum section containing the letter or letters identifying the datum system.

Figure 116 – ***Three sections of the tolerance indicator***



NOTE The datum section may have one or more compartments

7.5.2 Symbol section

The symbol section shall contain one of the symbols given in Table 17. This section shall always be present in a tolerance indicator. No additional modifiers shall be used in this section.

7.5.3 Tolerance zone section

The tolerance zone section shall contain the tolerance value in the unit used for linear dimensions. This section shall always be present in a tolerance indicator.

The tolerance value shall be preceded by the symbol "Ø" if the tolerance zone is circular or cylindrical or by "SØ" if the tolerance zone is spherical, and may be followed by other modifiers such as CZ, SZ and Ⓜ.

Any modifier following the tolerance value shall be separated from the tolerance value and any other modifiers by a single space, except for modifiers consisting of letters contained in a circle, in which case no space is used.

7.5.4 Datum section

The datum section shall contain one or more datum feature identifiers. This section shall be present in the tolerance indicator when location and orientation requirements are defined.

7.6 Tolerance zones

The width of the tolerance zone shall apply normal to the specified geometry (see Figure 117), unless otherwise indicated with a direction feature, with the exception of roundness (see **8.4.4** for details on the direction feature and on roundness).

NOTE 1 The orientation alone of the leader line does not influence the definition of the tolerance.

According to the characteristic to be toleranced and the manner in which it is dimensioned, the tolerance zone shall be one of the following:

- the space within a circle;
- the space between two concentric circles;
- the space between two equidistant lines or two parallel straight lines;
- the space within a cylinder;
- the space between two coaxial cylinders;
- the space between two equidistant surfaces or two parallel planes; or
- the space within a sphere.

Figure 117 – *Width of tolerance zone applying to the specified geometry*

Key

a Datum axis

a) Drawing indication

b) Interpretation

7.7 Toleranced features

The tolerance frame shall be connected to the toleranced feature by a leader line starting from either side of the frame and ending with a terminator (arrowhead or filled dot) in one of the following ways:

- on the outline of the feature or an extension of the outline (but clearly separated from the dimension line), with a leader line terminating in an arrowhead, when the tolerance refers to the line or surface itself [see Figure 118a) and Figure 118b)];

  NOTE 1 The arrowhead could be placed on a reference line using a leader line to point to the surface [see example of Figure 118c)].

- on the surface of a feature, with a leader line ending in a filled dot on the surface, when the tolerance refers to that surface [see Figure 118d)];
- as an extension of the dimension line when the tolerance refers to the median line or median surface, or a point defined by the feature so dimensioned [see Figure 119)].

Figure 118 – ***Placement of tolerance indicator when applied to a surface***

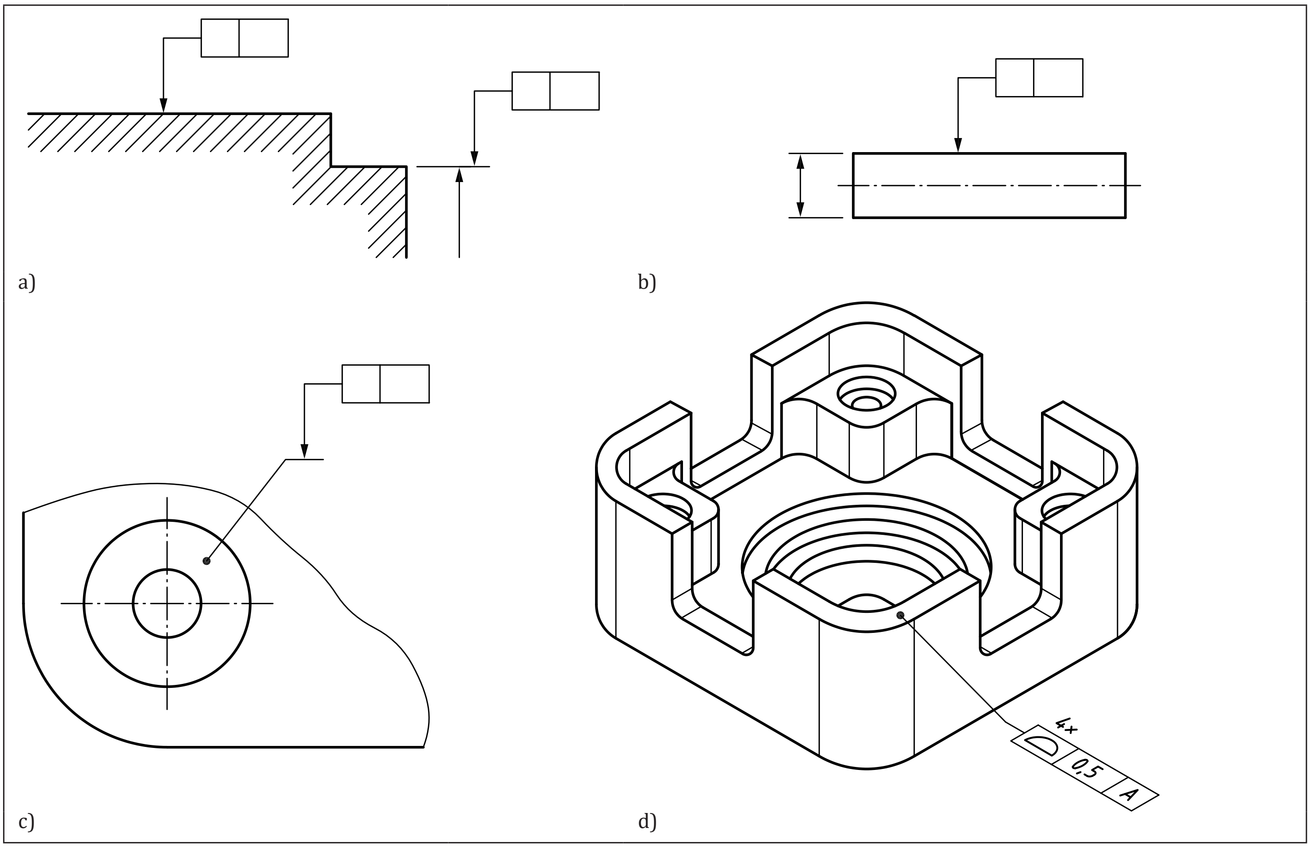
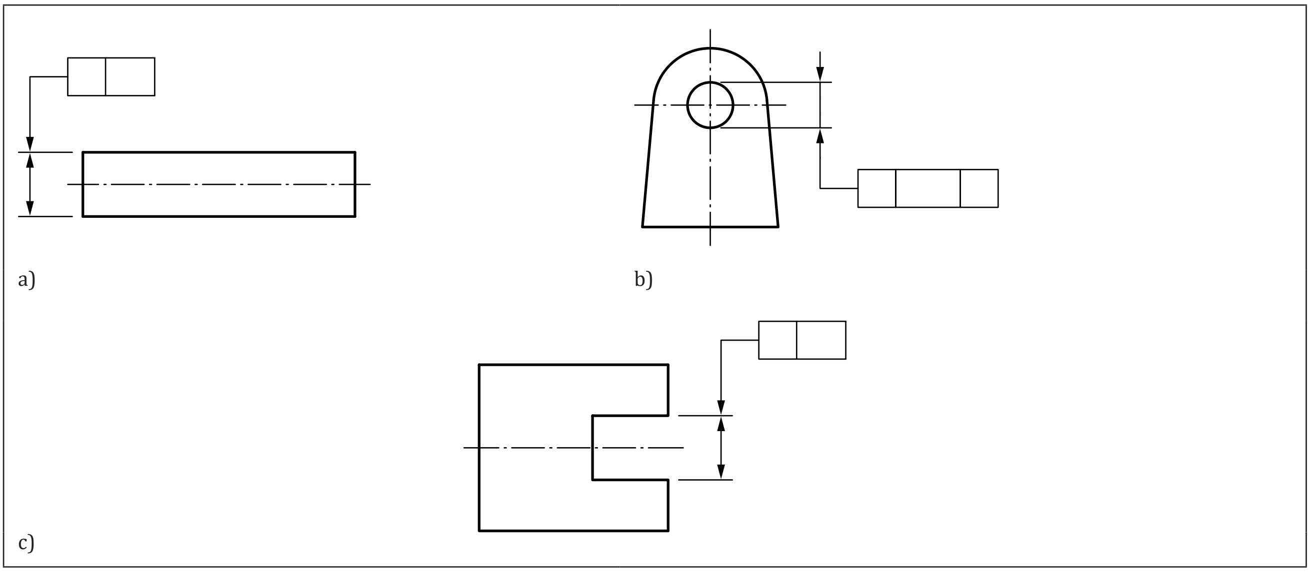


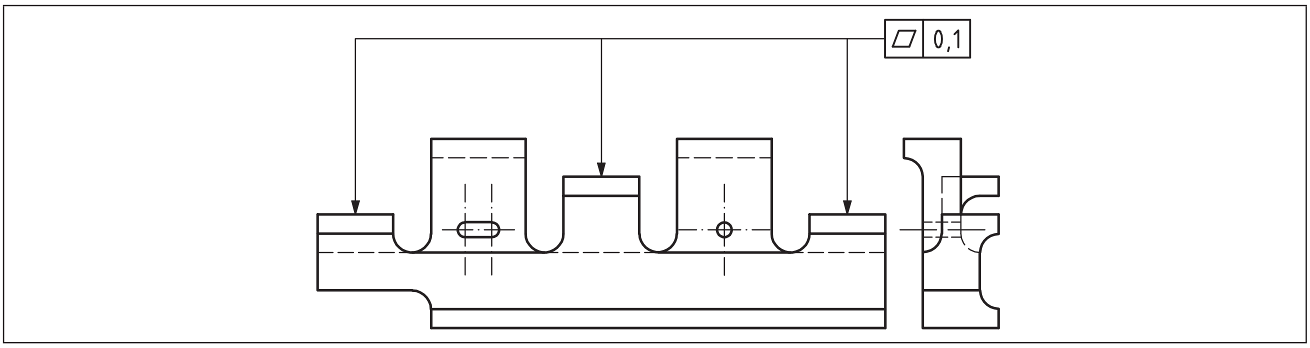
Figure 119 – *Placement of tolerance indicator when applied to a derived feature*



When a tolerance applies to more than one feature this shall be indicated by one of the following methods:

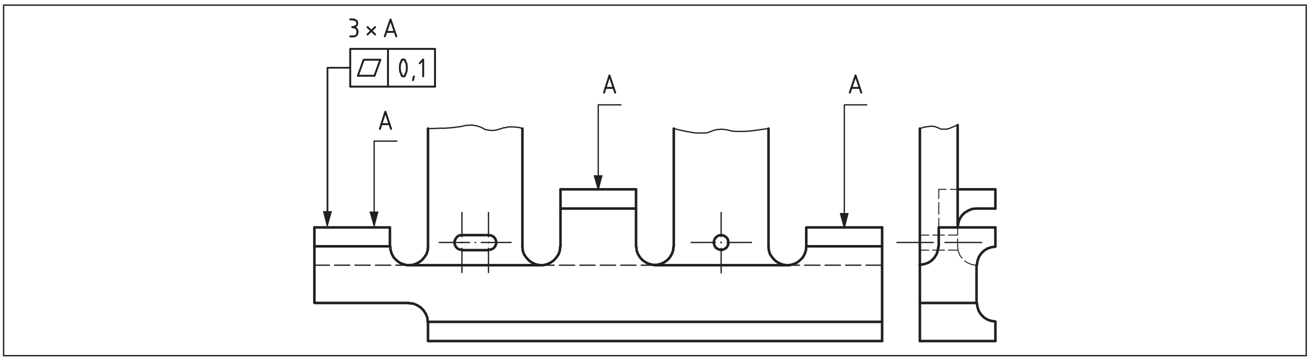
- separate leader lines going to each of the features (see Figure 120);

Figure 120 – *Specification applying to several features individually*



- indicating *n*× above the tolerance indicator, where *n* is the number of features;
- in cases where the *n*× indication would not be sufficiently clear, by placing the indication *n*× A above the tolerance indicator, where *n* is the number of features and A is a letter used to identify each instance (see Figure 121);

Figure 121 – *Specification applying to several features individually*



- use of the between symbol (see **8.1.1**);
- use of the all-around symbol (see **8.1.2**); and
- use of the all-over symbol (see **8.1.3**).

NOTE 2 If it is necessary to specify more than one geometrical specification for a feature, the tolerance frames may be stacked, one under the other, and attached to the feature with a single leader line (see example of Figure 122 and Figure 123).

NOTE 3 Figure 124 gives examples of geometrical tolerances and requirements associated with them.

Figure 122 – *Tolerance applying to more than one feature*

Figure 123 – *Requirements given in tolerance frames one under the other*

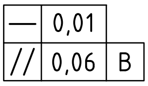
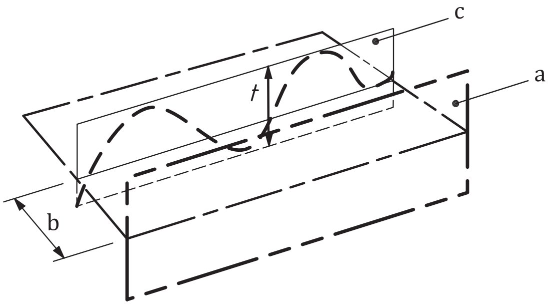
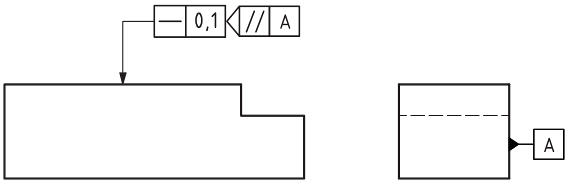
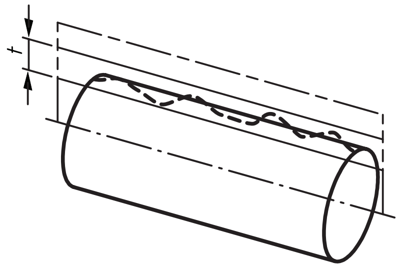
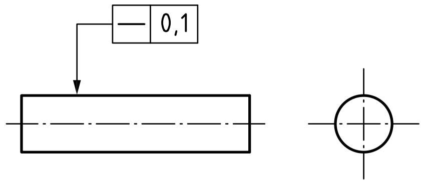
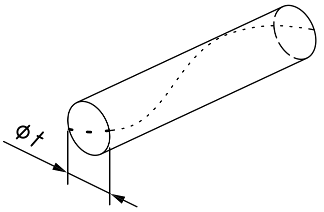
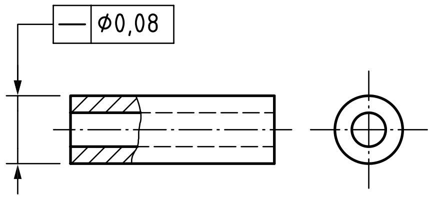
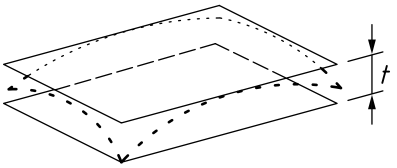
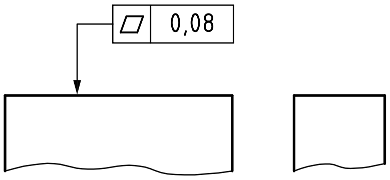
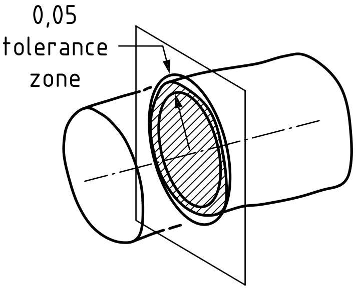
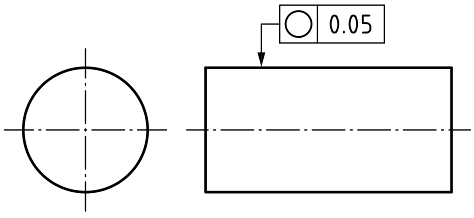


Figure 124 – *Examples of geometrical tolerancing*

Symbol	Definition of the tolerance zone	Indication and explanation in 2D
	Straightness tolerance	
—	The tolerance zone in the considered plane shall be limited by two parallel straight lines a distance *t* apart, where *t* is the tolerance value, in each plane defined by the intersection plane indicator.	Any extracted line on the upper surface, within a plane parallel to datum A, shall be contained between two parallel straight lines 0,1 apart.
	 a Datum A b Any distance c Intersection plane parallel to datum A	

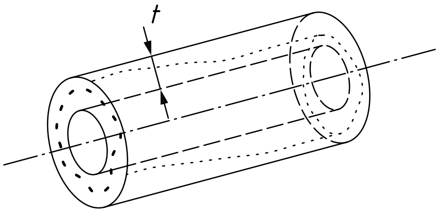
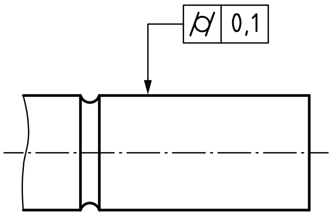
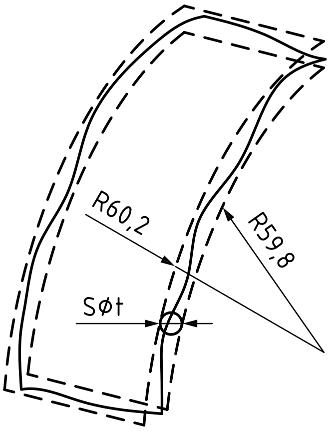
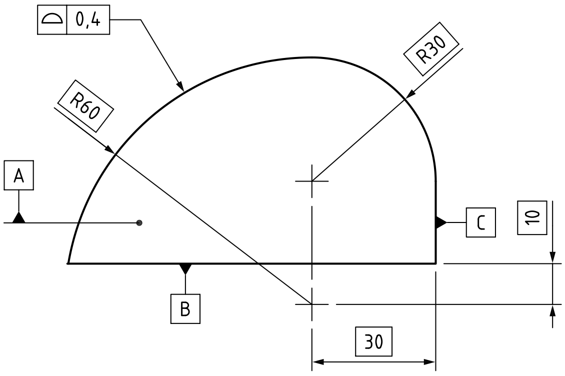
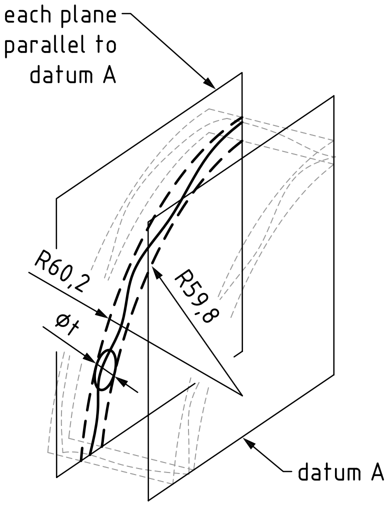
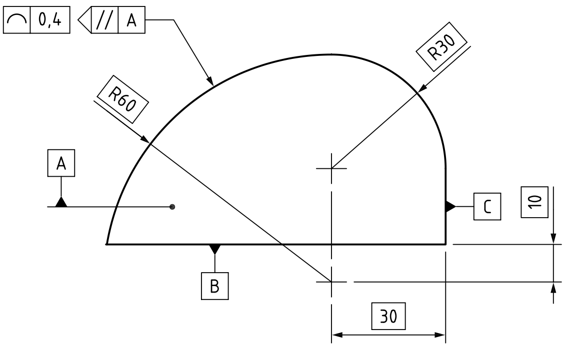
(Continues)

Figure 124 – ***Examples of geometrical tolerancing (Cont.)***

Symbol	Definition of the tolerance zone	Indication and explanation in 2D
—	The tolerance zone shall be limited by two parallel lines a distance *t* apart, where *t* is the tolerance value, in each plane that includes the axis of the associated cylinder.  	Any extracted line on the cylindrical surface, contained within a plane through the axis of the associated cylinder, shall be contained between two parallel lines 0,1 apart. 
—	The tolerance zone shall be limited by a cylinder of diameter *t*, where *t* is the tolerance value preceded by the symbol Ø.  	The extracted median line of the cylinder to which the tolerance applies shall be contained within a cylindrical zone of diameter 0,08. 
	Flatness tolerance	
▱	The tolerance zone shall be limited by two parallel planes a distance *t* apart.  	The extracted surface shall be contained between two parallel planes 0,08 apart. 
	Roundness tolerance	
○	The tolerance zone, in any cross-section normal to the axis, shall be limited by two concentric circles with a difference in radii of *t.*  	The extracted circumferential line in any cross-section of a cylindrical surface shall be contained between two coplanar concentric circles with a difference in radii of 0,05.  For any surface other than a cylinder, the direction in which the tolerance is applied shall always be defined with a direction symbol.

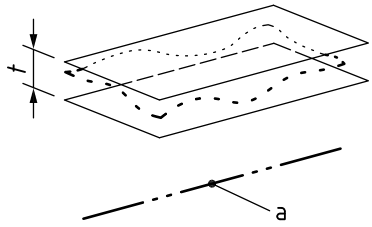
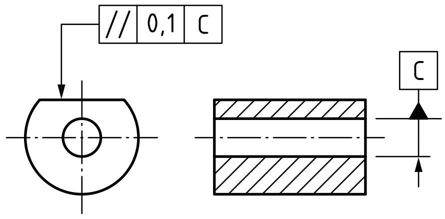
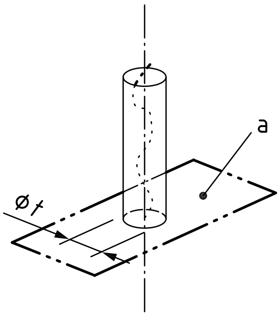
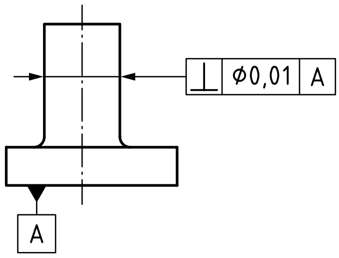
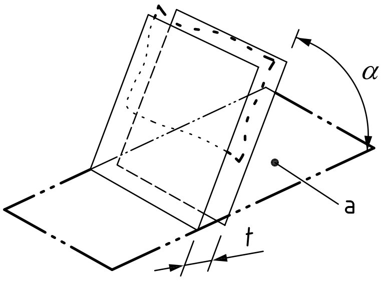
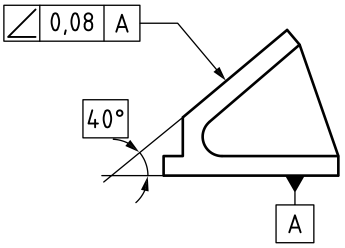
(Continues)

Figure 124 – ***Examples of geometrical tolerancing (Cont.)***

Symbol	Definition of the tolerance zone	Indication and explanation in 2D
	Cylindricity tolerance	
⌭	The tolerance zone shall be limited by two coaxial cylinders with a difference in radii of *t*. 	The extracted cylindrical surface shall be contained between two coaxial cylinders with a difference in radii of 0,1. 
	Surface profile with no datum system	
⌓	The tolerance zone shall be limited by two surfaces enveloping spheres of diameter *t*, the centres of which are situated on a line having the theoretically exact geometrical form, with each plane defined by the intersection plane indicator. 	The extracted surface shall be contained between two equidistant surfaces enveloping spheres of diameter 0,4, the centres of which are situated on a surface having the theoretically exact geometrical form. 
	Line profile with no datum system	
⌒	The tolerance zone shall be limited by two lines enveloping circles of diameter *t*, the centres of which are situated on a line having the theoretically exact geometrical form. 	An extracted line on the surface parallel to datum A shall be contained between two equidistant surfaces enveloping spheres of diameter 0,4, the centres of which are situated on a line having the theoretically exact geometrical form. 

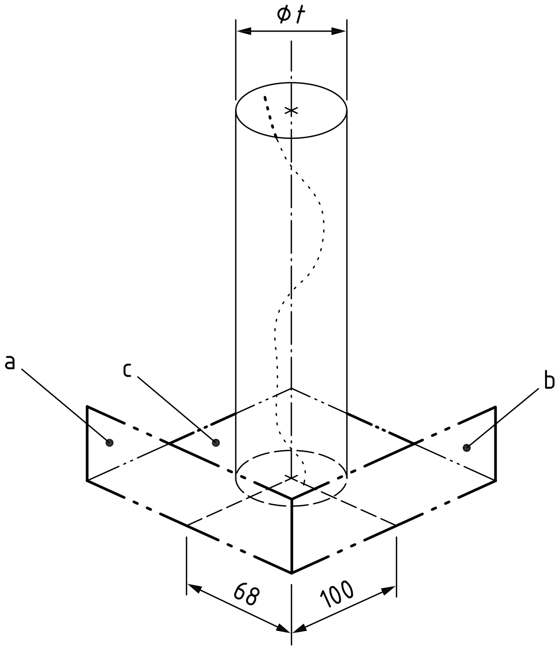
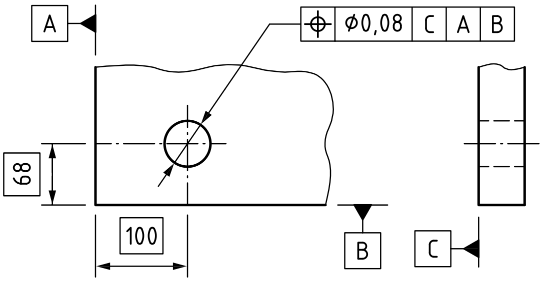
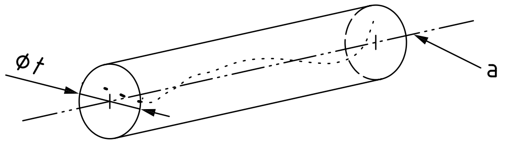
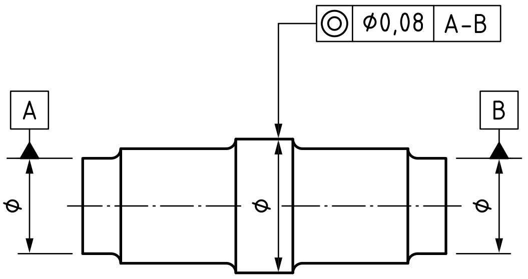
(Continues)

Figure 124 – *Examples of geometrical tolerancing* *(Cont.)*

Symbol	Definition of the tolerance zone	Indication and explanation in 2D
	Parallelism tolerance of a surface related to a datum line	
//	The tolerance zone shall be limited by two parallel planes a distance *t* apart and parallel to the datum (a = datum C).  	The extracted surface shall be contained between two parallel planes 0,1 apart, which are parallel to the datum axis C. 
	Perpendicularity tolerance of a feature of size related to a datum	
⊥	The tolerance zone shall be limited by a cylinder of diameter *t* perpendicular to the datum if the tolerance value is preceded by the symbol Ø (a = datum A).  	The extracted median line of the cylinder shall be within a cylindrical zone of diameter 0,01 perpendicular to datum plane A. 
	Angularity tolerance of a surface related to a datum	
∠	The tolerance zone shall be limited by two parallel planes a distance *t* apart and inclined at the specified angle to the datum (a = datum A).  	The extracted surface shall be contained between two parallel planes 0,08 apart that are inclined at a theoretically exact angle of 40° to datum plane A. 

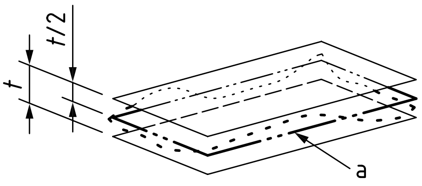
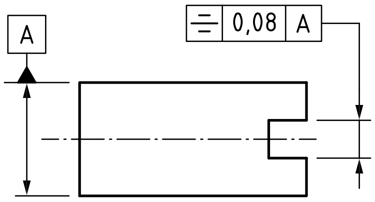
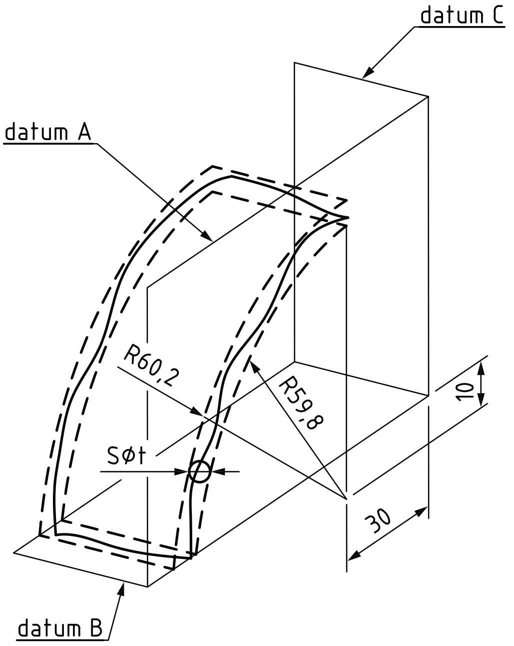
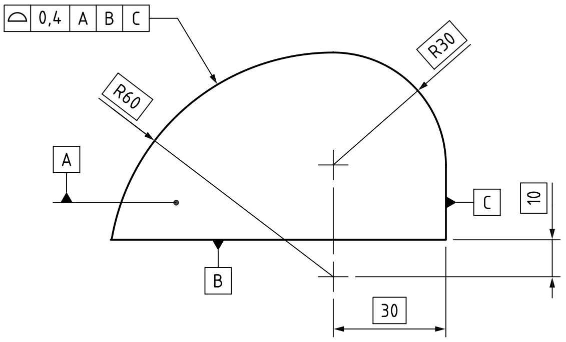
(Continues)

Figure 124 – *Examples of geometrical tolerancing* (Cont.)

Symbol	Definition of the tolerance zone	Indication and explanation in 2D
	Positional tolerance applied to a feature of size	
⌖	The tolerance zone shall be limited by a cylinder of diameter *t* if the tolerance value is preceded by the symbol Ø. The axis of the tolerance cylinder shall be fixed by theoretically exact dimensions with respect to the datum system. 	The extracted median line shall be within a cylindrical zone of diameter 0,08, the axis of which coincides with the theoretically exact position of the considered hole, with respect to datum planes C, A and B. 
	NOTE *a = datum A; b = datum B; c = datum C.*	
	Coaxiality tolerance	
◎	The tolerance zone shall be limited by a cylinder of diameter *t*; the tolerance value shall be preceded by the symbol Ø. The axis of the cylindrical tolerance zone coincides with the datum (a = datum A–B). 	An implied TED of zero exists between the datum feature and the toleranced feature. The extracted median line of the toleranced cylinder shall be within a cylindrical zone of diameter 0,08, the axis of which is the common datum straight line A–B. 

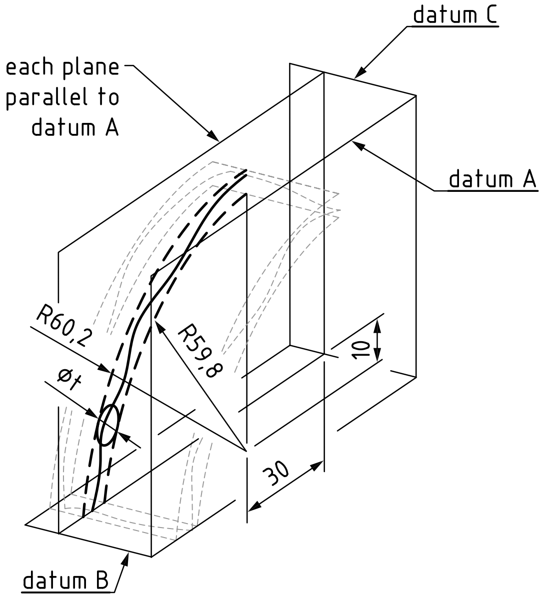
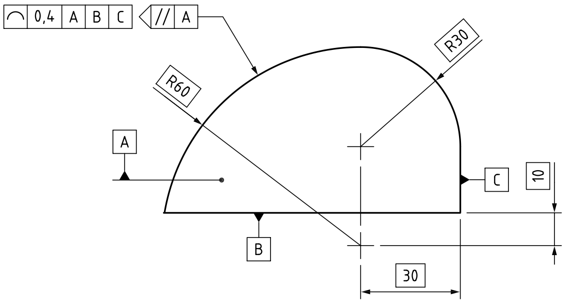
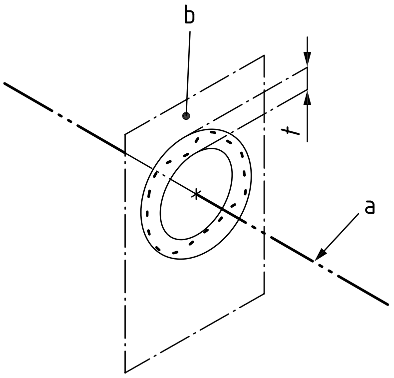
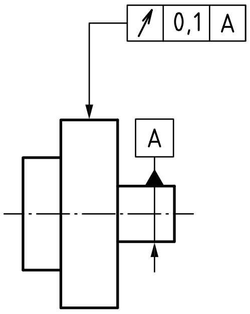
(Continues)

Figure 124 – *Examples of geometrical tolerancing* *(Cont.)*

Symbol	**Definition of the tolerance zone**	**Indication and explanation in 2D**
Symmetry tolerance of a feature of size		
⌯	The tolerance zone shall be limited by two parallel planes a distance *t* apart, symmetrically disposed about the median plane, with respect to the datum (a = datum). 	An implied TED of zero exists between the datum feature and the toleranced feature. The extracted median surface shall be contained between two parallel planes 0,08 apart, which are symmetrically disposed about the datum median plane A. 
Surface profile with a datum system		
⌓	The tolerance zone shall be limited by two surfaces enveloping spheres of diameter *t*, the centres of which are situated on a surface having the theoretically exact geometrical form with respect to the datum system. 	The extracted surface shall be contained between two equidistant surfaces enveloping spheres of diameter 0,4, the centres of which are situated on a surface having the theoretically exact geometrical form with respect to datum system A|B|C. 

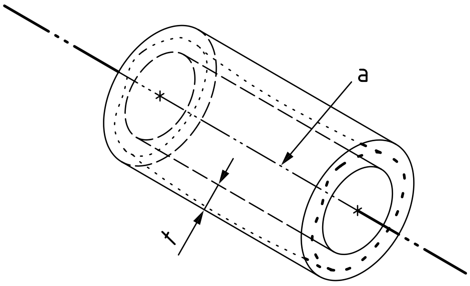
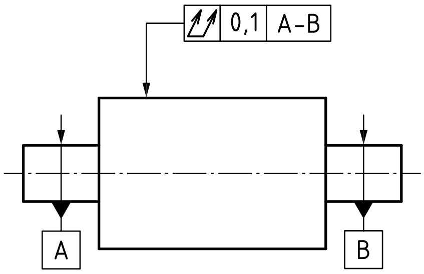
(Continues)

Figure 124 – *Examples of geometrical tolerancing (Cont.)*

Symbol	Definition of the tolerance zone	Indication and explanation in 2D
	Line profile with a datum system	
⌒	The tolerance zone shall be limited by two lines enveloping circles of diameter *t*, the centres of which are situated on a line having the theoretically exact geometrical form with respect to the datum system, within each plane defined by the intersection plane indicator.  	An extracted line on the surface, in a plane parallel to datum A, shall be contained between two equidistant lines enveloping circles of diameter 0,4, the centres of which are situated on a line having the theoretically exact geometrical form with respect to datum system A|B|C. 
	Circular run–out tolerance – Radial	
↗	The tolerance zone shall be limited within any cross-section perpendicular to the nominal surface by two concentric circles with a difference in radii of *t*, the centres of which coincide with the datum *a*.  NOTE a = datum; b = cross section plane.	The extracted line in any cross–section plane perpendicular to the nominal surface A shall be contained between two coplanar concentric circles with a difference in radii of 0,1 which are centred on datum axis A. 

(Continues)

Figure 124 – *Examples of geometrical tolerancing (Cont.)*

Symbol	Definition of the tolerance zone	Indication and explanation in 2D
	Total run-out tolerance – Radial	
⌰	The tolerance zone shall be limited by two coaxial cylinders with a difference in radii of *t*, the axes of which coincide with the datum (a = datum A–B).  	The extracted surface shall be contained between two coaxial cylinders with a difference in radii of 0,1 and the axes coincident with the common datum straight line A–B. 

7.8 Theoretically exact dimensions (TED)

COMMENTARY ON 7.8

A TED is a dimension, a linear or angular dimension used to define theoretically exact geometry.

A TED can be explicit or implicit. An implicit TED is not indicated. An implicit TED is one of the following: 0 mm, 0°, 90°, 180°, 270° and the angular distance between equally spaced features on a complete circle, when the location or orientation of the features concerned is controlled by a geometrical specification, or when the features are used as datum features.

When indicated on a TPD, a TED shall be shown with the dimension value enclosed in a rectangular frame. Any associated symbols such as ø or R shall also be contained within the frame. A multiplier indication such as *n*× shall be shown outside the frame.

NOTE 1 On a 3D model which is stated to consist of theoretically exact geometry, any dimension obtained through a query is a TED.

No tolerance shall be applied to the value of a TED, including individual or general tolerances.

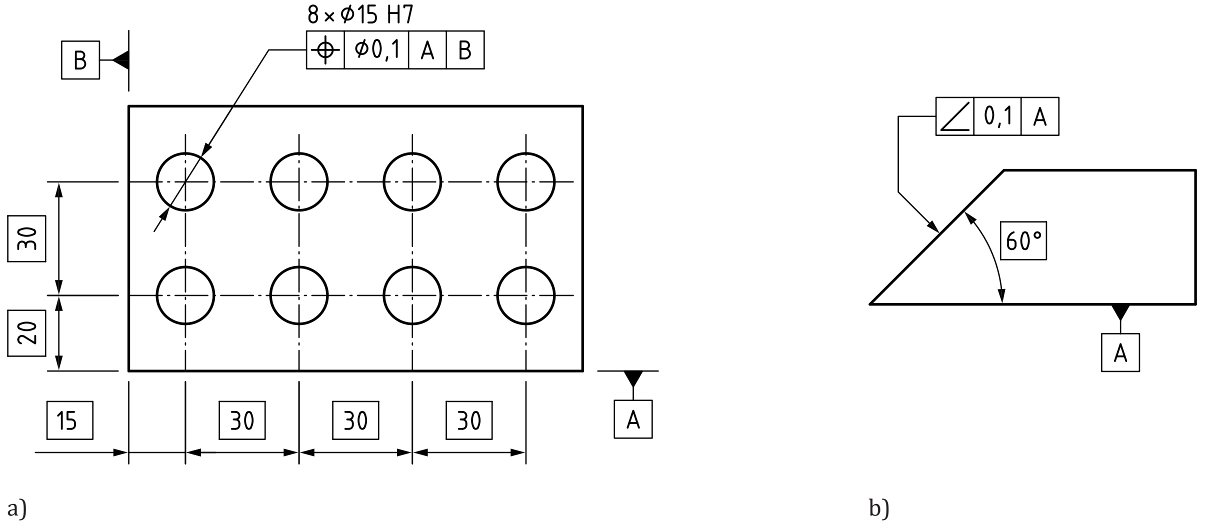
When a geometrical tolerance is used to tolerance the location of a feature (or group of features), the theoretically exact location of the feature(s) shall be defined with TEDs [see Figure 125a)].

When a geometrical tolerance is used to tolerance the orientation of a feature (or group of features), the theoretically exact orientation of the feature(s) shall be defined with TEDs [see Figure 125b)].

NOTE 2 Some examples are given in Figure 124.

When a profile tolerance is applied to a feature (or group of features), the theoretically exact profile or surface shall be defined with TEDs.

Figure 125 – *Use of theoretically exact dimensions (TEDs)*



7.9 Material condition modifiers

NOTE See BS EN ISO 2692 for additional information.

7.9.1 Maximum material requirement applied to the toleranced feature

When the maximum material requirement (MMR) is applied to a feature, it defines a maximum material virtual condition (MMVC) boundary for the feature, and the following requirements shall be met.

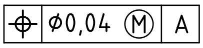
a) The upper size limit shall be satisfied.

b) The lower size limit shall be satisfied.

c) The surface of the feature shall not violate the MMVC boundary (an internal feature shall always remain outside its MMVC boundary, and an external feature shall always remain inside its MMVC boundary).

The MMR for the toleranced feature shall be invoked by placing the Ⓜ modifier immediately after the tolerance value in the tolerance zone section (see Figure 126).

Figure 126 – *Indication of the maximum material requirement*



The MMR shall only be applied to features of linear size.

The size of the MMVC boundary is known as the maximum material virtual size (MMVS) of the feature, and shall be calculated using the following:

1) for an internal feature of size (e.g. a bore): MMVS = MMS – Geometrical Tolerance; and

2) for an external feature of size (e.g. a shaft): MMVS = MMS + Geometrical Tolerance.

NOTE 1 MMS = maximum material size.

NOTE 2 The MMVC boundary is equivalent to a functional gauge limit, of fixed size, which can be used to verify the geometrical tolerance requirement.

COMMENTARY ON 7.9.1

The effect of an MMVC boundary can be approximated by the concept of "bonus tolerance". The tolerance value given in the tolerance indicator applies to the feature when it is at its maximum material size. When the feature departs from its MMS, the difference between the actual size and the MMS is considered a "bonus tolerance" which is added to the geometrical tolerance value as follows:

a) for an internal feature of size (e.g. a bore):

Geometrical tolerance allowance = Geometrical Tolerance value + (Actual Size – MMS);

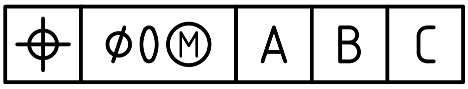
b) for an external feature of size (e.g. a shaft):

Geometrical tolerance allowance = Geometrical Tolerance value + (MMS – Actual Size).

7.9.2 Maximum material requirement applied with a zero tolerance

In order to maximize the range of features which can satisfy a tolerance using the MMR, the MMS of a feature shall be set equal to the MMVS for the feature and the geometrical specification shall be set to zero. This shall be indicated by using a zero tolerance value with the Ⓜ modifier in the geometrical tolerance indicator as shown in Figure 127, or an equivalent indication (see **7.9.8**).

Figure 127 – ***Indication of the maximum material requirement***



NOTE The same outcome can be achieved through the use of the reciprocity requirement (see 7.9.7).

7.9.3 Maximum material requirement applied to a datum reference in a tolerance indicator

When the MMR is applied to a datum reference in a tolerance indicator, the datum shall be derived from the MMVC boundary of the feature instead of the physical surface of the feature. In addition to any other requirements which have been directly defined for the datum feature, the following requirements shall be met.

a) The surface of the datum feature shall not violate the MMVC boundary (an internal feature shall always remain outside its MMVC boundary, and an external feature shall always remain inside its MMVC boundary).

b) The MMR for the datum feature shall be invoked by placing the Ⓜ modifier immediately after the datum name in the datum section of the tolerance indicator (see Figure 128).

Figure 128 – ***Indication of the maximum material requirement***

c) The MMR shall only be applied to datum features which are features of linear size.

NOTE 1 Use of the MMR for the datum feature is equivalent to basing the datum on a fixed size gauge limit for the datum feature rather than the actual surface of the datum feature.

NOTE 2 Where MMR is applied to a datum, care should be taken to ensure the datum feature specification either has MMR or the envelope requirement applied to define a boundary condition. If it does not have a boundary condition defined, the reference to the datum feature with the MMR will define a boundary condition which might be more restrictive than the tolerances applied directly to the datum feature. The result is a specification which appears to contradict itself.

COMMENTARY ON 7.9.3

The effect of applying the MMR to a datum feature is that when the datum feature departs from its MMVC limit, the datum can move relative to the datum feature by a corresponding amount. This is sometimes addressed in a similar manner to the "bonus tolerance" concept, with the potential movement of the datum described as a "datum shift" allowance and is calculated in the same way.

7.9.4 Least material requirement applied to the toleranced feature

When the least material requirement (LMR) is applied to a feature, it defines a least material virtual condition (LMVC) boundary for the feature, and the following requirements shall be met.

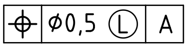
a) The upper size limit shall be satisfied.

b) The lower size limit shall be satisfied.

c) The surface of the feature shall not violate the LMVC boundary; an internal feature shall always remain inside its LMVC boundary, and an external feature shall always remain inside its LMVC boundary.

The LMR for the toleranced feature shall be invoked by placing the Ⓛ modifier immediately after the tolerance value in the tolerance zone section (see Figure 129).

Figure 129 – *Indication of the least material requirement*



The LMR shall only be applied to features of linear size.

The size of the LMVC boundary is known as the least material virtual size (LMVS) of the feature, and shall be calculated using the following:

1) for an internal feature of size (e.g. a bore): LMVS = LMS + Geometrical Tolerance; and

2) for an external feature of size (e.g. a shaft): LMVS = LMS – Geometrical Tolerance.

NOTE 1 LMS = Least material size.

NOTE 2 The LMVC boundary is inside the material of a feature which satisfies the tolerance requirement, so this requirement cannot be verified with a mechanical gauge.

COMMENTARY ON 7.9.4

The effect of an LMVC boundary can be approximated by the concept of "bonus tolerance". The tolerance value given in the tolerance indicator applies to the feature when it is at its LMS. When the feature departs from its LMS, the difference between the actual size and the LMS is considered a "bonus tolerance" which is added to the geometrical tolerance value.

a) For an internal feature of size (e.g. a bore):

Geometrical tolerance allowance = Geometrical Tolerance value + (LMS – Actual Size);

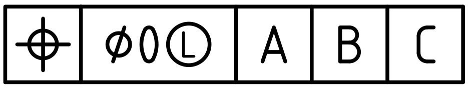
b) For an external feature of size (e.g. a shaft):

Geometrical tolerance allowance = Geometrical Tolerance value + (Actual Size – LMS).

7.9.5 Least material requirement applied with a zero tolerance

In order to maximize the range of features which can satisfy a tolerance using the LMR, the LMS of a feature shall be set equal to the LMVS for the feature and the geometrical specification shall be set to zero. This shall be indicated by using a zero tolerance value with the Ⓛ modifier in the geometrical tolerance indicator as shown in Figure 130, or an equivalent indication (see **7.9.8**).

Figure 130 – ***Indication of the least material requirement applied with a zero tolerance***



NOTE The same outcome can be achieved through the use of the reciprocity requirement (see 7.9.7).

7.9.6 Least material requirement applied to a datum reference in a tolerance indicator

When the LMR is applied to a datum reference in a tolerance indicator, the datum shall be derived from the LMVC boundary of the feature instead of the physical surface of the feature. In addition to any other requirements which have been directly defined for the datum feature, the surface of the datum feature shall not violate the LMVC boundary.

The LMR for the datum feature shall be invoked by placing the Ⓛ modifier immediately after the tolerance value in the tolerance zone section (see Figure 131).

Figure 131 – ***Least material requirement applied to a datum reference in a tolerance indicator***

The LMR shall only be applied to datum features which are features of linear size.

NOTE Use of the LMR for the datum feature means that the datum is based on a boundary inside the material of the feature, so the datum cannot be established by a mechanical gauge.

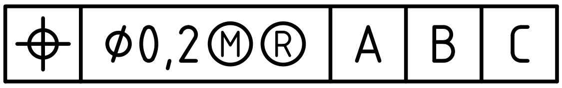
COMMENTARY ON 7.9.6

The effect of applying the LMR to a datum feature is that when the datum feature departs from its LMVC limit, the datum can move relative to the datum feature by a corresponding amount. This is sometimes addressed in a similar manner to the "bonus tolerance" concept, with the potential movement of the datum described as a "datum shift" allowance and is calculated in the same way.

7.9.7 Reciprocity requirement (RPR)

Where the reciprocity requirement (RPR) is used with a geometrical specification applied to a linear feature of size, it shall be indicated as a supplementary requirement to MMC or LMC by placing the Ⓡ symbol after the Ⓜ or Ⓛ symbol in the tolerance zone compartment as shown in Figure 132.

Figure 132 – ***Reciprocity requirement***



RPR shall not be indicated in the datum section.

RPR shall not be used when the geometrical tolerance value is given as zero value (see 7.9.2 and 7.9.5).

When the RPR modifier is used with MMR, the requirement to satisfy the MMS limit no longer applies. For an external feature, the size of the feature can be larger than the MMS, but the surface shall still be contained within the MMVC boundary. For an internal feature, the size of the feature can be smaller than the MMS, but the surface shall still lie outside the MMVC boundary.

When the RPR modifier is used with LMR, the requirement to satisfy the LMS limit no longer applies. For an external feature, the size of the feature can be smaller than the LMS, but the surface shall still remain outside the LMVC boundary. For an internal feature, the size of the feature can be larger than the LMS, but the surface shall still be contained within the LMVC boundary.

NOTE To achieve the same conditions as reciprocity it is common practice to apply all available tolerance to the linear size dimension and indicate the geometrical tolerance as 0 Ⓜ *or* 0 Ⓛ*. In this case reciprocity is not required.*

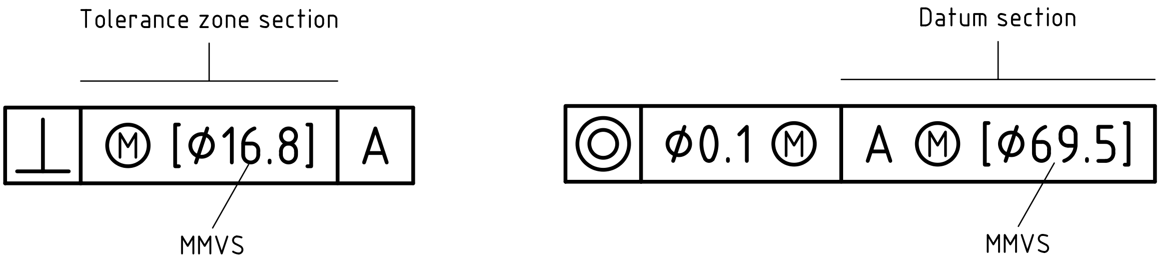
7.9.8 Direct indication of virtual size

NOTE 1 A planned future revision of BS EN ISO 2692 will introduce the option of indicating the MMVS and LMVS for a feature directly, rather than having to calculate these values.

The MMVS or LMVS value shall be indicated directly, in square brackets, after the material condition modifier.

NOTE 2 This method can be used in the tolerance zone section [see Figure 133a)] and in any appropriate compartment of the datum section of the tolerance indicator [see Figure 133b)].

Figure 133 – *Direct indication of virtual size*



a) Tolerance zone section

b) Datum section

NOTE 3 The next edition of BS EN ISO 2692 (which will be post 2020) will allow this option. If an organization wishes to make use of these indications prior to that, the meaning should be clearly explained in the TPD.

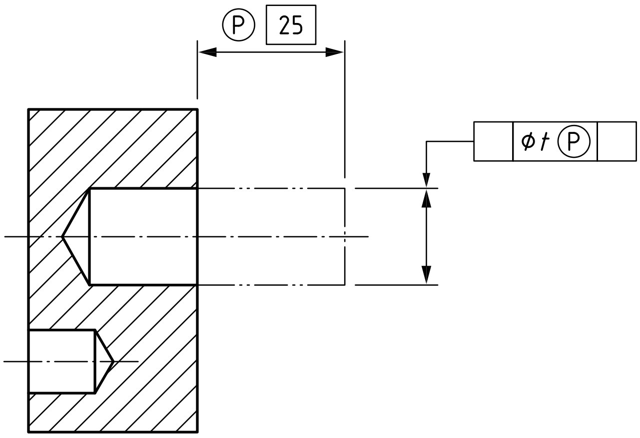
7.10 Additional indications relating to the toleranced feature

7.10.1 Projected tolerance zone

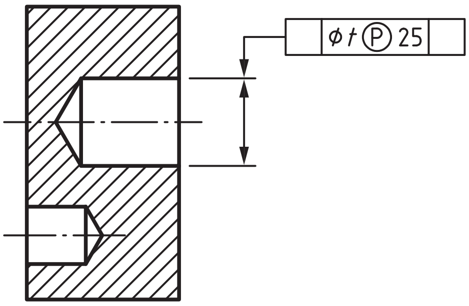
Projected tolerance zones shall be indicated by the specification modifier symbol Ⓟ in the tolerance value compartment (see example in Figure 134).

NOTE See BS EN ISO 1101 for additional information.

Figure 134 – ***Two ways of indicating a geometrical specification with projected tolerance modifier***



a) Direct indication of the length of the extension by a TED



b) Indirect indication of the length of the projected toleranced feature in the tolerance frame

7.10.2 Free-state condition

A geometrical tolerance which applies to a non-rigid part in its free-state condition (i.e. with no constraints or clamps) shall be indicated by the specification modifier symbol Ⓕ placed after the specified tolerance value (see Figure 135).

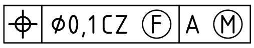
*NOTE 1 See also **3.1.8** for requirements relating to BS EN ISO 10579. See BS ISO 10579 and BS EN ISO 1101 for additional information.*

Figure 135 – ***Free-state condition***

a) b)

NOTE 2 Several specification modifiers, e.g. Ⓟ, Ⓜ, Ⓛ, Ⓕ and CZ could be used simultaneously in the same tolerance frame (see Figure 136).

Figure 136 – ***Use of several specification modifiers***



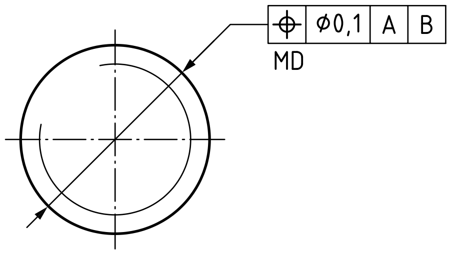
7.10.3 Tolerances applied to threaded features

Geometrical tolerances controlling the location or orientation of screw threads shall apply to the axis derived from the pitch cylinder of the thread, unless otherwise specified.

If the tolerance is to be applied to the axis of the major diameter of the thread, the modifier MD shall be used as an adjacent indication for the tolerance indicator (see example of Figure 137).

If the tolerance is to be applied to the axis of the minor diameter of the thread, the modifier LD shall be used as an adjacent indication for the tolerance indicator.

Figure 137 – ***Example of "MD"***



7.10.4 Tolerances applied to splines and gears

When a geometrical tolerance is used to control the location or orientation of a gear profile applied to a bore or shaft, or a spline applied to a bore or shaft, an adjacent indication for the tolerance indicator shall always be used to indicate whether the tolerance applies to the axis of the major diameter, minor diameter or pitch diameter, because there is no default.

If the tolerance is to be applied to the axis of the pitch diameter of the gear or spline, the modifier PD shall be used as an adjacent indication for the tolerance indicator.

If the tolerance is to be applied to the axis of the major diameter of the gear or spline, the modifier MD shall be used as an adjacent indication for the tolerance indicator.

If the tolerance is to be applied to the axis of the minor diameter of the thread, the modifier LD shall be used as an adjacent indication for the tolerance indicator.

7.11 Unequal tolerance zone (UZ)

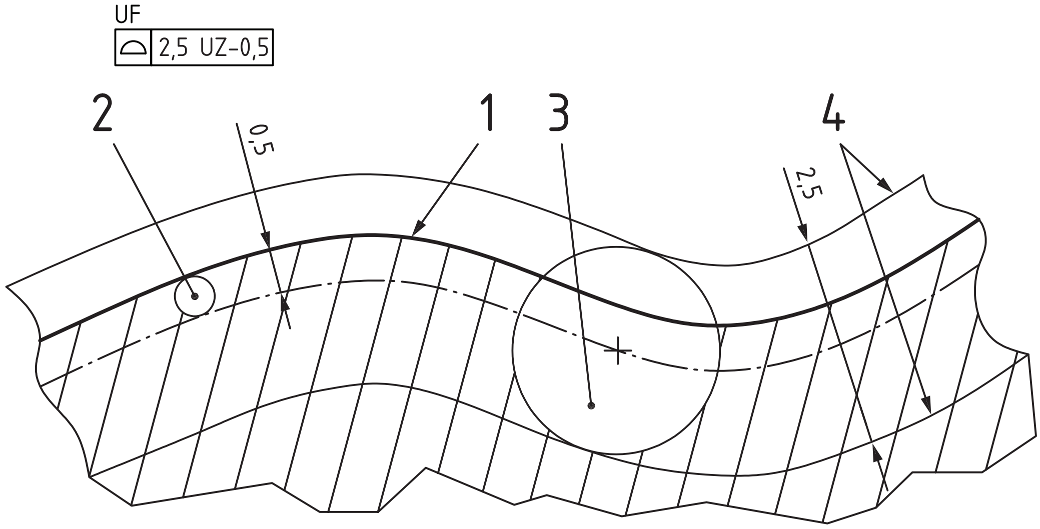
When a profile tolerance is applied to a surface, or when a position tolerance is applied to a plane, the tolerance zone shall be symmetrical around the theoretically exact feature (TEF), unless otherwise indicated; this is known as a bilateral tolerance zone.

If an unequal (offset) tolerance zone is required, the UZ modifier shall be placed after the tolerance value in the tolerance indicator. The value following the UZ shall indicate the magnitude of the offset of the tolerance zone. A positive value (+) shall indicate that the zone is offset in a direction away from the material of the TEF. A negative value (-) shall indicate that the zone is offset in a direction towards the material of the TEF.

The "+" symbol or "-" symbol shall always be included with the offset value

Example: In Figure 138, the 2,5 tolerance zone is offset into the material (-ve) by 0,5. The size of the zone outside of the material will be (2,5/2) – 0,5 = 0,75 and the size of zone inside of the material will be –(2,5/2) – 0,5 = –1,75.

Figure 138 – ***Example of an unequal tolerance zone***



Key

1	Single complex TEF; in this example, the material is below the feature
2	One of the infinite number of spheres defining the offset theoretical feaure, i.e. the reference feature
3	One of the infinite number of spheres defining the tolerance zone along the reference feature
4	Limits of the tolerance zone

For profile specifications of complex lines or surfaces, the UZ specification element can be used with or without datums

To specify a tolerance zone offset entirely into the material of the TEF, or entirely outside the material of the TEF, the value following the UF modifier shall be half the tolerance zone size value.

7.12 Restrictive specifications

7.12.1 Tolerance requirements applied to a specific restricted portion of a feature

To specify a tolerance requirement which is applied to a specific restricted portion of a feature, the restricted portion shall be defined using one of the following methods:

a) in a 2D view, by outlining the surface portions involved using a long-dashed dotted wide line (in accordance with BS ISO 128-24:1999, type 04.2), with the location and dimensions defined by TEDs [see Figure 139c)];

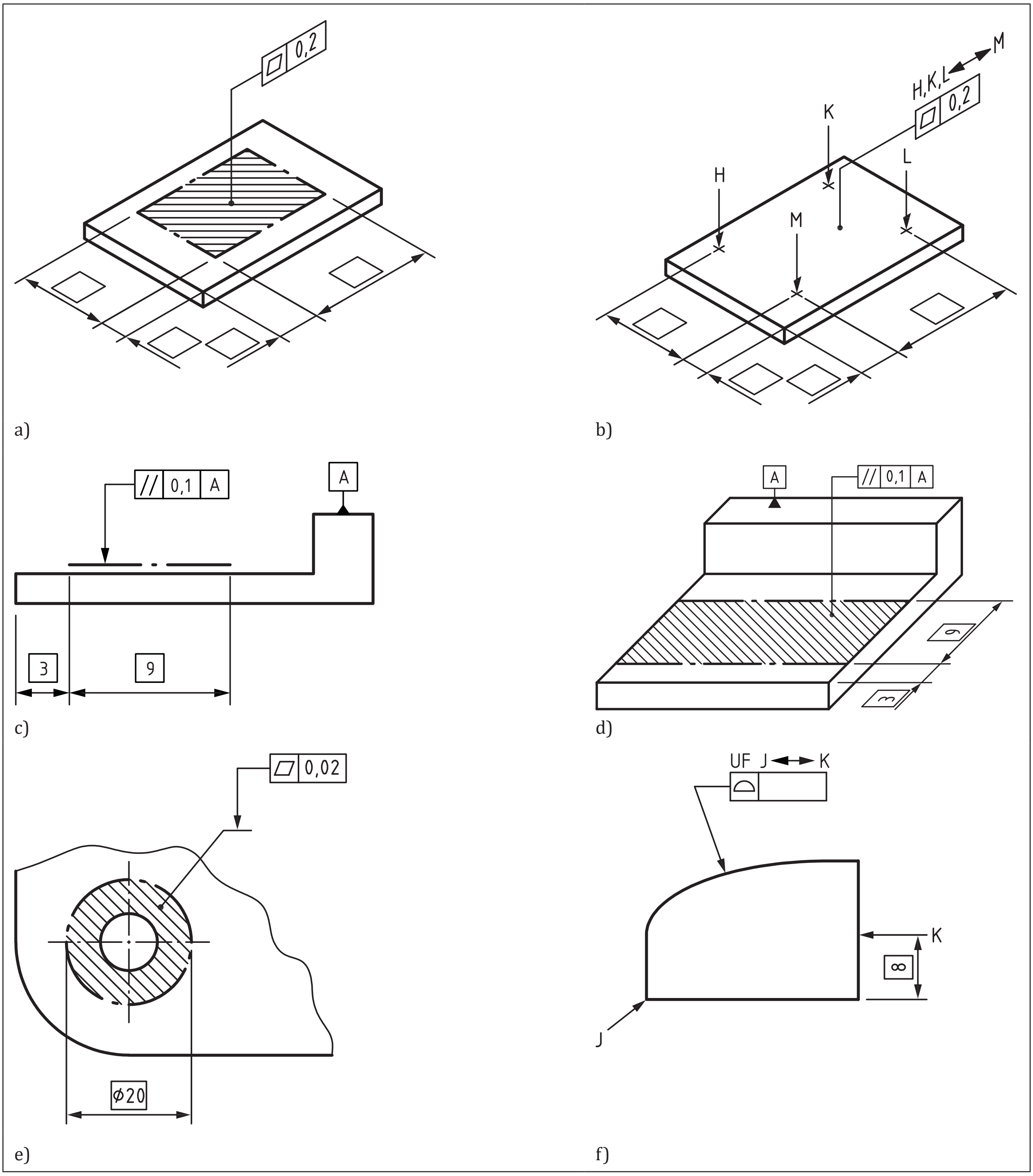
b) by a hatched area with its border indicated as a long-dashed dotted wide line (in accordance with BS ISO 128-24:1999, type 04.2), with the location and dimensions defined by TEDs [see Figure 139a), Figure 139d) and Figure 139e)];

c) by its corner points, indicated as crosses on the integral feature (the location of the points being defined by TEDs), identified by capital letters and leader lines terminated by an arrow, and the letters indicated above the tolerance indicator with a "between" symbol between the last two [see Figure 139b)]. The border is formed by connecting the corner points with straight segments;

d) by two straight border lines identified by capital letters and leader lines terminated by an arrow (the location of the border lines being defined by TEDs) combined with an indication using the "between" symbol [see Figures 139f)].

The leader line from the geometrical tolerance indication shall terminate on the restricted area.

Figure 139 – ***Tolerance applied to a restricted part of a feature***



7.12.2 Tolerance requirements applied to any given restricted portion of a feature

If a tolerance of the same characteristic is applied to a restricted length, lying anywhere within the total extent of the feature, the value of the restricted length shall be added after the tolerance value and separated from it by an oblique stroke [see Figure 140a)].

NOTE 1 If two or more tolerances of the same characteristic are to be indicated, they can be combined as shown in Figure 140b). The use of separate tolerance characteristic symbols or a single combined tolerance characteristic symbol are both permitted and have an identical meaning. Where a single combined tolerance symbol is used this is not equivalent to the composite tolerancing concept in ASME Y14.5 [1].

Figure 140 – *Examples of tolerances of the same characteristic*

a) Single tolerance

b) Two or more tolerances

If a tolerance of the same characteristic is applied to a restricted area, lying anywhere within the total extent of the feature, the value of the restricted area shall be added after the tolerance value and separated from it by an oblique stroke [see Figure 141)].

The restricted area shall be:

a) circular: indicated by a diameter symbol followed by the diameter value;

b) square or rectangular: indicated by the length and width values separated by “×”;

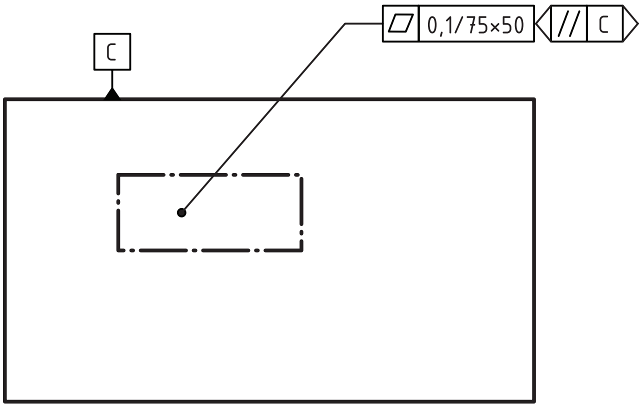
c) cylindrical segment: indicated by the axial length and angular circumferential values separated by “×”;

d) spherical segment: indicated by two perpendicular angular circumferential values separated by “×”.

Where it is necessary to define the orientation or alignment of the area, this shall be defined by an orientation plane indicator.

NOTE 2 Where two values are given to define the area, the orientation plane indicator applies to the first of the two values.

Figure 141 – *Tolerance applied to an area*



7.13 The use of profile tolerances applied to derived features

NOTE BS EN ISO 1660:2017 introduced the possibility of applying profile tolerances to derived features. This subclause sets out how this is applied.

7.13.1 General

Profile tolerances applied to derived features shall conform to BS EN ISO 1660.

*NOTE Subclause **7.13.2** specifies the rules for geometrical specifications of derived features using the line profile and surface profile characteristic symbols as defined in BS EN ISO 1660.*

7.13.2 Profile tolerancing

COMMENTARY ON 7.13.2

BS EN ISO 1660:2017 extended the use of profile tolerances to enable them to be applied to derived features as well as integral features. As this is a major change to the way in which profile tolerances can be used, this section has been included to explain their application to derived features.

The line profile symbol or the surface profile symbol shown in Table 19 shall be used in the characteristic section of the tolerance indicator (see BS EN ISO 1101).

Table 19 – *Symbols for geometrical characteristics*

Symbol	**Meaning**
⌒	Line profile symbol
⌓	Surface profile symbol

SOURCE: BS EN ISO 1660:2017, Table 2.

The use of profile tolerances shall be restricted to the features identified in Table 20.

Table 20 – *Valid geometrical characteristic symbol and nominal tolerance feature combinations*

Description	⌒	⌓
Integral straight line	×	
Derived straight line	×	
Integral non-straight line	×	
Derived non-straight line	×	
Integral flat surface		×
Derived flat surface		×
Integral non-flat surface		×
Derived non-flat surface		×

SOURCE: BS EN ISO 1660:2017, Table 3.

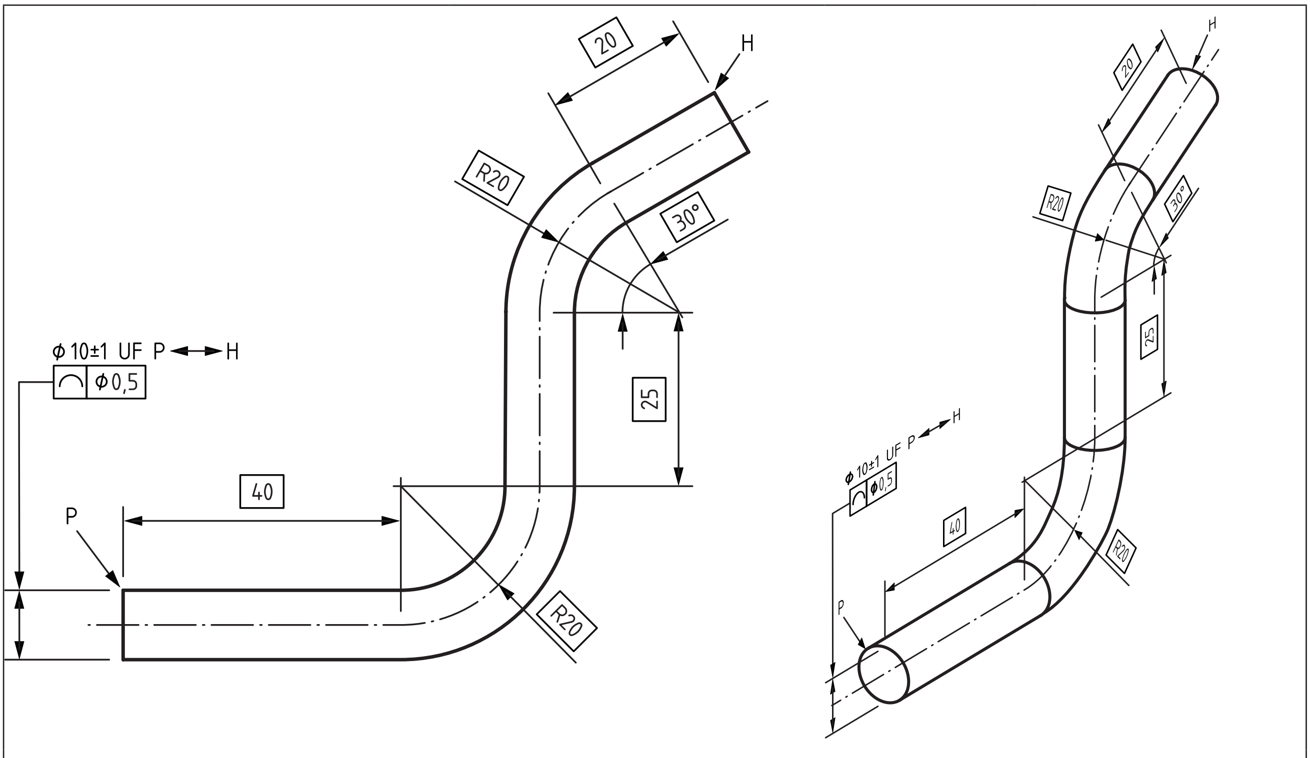
The drawing indications in Figure 142a) and b) shall be interpreted as follows:

- according to rule D in BS EN ISO 1660: because the “between” symbol and the UF modifier are used, the specification applies to a compound feature limited by the borders indicated in the “between” symbol – the compound feature is considered as one feature;
- according to rule A in BS EN ISO 1660, the theoretical exact feature (TEF) shall be defined with theoretically exact dimensions – in this case, the toleranced feature is defined as a compound line consisting of three straight segments joined by two circular segments of radius 20; one 90° segment and one 30° segment;
- according to rule B in BS EN ISO 1660, the toleranced feature is a line and, according to the indication rules given in BS EN ISO 1101:2017, Clause **8**, the toleranced feature is a derived line;
- according to rule C in BS EN ISO 1660, the tolerance zone is limited by a tube enveloping sphere with a diameter equal to the tolerance value, the centres of which are situated on the TEF; and because the specification does not reference datums, the tolerance zone is not constrained.

NOTE 1 The meaning would have been the same if the position characteristic symbol had been used instead of the line profile characteristic symbol.

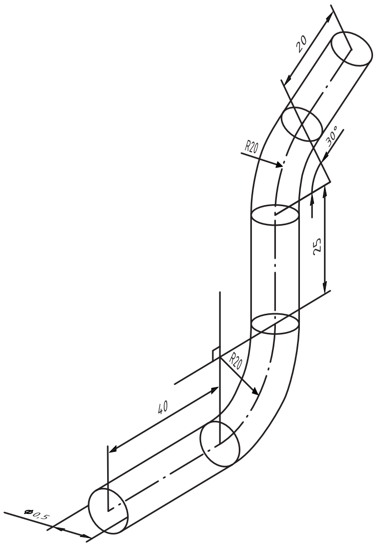
*NOTE 2 For rules A, B, C and D see BS EN ISO 1660:2017, **5.2**.*

Figure 142 – *Line profile specification for a compound derived feature*



a) 2D drawing indication

b) 3D drawing indication



c) Tolerance zone

The drawing indications in Figure 143a) and b) shall be interpreted as follows:

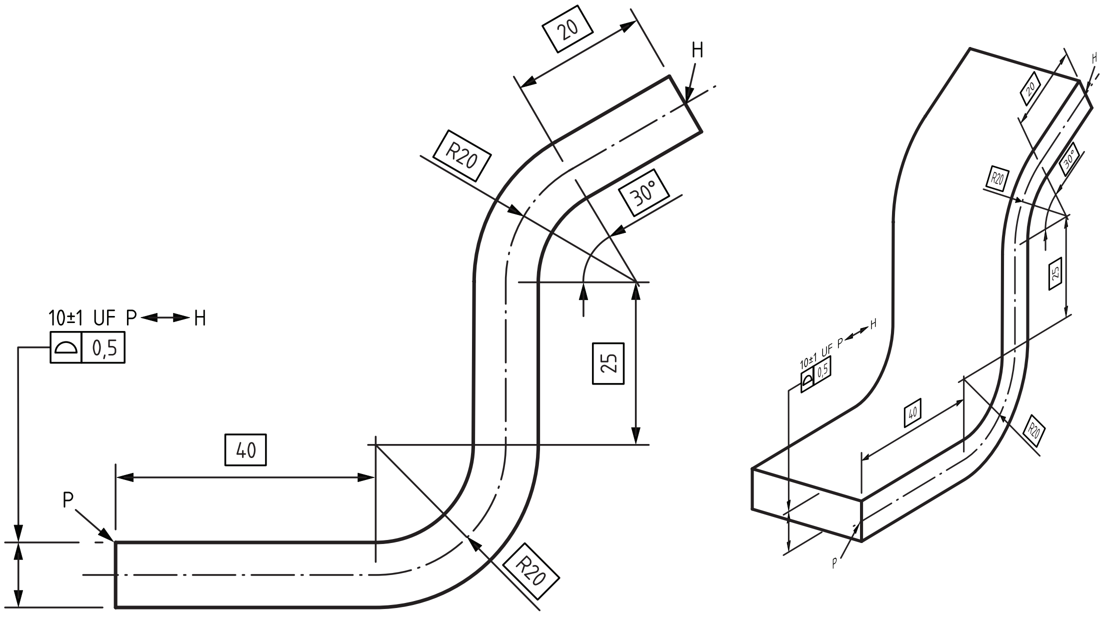
- according to rule D: because the “between” symbol and the UF modifier are used, the specification applies to a compound feature limited by the borders indicated in the “between” symbol – the compound feature is considered as one feature;
- according to rule A, the TEF shall be defined with theoretically exact dimensions – in this case, the toleranced feature is defined as a compound surface consisting of three planar segments joined by two curved segments of radius 20; one 90° segment and one 30° segment;
- according to rule B, the toleranced feature is a surface and, according to the indication rules given in BS EN ISO 1101:2017, Clause **8**, the toleranced feature is a derived surface;

- according to rule C, the tolerance zone is limited by two equidistant surfaces enveloping spheres with a diameter equal to the tolerance value, the centres of which are situated on the TEF; and
- because the specification does not reference datums, the tolerance zone is not constrained.

NOTE 3 The meaning would have been the same if the position symbol had been used.

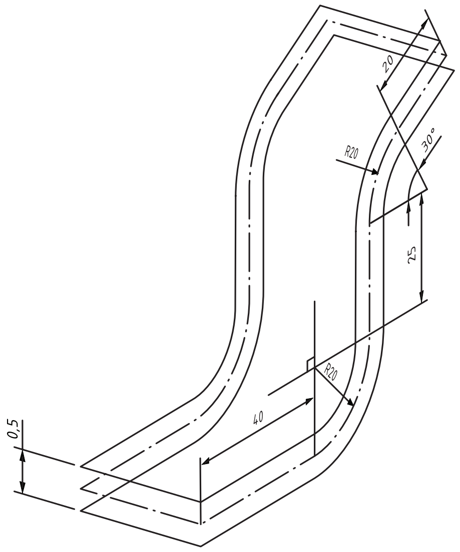
*NOTE 4 For rules regarding profile tolerances, see BS EN ISO 1660:2017, **5.2**.*

Figure 143 – *Surface profile specification for a compound derived feature*



a) 2D drawing indication

b) 3D drawing indication



c) Tolerance zone

7.14 Interrelationship of geometrical tolerances

Location tolerances of a feature shall control location deviation, orientation deviation and form deviation of this feature.

Orientation tolerances of a feature shall control orientation and form deviations of the feature.

Form tolerances of a feature shall only control form deviations of this feature.

COMMENTARY ON 7.14

For functional reasons, one or more characteristics can be toleranced to define the geometrical deviations of a feature. Certain types of tolerances which limit the geometrical deviations of a feature can also limit other types of deviations for the same feature.

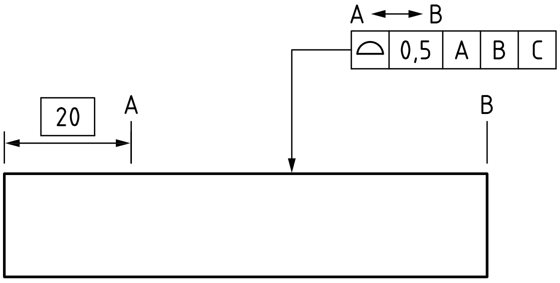
Section 8 Additional indications for geometrical tolerances

8.1 Restricted or compound tolerancing of features

8.1.1 Between

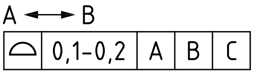
If a specification is not to apply to an entire feature it shall be restricted using the between symbol (◄►). The specification shall then only apply to a restricted portion of the feature. The start and end of the restricted element shall be identified using capital letters. The between symbol shall be used in combination with the letters and placed over the tolerance indicator (see Figure 144).

Figure 144 – *Use of between symbol*



The tolerance zone may also vary uniformly over the length of a toleranced feature or portion of a feature. This shall be done by identifying the start and finish value of the tolerance in the tolerance zone section with the two values separated by a hyphen (see Figure 145).

Figure 145 – *Use of between symbol*

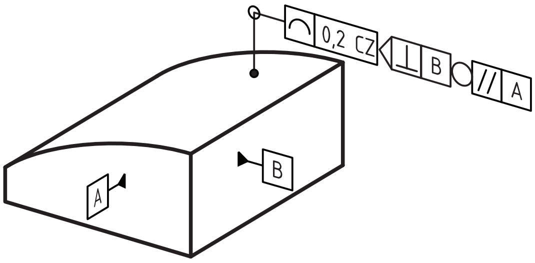


8.1.2 All-around

If a specification is to be applied to features represented by a closed outline or 2D sections of an outline, the all-around symbol (○—) shall be used, placed at the intersection of the leader line and reference line of the tolerance indicator (see Figure 146).

The all-around symbol shall always be accompanied by UF, SZ or CZ unless all degrees of freedom are locked and should be accompanied by a collection plane indicator.

Figure 146 – *Use of all-round symbol*



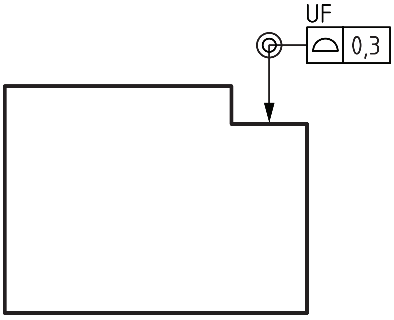
8.1.3 All-over

If a specification is to apply to an entire workpiece, the all-over symbol (◎——) shall be used (see Figure 147).

The all-over symbol shall always be accompanied by UF, SZ or CZ unless all degrees of freedom are locked.

NOTE Where features of different types are included in a workpiece, the use of all over can lead to ambiguity. In such cases additional definition or notes should be used to clarify the features included in the specification using the all over symbol.

Figure 147 – ***Use of all-over symbol***



8.2 United features

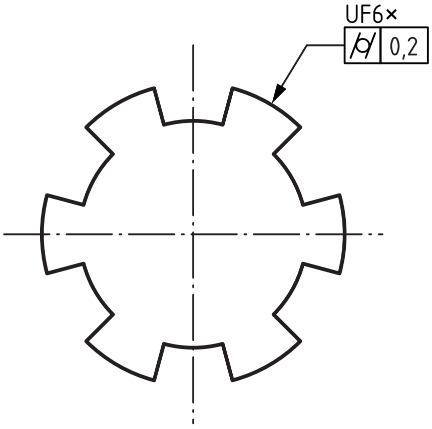
By default, a toleranced feature shall be a complete single feature. Where multiple features are to be considered as a single feature, the specification element UF shall be used. The features to be united shall be identified using multiple leader lines or *n*× and preceded by UF. The features shall then be considered and toleranced as a single feature.

The features being united shall combine to create a single compound integral feature.

NOTE 1 For example, a collection of curved surfaces may be united using UF and toleranced as a single cylindrical feature e.g. using cylindricity. This would not be possible if treating them as several individual features (see Figure 148).

NOTE 2 Coaxial cylinders with identical diameters may be considered as a single cylinder using UF. Non–coaxial cylinders or cylinders with different diameters cannot. Multiple planar features on a common plane may be treated as a united feature. Planar features that do not reside on a common plane cannot.

Figure 148 – ***Example united feature***



8.3 Feature patterns (CZ, CZR, SZ and SIM)

8.3.1 Internal and external constraints

NOTE 1 Requirements for pattern specifications are specified in BS EN ISO 5458.

A pattern specification shall be defined when a geometrical specification is applied to a set of identical features, and the tolerance zones shall have their locations and/or orientations locked together.

The relationships between the tolerance zones in the pattern are known as "internal constraints", each of which shall consist of one of the following:

a) location and orientation constraints;

b) only orientation constraints.

NOTE 2 If there are no internal constraints, there is no pattern.

The relationships between the pattern of tolerance zones and the datum system are known as "external constraints", each of which shall consist of one of the following:

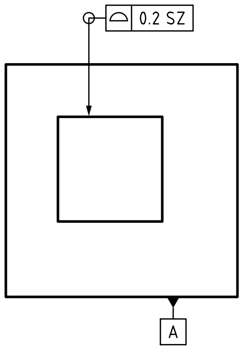
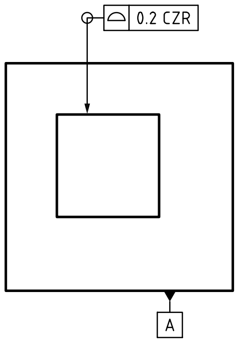
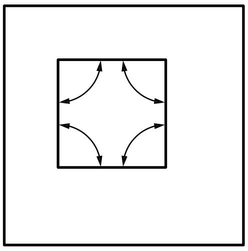
1) location and orientation constraints;

2) only orientation constraints.

*NOTE 3 If a requirement arises to group dissimilar features together, this can be achieved through the use of the SIM indicator (see **8.3**).*

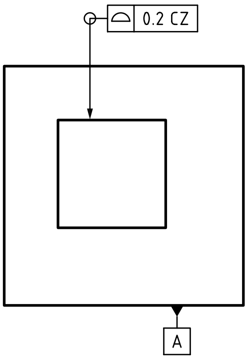
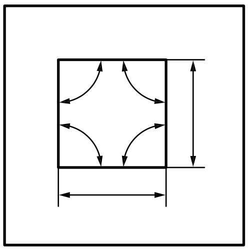
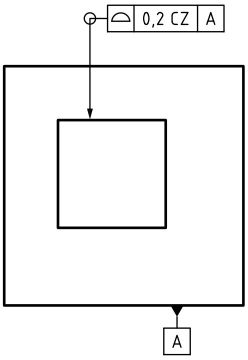
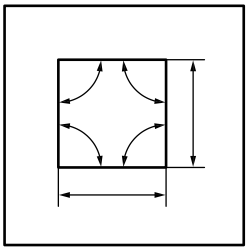
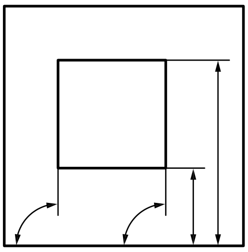
NOTE 4 Examples of internal and external constraints are given in Table 21.

Table 21 — *Example showing internal and external constraints*

Specification	Internal constraints	External constraints
a)  TEDs defined in CAD model 123 rev. C.	None. No pattern is specified.	None. No datum or datum system(s) is referenced by the geometrical specification.
b)  TEDs defined in CAD model 123 rev. C.	CZR introduces internal orientation constraints. 	None. No datum or datum system(s) is referenced by the geometrical specification.

(Continues)

Table 21 — *Example showing internal and external constraints (Cont.)*

Specification	Internal constraints	External constraints
c)  TEDs defined in CAD model 123 rev. C.	CZ introduces internal location and orientation constraints 	None. No datum or datum system(s) is referenced by the geometrical specification.
d)  TEDs defined in CAD model 123 rev. C.	CZ introduces internal location and orientation constraints. 	The reference to datum A introduces external constraints to datum A. The type of constraints depends on the tolerance characteristic. 

8.3.2 Specifying a pattern requirement

The geometrical specification shall be applied to a set of identical features by one of the following methods:

a) separate leader lines to each feature;

b) the indication *n*× where *n* is the number of features;

c) the “all-around” symbol;

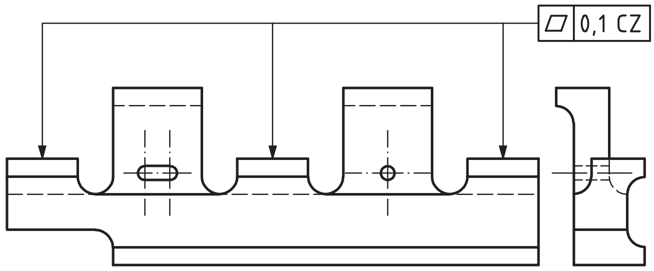
d) the “between” symbol.

The internal constraints shall be defined using either the CZ or the CZR modifier immediately after the tolerance value, separated by a space:

1) the CZ modifier indicates that both location and orientation constraints apply (see Figure 149 and Figure 150); and

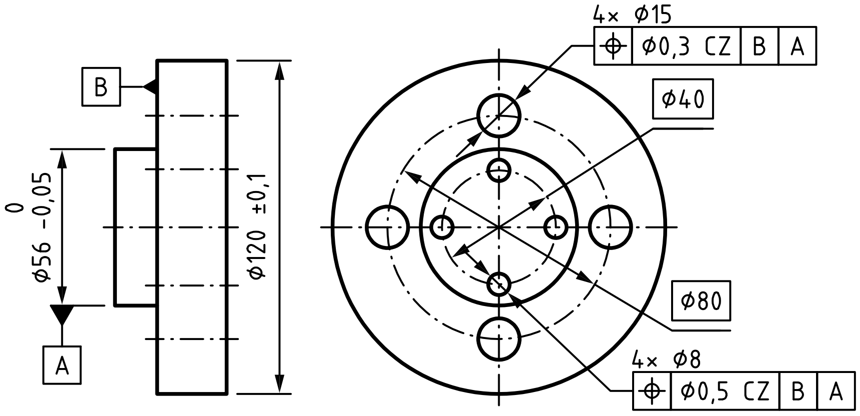
2) the CZR modifier indicates that only orientation constraints apply.

Figure 149 – *Pattern of three flat faces subject to a combined zone flatness tolerance*



NOTE Location constraints (implied distance of 0) and orientation constraints (implied parallelism) apply to the three tolerance zones.

Figure 150 – *Two independent patterns, one of four Ø15 holes, and one of four Ø8 holes*



NOTE Within each group location constraints (pitch diameter and implied 90° spacing) and orientation constraints (implied parallelism) apply. No constraints exist between the two patterns; the group of Ø8 holes does not have to be rotationally aligned with the group of Ø15 holes (this is a change from BS EN ISO 5458:1999).

8.3.3 Nested pattern requirements

NOTE 1 A nested pattern specification is defined when a geometrical specification is applied to a set of identical patterns and the sets of tolerance zones from each pattern have their locations and/or orientations locked together.

If there are *k* groups, each composed of *n* individual features, these shall be identified using one of the following methods:

a) *k*× indicated in front of *n*× in the adjacent indication area with a slash as separator and a space on both sides of the slash. The *k*× and *n*× shall be followed by a space and the identifier letter or symbol to avoid ambiguities (e.g. 4× / 2× or 4× A / 2× B). The identification letter can be used to establish a link with individual integral features, or with a group of integral features. When used to identify a group of features, the group may be indicated on a drawing by surrounding the features with a long-dashed double–dotted narrow line (line type 05.1 according to BS EN ISO 128–24:1999) (see Figure 151); or

b) *n* leader lines connecting the tolerance indicator to the *n* geometrical features and *k*× indicated in the adjacent indication area; or

c) an "all-around" symbol (covering *n* features) defining one group and *k*× indicated in the adjacent indication area; or

d) if the integral feature related to the toleranced feature is a feature of size, the number of groups shall be indicated followed by a space and the group identifier letter if applicable, followed by a space, a slash and a space, followed by the number of features and a space and the nominal size and its specification (general or individual), followed by a space and the feature identifier letter if applicable (e.g. 3× B / 2× 10±0,05 A or 3× / 2× 10±0,05 or 3× / 2× 10).

The nested pattern requirement shall be defined in the geometrical specification using a sequence composed of the CZ, SZ and CZR modifiers (see Table 22).

NOTE 2 Some combinations will not result in meaningful requirements.

NOTE 3 If all the elements of the sequence are SZ, then:

i) this specification does not define a pattern specification;

ii) the specification consists of a set of k×n independent tolerance zones, each applying to one geometrical feature, and defining k×n geometrical characteristics.

Figure 151 – *Eight independent specifications*

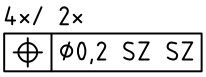
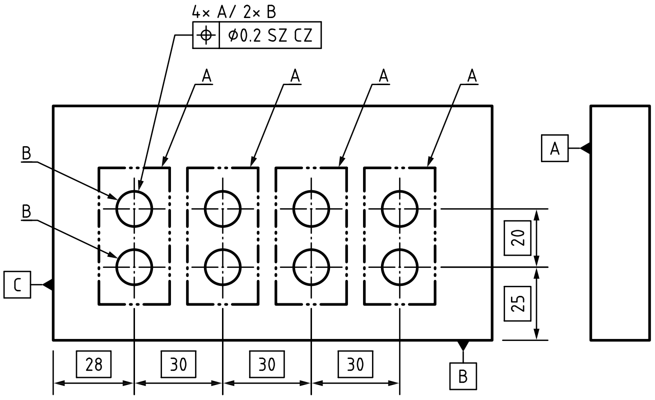
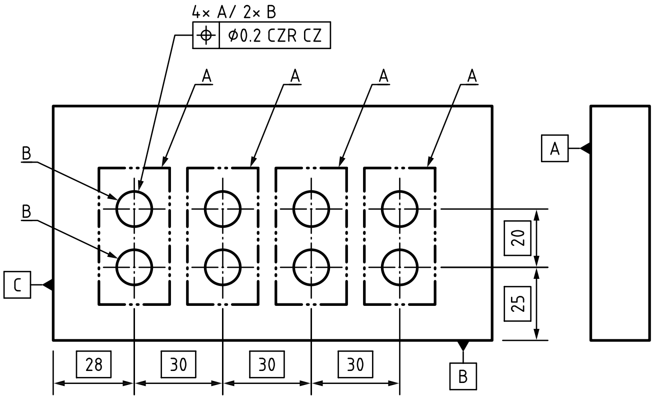
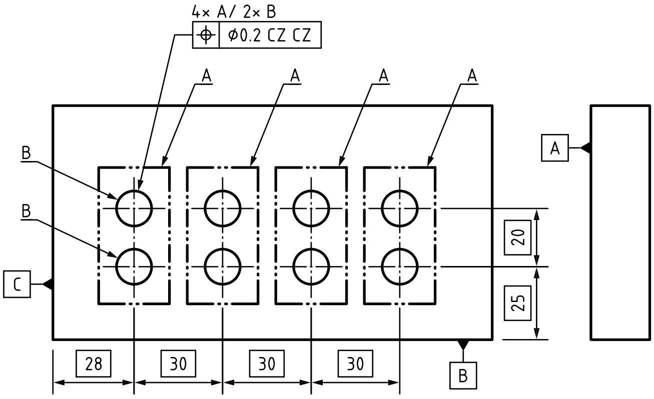
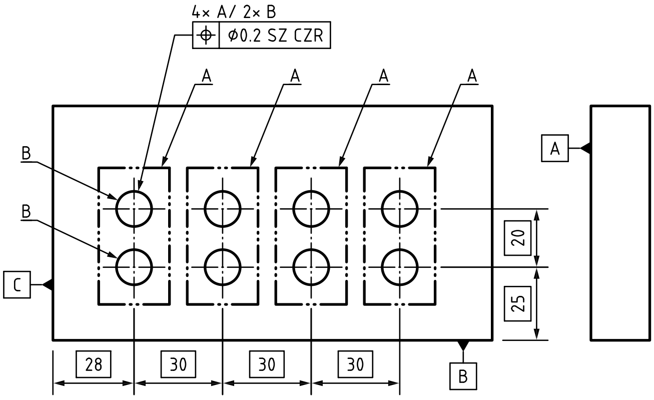
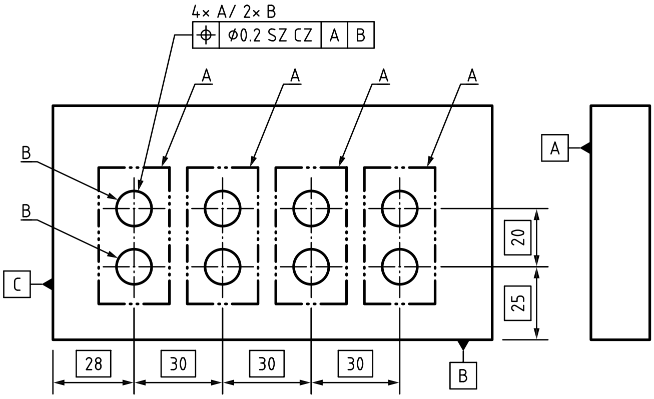


Table 22 — *Use of CZ, SZ and CZR modifiers*

Indication	**Meaning**
	**SZ CZ with no datum references** Four independent patterns: for each pattern: • toleranced feature = the group of two median lines; • tolerance zone pattern = two Ø0,2 tolerance zones with an internal location constraint (20 mm) and internal orientation constraint (parallelism).
	**CZR CZ with no datum references** Four dependent patterns: for each pattern: • toleranced feature = the group of two median lines; • tolerance zone pattern = two Ø0,2 tolerance zones with internal and location constraint (20 mm) and internal orientation constraint (parallelism); • there are also orientation constraints between the four patterns, but no location constraints.

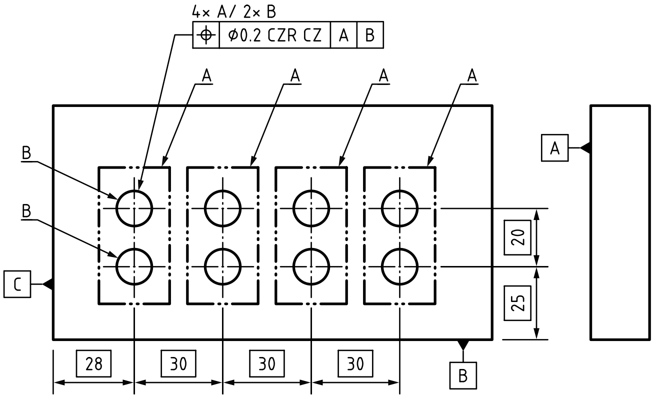
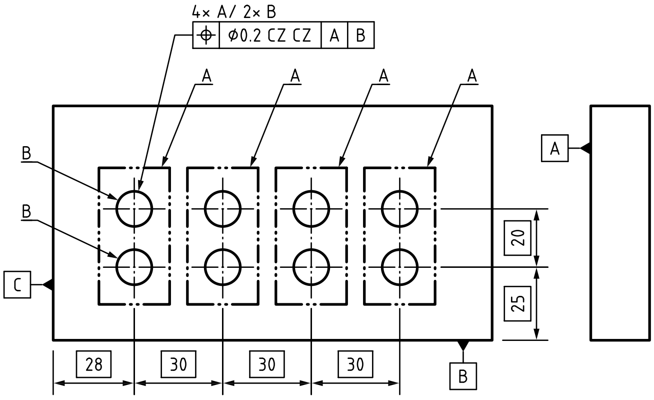
(Continues)

Table 22 — *Use of CZ, SZ and CZR modifiers (Cont.)*

Indication	Meaning
	**CZ CZ with no datum references** Four fully dependent patterns result in a single pattern specification: • toleranced feature = the group of eight median lines; • tolerance zone pattern = eight Ø0,2; tolerance zones with internal location constraints (30 mm horizontally and 20 mm vertically) and orientation constraints (parallelism).
	**SZ CZR with no datum references** Four independent patterns. For each pattern: • toleranced feature = the group of two median lines; • tolerance zone pattern = two Ø0,2 tolerance zones with an internal orientation constraint (parallelism) only. The 20 mm distance between the median lines does not apply.
	**SZ CZ with datum references** Four independent patterns constrained in orientation to datum A and location to datum B (see Note 1). For each pattern: • toleranced feature = the group of two median lines; • tolerance zone pattern = two Ø0,2 tolerance zones with internal location constraint (20 mm) and internal orientation constraint (parallelism); • there are also external orientation constraints to datum A (perpendicularity) and location constraints to datum B [theoretically exact distances from datum B of 25 mm and 45 mm (25±20)]; datum B will not influence the orientation (parallelism) of the tolerance zones because perpendicularity to datum A takes precedence.

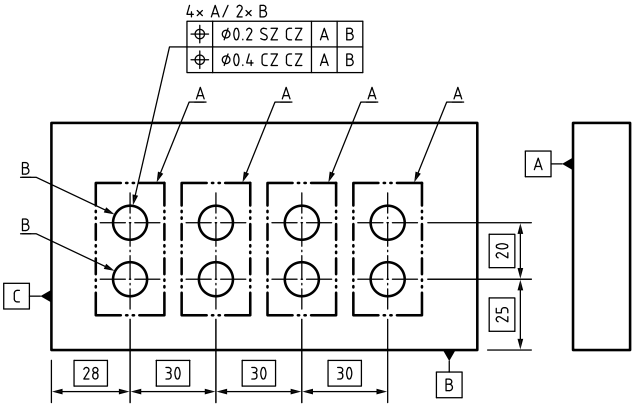
(Continues)

Table 22 — *Use of CZ, SZ and CZR modifiers (Cont.)*

Indication	Meaning
	**CZR CZ with datum references** Four dependent patterns constrained in orientation to datum A and location to datum B (see Note 1). For each pattern: • toleranced feature = the group of two median lines; • tolerance zone pattern = two Ø0,2 tolerance zones with internal location constraints (20 mm) and internal orientation constraints (parallelism); • there are orientation constraints between the four patterns, but no location constraints; • there are also external orientation constraints to datum A (perpendicularity) and location constraints to datum B [theoretically exact distances from datum B of 25 mm and 45 mm (25±20)]; datum B will not influence the orientation (parallelism) of the tolerance zones because perpendicularity to datum A takes precedence.
	**CZ CZ with datum references** Four dependent patterns constrained in orientation to datum A and location to datum B (see Note 1) results in one pattern specification: • toleranced feature = the group of eight median lines; • tolerance zone pattern = eight Ø0,2 tolerance zones with internal location constraint (30 × 20) mm and internal orientation constraint (parallelism); • there are also external orientation constraints to datum A (perpendicularity) and location constraints to datum B [theoretically exact distances from datum B of 25 mm and 45 mm (25±20)]. Datum B will not influence the orientation (parallelism) of the tolerance zones because perpendicularity to datum A takes precedence.

(Continues)

Table 22 — ***Use of CZ, SZ and CZR modifiers (Cont.)***

Indication	Meaning
	These requirements can be stacked. In this example, each pair of holes is located to each other, and to datums A and B, by the Ø0,2 tolerance. In addition, the four groups A are located to each other by the Ø0,4 tolerance.

NOTE 1 The datum system A|B does not constrain all six degrees of freedom. A datum system such as A|B|C which did constrain all six degrees of freedom would make no sense for nested patterns, as the individual features would be fully constrained to the datum system.

NOTE 2 Not all cases illustrated would result in a complete definition for the workpiece. Additional indications could be required.

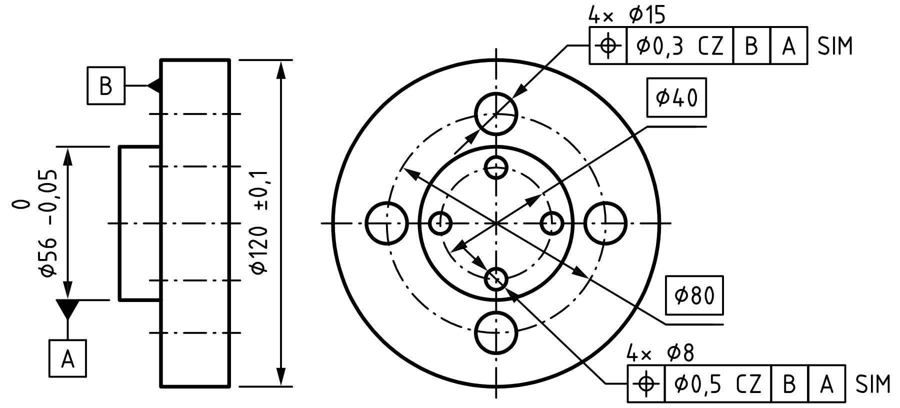
8.3.4 Combined geometrical specifications

NOTE 1 Individual geometrical specifications and pattern specifications can be combined together using the simultaneous requirement invoked by the SIM modifier. This makes it possible to create groups of non-identical features.

The SIM modifier shall be placed adjacent to two or more tolerance indicators when two or more tolerances are to be combined into a single requirement.

If there is only one simultaneous requirement on the TPD, the modifier shall be used without an index number (see Figure 152). If there are multiple simultaneous requirements on the TPD, the modifier is used with an index number.

Figure 152 – ***Two patterns locked together by the SIM indicator***



NOTE Two independent patterns, one of four Ø15 holes and one of four Ø8 holes, which are then locked together by the SIM indication. Within each group location constraints (pitch diameter and implied 90° spacing) and orientation constraints (implied parallelism) apply. Location and orientation constraints also exist between the two patterns, so the group of Ø8 holes has to be rotationally aligned with the group of Ø15 holes.

NOTE 2 In Figure 153a), the SIM modifier adjacent to the two tolerance indicators means that the two tolerance zone patterns are combined into a single requirement. All five tolerance zones are locked together by location and orientation constraints.

In Figure 153b), the SIM1 modifier creates one simultaneous requirement, and the SIM2 modifier creates a separate simultaneous requirement. The SIM1 and SIM2 requirements are unrelated to each other.

Figure 153 – *Examples of indication of simultaneous requirements from two separate specification*

a) Example 1

b) Example 2

8.4 Plane and feature indicators

8.4.1 Intersection plane indicator

8.4.1.1 Purpose of intersection plane indicators

The intersection plane indicator shall be used to identify the orientation of a geometrical specification applied to line elements on a surface, such as a straightness tolerance or a line profile tolerance.

NOTE The intersection plane indicator replaces the former practice of using the drawing view to determine the plane in which a geometrical specification applies, and also enables a geometrical specification to be applied to line elements in 3D specifications where there is no drawing view.

An intersection plane indicator shall only be used in conjunction with the following types of geometrical specification:

a) straightness;

b) roundness;

c) line profile; and

d) circular runout,

and is not required for the following:

1) a straightness tolerance applied to a cylinder or cone;

2) a roundness tolerance applied to a cylinder, cone or sphere; and

3) a geometrical specification that is applied to a complete integral feature.

8.4.1.2 Intersection plane indicator symbol

The intersection plane indicator symbol shall consist of a rectangular frame of two or more compartments, with a triangle attached to the left-hand end as shown in Figure 154 and:

a) the first compartment containing one of the geometrical tolerance symbols set out in Table 23; and

b) the remaining compartments contain one or more datum references (i.e. a datum system).

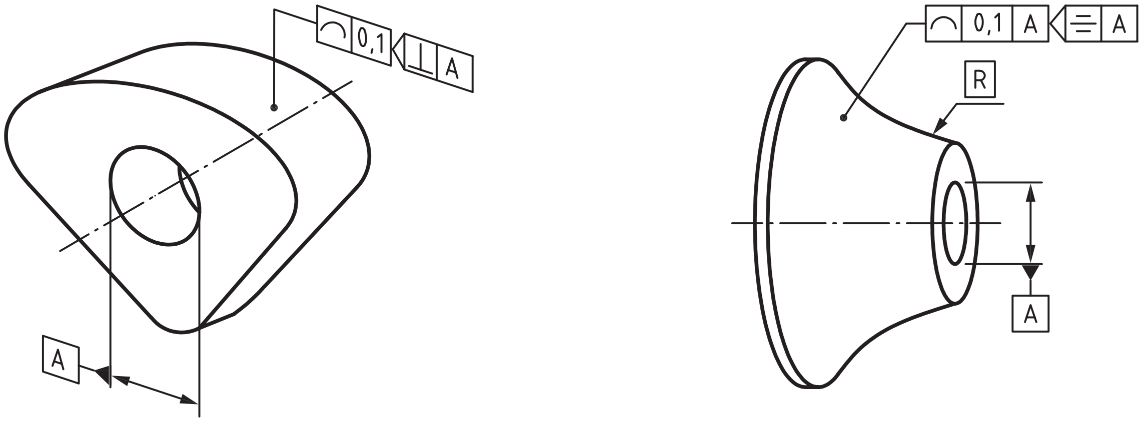
Figure 154 – ***Intersection plane indicator symbol***

Table 23 — ***Geometrical tolerance symbols***

Symbol	Meaning
//	The tolerance applies in each plane parallel to the datum system.
⊥	The tolerance applies in each plane perpendicular to the datum system.
∠	The tolerance applies in each plane at an angle (defined with a TED) to the datum system.
⌯	The tolerance applies in each plane including the datum (can only be a datum axis).

NOTE Example uses of intersection plane indicators are shown in Figure 155.

Figure 155 – ***Example uses of intersection plane indicators***



a) The line profile specification applies in each plane perpendicular to datum A

b) The line profile specification applies in each plane through (including) datum axis A

8.4.2 Orientation plane indicator

8.4.2.1 Purpose of orientation plane indicators

The orientation plane indicator shall be used to identify the orientation of a geometrical specification applied to a median line or a median point, where the specification meaning would not otherwise be fully defined, e.g. if a geometrical specification is applied to a median line, and the tolerance zone consists of two parallel planes, or if a geometrical specification is applied to a median point, and the tolerance zone is cylindrical.

NOTE The orientation plane indicator replaces the former practice of using the direction of the leader line to indicate the orientation of the tolerance zone.

The orientation plane indicator shall also be used for orientating a restricted area application of a geometrical specification.

8.4.2.2 Orientation plane indicator symbol

The orientation plane indicator symbol shall consist of a rectangular frame of two or more compartments, with a triangle attached to each end as shown in Figure 156 and:

a) the first compartment containing one of the geometrical tolerance symbols set out in Table 24; and

b) the remaining compartments contain one or more datum references (i.e. a datum system).

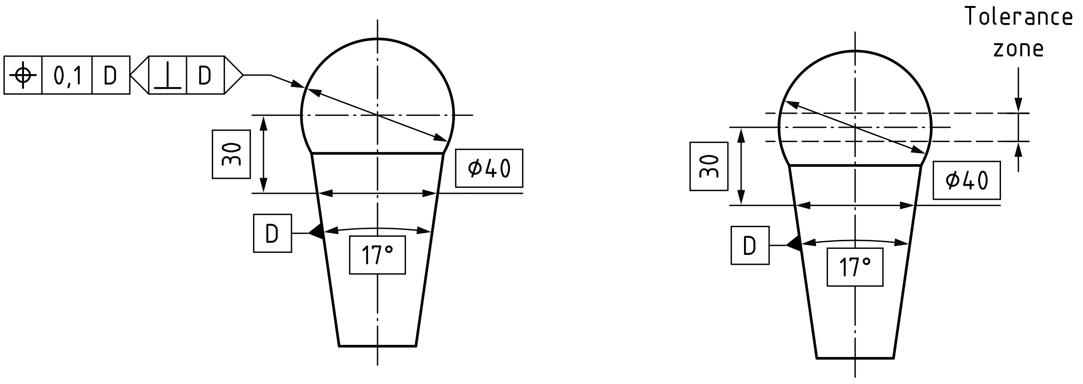
Figure 156 – *Orientation plane indicator symbol*

Table 24 — *Geometrical tolerance symbols*

Symbol	Meaning
//	For a tolerance zone composed of two parallel planes, the two parallel planes are parallel to the datum system. For a cylindrical tolerance zone, the axis of the tolerance zone is parallel to the datum system.
⊥	For a tolerance zone composed of two parallel planes, the two parallel planes are perpendicular to the datum system. For a cylindrical tolerance zone, the axis of the tolerance zone is perpendicular to the datum system.
∠	For a tolerance zone composed of two parallel planes, the two parallel planes are at an angle (defined with TEDs) to the datum system. For a cylindrical tolerance zone, the axis of the tolerance zone is at an angle (defined with TEDs) to the datum system.

NOTE The orientation plane indicator is not required when the orientation of the tolerance zone is unambiguously defined by the datum system in the tolerance indicator. Example uses of intersection plane indicators are shown in Figure 157.

Figure 157 – *Example uses of intersection plane indicators*



The centre of the sphere lies between two parallel planes, 0,1 mm apart, centred 30 mm from the Φ40 mm section of the conical datum feature, and perpendicular to datum D.

a) Indication

b) Interpretation

8.4.3 Collection plane

8.4.3.1 Purpose of collection planes

The collection plane indicator shall be used with the “all-around” symbol in order to identify the set of features to which the “all-around” requirement applies.

When a geometrical specification is applied with the “all-around” symbol, the collection plane symbol shall also be used.

NOTE 1 The geometrical specification applies to the set of features which create a continuous closed line of intersection with the plane that also passes through the point at which the leader line for the specification is applied.

NOTE 2 The collection plane indicator replaces the former practice of using the drawing view to define the plane in which the “all-around” symbol applied. This allows the “all-around” symbol to be used in 3D indications, and to be applied to a wider range of geometries.

8.4.3.2 Collection plane symbol

The collection plane symbol shall consist of a rectangular frame of two or more compartments, with a circle attached to the left-hand end as shown in Figure 158 and

a) the first compartment containing one of the geometrical tolerance symbols set out in Table 25; and

b) the remaining compartments containing one or more datum references (i.e. a datum system).

Figure 158 – ***Collection plane indicator symbol***

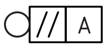
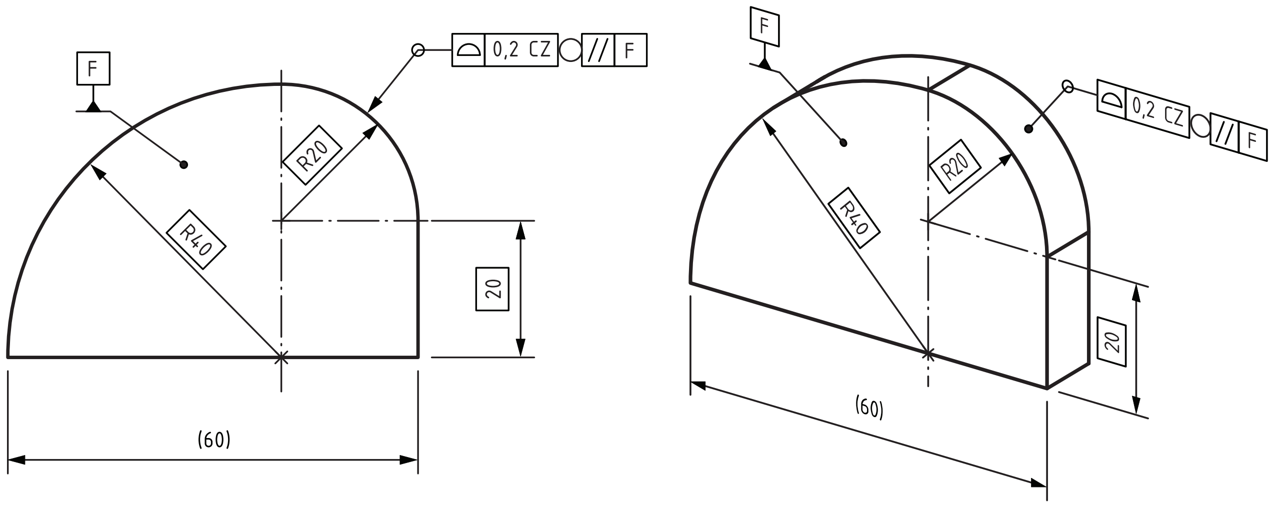


Table 25 — ***Geometrical tolerance symbols***

Symbol	Meaning
//	The “all-around” requirement applies in a plane parallel to the datum system.
⊥	The “all-around” requirement applies in a plane perpendicular to the datum system.
∠	The “all-around” requirement applies in a plane at an angle (defined with a TED) to the datum system.

NOTE Example uses of intersection plane indicators are shown in Figure 159. The surface profile tolerance applies to all the features which make up the continuous profile shown in the 2D indication in Figure 159a. Figure 159b shows the same requirement identified in a 3D CAD specification.

Figure 159 – *Example uses of collection plane indicators*



a) 2D drawing indication

b) 3D drawing indication

8.4.4 Direction feature

8.4.4.1 Purpose of direction features

The direction feature shall be used when it is necessary to identify the direction in which a geometrical specification is applied. In most cases, the default shall be that a geometrical specification applies in a direction perpendicular to the surface of the feature, and no direction feature symbol is required in these cases. A direction feature shall be used if the geometrical specification requirement is to be applied in a direction which is not perpendicular to the surface.

In the case of roundness, the default shall be that the tolerance applies perpendicular to the axis of the feature, rather than perpendicular to the surface. The direction feature shall be used to identify the direction in which a roundness tolerance applies in all cases, except when the tolerance is applied perpendicularly to a cylindrical or spherical surface.

NOTE The main use of this symbol is for roundness tolerances.

8.4.4.2 Direction feature symbol

The direction feature symbol shall consist of a rectangular frame of two or more compartments, with a short leader terminating in a filled arrowhead attached to the left-hand end as shown in Figure 160 and:

a) the first compartment containing one of the geometrical tolerance symbols set out in Table 26; and

b) the remaining compartments containing one or more datum references (i.e. a datum system).

Figure 160 – *Intersection plane indicator symbol*

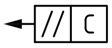
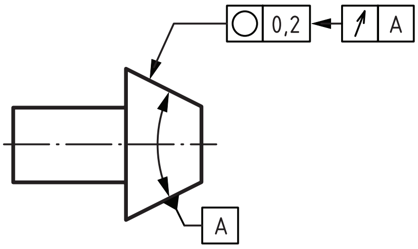


Table 26 — *Geometrical tolerance symbols*

Symbol	Meaning
//	The width of the tolerance zone is parallel to the datum system.
⊥	The width of the tolerance zone is perpendicular to the datum system.
∠	The width of the tolerance zone is at an angle (defined with a TED) to the datum system.
↗	The width of the tolerance zone is normal to the surface as for a run-out tolerance.

NOTE An example use of the direction feature is shown in Figure 161. The roundness tolerance applies perpendicular to the surface of the conical feature, instead of perpendicular to the axis of the conical feature.

Figure 161 – *An example use of the direction feature*



Section 9 Statistical tolerancing

COMMENTARY ON ***SECTION 9***

Tolerances can be applied to features of a TPS to facilitate the prediction of assembly conditions in the final product, e.g. clashes, gaps. The process of assigning tolerances to individual features involves identifying the final functional limits and then arithmetically sub-dividing and sharing this tolerance to the sub-assemblies and component parts.

Statistical methods of assigning tolerances, such as square root of the sum of the squares (RSS), can be used to sub-divide the allowable assembly tolerance amongst the sub-assemblies and components. Differentiating statistically derived tolerances via symbology and/or a note identifies that the tolerance has been used in statistical analysis and, as such, when added to other contributory tolerances, could exceed the overall product requirement specification limits.

A statistical control of the manufactured population, as well as the individual feature, might also therefore be required to guarantee the desired output for the final product. The success of statistical tolerancing is dependent on the manufacturing processes producing the features being capable and stable within the parameters defined by the statistical control.

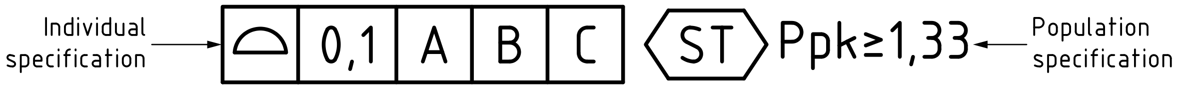
By default, a GPS specification is an individual specification. Where a tolerance is to be controlled using statistical methods, the population specification shall be identified using the statistical tolerance symbol (see Figure 162).

Figure 162 – ***Statistical tolerance symbol***

The symbol shall be placed after the tolerance value, tolerance indicator or any linear, size or angle dimension for an individual feature specification, and before the population specification (statistical requirement) (see Figure 163). Both the individual and population specifications shall be satisfied independently to demonstrate conformity of the workpiece.

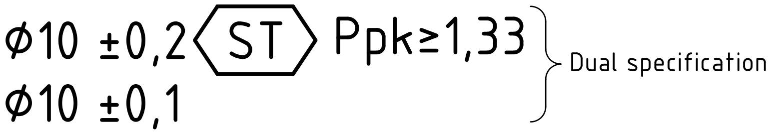
NOTE The statistical control criteria apply to a set of values which consist of the same individual characteristic taken from a sample of individual workpieces in a population.

Figure 163 – ***Population specification***



Where two individual specifications are given for same feature with different tolerance values and where one also has a complimentary statistical population specification (dual specification), either the individual specification with an associated population specification OR the more restrictive individual specification can be used to prove conformity (see Figure 164).

Figure 164 – ***Dual specification***



The following shall be agreed between the supplier and customer:

a) the statistical control criteria e.g. Ppk and Cpk, for demonstrating conformity to the population specification; and

b) the population sample size to be tested, method of assessing conformity and any physical validation required.

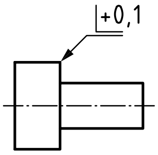
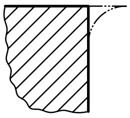
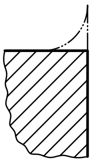
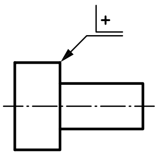
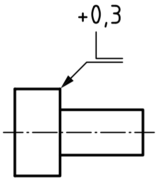
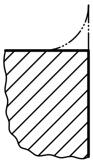
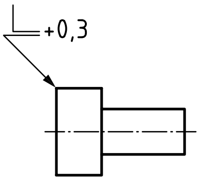
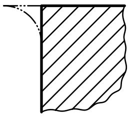
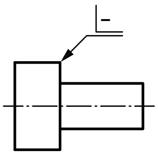
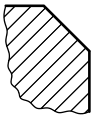
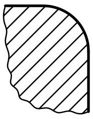
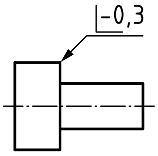
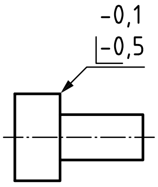
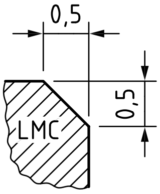
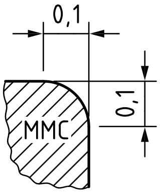
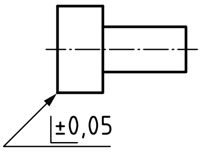
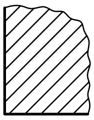
Care shall be taken in the application of statistical tolerances as knowledge in the field of statistical analysis and control is required to ensure the appropriate population controls are applied in any specific application. Statistical tolerances shall only be applied to product features where the variation and distribution can be assessed and monitored on the shop floor to ensure the physical output aligns with the statistical assumptions made in any tolerance analysis.

Section 10 Edges

Edges shall be specified in accordance with BS EN ISO 13715.

NOTE Examples of how edges can be specified are given in Table 27. For full details, though, see BS EN ISO 13715.

Table 27 — *Examples of indication of edges*

Indication	Meaning	Explanation
	 or 	External edge with burr permitted up to 0,1mm; burr direction undefined.
		External edge with permitted burr; size and direction of burr undefined.
		External edge with burr acceptable up to 0,3 mm; burr direction defined.
		
	 	External edge without burr; undercut permitted, size undefined.
		External edge without burr; undercut up to 0,3 mm.
	 or 	External edge without burr; undercut in the zone from 0,1 mm to 0,5 mm.
		External edge with burr permitted up to 0,05 mm or undercut down to 0,05 mm (sharp edge); burr direction undefined.

Abbreviations: LMC = least material condition; MMC = maximum material condition.

Section 11 Welding symbols

11.1 The use of welding symbols on a TPS

The use of welding symbols and how they are displayed in a TPS shall conform to BS EN ISO 2553.

11.2 Welding symbol

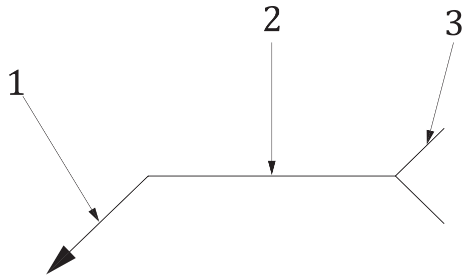
A reference line and arrow line shall be used. Additional elements shall be included where necessary to convey specific information.

NOTE It is preferable that the welding symbol is shown on the same side of the joint that the weld is to be made, i.e. the arrow side. See Figure 165 for examples of the use of welding symbols.

The thickness of the arrow lines, reference line, elementary symbols and lettering shall be in accordance with BS EN ISO 128 (all parts) and BS EN ISO 3098-2.

In order not to overburden drawings, reference shall be made to notes in the drawing or other design-related documents.

Figure 165 – *Basic welding symbol (joint details and type not specified)*



Key

1	Arrow line	3	Tail
2	Continuation leader line		

NOTE This symbol is often used to indicate the location of tack welds.

11.3 Arrow line

An arrow line shall be used to indicate the joint to be welded.

The arrow line shall:

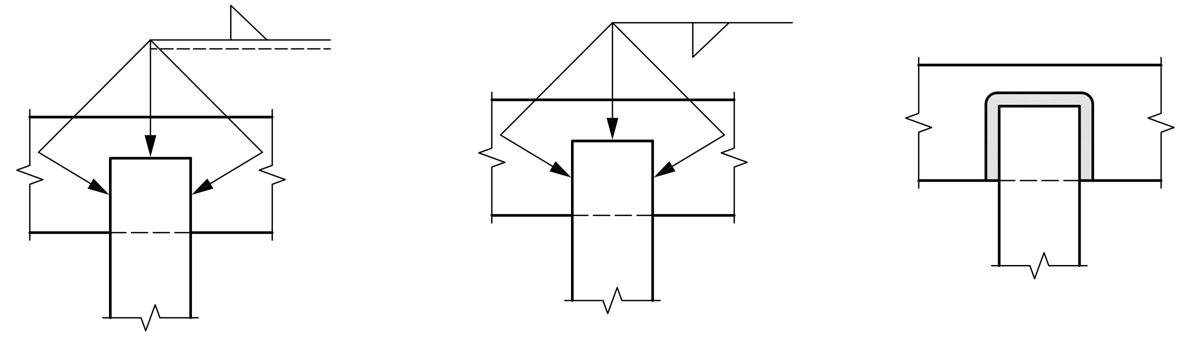
- point to and be in contact with a solid line comprising part of the joint on the drawing (visible line); and
- be drawn at an angle to and joined to a reference line and completed with a closed filled arrowhead.

NOTE The arrow line can be joined to either end of the reference line.

*COMMENTARY ON **11.3***

Two or more arrow lines can be combined with a single reference line to indicate the locations of identical welds (see Figure 166).

Figure 166 – ***Use of multiple arrow lines***



a) System A welding symbol b) System B welding symbol c) Weld produced using either system

11.4 Reference line

The reference line, when combined with elementary symbols, shall be used to indicate the side of the joint on which the weld is to be made.

NOTE 1 The reference line can be drawn parallel to the side edge of the drawing (whole welding symbol rotated by 90°) but this should only be done when space does not permit drawing parallel to the bottom edge.

Reference line — System A: The reference line shall consist of two parallel lines of equal length: a continuous line and a dashed line (see Figure 167).

The dashed line may be drawn above or below the continuous line but shall preferably be drawn below.

The dashed line shall be omitted for symmetrical welds and for spot and seam welds made at the interface between two components.

Reference line — System B: The reference line shall be drawn as a continuous line (see Figure 167).

NOTE 2 The System A approach is based on BS EN ISO 2553:1992. The System B approach is based on standards used by Pacific Rim countries and used by exception. For further information on System A/System B, including examples, see BS EN ISO 2553:2019.

System A and B shall not be mixed and drawings shall clearly indicate which system is used, including units of measurement in accordance with BS ISO 129‑1.

Weld location arrow side/other side

The arrow side shall be the side of the joint to which the arrowhead is pointing.

The other side shall be the opposite side of the joint to which the arrowhead is pointing. The arrow side and other side shall always form part of the same joint.

The other side of a joint shall not be confused with a hidden weld forming part of a different joint.

Arrow side/other side — System A

Elementary symbols shall be located on the continuous line when the weld is to be made on the arrow side of the joint (see Figure 167).

Elementary symbols shall be located on the dashed (identification) line when the weld is to be made on the other side of the joint.

Arrow side/other side — System B

Elementary symbols shall be located below the reference line when the weld is to be made on the arrow side of the joint (see Figure 167).

Elementary symbols shall be located above the reference line when the weld is to be made on the other side of the joint.

NOTE 3 In System A, the component of the reference line on which the elementary symbol is placed determines the side of the joint which is to be welded – the dashed line can be drawn above or below the solid line.

NOTE 4 In System B, the position of the elementary symbol above or below the reference line determines the side of the joint on which the weld is made

NOTE 5 The other side option is typically used where space is restricted in the TPS and it is not possible to position the welding symbol adjacent to the joint.

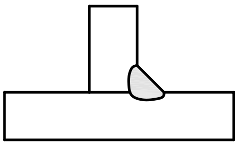
Figure 167 – *Reference lines illustrating arrow side and other side for System A and System B*

a) System A: Arrow side (symbol on solid component of reference line)

b) System B: Arrow side (symbol below reference line)

c) System A: Other side (symbol on dashed component of reference line)

d) System B: Other side (symbol above reference line)



e) Same weld produced using four options a) to d)

11.5 Tail

The tail, an optional element, shall be added to the end of the continuous reference line (see Figure 167) where additional complementary information is included as part of the welding symbol, for example:

a) quality level, for example in accordance with BS EN ISO 5817, BS EN ISO 10042, BS EN ISO 13919 (both parts);

b) the welding process reference number in accordance with BS EN ISO 4063, or abbreviation;

c) filler material for example in accordance with BS EN ISO 14171, BS EN ISO 14341, etc.;

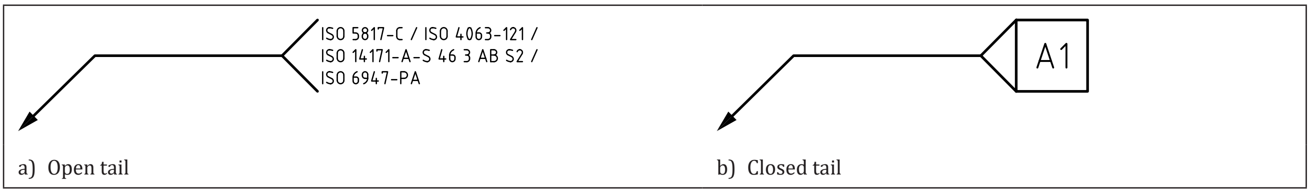
d) welding position, for example in accordance with BS EN ISO 6947; and

e) supplementary information to be considered when making the joint the information shall be listed and separated by a forward slash (/), [see Figure 168a)].

A closed tail shall only be used to indicate reference to a specific instruction, e.g. reference to a welding procedure specification (WPS), welding procedure qualification record (WPQR) or other document [see Figure 168b)].

Repetition of additional information on symbols on a drawing shall be avoided. A single general note on the drawing shall be used instead.

Figure 168 – *Examples of the use of a tail on welding symbols*



a) Open tail b) Closed tail

11.6 Elementary symbols

Elementary symbols, in accordance with Table 28, shall be added to the reference line in both systems A and B to indicate the type of weld to be made.

Elementary symbols form part of the welding symbol and shall be drawn attached to the reference line generally at the mid-point.

NOTE 1 Elementary symbols can be complemented by:

a) supplementary symbols;

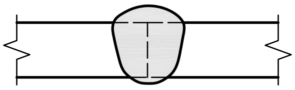
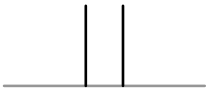
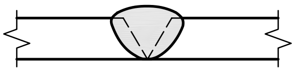
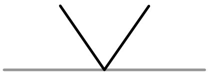
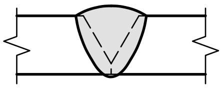
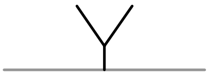
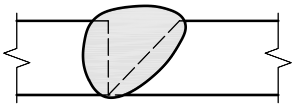
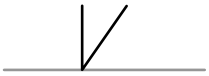
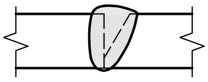
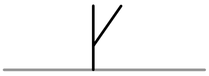
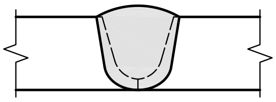
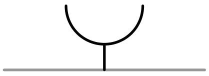
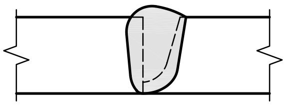
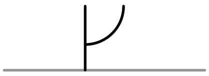
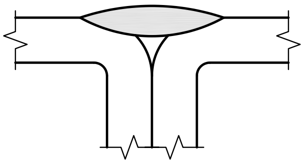
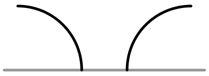
b) dimensions; and

c) complementary information.

The orientation of the elementary symbols shall not be changed to that shown.

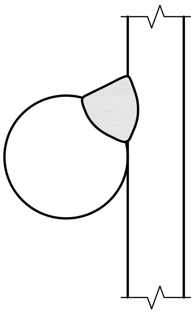
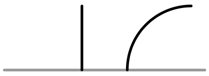
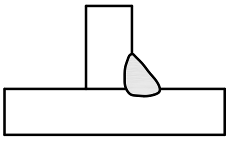
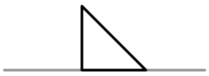
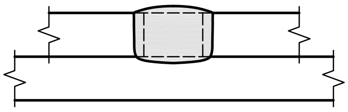
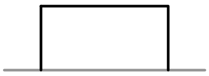
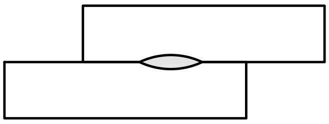
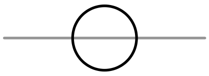
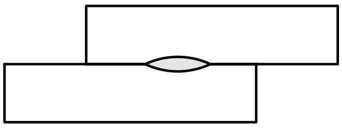
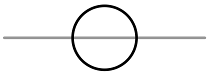
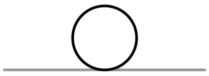
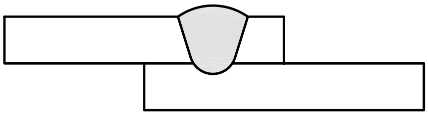
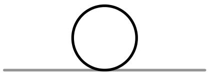
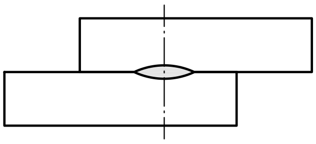
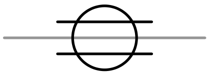
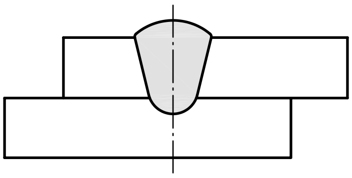
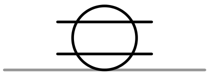
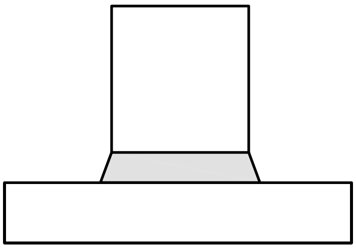
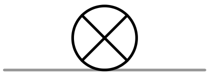
NOTE 2 Combinations of the above elementary symbols are shown in accordance with BS EN ISO 2553.

Table 28 — *Elementary symbols*

Designation	**Illustration of weld type[A)]**	**Symbol[B)]**
Square butt[C)]		
Single-V butt[C)]		
Single-V butt with broad root face[C)]		
Single-bevel butt[C)]		
Single-bevel butt with broad root face[C)]		
Single-U butt[C)]		
Single-J butt[C)]		
Flare V		

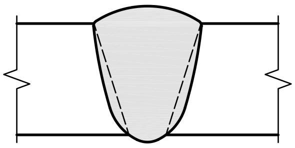
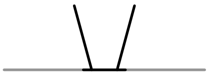
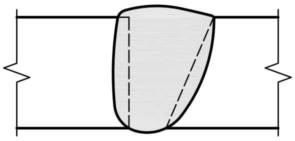
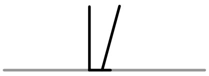
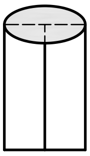
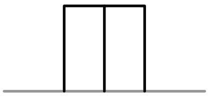
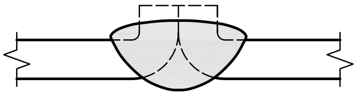
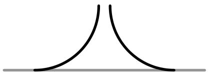
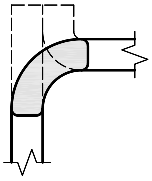
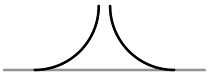
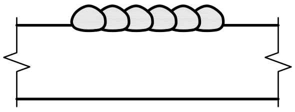
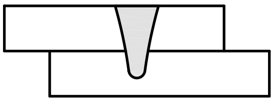
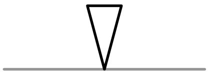
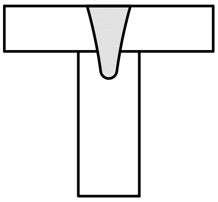
(Continues)

Table 28 — *Elementary symbols (Cont.)*

Designation	Illustration of weld type^A)	Symbol^B)
Flare bevel		
Fillet		
Plug		
Resistance spot^D)		
Projection		System A  System B 
Fusion spot		
Resistance seam^D)		
Fusion seam		
Stud		

(Continues)

Table 28 — *Elementary symbols (Cont.)*

Designation	**Illustration of weld type**[A]	**Symbol**[B]
Steep-flanked single-V butt[C]		
Steep-flanked single-bevel butt[C]		
Edge[D]		
Flanged butt		
Flanged corner		
Overlay		
Stake[D]		
		

[A] Dashed lines show joint preparation prior to welding.

[B] The grey line is not part of the symbol. It indicates the position of the reference line.

[C] Butt welds are full penetration unless otherwise indicated by dimensions on the welding symbol or by reference to other information, for example the WPS.

[D] Symbol can also be used for joints with more than two members.

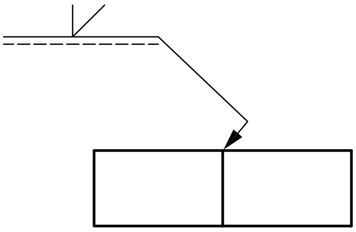
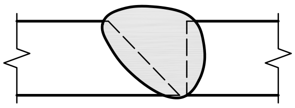
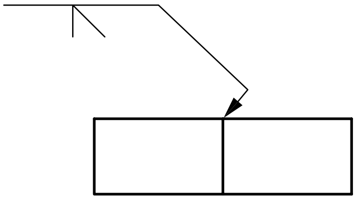
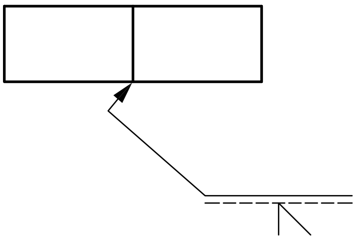
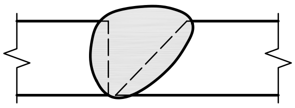
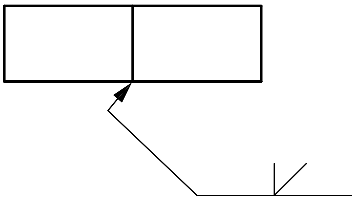
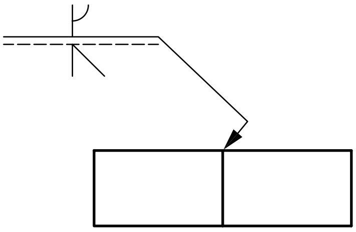
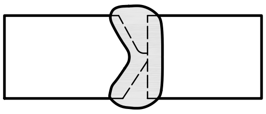
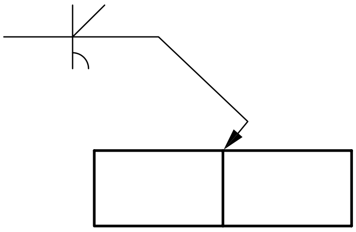
11.7 Broken arrow lines

For butt welds in plates (excluding T–butt welds) when a specific joint member is required to be prepared (e.g. single-bevel or single-J butt welds), the arrow line shall have a break and point toward that member.

NOTE 1 The arrow line need not be broken if it is obvious or if there is no preference as to which member is to be prepared.

NOTE 2 Examples of the use of broken arrow lines are given in the Table 29.

Table 29 – *Example uses of broken arrow line*

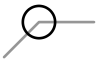
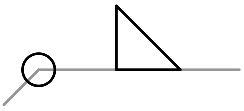
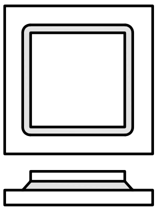
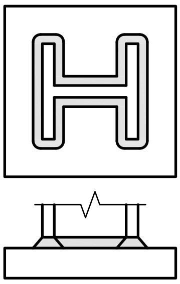
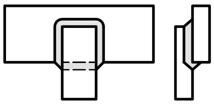
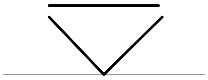
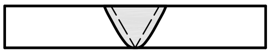
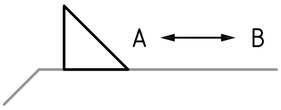
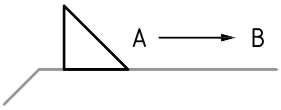
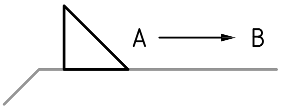
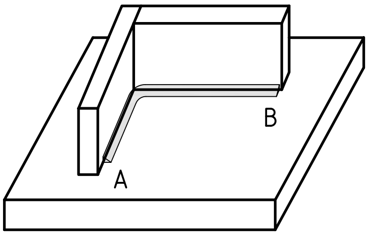
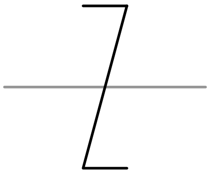
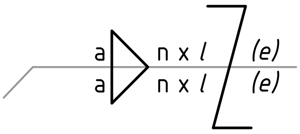
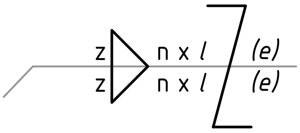
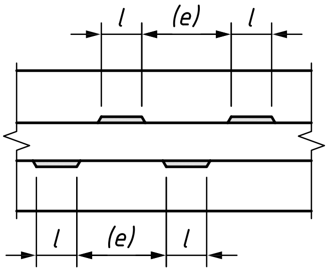
System A welding symbol	Illustration of weld type	System B welding symbol
		
		
		

11.8 Supplementary symbols

Additional information concerning the required joint shall be provided by the use of supplementary symbols, where required, for example, information about the shape of the weld or how the welded joint is to be made.

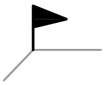
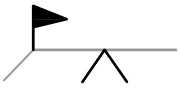
NOTE The most commonly used symbols are given in Table 30. For the full list of supplementary symbols, descriptions and diagrams available, see BS EN ISO 2553:2019, Table 3.

Table 30 – *Supplementary symbols*

Designation	**Symbol**[A]	**Application**[A]	**Illustration of weld**
Weld all-round			 Example A  Example B  Example C
Flat-finished flush[B]	—		
Weld between two points	⟷	 or  or 	
Staggered intermittent weld		 or 	

(Continues)

Table 30 – *Supplementary symbols (Cont.)*

Designation	Symbol[A)]	Application[A)]	Illustration of weld
Field weld			No example

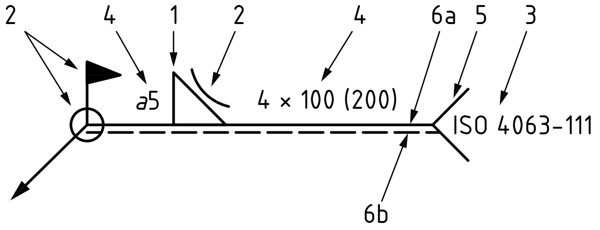
[A)] The grey line is not part of the symbol and is included to show the position of symbol on reference line and/or the arrow line only.

[B)] Welds that require approximately flush, convex or concave faces without post-weld finishing are specified by use of the flush, convex or concave contour symbol.

Example of the use of welding symbols

The example given in Figure 169 is illustrative only and is intended to demonstrate the correct application of drawing principles. It is not intended to represent good design practices or to replace code or specification requirements. For more examples of the use of welding symbols see BS EN ISO 2553.

Figure 169 – *Comprehensive welding symbols showing the location of weld elements*



Key

1	Elementary symbol (fillet weld)
2	Supplementary symbol (concave fillet weld contour, field weld, weld all-around)
3	Complementary information [shielded metal arc welding (SMAW)/process 111 in accordance with BS EN ISO 4063]
4	Dimensions (5 mm nominal throat thickness intermittent fillet weld, comprising 4 weld elements 100 mm in length with 200 mm spacing between elements)
5	Tail
6a	Reference line (continuous)
6b	Dashed line (identification line) — system A only

Section 12 Surface texture specification

*COMMENTARY ON SECTION **12***

Although the surface of a workpiece might appear smooth, there is a complex surface structure or texture comprising peaks and valleys, with its form and orientation derived from manufacturing and surface treatment processes. The texture of surfaces is critical to the durability and performance of interacting systems, particularly in the case of lubricated sliding and rolling contacts. It is therefore useful to be able to specify the surface texture and orientation required for the desired functionality.

BS 1134 offers an excellent introduction to surface texture specification.

12.1 General

Indication of profile surface texture shall conform to the following standards, as appropriate.

BS 1134	*Assessment of surface texture – Guidance and general information*
BS EN ISO 1302	*Geometrical Product Specifications (GPS) – Indication of surface texture in technical product documentation*

NOTE 1 A new series of ISO standards for profile surface texture, the ISO 21910 series, are in development and will superseded many of the standards listed below.

NOTE 2 The correct application of BS EN ISO 1302 requires the use of the following standards.

BS EN ISO 8785	*Geometrical product specification (GPS) – Surface imperfections – Terms, definitions and parameters*
BS EN ISO 3274	*Geometrical Product Specifications (GPS) – Surface texture: Profile method – Nominal characteristics of contact (stylus) instruments*
BS EN ISO 4287	*Geometrical product specification (GPS) – Surface texture: Profile method – Terms, definitions and surface texture parameters*
BS EN ISO 4288	*Geometric Product Specification (GPS) – Surface texture – Profile method: Rules and procedures for the assessment of surface texture*
BS EN ISO 10135	*Geometrical product specifications (GPS) – Drawing indications for moulded parts in technical product documentation (TPD)*
BS EN ISO 16610–21	*Geometrical product specifications (GPS) – Filtration – Part 21: Linear profile filters: Gaussian filters*
BS EN ISO 12085	*Geometrical Product Specifications (GPS) – Surface texture: Profile method – Motif parameters*
BS EN ISO 13565–1	*Geometric Product Specifications (GPS) – Surface texture: Profile method – Surfaces having stratified functional properties – Part 1: Filtering and general measurement conditions*
BS EN ISO 13565–2	*Geometrical Product Specifications (GPS) – Surface texture: Profile method – Surfaces having stratified functional properties – Part 2: Height characterization using the linear material ration curve*
BS EN ISO 13565–3	*Geometrical product specifications (GPS) – Surface texture: Profile method – Surfaces having stratified functional properties – Part 3: Height characterization using the material probability curve*

BS EN ISO 14253–1	*Geometrical Product Specifications (GPS) – Inspection by measurement of workpieces and measuring equipment – Part 1: Decision rules for verifying conformity or non–conformity with specifications*
BS EN ISO 14660–1	*Geometrical Product Specifications (GPS) – Geometrical features – Part 1: General terms and definitions*
BS EN ISO 81714–1	*Design of graphical symbols for use in the technical documentation of products – Part 1: Basic rules*

Indication of area surface texture shall conform to the following standards, as appropriate.

BS EN ISO 25178-1	*Geometrical product specifications (GPS) – Surface texture: Areal – Part 1: Indication of surface texture*
BS EN ISO 25178-2	*Geometrical product specifications (GPS) – Surface texture: Areal – Part 2: Terms, definitions and surface texture parameters*
BS EN ISO 25178-3	*Geometrical product specifications (GPS) – Surface texture: Areal – Part 3: Specification operators*
BS EN ISO 25178-70	*Geometrical product specifications (GPS) – Surface texture: Areal – Part 70: Material measures*
BS EN ISO 25178-71	*Geometrical product specifications (GPS) – Surface texture: Areal – Part 71: Software measurement standards*
BS EN ISO 25178-72	*Geometrical product specifications (GPS) – Surface texture: Areal – Part 72: XML file format x3p*
BS EN ISO 25178-73	*Geometrical product specifications (GPS) – Surface texture: Areal – Part 73: Terms and definitions for surface defects on material measures*
BS EN ISO 25178-600	*Geometrical product specifications (GPS) – Surface texture: Areal – Part 600: Metrological characteristics for areal topography measuring methods*
BS EN ISO 25178-601	*Geometrical product specifications (GPS) – Surface texture: Areal – Part 601: Nominal characteristics of contact (stylus) instruments*
BS EN ISO 25178-602	*Geometrical product specifications (GPS) – Surface texture: Areal – Part 602: Nominal characteristics of non-contact (confocal chromatic probe) instruments*
BS EN ISO 25178-603	*Geometrical product specifications (GPS) – Surface texture: Areal – Part 603: Nominal characteristics of non-contact (phase-shifting interferometric microscopy) instruments*
BS EN ISO 25178-604	*Geometrical product specifications (GPS) – Surface texture: Areal – Part 604: Nominal characteristics of non-contact (coherence scanning interferometry) instruments*
BS EN ISO 25178-605	*Geometrical product specifications (GPS) – Surface texture: Areal – Part 605: Nominal characteristics of non-contact (point autofocus probe) instruments*
BS EN ISO 25178-606	*Geometrical product specifications (GPS) – Surface texture: Areal – Part 606: Nominal characteristics of non-contact (focus variation) instruments*

BS EN ISO 25178-607	*Geometrical product specifications (GPS) – Surface texture: Areal – Part 607: Nominal characteristics of non-contact (confocal microscopy) instruments*
BS EN ISO 25178-701	*Geometrical product specifications (GPS) – Surface texture: Areal – Part 701: Calibration and measurement standards for contact (stylus) instruments*

NOTE 3 Further standards in the BS EN ISO 25178 series for areal surface texture are in development.

NOTE 4 The following documents detail different aspects of filtering which could be utilized to process surface texture readings.

BS EN ISO 16610–1	*Geometrical product specifications (GPS) – Filtration – Part 1: Overview and basic concepts*
BS EN ISO 16610–20	*Geometrical product specifications (GPS) – Filtration – Part 20: Linear profile filters: Basic concepts*
BS EN ISO 16610–22	*Geometrical product specifications (GPS) – Filtration – Part 22: Linear profile filters: Spline filters*
BS EN ISO 16610–29	*Geometrical product specifications (GPS) – Filtration – Part 29: Linear profile filters: Spline wavelets*
BS EN ISO 16610–40	*Geometrical product specifications (GPS) – Filtration – Part 40: Morphological profile filters: Basic concepts*
BS EN ISO 16610–41	*Geometrical product specifications (GPS) – Filtration – Part 41: Morphological profile filters: Disk and horizontal line–segment filters*
BS EN ISO 16610–49	*Geometrical product specifications (GPS) – Filtration – Part 49: Morphological profile filters: Scale space techniques*

12.2 Characterization

*COMMENTARY ON **12.2***

BS 1134 provides guidance and general information on the assessment of surface texture.

12.2.1 General

The surface profile shall be measured to establish a range of geometrical parameters and is usually taken perpendicular to the direction of the lay unless otherwise indicated.

NOTE Typically, a stylus device is drawn across the surface in the X direction, where Y is nominally parallel to the lay of the surface texture and Z is normal to the surface as shown in Figure 170a). The measuring device generates the total profile prior to the application of filters for the separation of roughness from waviness, form and signal noise.

12.2.2 Profiles

*COMMENTARY ON **12.2.2***

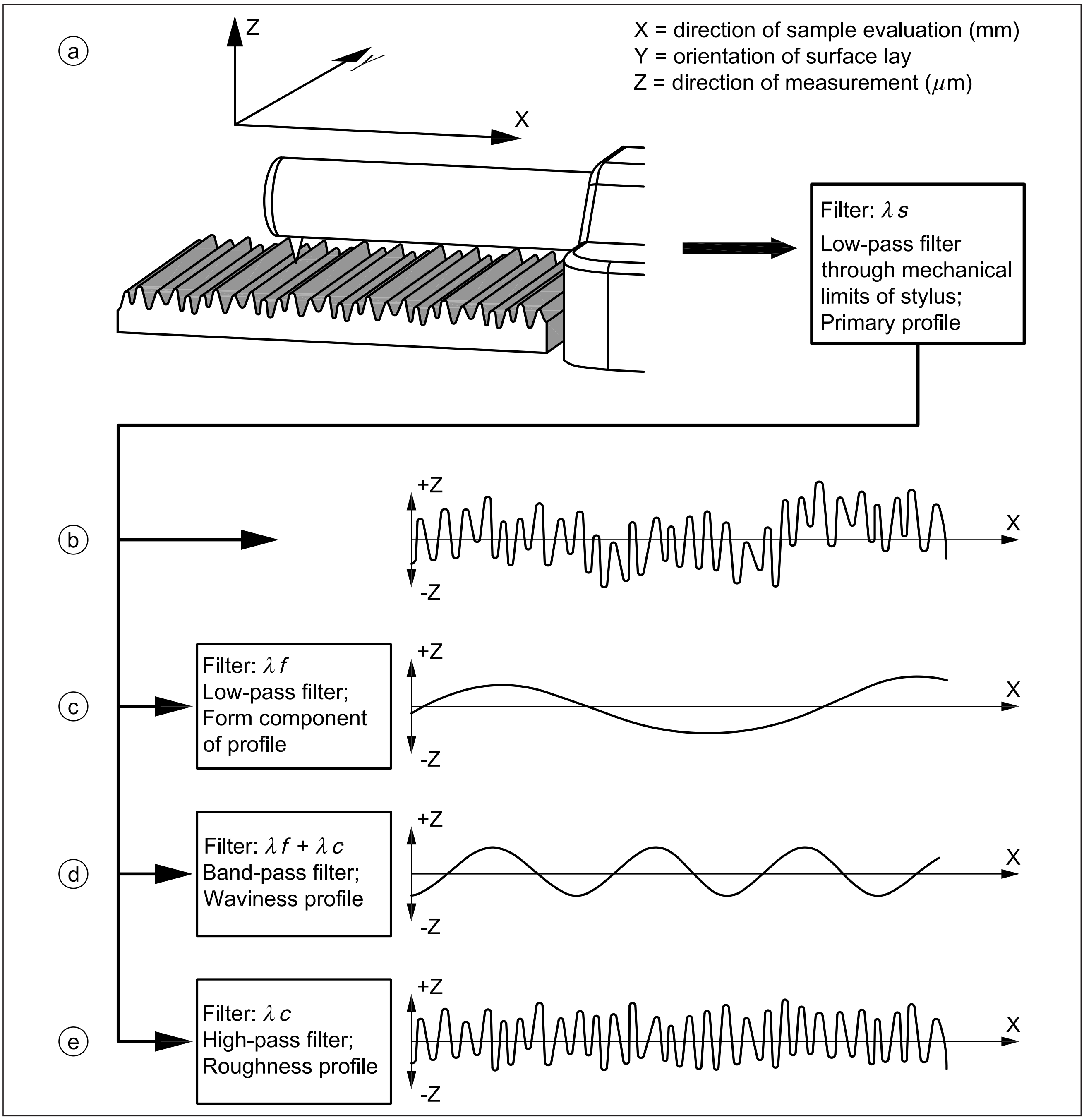
The primary profile is the total profile after application of the low pass filter λs. In practice, this filter is often the mechanical limit of the measurement stylus. The profile includes waviness, roughness and longer wavelength components of form as shown in Figure 170b). The application of a low pass filter λf supresses the waviness and roughness profiles while retaining the form component as shown in Figure 170c).

The waviness profile shall be derived from the primary profile, the through application of a band pass filter where the λc supresses the roughness and the λf supresses the form component as shown in Figure 170d).

The roughness profile shall be derived from the primary profile through the application of a high pass filter where the λ*c* filter supresses both form and waviness as shown in Figure 170e).

The filter settings and derived evaluation length have a significant effect upon the measured result and shall be taken into account when specifying surface roughness parameters.

Figure 170 – *Use of several specification modifiers*

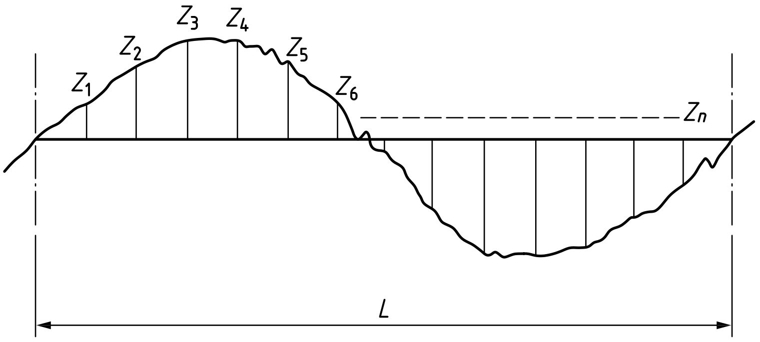


12.2.3 Roughness specifications

COMMENTARY ON 12.2.3

A wide range of surface roughness parameters can be specified and are described in BS 1134. The most commonly used are the arithmetical mean deviation (Ra) and the root mean square deviation (Rq) as shown in Figure 171.

Figure 171 – *Mathematical derivation of Ra and Rq*



$$Ra\frac{|Z_1|+|Z_2|...+|Z_n|}{n}$$

$$Rq=\sqrt{\frac{Z_1^2+Z_2^2+Z_3^2...Z_n^2}{n}}$$

The maximum peak height (Rp) and maximum valley depth (Rv) can be combined to produce the maximum profile height (Rz) (see Figure 172) while surface skewness can be evaluated using Rsk (see Figure 173).

Figure 172 – *Peak height (Rp), valley depth (Rv) and maximum height of profile (Rz)*

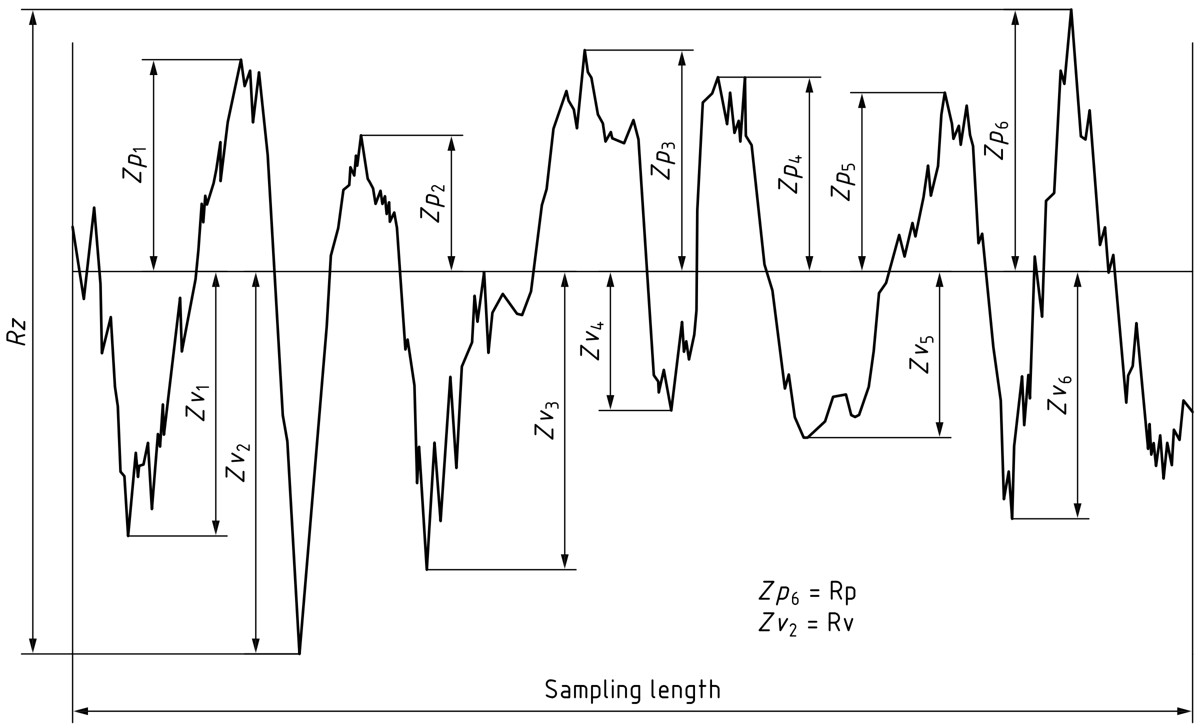
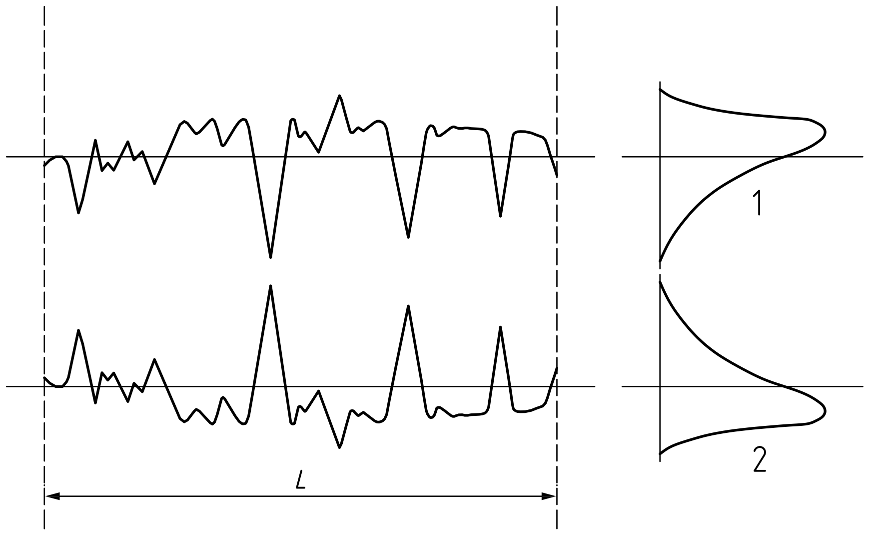


Figure 173 – *Mathematical derivation of surface skewness (Rsk)*



$$Rsk = \frac{1}{Rq^3}\left(\frac{1}{n}\sum_{i=1}^{n} Z_i^3\right)$$

Key

1 *Rsk* is negative

2 *Rsk* is positive

Although it is common to indicate only the desired surface texture specification, the filter and evaluation length settings shall also be specified.

NOTE 1 Failure to provide these settings will result in a lengthy procedure prior to establishing the correct settings for the sampled surface, with subsequent cost implications.

For non–periodic surfaces, measuring *Ra* and *Rq* the standard cut–off settings shall be as shown in Table 31, while those for *R*z shall be as shown in Table 32.

NOTE 2 Cut–off settings for periodic surfaces when measuring any surface roughness parameter are dependent upon the mean width of profile elements (RSm) value as shown in Table 33 and Figure 174 prior to selection from Table 31.

Table 31 – *Estimates for choosing roughness sampling lengths for the measurement of non–periodic profiles (Ra)*

*R*a Ra/µm	**Roughness sampling length** l_r/mm	**Roughness evaluation length** l_n/mm
0,006 < *Ra* ≤ 0,02	0,08	0,40
0,02 < *Ra* ≤ 0,1	0,25	1,25
0,1 < *Ra* ≤ 2	0,80	4,00
2 < *Ra* ≤ 10	2,50	12,50
10 < *Ra* ≤ 80	8,00	40,00

Table 32 – *Estimates for choosing roughness sampling lengths for the measurement of non–periodic profiles (Rz)*

Rz *Rz*1max/μm	**Roughness sampling length** l_r/mm	**Roughness evaluation length** l_n/mm
0,025 < *Rz Rz*1max ≤ 0,1	0,08	0,40
0,1 < *Rz Rz*1max ≤ 0,5	0,25	1,25
0,5 < *Rz Rz*1max ≤ 10	0,80	4,00
10 < Rz *Rz*1max ≤ 50	2,50	12,50
50 < Rz *Rz*1max ≤ 200	8,00	40,00

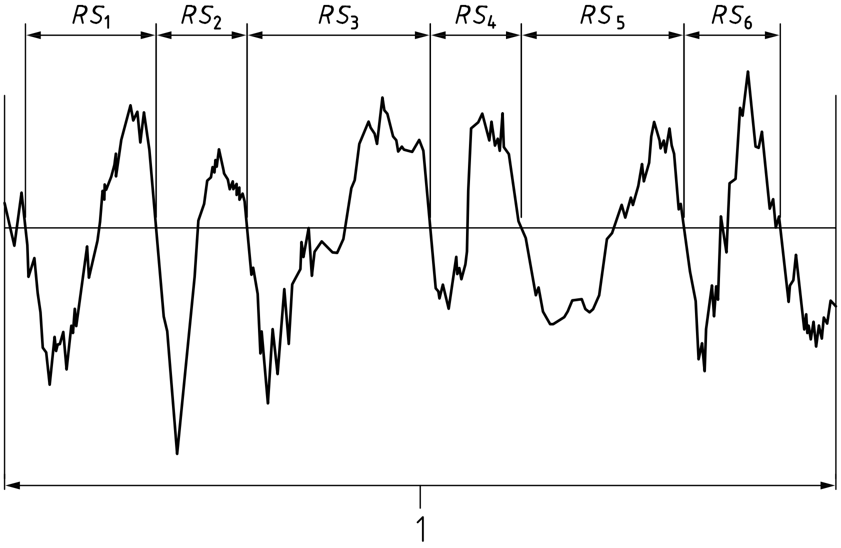
NOTE 1 Rz is used when measuring Rz, Rv, Rp, Rc and Rt.

NOTE 2 Rz1max is used only when measuring Rz1max, Rv1max, Pr1max and Rc1max.

Table 33 – *Estimates for choosing roughness sampling lengths for the measurement of periodic profiles*

***RS*$_m$** RS_m/mm	**Roughness sampling length** l_r/mm	**Roughness evaluation length** l_n/mm
0,013 < RS_m ≤ 0,04	0,08	0,40
0,04 < RS_m ≤ 0,13	0,25	1,25
0,013 < RS_m ≤ 0,4	0,80	4,00
0,4 < RS_m ≤ 1,3	2,50	12,50
1,3 < RS_m ≤ 4	8,00	40,00

Figure 174 – *Derivation of mean width of profile element (RS_m)*



$$RS_\mathrm{m} = \frac{1}{n}\sum_{i-n}^{n} Rs_\mathrm{i}$$

Key

1	Evaluation length

12.2.4 The 16% rule

When sampling surface texture not more than 16% of samples shall fall outside of the specification. If the max. or min. indication is made, no samples shall fall outside of the specification.

NOTE BS EN ISO 4288 provides further guidance on a simplified procedure for roughness inspection.

Where the 16% rule is employed the test shall be stopped if:

- the first measurement does not exceed 70% of the specified value;
- the first three measurements are within specification;
- not more than one of the first six measurements exceeds the specification;
- not more than two of the first twelve measurements exceed the specification.

12.3 Drawing indication of surface texture specification

NOTE BS EN ISO 1302 provides a full description for the indication of surface specification.

The surface specification shall be indicated through the use of the graphical symbol shown in Figure 175, while the positions of various requirements shall be indicated as shown in Figure 176.

Figure 175 – *Surface texture graphical symbol*

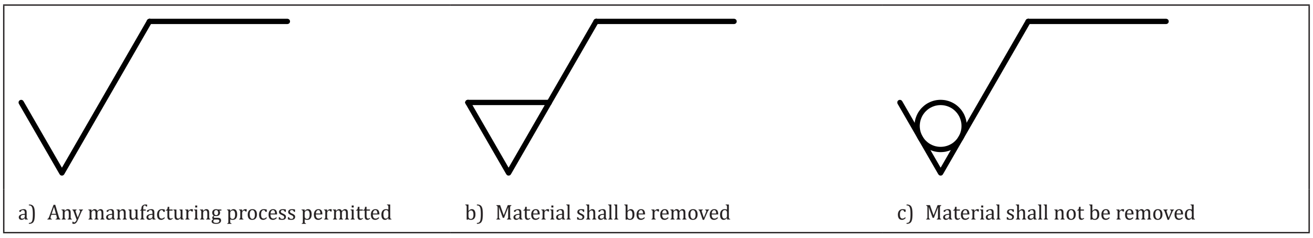
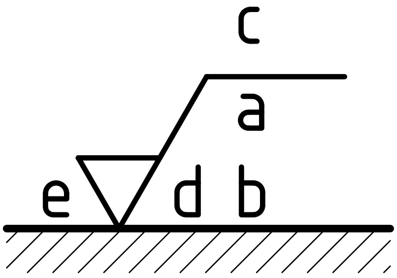

a) Any manufacturing process permitted
b) Material shall be removed
c) Material shall not be removed

Figure 176 – *Positions (a to e) for location of texture requirements*



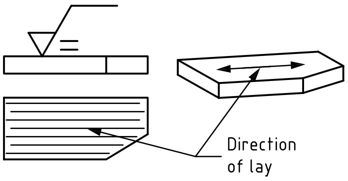
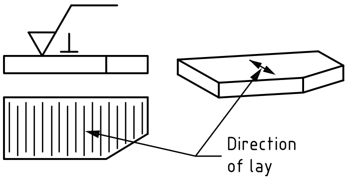
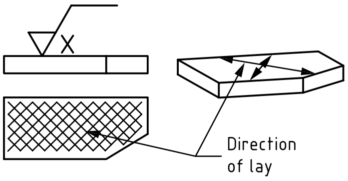
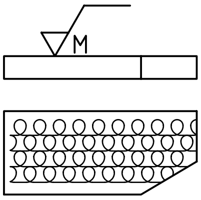
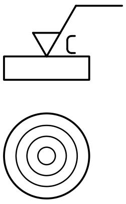
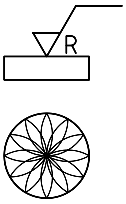
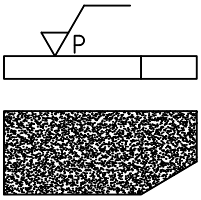
The surface texture requirement shall be in position “a” and indicate the sampling length or transmission band, separated by oblique (/), then texture parameter designation (with limit indicator if required), separation by double space and numerical limit. If a second specification is required, this shall be in position “b”; further specifications require the enlargement of the symbol.

The manufacturing method, treatment or other process, if required, shall be in position “c”.

Surface lay and orientation (as shown in Table 34) relative to the plane of projection shall be shown in position “d”.

Indication of any machining allowance as a numerical value in mm shall be shown in position “e”.

Table 34 – *Indication of surface lay*[A)]

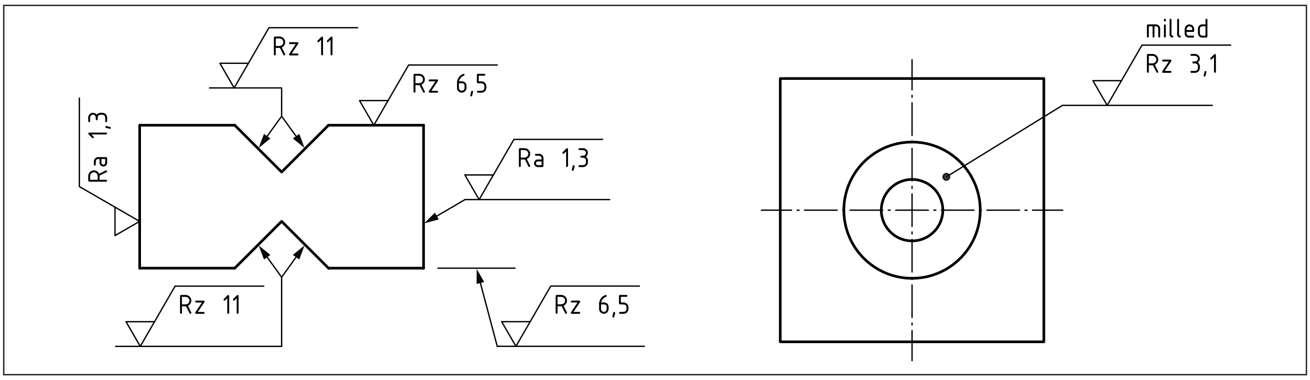
Graphical symbol	Interpretation and example	
=	Parallel to plane of projection of view in which symbol is used	
⊥	Perpendicular to plane of projection of view in which symbol is used	
X	Crossed in two oblique directions relative to plane of projection of view in which symbol is used	
M	Multi–directional	
C	Approximately circular relative to centre of surface to which symbol applies	
R	Approximately radial relative to centre of surface to which symbol applies	
P	Lay is particulate, non–directional, or protuberant	

[A)] If it is necessary to specify a surface pattern which is not clearly defined by these symbols, this shall be achieved by the addition of a suitable note to the drawing.

12.4 Position and orientation of graphical symbol

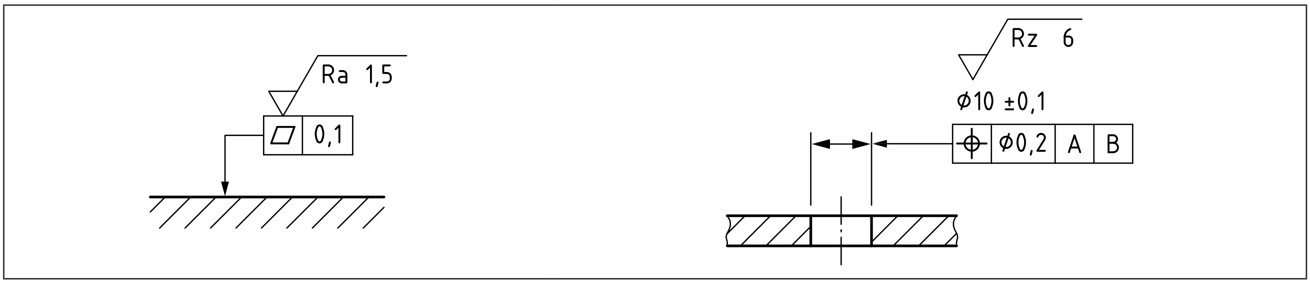
The graphical symbol and information shall be orientated so that they are readable from the bottom or right-hand side of the drawing in accordance with BS ISO 129–1. The symbol or terminating leader shall point at the surface from outside the workpiece, either to the outline or to an extension of it (see Figure 177).

Figure 177 – *Attachment of surface texture requirement to workpiece*



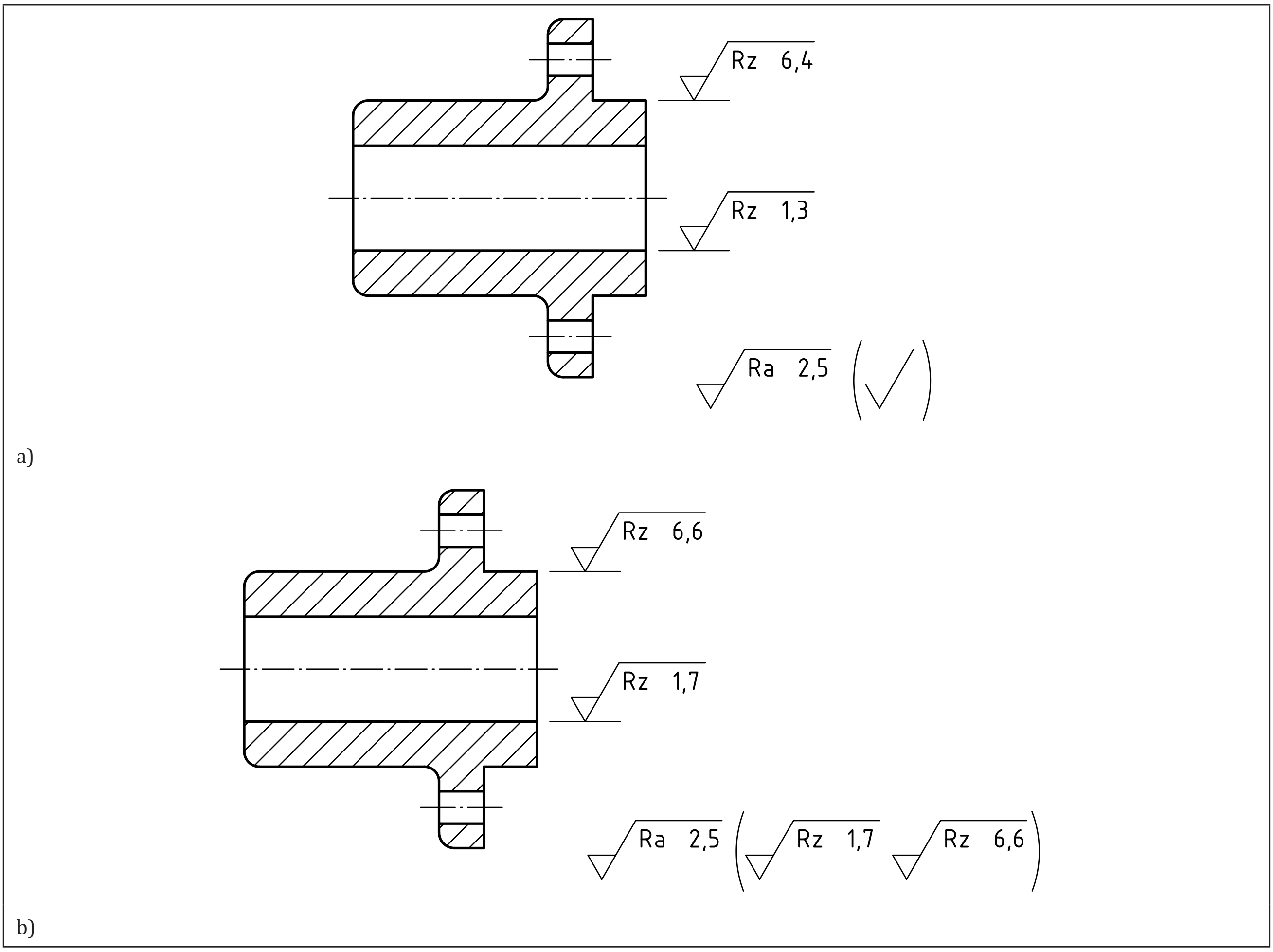
NOTE 1 The surface texture requirement can also be attached to a toleranced feature indicator where geometrical tolerancing is applied to BS EN ISO 1101, as shown in Figure 178.

Figure 178 – *Surface texture requirement: Geometrical tolerance indication*



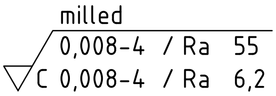
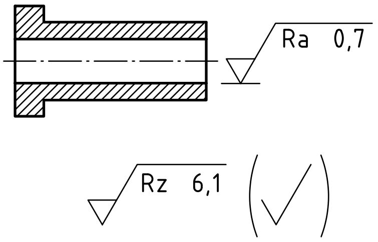
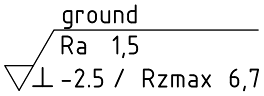
A surface texture specification may be indicated as a general requirement when it applies to several or all surfaces on a workpiece. The symbol representing general requirement shall be followed by either a basic symbol [Figure 179a)] or the actual requirements [Figure 179b)] in parentheses in order to indicate that other surface texture requirements are specified. A surface texture specification may also be applied using the "all-around" or "all-over" symbols.

Figure 179 – ***Simplified indication: Majority of surfaces with same surface texture requirement***



NOTE 2 BS EN ISO 1302 provides a full description of the application of surface texture requirements for engineering drawings with extensive examples of the application of surface texture specification, some of which are reproduced in Table 35.

Table 35 – *Example application of surface texture*

Requirement	**Example**
Surface roughness: – bilateral specification; – upper specification limit *Ra* = 55 µm; – lower specification limit *Ra* = 6,2 µm; – both "16 %–rule", default (BS EN ISO 4288); – both transmission band 0,008–4 mm; – default evaluation length (5 mm × 4 mm = 20 mm) (BS EN ISO 4288); – surface lay approximately circular around the centre; – manufacturing process, milling. *NOTE U and L are not stated because there is no doubt.*	
One surface has a specific requirement for a surface roughness: – one single, unilateral/upper specification limit; – *Ra* = 0,7 µm; – "16 %–rule", default; – default transmission band (BS EN ISO 4288 and BS EN ISO 3274); – default evaluation length (5 × λc) (BS EN ISO 4288); – surface lay, no requirement; – manufacturing process shall remove material. Unspecified surfaces have a roughness requirement of: – one single, unilateral/upper specification limit; – *Rz* = 6,1 µm; – "16 %–rule", default (BS EN ISO 4288); – default transmission band (BS EN ISO 4288 and BS EN ISO 3274); – default evaluation length (5 × λc) (BS EN ISO 4288); – surface lay, no requirement; – manufacturing process shall remove material.	
Surface roughness: – two, unilateral/upper specification limits: 1) *Ra* = 1,5 µm; 2) "16 %–rule", default (BS EN ISO 4288); 3) default transmission band (BS EN ISO 4288 and BS EN ISO 3274); 4) default evaluation length (5 × λc) (BS EN ISO 4288); 5) *Rz* max = 6,7 µm; 6) max-rule; 7) transmission band – 2,5 mm (BS EN ISO 3274); 8) evaluation length default (5 × 2,5 mm); – surface lay approximately perpendicular on the projection plane; – manufacturing process, grinding.	

12.5 Surface texture and surface characterization

12.5.1 Surface texture: Areal

Areal surface texture requirements shall conform to BS EN ISO 25178 (all parts).

*COMMENTARY ON **12.5.1***

BS EN ISO 25178 (all parts) provides a detailed set of standards for understanding the derivation, application, indication and measurement of areal surface texture.

Surface texture has traditionally been defined from profiles; this reflects the available technology limited to profile measuring instruments. Advances in technology have made areal instruments widely available and led to the development of areal-surface-texture standards.

Areal parameters that have a direct profile equivalent can be compared, e.g. Sq can be compared with Rq. However, surface textures tend to have larger measured values using areal parameters than with the equivalent profile parameters.

12.5.2 Surface characterization: Filtration

Filtration of surface texture requirements shall conform to BS EN ISO 25178-2 and BS EN ISO 25178-3.

*COMMENTARY ON **12.5.2***

BS EN ISO 25187-2 provides detailed descriptions and explanations of the terms definitions and parameters used for areal surface texture. BS EN ISO 25178-3 provides detailed description for the specification operator, in particular, filtration.

*The primary extracted surface is a finite set of points extracted from the primary surface. Typically, this results in the application of an S-Filter which removes the small-scale surface components. In common with surface texture profiles (see **12.2.2**) further surface filters are applied to derive scale-limited surfaces.*

S-F surfaces are scale-limited surfaces derived from the primary surface by using an F-operation to remove form.

S-L surfaces are scale-limited surfaces derived from the primary surface by using an L-filter to remove large scale components from the either the primary extracted surface of the S-F surface.

12.5.3 Surface characterization: Field parameters

Field parameters are categorized as height parameters, spatial parameters, hybrid parameters, functions and related parameters, and miscellaneous parameters, and are described in detail in BS EN ISO 25178-2.

The most common height parameters for areal surface texture are:

a) Sa, Arithmetical mean height of the scale-limited surface;

b) Sp, Maximum peak height of the scale-limited surface;

c) Sv, Maximum pit height of the scale-limited surface;

d) Sz, Maximum height of the scale limited-surface;

e) Sq, Root mean square height of the scale-limited surface; and

f) Ssk, Skewness of the scale-limited surface.

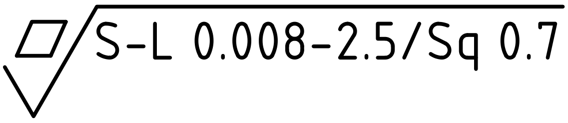
12.5.4 Drawing indication

Drawing indication shall utilize the same graphical symbol and structure defined in BS EN ISO 1302 and illustrated in Figure 149 and Figure 150) with the addition of a rhomb. The parameter specification shall be in location “a”; if a second specification is required it shall be indicated at location “b”, and if further parameters are required the symbols shall be enlarged.

The surface texture parameter shall be supplemented with the information correct and unambiguous specification.

NOTE Normally, this includes filters and nesting indices, parameters and values, and non-defaults. An example of an S-L surface specification is shown in Figure 180.

Figure 180 – *Example of Areal surface texture indication*



Interpretation: Surface with no manufacturing requirements, S-L surface, S-filter nesting index = 0,008 mm, L-filter nesting index = 2,5 mm, field parameter is Sq with an upper limit of 0,7 µm.

Section 13 Symbols and abbreviations

13.1 Symbols and abbreviations

13.1.1 General

Abbreviations (text equivalents) used in a TPS shall be the same in the singular and the plural. A full stop shall not be used except where an abbreviation forms a word (e.g. NO. as an abbreviation for "number").

NOTE Where possible, abbreviations should be avoided (see ***13.1.2****).*

Symbols used for physical quantities and units of measurement shall conform to the following standards, as appropriate.

BS EN ISO 80000‑1	*Quantities and units – Part 1: General*
BS EN ISO 80000‑2	*Quantities and units – Part 2: Mathematics*
BS EN ISO 80000‑3	*Quantities and units – Part 3: Space and time*
BS EN ISO 80000‑4	*Quantities and units – Part 4: Mechanics*
BS EN ISO 80000‑5	*Quantities and units – Part 5: Thermodynamics*
BS EN 80000‑6	*Quantities and units – Part 6: Electromagnetism*
BS ISO 80000‑7	*Quantities and units – Part 7: Light and radiation*
BS EN ISO 80000‑8	*Quantities and units – Part 8: Acoustics*
BS EN ISO 80000‑9	*Quantities and units – Part 9: Physical chemistry and molecular physics*
BS EN ISO 80000‑10	*Quantities and units – Part 10: Atomic and nuclear physics*
BS EN ISO 80000‑11	*Quantities and units – Part 11: Characteristic numbers*
BS EN ISO 80000‑12	*Quantities and units – Part 12: Condensed matter physics*
BS EN 80000‑13	*Quantities and units – Part 13: Information science and technology*
BS EN 80000‑14	*Quantities and units – Part 14: Telebiometrics related to human physiology*

13.1.2 Standard symbols and abbreviations

COMMENTARY ON ***13.1.2***

In the existing environment of outsourcing across national borders, every effort is being made to make the use of GPS independent of language through the adoption of standard symbology. It is for this reason that the continued use of abbreviations is deprecated.

Where particular specification requirements cannot be expressed using the available GPS system, full text description shall be employed.

NOTE 1 It is suggested that, where such a requirement occurs frequently, this be drawn to the attention of the relevant ISO committee through the appropriate BSI Technical Committee.

For diagrams used in technical applications, a library of harmonized graphical symbols has been developed with close cooperation between ISO and IEC. This is published in the following series of standards and these symbols shall be applied wherever practicable to improve the universal applicability of the TPS.

BS ISO 14617–1	*Graphical symbols for diagrams – Part 1: General information and indexes*
BS ISO 14617–2	*Graphical symbols for diagrams – Part 2: Symbols having general application*
BS ISO 14617–3	*Graphical symbols for diagrams – Part 3: Connections and related devices*
BS ISO 14617–4	*Graphical symbols for diagrams – Part 4: Actuators and related devices*
BS ISO 14617–5	*Graphical symbols for diagrams – Part 5: Measurement and control devices*
BS ISO 14617–6	*Graphical symbols for diagrams – Part 6: Measurement and control functions*
BS ISO 14617–7	*Graphical symbols for diagrams – Part 7: Basic mechanical components*
BS ISO 14617–8	*Graphical symbols for diagrams – Part 8: Valves and dampers*
BS ISO 14617–9	*Graphical symbols for diagrams – Part 9: Pumps, compressors and fans*
BS ISO 14617–10	*Graphical symbols for diagrams – Part 10: Fluid power converters*
BS ISO 14617–11	*Graphical symbols for diagrams – Part 11: Devices for heat transfer and heat engines*
BS ISO 14617–12	*Graphical symbols for diagrams – Part 12: Devices for separating, purification and mixing*

Symbols appropriate to TPS are provided and detailed throughout the suite of documents cross–referenced from this British Standard, and these shall be used where appropriate.

NOTE 2 It is strongly recommended that abbreviations not be used.

Where, in particular technical fields, certain abbreviations are in common use and generally understood, it is accepted that these can continue to be used, but new abbreviations shall not be introduced.

NOTE 3 Former practice has resulted in certain abbreviations becoming accepted as symbols and these should not be considered to provide precedence for the proliferation of abbreviations.

Section 14 Document handling

14.1 Types of documentation

14.1.1 General

COMMENTARY ON 14.1.1

The careful targeting of TPD to known or intended users can greatly assist the accuracy with which the specification is converted into the final product.

While precision and avoidance of ambiguity are always paramount, the means employed to convey this information shall be shown to match the capability, or potential capability, of the available or achievable manufacturing facility.

NOTE Specification beyond this level is unlikely to produce satisfactory results and can often prove expensive, both in terms of the cost of the over–specification itself and in terms of inadequate or unacceptable product.

14.1.2 Presentation media

14.1.2.1 General

The presentation of the drawings shall conform to the following standards, as appropriate.

BS EN ISO 5457	*Technical product documentation – Sizes and layout of drawing sheets*
BS EN ISO 7200	*Technical product documentation – Data fields in title blocks and document headers*
BS ISO 7573	*Technical product documentation – Parts lists*

14.1.2.2 Format

Drawing sheets and other documents shall be presented in one of the following formats:

a) landscape: intended to be viewed with the longest side of the sheet horizontal;

b) portrait: intended to be viewed with the longest side of the sheet vertical.

NOTE Contrary to BS EN ISO 5457, A4 sheets can be used in landscape or portrait mode.

14.2 Security

14.2.1 Introduction

Many TPSs have minimal requirements for security, other than that provided by general handling and storage procedures. However, where specific need for a general level of security is identified, the requirements in **14.2.2**, **14.2.3** and **14.2.4** shall be met.

14.2.2 General security

Procedures for ensuring the security of TPDs and TPSs shall conform to the following standard.

BS EN ISO 11442	*Technical product documentation – Document management*

14.2.3 Enhanced security

Where enhanced security is claimed, the requirements of Annex D shall be met, in addition to those in **14.2.4**.

14.2.4 Security level identification

The level of security attributed to any given TPS shall be clearly identified by the relevant marking placed adjacent to the title or title block of every TPD making up that TPS.

14.3 Storage

Methods for storage and retrieval of the document shall conform to the following standards, as appropriate.

BS EN ISO 6428	*Technical drawings – Requirements for microcopying*
BS EN ISO 11442	*Technical product documentation – Document management*

14.4 Marking

14.4.1 General

Technical product documents shall be marked to indicate which standards, or system of standards, are to govern their interpretation in a prominent location.[7)]

Technical product documents prepared in accordance with this British Standard shall be prepared in accordance with the ISO system.

*NOTE The marking of a TPD or TPS with the number of this standard constitutes a claim that the appropriate requirements of all relevant cross–referenced standards, in addition to the requirements directly specified in this British Standard, have been met. Attention is drawn to the date of issue principle (**3.1.3**).*

14.4.2 Enhanced security

TPDs conforming to Annex D, shall be marked in a prominent location with the number of this standard, followed by the suffix "/D", i.e.:

CONFORMS TO BS 8888/D

14.5 Protection notices

Where appropriate to place restrictions on the use of TPD, the following standard shall be applied.

BS ISO 16016	*Technical product documentation – Protection notices for restricting the use of documents and products*

[7)] Marking BS 8888:2020 on or in relation to a product represents a manufacturer's declaration of conformity, i.e. a claim by or on behalf of the manufacturer that the product meets the requirements of the standard. The accuracy of the claim is solely the claimant's responsibility. Such a declaration is not to be confused with third party certification of conformity.

Annex A (normative) Normative references

The following referenced documents are indispensable for the application of this document. For dated references, only the edition cited applies. For undated references, the latest edition of the referenced document (including any amendments) applies.

BS 1134, *Assessment of surface texture – Guidance and general information*

BS 1916–1, *Limits and fits for engineering – Part 1: Guide to limits and tolerances*

BS 1916–2, *Limits and fits for engineering – Part 2: Guide to the selection of fits in BS 1916–1*

BS 1916–3, *Limits and fits for engineering – Part 3: Guide to tolerances, limits and fits for large diameters*

BS 3238–1, *Graphical symbols for components of servo–mechanisms – Part 1: Transductors and Magnetic Amplifiers*

BS 3238–2, *Graphical symbols for components of servo–mechanisms – Part 2: General Servo–mechanisms*

BS 3734–1, *Rubber – Tolerances for products – Part 1: Dimensional tolerances*[8)]

BS 4500, *Limits and fits – Guidance for system of cone (taper) fits and tolerances for cones from C = 1:3 to 1:500, lengths from 6 mm to 630 mm and diameters up to 500 mm*

BS 7010, *Code of practice for a system of tolerances for the dimensions of plastic mouldings*

BS EN 80000–6, *Quantities and units – Part 6: Electromagnetism*

BS EN 80000–13, *Quantities and units – Part 13: Information science and technology*

BS EN 80000–14, *Quantities and units – Part 14: Telebiometrics related to human physiology*

BS EN ISO 1, *Geometrical product specifications (GPS) – Standard reference temperature for the specification of geometrical and dimensional properties*

BS EN ISO 128–20, *Technical drawings – General principles of presentation – Part 20: Basic conventions for lines*

BS EN ISO 128–21, *Technical drawings – General principles of presentation – Part 21: Preparation of lines by CAD systems*

BS EN ISO 286–1, *Geometrical product specifications (GPS) – ISO code system for tolerances on linear sizes – Part 1: Basis of tolerances, deviations and fits*

BS EN ISO 286–2, *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*

BS EN ISO 1101:2017, *Geometrical product specifications (GPS) – Geometrical tolerancing – Tolerances of form, orientation, location and run–out*

BS EN ISO 1119, *Geometrical product specifications (GPS) – Series of conical tapers and taper angles*

BS EN ISO 1302, *Geometrical Product Specifications (GPS) – Indication of surface texture in technical product documentation*

[8)] Withdrawn.

BS EN ISO 1660, *Geometrical product specifications (GPS) – Geometrical tolerancing – Profile tolerancing*[9)]

BS EN ISO 2162–1, *Technical product documentation – Springs – Part 1: Simplified representation*

BS EN ISO 2162–2, *Technical product documentation – Springs – Part 2: Presentation of data for cylindrical helical compression springs*

BS EN ISO 2203, *Technical drawings – Conventional representation of gears*

BS EN ISO 2553, *Welded and allied processes – Symbolic representation on drawings –Welded joints*[10)]

BS EN ISO 2692, *Geometrical product specifications (GPS) – Geometrical tolerancing –Maximum material requirement (MMR), least material requirement (LMR) and reciprocity requirement (RPR)*

BS EN ISO 3040, *Geometrical product specifications (GPS) – Dimensioning and tolerancing – Cones*

BS EN ISO 3098–0, *Technical Product Documentation – Lettering – Part 0: General requirements*[11)]

BS EN ISO 3098–2, *Technical product documentation – Lettering – Part 2: Latin alphabet, numerals and marks*

BS EN ISO 3098–3, *Technical product documentation – Lettering – Part 3: Greek alphabet*

BS EN ISO 3098–4, *Technical product documentation – Lettering – Part 4: Diacritical and particular marks for the Latin alphabet*

BS EN ISO 3098–5, *Technical product documentation – Lettering – Part 5: CAD lettering of the Latin alphabet, numerals and marks*

BS EN ISO 3098–6, *Technical product documentation – Lettering – Part 6: Cyrillic alphabet*

BS EN ISO 3274, *Geometrical Product Specifications (GPS) – Surface texture: Profile method – Nominal characteristics of contact (stylus) instruments*

BS EN ISO 4063, *Welding and allied processes – Nomenclature of processes and reference numbers*

BS EN ISO 4287, *Geometrical product specification (GPS) – Surface texture: Profile method – Terms, definitions and surface texture parameters*

BS EN ISO 4288, *Geometric Product Specification (GPS) – Surface texture – Profile method: Rules and procedures for the assessment of surface texture*

BS EN ISO 5261, *Technical drawings – Simplified representation of bars and profile sections*

BS EN ISO 5456–2, *Technical drawings – Projection methods – Part 2: Orthographic representations*

BS EN ISO 5456–3, *Technical drawings – Projection methods – Part 3: Axonometric representations*

BS EN ISO 5457, *Technical product documentation – Sizes and layout of drawing sheets*

BS EN ISO 5458, *Geometrical Product Specifications (GPS) – Geometrical tolerancing – Pattern and combined geometrical specification*

BS EN ISO 5459, *Geometrical product specifications (GPS) – Geometrical tolerancing – Datums and datum systems*[12)]

BS EN ISO 5845–1, *Technical drawings – Simplified representation of the assembly of parts with fasteners – Part 1: General principles*

[9)] This standard also gives an informative reference to BS EN ISO 1660:2015.

[10)] This standard also gives an informative reference to BS EN ISO 2553:2019.

[11)] Withdrawn.

[12)] This standard also gives an informative reference to BS EN ISO 5459:2011.

BS EN ISO 5817, *Welding – Fusion-welded joints in steel, nickel, titanium and their alloys (beam welding excluded) – Quality levels for imperfections*

BS EN ISO 6410–1, *Technical drawings – Screw threads and threaded parts – Part 1: General conventions*

BS EN ISO 6410–2, *Technical drawings – Screw threads and threaded parts – Part 2: Screw thread inserts*

BS EN ISO 6410–3, *Technical drawings – Screw threads and threaded parts – Part 3: Simplified representation*

BS EN ISO 6411, *Technical drawings – Simplified representation of centre holes*

BS EN ISO 6412–1, *Technical product documentation – Simplified representation of pipelines – Part 1: General rules and orthogonal representation*

BS EN ISO 6412–2, *Technical product documentation – Simplified representation of pipelines – Part 2: Isometric projection*

BS EN ISO 6412–3, *Technical product documentation – Simplified representation of pipelines – Part 3: Terminal features of ventilation and drainage systems*

BS EN ISO 6413, *Technical drawings – Representation of splines and serrations*

BS EN ISO 6428, *Technical drawings – Requirements for microcopying*

BS EN ISO 6947, *Welding and allied processes – Welding positions*

BS EN ISO 7083, *Technical drawings – Symbols for geometrical tolerancing – Proportions and dimensions*

BS EN ISO 7200, *Technical product documentation – Data fields in title blocks and document headers*

BS EN ISO 8015, *Geometrical product specifications (GPS) – Fundamentals – Concepts, principles and rules*

BS EN ISO 8062–1, *Geometrical product specifications (GPS) – Dimensional and geometrical tolerances for moulded parts – Part 1: Vocabulary*

BS EN ISO 8062–3, *Geometrical product specifications (GPS) – Dimensional and geometrical tolerances for moulded parts – Part 3: General dimensional and geometrical tolerances and machine allowances for castings*

BS EN ISO 8785, *Geometrical product specification (GPS) – Surface imperfections – Terms, definitions and parameters*

BS EN ISO 8826–1, *Technical drawings – Roller bearings – Part 1: General simplified representation*

BS EN ISO 8826–2, *Technical drawings – Roller bearings – Part 2: Detailed simplified representation*

BS EN ISO 9222–1, *Technical drawings – Seals for dynamic application – Part 1: General simplified representation*

BS EN ISO 9222–2, *Technical drawings – Seals for dynamic application – Part 2: Detailed simplified representation*

BS EN ISO 10135, *Geometrical product specifications (GPS) – Drawing indications for moulded parts in technical product documentation (TPD)*

BS EN ISO 10209, *Technical product documentation – Vocabulary – Terms relating to technical drawings, product definition and related documentation*

BS EN ISO 10042, *Welding – Arc-welded joints in aluminium and its alloys – Quality levels for imperfections*

BS EN ISO 10579, *Geometrical product specifications (GPS) – Dimensioning and tolerancing – Non-rigid parts*

BS EN ISO 11442, *Technical product documentation – Document management*

BS EN ISO 12085, *Geometrical Product Specifications (GPS) – Surface texture: Profile method – Motif parameters*

BS EN ISO 12180–1, *Geometrical product specifications (GPS) – Cylindricity – Part 1: Vocabulary and parameters of cylindrical form*

BS EN ISO 12180–2, *Geometrical product specifications (GPS) – Cylindricity – Part 2: Specification operators*

BS EN ISO 12181–1, *Geometrical product specifications (GPS) – Roundness – Part 1: Vocabulary and parameters of roundness*

BS EN ISO 12181–2, *Geometrical product specifications (GPS) – Roundness – Part 2: Specification operators*

BS EN ISO 12780–1, *Geometrical product specifications (GPS) – Straightness – Part 1: Vocabulary and parameters of straightness*

BS EN ISO 12780–2, *Geometrical product specifications (GPS) – Straightness – Part 2: Specification operators*

BS EN ISO 12781–1, *Geometrical product specifications (GPS) – Flatness – Part 1: Vocabulary and parameters of flatness*

BS EN ISO 12781–2, *Geometrical product specifications (GPS) – Flatness – Part 2: Specification operators*

BS EN ISO 13565–1, *Geometric Product Specifications (GPS) – Surface texture: Profile method – Surfaces having stratified functional properties – Part 1: Filtering and general measurement conditions*

BS EN ISO 13565–2, *Geometrical Product Specifications (GPS) – Surface texture: Profile method – Surfaces having stratified functional properties – Part 2: Height characterization using the linear material ration curve*

BS EN ISO 13565–3, *Geometrical product specifications (GPS) – Surface texture: Profile method – Surfaces having stratified functional properties – Part 3: Height characterization using the material probability curve*

BS EN ISO 13715, *Technical product documentation – Edges of unidentified shape – Indication and dimensioning*

BS EN ISO 13919 (both parts), *Electron and laser-beam welded joints*

BS EN ISO 13920, *Welding – General tolerances for welded constructions – Dimensions for lengths and angles – Shape and position*

BS EN ISO 14171, *Welding consumables – Solid wire electrodes, tubular cored electrodes and electrode/flux combinations for submerged arc welding of non alloy and fine grain steels – Classification*

BS EN ISO 14253–1, *Geometrical product specifications (GPS) – Inspection by measurement of workpieces and measuring equipment – Part 1: Decision rules for verifying conformity or non–conformity with specifications*

BS EN ISO 14341, *Welding consumables – Wire electrodes and weld deposits for gas shielded metal arc welding of non alloy and fine grain steels – Classification*

BS EN ISO 14405–1, *Geometrical product specifications (GPS) – Dimensional tolerancing – Part 1: Linear sizes*

BS EN ISO 14405–2, *Geometrical product specifications (GPS) – Dimensional tolerancing – Part 2: Dimensions other than linear sizes*

BS EN ISO 14405-3, *Geometrical product specifications (GPS) – Dimensional tolerancing – Part 3: Angular sizes*

BS EN ISO 14660–1, *Geometrical Product Specifications (GPS) – Geometrical features – Part 1: General terms and definitions*

BS EN ISO 14660–2, *Geometrical Product Specifications (GPS) – Geometrical features – Part 2: Extracted median line of a cylinder and a cone, extracted median surface, local size of an extracted feature*[13)]

BS EN ISO 15785, *Technical drawings – Symbolic presentation and indication of adhesive, fold and pressed joints*

BS EN ISO 16610 (all parts), *Geometrical product specifications (GPS) – Filtration*

BS EN ISO 16610–1, *Geometrical product specifications (GPS) – Filtration – Part 1: Overview and basic concepts*

BS EN ISO 16610–20, *Geometrical product specifications (GPS) – Filtration – Part 20: Linear profile filters: Basic concepts*

BS EN ISO 16610–21, *Geometrical product specifications (GPS) – Filtration – Part 21: Linear profile filters: Gaussian filters*

BS EN ISO 16610–22, *Geometrical product specifications (GPS) – Filtration – Part 22: Linear profile filters: Spline filters*

BS EN ISO 16610–29, *Geometrical product specifications (GPS) – Filtration – Part 29: Linear profile filters: Spline wavelets*

BS EN ISO 16610–40, *Geometrical product specifications (GPS) – Filtration – Part 40: Morphological profile filters: Basic concepts*

BS EN ISO 16610–41, *Geometrical product specifications (GPS) – Filtration – Part 41: Morphological profile filters: Disk and horizontal line–segment filters*

BS EN ISO 16610–49, *Geometrical product specifications (GPS) – Filtration – Part 49: Morphological profile filters: Scale space techniques*

BS EN ISO 17450-1, *Geometrical product specifications (GPS) – General concepts – Part 1: Model for geometrical specification and verification*

BS EN ISO 17450-2, *Geometrical product specifications (GPS) – General concepts – Part 2: Basic tenets, specifications, operators, uncertainties and ambiguities*

BS EN ISO 17450-3, *Geometrical product specifications (GPS) – General concepts – Part 3: Toleranced features*

BS EN ISO 17450-4, *Geometrical product specifications (GPS) – Basic concepts – Part 4: Geometrical characteristics for quantifying GPS deviations*

BS EN ISO 22432, *Geometrical product specifications (GPS) – Features utilized in specification and verification*

[13)] Withdrawn.

BS EN ISO 25178-1, *Geometrical product specifications (GPS) – Surface texture: Areal – Part 1: Indication of surface texture*

BS EN ISO 25178-2, *Geometrical product specifications (GPS) – Surface texture: Areal – Part 2: Terms, definitions and surface texture parameters*

BS EN ISO 25178-3, *Geometrical product specifications (GPS) – Surface texture: Areal – Part 3: Specification operators*

BS EN ISO 25178-70, *Geometrical product specifications (GPS) – Surface texture: Areal – Part 70: Material measures*

BS EN ISO 25178-71, *Geometrical product specifications (GPS) – Surface texture: Areal – Part 71: Software measurement standards*

BS EN ISO 25178-72, *Geometrical product specifications (GPS) – Surface texture: Areal – Part 72: XML file format x3p*

BS EN ISO 25178-73, *Geometrical product specifications (GPS) – Surface texture: Areal – Part 73: Terms and definitions for surface defects on material measures*

BS EN ISO 25178-600, *Geometrical product specifications (GPS) – Surface texture: Areal – Part 600: Metrological characteristics for areal topography measuring methods*

BS EN ISO 25178-601, *Geometrical product specifications (GPS) – Surface texture: Areal – Part 601: Nominal characteristics of contact (stylus) instruments*

BS EN ISO 25178-602, *Geometrical product specifications (GPS) – Surface texture: Areal – Part 602: Nominal characteristics of non-contact (confocal chromatic probe) instruments*

BS EN ISO 25178-603, *Geometrical product specifications (GPS) – Surface texture: Areal – Part 603: Nominal characteristics of non-contact (phase-shifting interferometric microscopy) instruments*

BS EN ISO 25178-604, *Geometrical product specifications (GPS) – Surface texture: Areal – Part 604: Nominal characteristics of non-contact (coherence scanning interferometry) instruments*

BS EN ISO 25178-605, *Geometrical product specifications (GPS) – Surface texture: Areal – Part 605: Nominal characteristics of non-contact (point autofocus probe) instruments*

BS EN ISO 25178-606, *Geometrical product specifications (GPS) – Surface texture: Areal – Part 606: Nominal characteristics of non-contact (focus variation) instruments*

BS EN ISO 25178-607, *Geometrical product specifications (GPS) – Surface texture: Areal – Part 607: Nominal characteristics of non-contact (confocal microscopy) instruments*

BS EN ISO 25178-701, *Geometrical product specifications (GPS) – Surface texture: Areal – Part 701: Calibration and measurement standards for contact (stylus) instruments*

BS EN ISO 25378, *Geometrical product specifications (GPS) – Characteristics and conditions – Definitions*

BS EN ISO 26909, *Springs – Vocabulary*

BS EN ISO 80000–1, *Quantities and units – Part 1: General*

BS EN ISO 80000–2, *Quantities and units – Part 2: Mathematics*

BS EN ISO 80000–3, *Quantities and units – Part 3: Space and time*

BS EN ISO 80000–4, *Quantities and units – Part 4: Mechanics*

BS EN ISO 80000–5, *Quantities and units – Part 5: Thermodynamics*

BS EN ISO 80000–8, *Quantities and units – Part 8: Acoustics*

BS EN ISO 80000–9, *Quantities and units – Part 9: Physical chemistry and molecular physics*

BS EN ISO 80000–10, *Quantities and units – Part 10: Atomic and nuclear physics*

BS EN ISO 80000–11, *Quantities and units – Part 11: Characteristic numbers*

BS EN ISO 80000–12, *Quantities and units – Part 12: Condensed matter physics*

BS EN ISO 81714–1, *Design of graphical symbols for use in the technical documentation of products – Part 1: Basic rules*

BS ISO 128–22, *Technical drawings – General principles of presentation – Part 22: Basic conventions and applications for leader lines and reference lines*

BS ISO 128–23, *Technical drawings – General principles of presentation – Part 23: Lines on construction drawings*

BS ISO 128–24:1999, *Technical drawings – General principles of presentation – Part 24: Lines on mechanical engineering drawings*[14)]

BS ISO 128–25, *Technical drawings – General principles of presentation – Part 25: Lines on shipbuilding drawings*

BS ISO 128–30, *Technical drawings – General principles of presentation – Part 30: Basic conventions for views*

BS ISO 128–34, *Technical drawings – General principles of presentation – Part 34: Views on mechanical engineering drawings*

BS ISO 128–40, *Technical drawings – General principles of presentation – Part 40: Basic conventions for cuts and sections*

BS ISO 128–44, *Technical drawings – General principles of presentation – Part 44: Sections on mechanical engineering drawings*

BS ISO 128–50, *Technical drawings – General principles of presentation – Part 50: Basic conventions for representing areas on cuts and sections*

BS ISO 129–1, *Technical product documentation (TPD) – Presentation of dimensions and tolerances – Part 1: General principles*

BS ISO 1219–1, *Fluid power systems and components – Graphical symbols and circuit diagrams – Part 1: Graphical symbols for conventional use and data–processing applications*

BS ISO 5456–4, *Technical drawings – Projection methods – Part 4: Central projection*

BS ISO 7573, *Technical product documentation – Parts lists*

BS ISO 10579, *Geometrical product specifications (GPS) – Dimensioning and tolerancing – Non–rigid parts*

BS ISO 14617–1, *Graphical symbols for diagrams – Part 1: General information and indexes*

BS ISO 14617–2, *Graphical symbols for diagrams – Part 2: Symbols having general application*

BS ISO 14617–3, *Graphical symbols for diagrams – Part 3: Connections and related devices*

BS ISO 14617–4, *Graphical symbols for diagrams – Part 4: Actuators and related devices*

BS ISO 14617–5, *Graphical symbols for diagrams – Part 5: Measurement and control devices*

[14)] This standard also gives informative references to BS ISO 128-24:1999.

BS ISO 14617–6, *Graphical symbols for diagrams – Part 6: Measurement and control functions*

BS ISO 14617–7, *Graphical symbols for diagrams – Part 7: Basic mechanical components*

BS ISO 14617–8, *Graphical symbols for diagrams – Part 8: Valves and dampers*

BS ISO 14617–9, *Graphical symbols for diagrams – Part 9: Pumps, compressors and fans*

BS ISO 14617–10, *Graphical symbols for diagrams – Part 10: Fluid power converters*

BS ISO 14617–11, *Graphical symbols for diagrams – Part 11: Devices for heat transfer and heat engines*

BS ISO 14617–12, *Graphical symbols for diagrams – Part 12: Devices for separating, purification and mixing*

BS ISO 16016, *Technical product documentation – Protection notices for restricting the use of documents and products*

BS ISO 16792:2015, *Technical product documentation – Digital product definition data practices*15)

BS ISO 80000–7, *Quantities and units – Part 7: Light and radiation*

ISO/IEC Guide 98–3, *Guide to the expression of uncertainty in measurement (GUM)*

ISO/IEC Guide 99, *International vocabulary of metrology – Basic and general concepts and associated terms (VIM)*

PD CEN ISO/TS 8062–2, *Geometrical Product Specifications (GPS) – Dimensional and geometrical tolerances for moulded parts – Part 2: Rules*

---

15) This standard also gives an informative reference to BS ISO 16792:2015.

Annex B (informative) Bibliography

Standards publications

For dated references, only the edition cited applies. For undated references, the latest edition of the referenced document (including any amendments) applies.

BS 308, *Engineering drawing practice*[16]

BS 1192 (all parts), *Construction drawing practice*

BS 3643–1, *ISO metric screw threads – Part 1: Principles and basic data*

BS 3643–2, *ISO metric screw threads – Part 2: Specification for selected limits of size*

BS 4235–1, *Specification for metric keys and keyways – Part 1: Parallel and taper keys*

BS 4235–2, *Specification for metric keys and keyways – Part 2: Woodruff keys and keyways*

BS 4827, *Specification for ISO miniature screw threads – Metric series*

BS 8887 (all parts), *Design for manufacture, assembly, disassembly and end–of–life processing (MADE)*

BS 8889, *Technical product verification – Inspection of size, form and surface texture in relation to function – Specification*[16]

BS EN 22768–1, *General tolerances – Part 1: Tolerances for linear and angular dimensions without individual tolerance indications*

BS EN 22768–2, *General tolerances – Part 2: Geometrical tolerances for features without individual tolerance indications*

BS EN ISO 216, *Writing paper and certain classes of printed matter – Trimmed sizes – A and B series, and indication of machine direction*

BS EN ISO 5455, *Technical drawings – Scales*

BS EN ISO 5456–1, *Technical drawing – Projection methods – Part 1: Synopsis*

BS EN ISO 14638, *Geometrical product specification (GPS) – Matrix model*

BS ISO 129–4, *Technical product documentation (TPD) – Indication of dimensions and tolerances – Part 4: Dimensioning of shipbuilding drawings*

BS ISO 261, *ISO general purpose metric screw threads – General plan*

BS ISO 262, *ISO general purpose metric screw threads – Selected sizes for screws, bolts and nuts*

BS ISO 965–1, *ISO general purpose metric screw threads – Tolerances – Part 1: Principles and basic data*

BS ISO 15786, *Technical drawings – Simplified representation and dimensioning of holes*

BS ISO 29845, *Technical product documentation — Document types*

PD 68888, *Objectives and learning outcomes for BS 8888 training*

DIN 7167, *Relationship between tolerances of size, form, and parallelism; envelope requirement without individual indication on the drawing*[16]

---

[16] Withdrawn.

Other publications

[1] AMERICAN SOCIETY OF MECHANICAL ENGINEERS (ASME). *Dimensioning and tolerancing (Y14.5)*. New York: ASME, 2009.

Annex C (informative) Associations

C.1 Association operations

The association operation is the method used to “fit” the ideal geometry to the non–ideal feature. Traditionally, when this was achieved using a mechanical device such as a surface table, angle plate or chuck, there was no need to define an association operation, as there was only one possibility.

Advances in metrological equipment have meant that devices such as coordinate measuring machines (CMMs), articulated arms and scanners of various kinds can offer a multiplicity of different association operations. Some of these are used when defining datums, while others are used when evaluating tolerances; some work better with some kinds of geometry, and others with different kinds of geometry. The term “best fit” is frequently used, often very loosely, and can refer to any of a number of association operations, including the following:

a) minimax (also known as the $L_{\infty \text{ norm}}$): the associated feature is arranged to minimize the maximum distance between the associated feature and the datum feature;

b) minimum circumscribed (also known as “Chebyshev”): the associated feature is the smallest ideal feature which fully envelops the datum feature;

c) maximum inscribed (also known as “Chebyshev”): the associated feature is the largest ideal feature which can be fully enveloped by the datum feature;

d) least squares (also known as “Gaussian” or $L_{2 \text{ norm}}$): the associated feature is an ideal feature fitted to the datum feature using the total least squares algorithm.

NOTE The name is usually taken from the mathematical algorithm on which the operation is based.

In addition, the following factors can affect the association.

1) Location and orientation constraints

If the association is being performed in the definition of a secondary or tertiary datum, it will include additional orientation or location constraints to ensure that the correct relationship with the preceding datums is maintained.

2) Material constraints

In some cases, the associated feature can be established entirely outside the material of the workpiece, or entirely inside the material of the workpiece.

3) Intrinsic constraints

Some datum features can be defined either with a toleranced dimension or with a theoretically exact dimension. In some cases, the associated feature could be defined with either a toleranced dimension or a theoretically exact dimension.

C.2 Default association operations

When defining datums, default association operations be used unless otherwise stated (see Table C.1).

Table C.1 – Default association operations used when defining datums

Datum feature	Internal or external	Default association operation	Default material constraint	Intrinsic characteristic	Default intrinsic constraint	Situation feature
Plane	–	Minimax	Outside the material	–	–	Plane
Two parallel opposed planes	External (flange)	Minimum circumscribed[A)]	Outside the material	Width (distance between the two parallel planes)	Variable	Plane
	Internal (slot)	Maximum inscribed[A)]				
Cylinder	External (shaft)	Minimum circumscribed[A)]	Outside the material	Diameter	variable	Line (axis)
	Internal (hole)	Maximum inscribed[A)]				
Sphere	External (ball)	Minimum circumscribed[A)]	Outside the material	Diameter	Variable	Point
	Internal (socket)	Maximum inscribed[A)]				
Cone	External	Minimax	Outside the material	Angle	Fixed	Line and point
	Internal	Minimax				
Wedge	External	Minimax	Outside the material	Angle	Fixed	Plane and line
	Internal	Minimax				
Prismatic	–	Minimax	Outside the material	–	–	Plane and line
Complex	–	Minimax	Outside the material	–	–	Plane, line and point

[A)] In cases where the linear size (of the feature of size) is considered variable, the result of the association can lead to several solutions with the same datum feature ("unstable association"). In this case, the following alternative association criterion are used: minimize the maximum distance normal to the associated feature between the associated feature and the datum feature or between the two associated features and the two datum features (in the case of two parallel planes).

NOTE The next planned revision of BS EN ISO 5459 will change the default association criterion to the Gaussian (least squares) objective function with the "outside the material" constraint. This will result in datum definitions which more closely correspond to the "real life" behaviour of mechanical parts and have improved reproducibility.

C.3 Non–default association operations

It is also possible to use alternative association operations. These can be invoked by using the following modifiers next to the datum reference in a tolerance frame. As these are yet to be standardized in a published ISO standard, it is imperative that the use of any of these modifiers is fully explained with a key or a description on the specification.

Non–default association algorithms are invoked with the following symbols:

Algorithm	Symbol
Least squares (Gaussian)	G
Minimax	C
Minimum circumscribed (Chebyshev)	N
Maximum inscribed (Chebyshev)	X
Minimal volume	K
Minimizing p–norm distance	L_1, L_2 or L_∞

The symbol for the non-default association should be accompanied with a symbol defining the material constraint to be used for the association criteria.

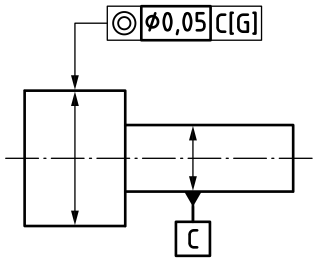
Material constraints options can be invoked with the following symbols:

Material constraint	Symbol
Outside material	E
Inside material	I
No material constraint	M
Shifted tangent outside material	+
Shifted tangent inside material	-

NOTE 1 GE should not be confused with G+. For G+, a first associated feature is established without material constraint (GM) and then shifted or scaled to just contact the material. For GE, the associated feature is established one time with outside material constraint. The GE and G+ association criteria generally yield different associated features.

An example of an individual specification operator for a datum association is given in Figure C.1.

Figure C.1 – *Example of individual specification operator for a datum association*



Non–default intrinsic constraints can be invoked with the following symbols:

Intrinsic constraint	Symbol
Size fixed	[SF]
Size variable	[SV]

C.4 Examples of datums obtained from different datum features

C.4.1 Datum based on a planar datum feature

The datum feature is identified on the TPD as a nominally flat plane.

The real datum feature on the manufactured workpiece is a non-ideal surface.

By default, the associated feature is a flat plane in contact with the datum feature, outside the material, arranged such as to minimize the maximum distances between the datum plane and the datum feature (see Figure C.2).

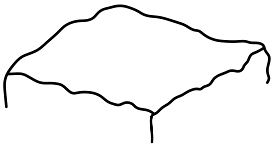
The datum is the situation feature for the associated feature, which is a flat plane.

A primary datum based on the datum feature is a plane, constructed with no additional constraints. This can control three degrees of freedom (see Figure C.3).

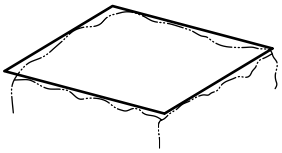
A secondary datum based on the datum feature is a plane, constructed with location and orientation constraints to the primary datum.

A tertiary datum based on the datum feature is a plane, constructed with location and orientation constraints to the primary and secondary datums.

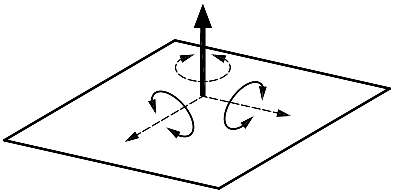
Figure C.2 – *Primary datum based on a planar datum feature*



a) **Real datum feature:** The datum feature is a plane. The real datum feature is the manufactured surface. The extracted datum feature is the data set obtained by sampling the surface.



b) **Associated feature:** The associated feature is a flat plane. The situation feature is a flat plane.



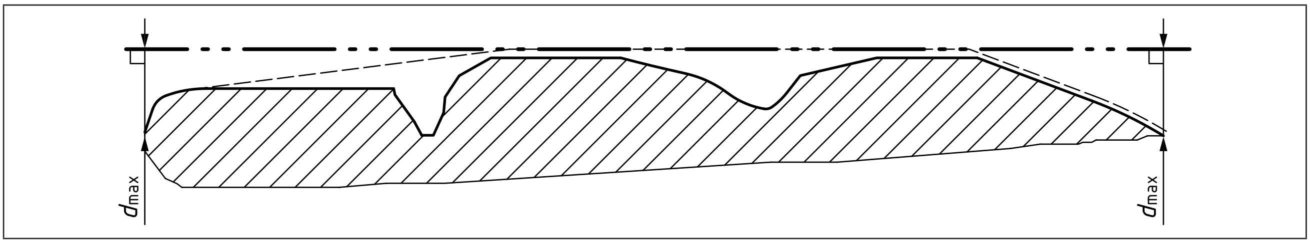
c) **Associated feature:** This datum can constrain three degrees of freedom:

- translation along an axis perpendicular to the plane; and
- rotation about two perpendicular axes aligned with the plane.

Key

⬆ Locked

Figure C.3 – ***Associated feature***



C.4.2 Datum based on two parallel, opposed planes (external)

The datum feature is identified on the TPD as two parallel opposed planes (e.g. a flange).

The real datum feature on the manufactured workpiece is composed of two non-ideal surfaces.

By default, the associated feature is a minimum circumscribed feature in contact with the datum feature, outside the material (see Figure C.4).

The datum is the situation feature for the associated feature, which is a median plane.

A primary datum based on the datum feature is a median plane, constructed with no additional constraints. This can control three degrees of freedom (see Figure C.4).

A secondary datum based on the datum feature is a median plane, constructed with location and orientation constraints to the primary datum.

A tertiary datum based on the datum feature is a median plane, constructed with location and orientation constraints to the primary and secondary datums.

C.4.3 Datum based on two parallel, opposed planes (internal)

The datum feature is identified on the TPD as two parallel opposed planes (e.g. a slot).

The real datum feature on the manufactured workpiece is composed of two non-ideal surfaces.

By default, the associated feature is a maximum inscribed feature in contact with the datum feature, outside the material.

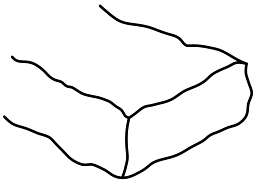
The datum is the situation feature for the associated feature, which is a median plane.

A primary datum based on the datum feature is a median plane, derived from an associated feature constructed with no additional constraints. This can control three degrees of freedom.

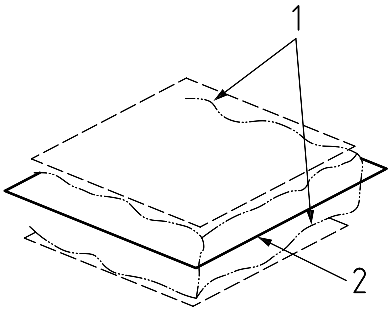
A secondary datum based on the datum feature is a median plane, derived from an associated feature constructed with location and orientation constraints to the primary datum.

A tertiary datum based on the datum feature is a median plane, derived from an associated feature constructed with location and orientation constraints to the primary and secondary datums.

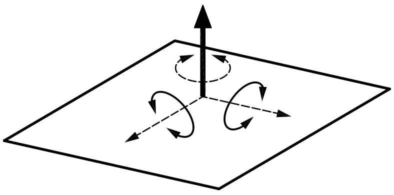
Figure C.4 – ***Primary datum based on two parallel opposed planes***



a) **Real datum feature:** The datum feature is two parallel opposed planes. The real datum feature is the manufactured surfaces. The extracted datum feature is the data set obtained by sampling the surfaces.



b) **Associated feature:** The associated feature is two parallel opposed planes. The situation feature is a median plane.



c) **Degrees of freedom that can be controlled:** This datum can constrain three degrees of freedom:

- translation along an axis perpendicular to the median plane; and
- rotation about two perpendicular axes aligned with the median plane.

Key

↑	Locked
1	Associated feature
2	Median plane

C.4.4 Datum based on an external cylindrical feature

The datum feature is identified on the TPD as a nominally cylindrical external feature.

The real datum feature on the manufactured workpiece is a non-ideal cylinder.

The associated feature is a minimum circumscribing cylinder constructed around the datum feature (unless otherwise indicated).

The datum is the situation feature for the minimum circumscribing cylinder, which is its axis.

A primary datum based on the cylindrical datum feature is an axis, derived from the minimum circumscribing cylinder with no additional constraints. This can control four degrees of freedom (see Figure C.5).

A secondary datum based on the datum feature is an axis, derived from the minimum circumscribing cylinder, which is constructed with location and orientation constraints to the primary datum.

A tertiary datum based on the datum feature is an axis, derived from the minimum circumscribing cylinder, which is constructed with location and orientation constraints to the primary and secondary datums.

C.4.5 Datum based on an internal cylindrical feature

The datum feature is identified on the TPD as a nominally cylindrical internal feature.

The real datum feature on the manufactured workpiece is a non-ideal cylinder.

The associated feature will be a maximum inscribing cylinder constructed within the datum feature (unless otherwise indicated).

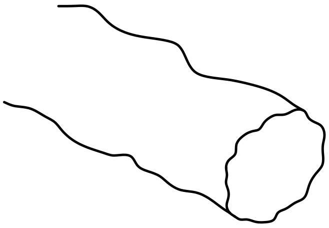
The datum is the situation feature for the maximum inscribing cylinder, which is its axis.

A primary datum based on the cylindrical datum feature is an axis, derived from a maximum inscribing cylinder constructed with no additional constraints. This can control four degrees of freedom.

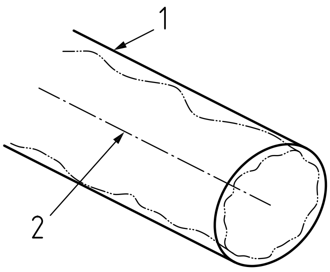
A secondary datum based on the datum feature is an axis, derived from a maximum inscribing cylinder constructed with location and orientation constraints to the primary datum.

A tertiary datum based on the datum feature is an axis, derived from the maximum inscribing cylinder constructed with location and orientation constraints to the primary and secondary datums.

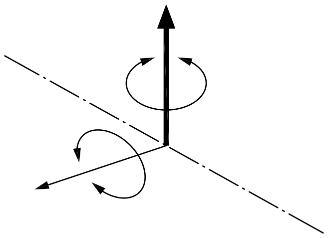
Figure C.5 – ***Primary datum based on an external cylindrical feature***



a) **Real datum feature:** The datum feature is an external cylinder. The real datum feature is the manufactured surface. The extracted datum feature is the data set obtained by sampling the surface.



b) **Associated feature:** The associated feature is the minimum circumscribing cylinder. The situation feature for the associated feature is its axis.



c) **Degrees of freedom that can be controlled:** This datum can constrain four degrees of freedom:

- translation along two perpendicular axes which are also perpendicular to the axis of the associated feature; and
- rotation about two perpendicular axes which are also perpendicular to the axis of the associated feature.

Key

↑	Locked
1	Associated feature
2	Axis

C.4.6 Datum based on an external spherical feature

The datum feature is identified on the TPD as a nominally spherical external feature.

The real datum feature on the manufactured workpiece is a non-ideal sphere.

The associated feature is a minimum circumscribing sphere constructed around the datum feature (unless otherwise indicated).

The datum is the situation feature for the minimum circumscribing sphere, which is its centre point.

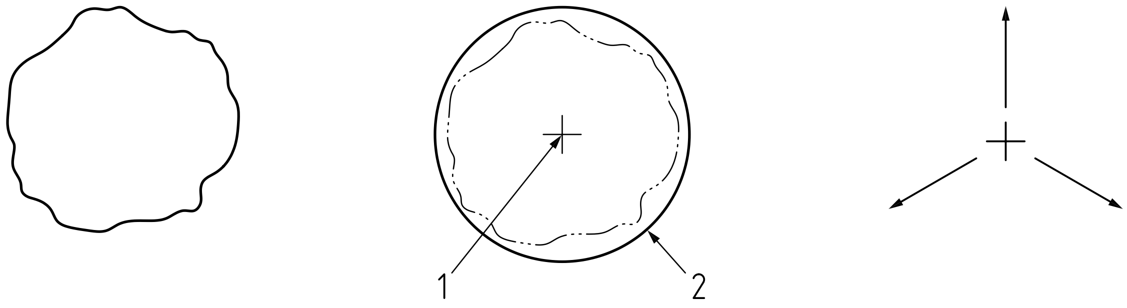
A primary datum based on the spherical datum feature is a point, derived from the minimum circumscribing sphere constructed with no additional constraints. This can control three degrees of freedom. (See Figure C.6).

A secondary datum based on the datum feature is a point, derived from the minimum circumscribing sphere, constructed with location constraints to the primary datum (a sphere cannot be orientated).

A tertiary datum based on the datum feature is a point, derived from the minimum circumscribing sphere, constructed with location constraints to the primary and secondary datums.

Figure C.6 – ***Primary datum based on an external spherical feature***

Real datum feature | **Associated feature** | **Degrees of freedom that can be controlled**



a) The datum feature is an external sphere. The real datum feature is the manufactured surface. The extracted datum feature is the data set obtained by sampling the surface.

b) The associated feature is the minimum circumscribing sphere. The situation feature for the associated feature is its centre point.

c) This datum can constrain three degrees of freedom:

- translation along each of three mutually perpendicular axes which have their origin located at the centre point of the associated feature.

Key

1 Centre point

2 Associated feature

C.4.7 Datum based on an internal spherical feature

The datum feature is identified on the TPD as a nominally spherical internal feature.

The real datum feature on the manufactured workpiece is a non-ideal sphere.

The associated feature is a maximum inscribing sphere constructed within the datum feature (unless otherwise indicated).

The datum is the situation feature for the maximum inscribing sphere, which is its centre point.

A primary datum based on the spherical datum feature is a point derived from the maximum inscribing sphere with no additional constraints. This can control three degrees of freedom. (See Figure C.6).

A secondary datum based on the datum feature is a point derived from the maximum inscribing sphere constructed with location constraints to the primary datum (a centre point cannot be orientated).

A tertiary datum based on the datum feature is a point derived from the maximum inscribing sphere constructed with location constraints to the primary and secondary datums.

C.4.8 Datum based on an external or internal cone

The datum feature is identified on the TPD as a nominally conical feature.

The real datum feature on the manufactured workpiece is a non-ideal conical surface.

The associated feature is a minimum circumscribing cone constructed around an external cone or a maximum inscribing cone constructed within an internal cone (unless otherwise indicated).

The datum is the situation feature for the associated cone, which is its centreline and a point.

A primary datum based on the conical datum feature is a centreline and a point derived from an associated cone constructed with no additional constraints. This can control five degrees of freedom (see Figure C.7).

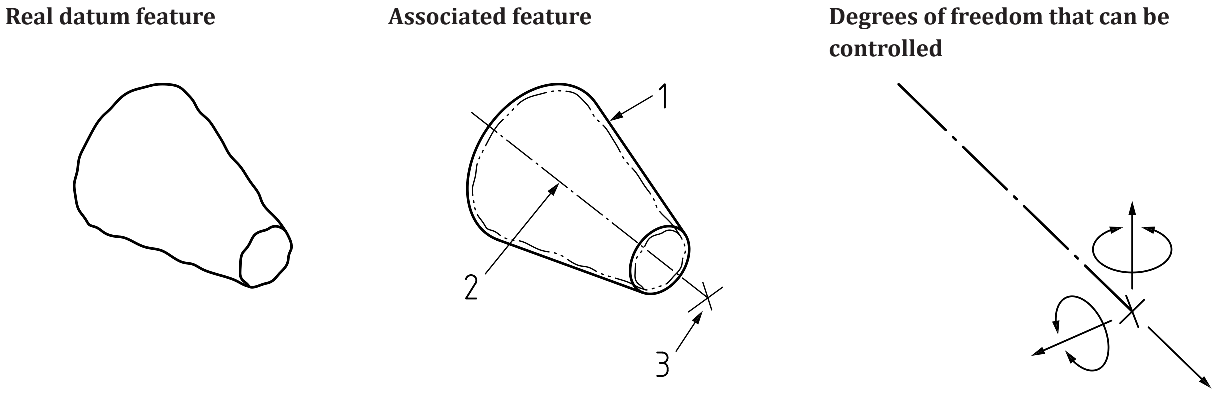
A secondary datum based on the conical datum feature is a centreline and a point derived from an associated cone constructed with location and orientation constraints to the primary datum.

A tertiary datum based on the conical datum feature is a centreline and a point derived from an associated cone constructed with location and orientation constraints to the primary and secondary datums.

The point situation feature needs to lie on the axis situation feature but has no unique location on that axis. The point can be taken to be at the apex of the cone or at any another specified position along the axis.

NOTE If the cone is defined with a theoretically exact angle, the angle of the associated feature is fixed at the same angle. If the cone is defined with a tolerance applied to the cone angle, the angle of the associated feature is variable.

Figure C.7 – *Primary datum feature based on a conical feature*



Real datum feature

Associated feature

Degrees of freedom that can be controlled

a) The datum feature is a cone. The real datum feature is the manufactured surface. The extracted datum feature is the data set obtained by sampling the surface.

b) The associated feature is the minimum circumscribing or maximum inscribing cone. The situation feature for the associated feature is its centreline and a point.

c) This datum can constrain five degrees of freedom:

- translation along all three axes; and
- rotation about two perpendicular axes which are also perpendicular to the axis of the associated feature.

Key

1	Associated feature	3	Point
2	Axis		

C.4.9 Datum based on an external or internal wedge

The datum feature is identified on the TPD as two planes which are not parallel.

The real datum feature on the manufactured workpiece is a non-ideal wedge.

The associated feature is a minimum circumscribing wedge constructed around an external feature or a maximum inscribing wedge constructed within an internal feature (unless otherwise indicated).

The datum is the situation feature for the associated wedge, which is its centre-plane and a line.

A primary datum based on the wedge datum feature is a centre-plane and a line derived from an associated wedge constructed with no additional constraints. This can control five degrees of freedom (see Figure C.8).

A secondary datum based on the wedge datum feature is a centre-plane and a line derived from an associated wedge constructed with location and orientation constraints to the primary datum.

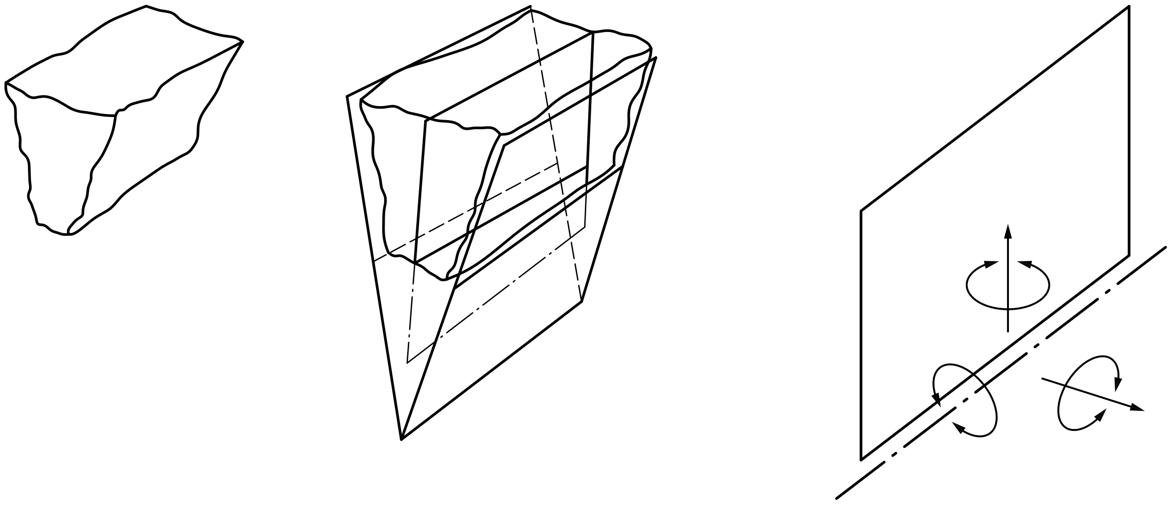
A tertiary datum based on the wedge datum feature is a centre-plane and a line derived from an associated wedge constructed with location and orientation constraints to the primary and secondary datums.

The line situation feature needs to lie within the plane situation feature and be parallel to the apex of the wedge but has no unique location on that plane. The line can be taken to be at the apex of the wedge or at any another specified position within the plane and parallel to the apex.

NOTE If the wedge is defined with a theoretically exact angle, the angle between the planes of the associated feature is fixed at the same angle. If the wedge is defined with a toleranced value for the angle, the angle between the planes of the associated feature is variable.

Figure C.8 – *Primary datum based on a wedge*

Real datum feature **Associated feature** **Degrees of freedom that can be controlled**



a) The datum feature is a wedge. The real datum feature is the manufactured surface. The extracted datum feature is the data set obtained by sampling the surface.

b) The associated feature is the minimum circumscribing or maximum inscribing wedge. The situation feature for the associated feature is its centre-plane and a line.

c) This datum can constrain five degrees of freedom:

- translation along an axis perpendicular to the median plane, and translation along an axis within the median plane but perpendicular to the situation feature axis; and
- rotation about all three axes.

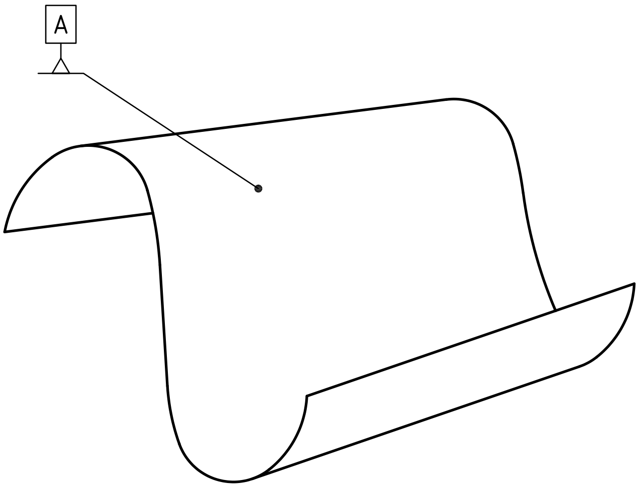
Key

1	Associated feature	3	Axis
2	Median plane		

C.4.10 Prismatic feature

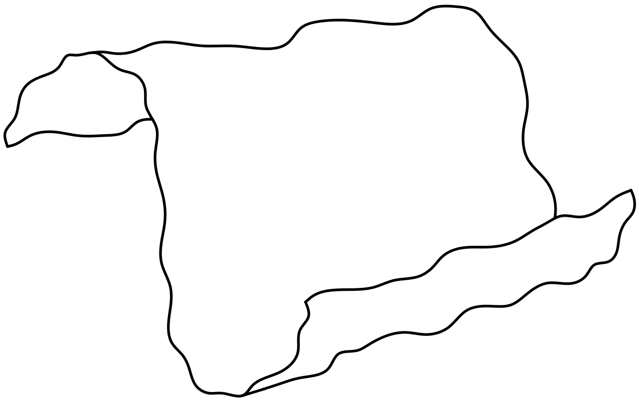
The datum feature is identified on the TPD as a prismatic surface (see Figure C.9).

Figure C.9 – ***Datum for prismatic feature***



The real datum feature on the manufactured workpiece is a non-ideal surface (see Figure C.10).

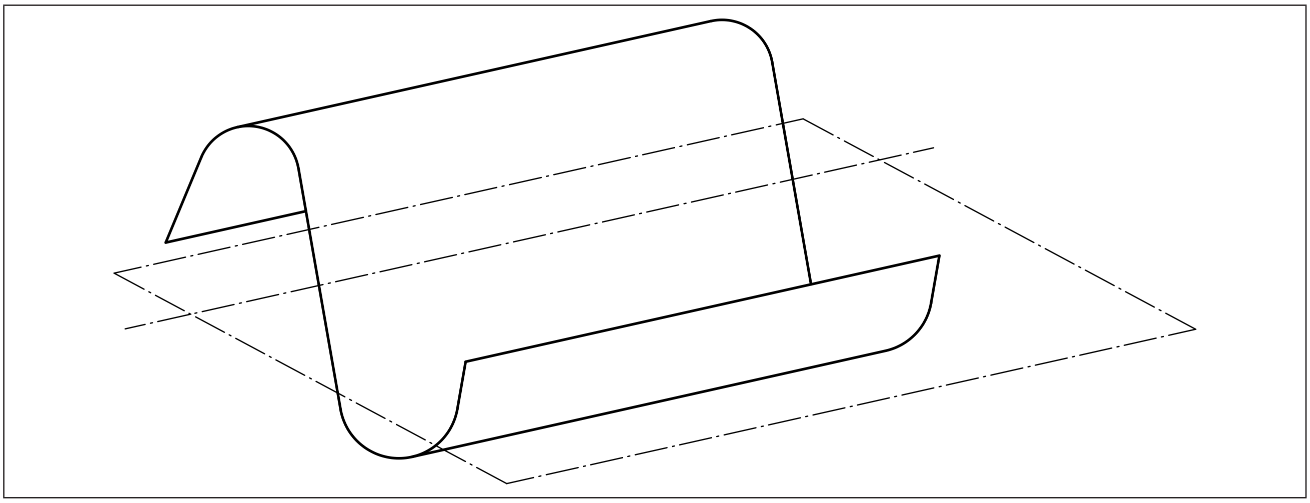
Figure C.10 – ***Non–ideal prismatic surface***



The associated feature is an ideal version of the prismatic surface, constructed to be in contact with the real datum feature, outside the material, and arranged such that the maximum distances between the associated feature and the real surface are minimized.

The datum is the situation feature for the associated prismatic surface, which is a plane and a line (see Figure C.11).

Figure C.11 – *Situation features for prismatic surface*



A primary datum based on the prismatic datum feature is a plane and a line based on an associated feature constructed with no additional constraints. This can control five degrees of freedom (see Figure C.12).

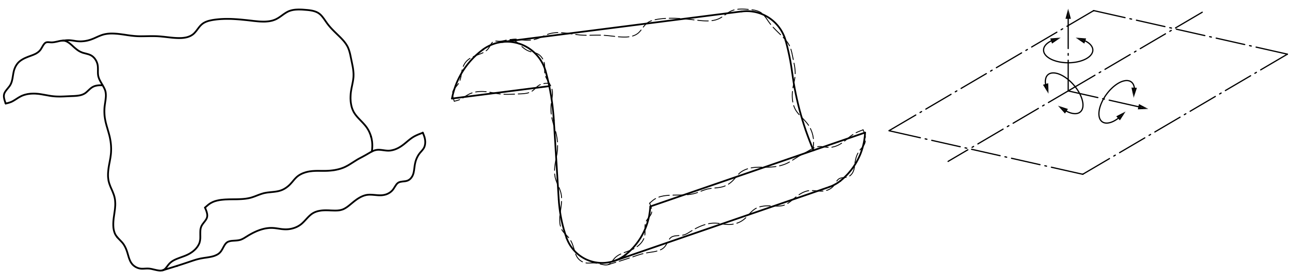
A secondary datum based on the prismatic datum feature is a plane and a line based on an associated feature constructed with location and orientation constraints to the primary datum.

A tertiary datum based on the prismatic datum feature is a plane and a line based on an associated feature constructed with location and orientation constraints to the primary and secondary datums.

The plane situation feature may have no unique location, although it needs to be aligned with the "direction of extrusion" of the prismatic feature. The line situation feature needs to lie within the plane situation feature and be aligned with the "direction of extrusion" of the prismatic feature but has no unique location on that plane. The line can be taken to be at any specified location within the plane and parallel to the direction of extrusion.

NOTE For a prismatic feature like the one shown in Figure C.12, a Gaussian (least squares) association with no material constraint might make more sense.

Figure C.12 – ***Primary datum based on a prismatic surface***



Real datum feature

a) The datum feature is a prismatic surface. The real datum feature is the manufactured surface. The extracted datum feature is the data set obtained by sampling the surface.

Associated feature amend figure to show situation features

b) By default, the associated feature is a minimax associated surface, but a least-squares associated surface might well make more sense. The situation feature for the associated feature is a plane and a line.

Degrees of freedom that can be controlled

c) This datum can constrain five degrees of freedom:

- translation along an axis perpendicular to the situation feature plane, and translation along an axis within the situation feature plane but perpendicular to the situation feature line; and
- rotation about all three axes.

Key

1	Associated feature	3	Axis
2	Median plane		

C.4.11 Complex surface

The datum feature is identified on the TPD as a complex surface.

The real datum feature on the manufactured workpiece is a non-ideal surface.

The associated feature is an ideal version of the complex surface, constructed to be in contact with the real datum feature, outside the material, and arranged such that the maximum distances between the associated feature and the real surface are minimized.

The datum is the situation feature for the associated complex surface, which is a plane, a line and a point.

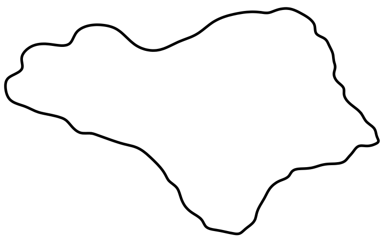
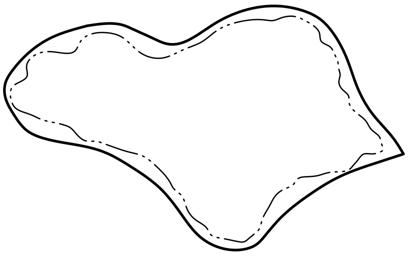
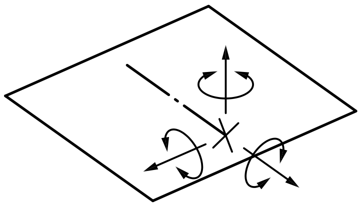
A primary datum based on the complex datum feature is a plane, a line and a point based on an associated feature constructed with no additional constraints. This can control six degrees of freedom (see Figure C.13).

A complex surface is not normally used as a secondary or tertiary datum feature.

The plane situation feature may have no unique location at all, so it may be placed at the discretion of the specifier. The line situation feature needs to lie within the plane situation feature, but has no unique location on that plane, so may be placed at any defined location within that plane. The point situation feature needs to lie on the line situation feature, but has no unique location on that line, so may be placed at any defined point on that line.

NOTE For a complex feature like the one shown, a Gaussian (least squares) association with no material constraint may make more sense.

Figure C.13 – *Primary datum based on a complex surface*

Real datum feature	**Associated feature amend figure to show situation features**	**Degrees of freedom that can be controlled**
		
a) The datum feature is a complex surface. The real datum feature is the manufactured surface. The extracted datum feature is the data set obtained by sampling the surface.	b) By default, the associated feature is a minimax associated surface, but a least-squares associated surface might well make more sense. The situation feature for the associated feature is a plane, a line and a point.	c) This datum can constrain six degrees of freedom: • translation along all three axes; and • rotation about all three axes.

Annex D (normative)
Document security – Enhanced

D.1 Enhanced security

Where requirements for enhanced security are known to exist, the procedures identified in this annex shall be applied in addition to those specified in 14.2.3.

D.2 Identification of security classification

Any required security classification and/or caveat shall be inserted in the TPS immediately after classified information is incorporated.

Each sheet shall be classified according to its content.

The security classification shall always appear at the top and bottom of A4 sheets and at the top left and bottom right-hand corners of sheets larger than A4.

The security classification shall either be:

a) larger than the largest text used in the TPS; or

b) bolder and the same size as the largest text used in the TPS.

D.3 Marking for enhanced security

Technical product document sets, prepared in accordance with the requirements of D.1 and D.2, shall be identified by the addition of the suffix "/D" to the number of this standard, i.e. "BS 8888/D", in a prominent location.

Annex E (informative)
Implementation in CAD

E.1 General

Most CAD software:

- is limited in its ability to implement the full requirements of ISO standards;
- does not correctly distinguish between datums and datum features; and
- is not currently capable of dealing with datums consisting of more than one situation feature.

To implement these concepts in CAD, it might be necessary to deviate from the rules and methods specified in BS 8888 and ISO standards.

The following subclause does not replace or override the requirements of BS 8888 and ISO standards; they are simply offered to assist with the practical implementation of some of the datum requirements described in this section.

E.2 Implementation according to BS 8888 and BS EN ISO 5459

In Figure E.1:

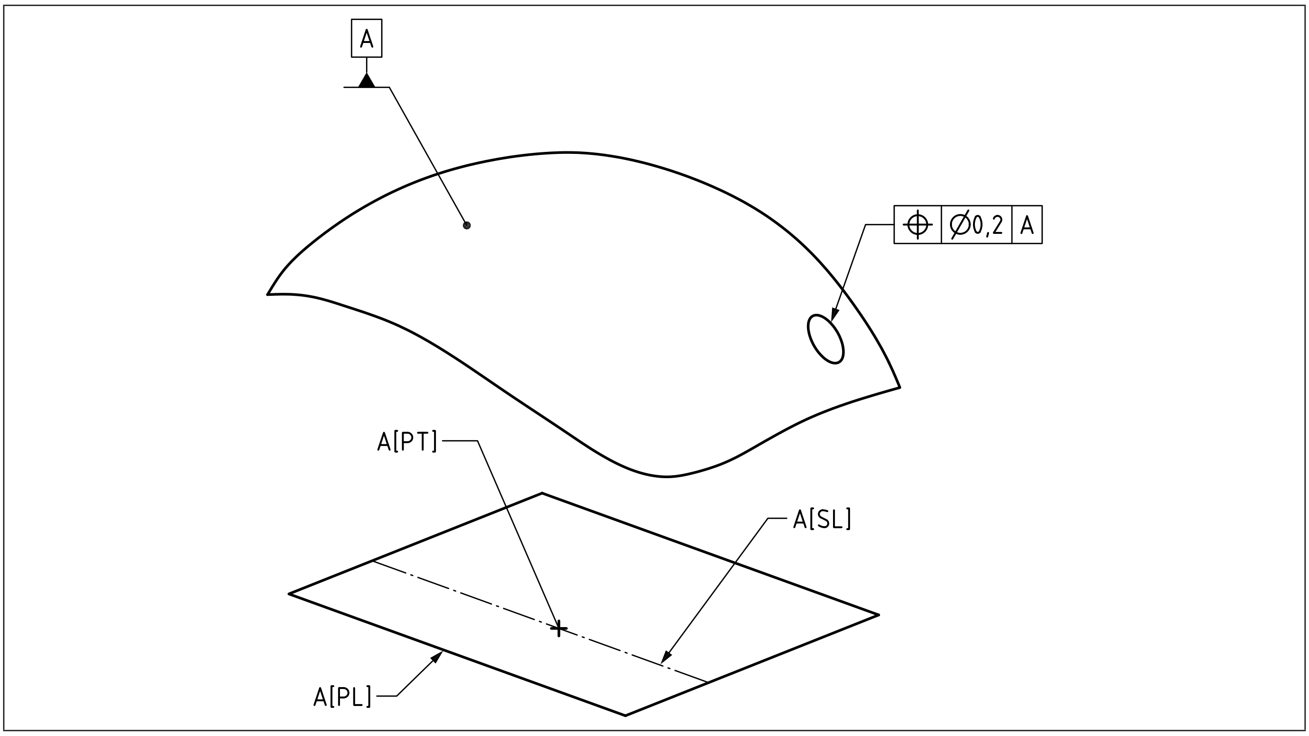
a) datum A is a single datum;

b) the datum feature is a complex surface;

c) datum A consists of a plane, line and point; and

d) if a position tolerance is applied to a hole in the part, a reference to datum A is sufficient to control all six degrees of freedom for the tolerance zone.

Figure E.1 – *Implementation according to BS 8888 and BS EN ISO 5459*



E.3 Implementation according to most CAD software

Where necessary, each situation feature can be treated as a separate datum. For example, in Figure E.2:

a) the datum system consists of three mutually perpendicular planes, each of which has a separate datum name;

b) the datum feature is the complex surface.

Because the CAD system has been used to allocate datum names individually to the three planes, it is not usually possible to use a datum feature identifier on the complex surface, and it shall be labelled with a note instead.

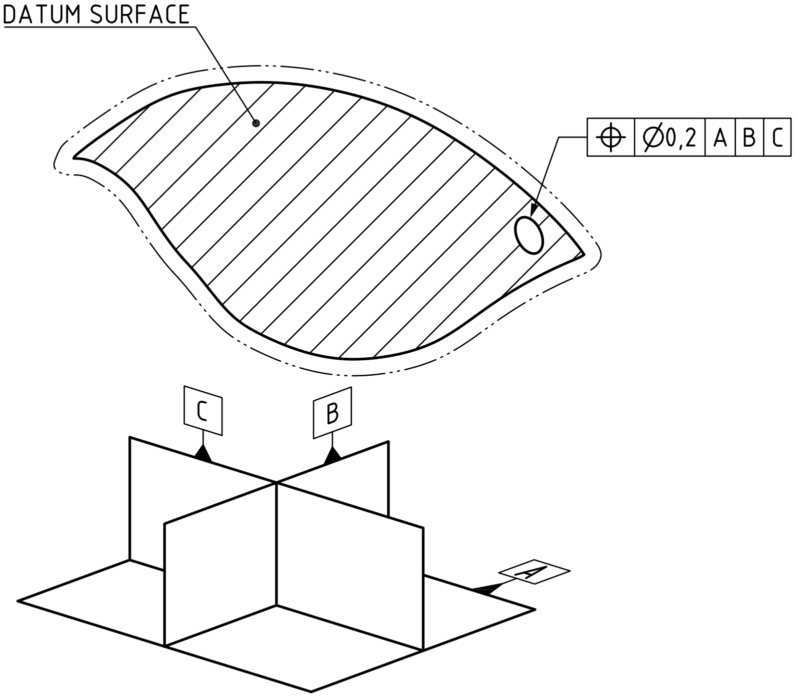
It might be necessary to highlight the datum feature surfaces in some way to clearly distinguish them.

It might also be necessary to clarify the "best fit" method to be used with a note.

EXAMPLE:

DATUM SYSTEM A|B|C IS DEFINED USING LEAST SQUARES BEST FIT TO DATUM SURFACE INDICATED.

Figure E.2 – *Implementation according to most CAD software*

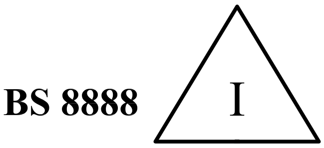


Annex F (informative)
Tolerancing system: Former practice

It was former practice to mark a TPS with the indication "BS 8888", supplemented by the letter "I" contained within an equilateral triangle (see Figure F.1) or the letter "D" contained within an equilateral triangle (see Figure F.2).

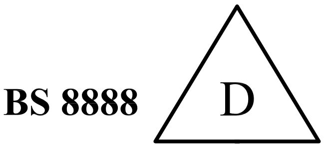
The triangle "I" symbol (see Figure F.1) was taken to indicate that the principle of independency was to be used to govern the interpretation of size and form requirements. While this had the same meaning as "TOLERANCING ISO 8015", its meaning might not be apparent to an interpreter who was familiar with ISO standards, but not BS 8888, so its use is no longer recommended.

Figure F.1 – ***BS 8888 independency system symbol***



The triangle "D" symbol (see Figure F.2) was taken to indicate that the principle of dependency was to be used to govern the interpretation of size and form requirements. While this was included to maintain consistency with earlier versions of BS 8888 and BS 308, the interpretation of the principle of dependency as a general requirement is not fully defined within the ISO system, so there might be ambiguities in its interpretation. In view of such possible ambiguities, and the fact that the symbol might not be understood by interpreters who were familiar with the ISO system but not BS 8888, the use of this symbol is no longer recommended.

Figure F.2 – ***BS 8888 dependency system symbol***



Annex G (informative)
Selected ISO fits: Hole and shaft basis

Table G.1 gives selected ISO fits for holes and shafts.

Table G.1 – *Selected ISO fits: Hole basis*

		Clearance fits												Transition fits				Interference fits					
Nominal sizes		**Tolerances**		**Tolerances**		**Tolerances**		**Tolerances**		**Tolerances**		**Tolerances**		**Tolerances**		**Tolerances**		**Tolerances**		**Tolerances**		**Nominal sizes**	
Over	To	H11	c11	H9	d10	H9	e9	H8	f7	H7	g6	H7	h6	H7	k6	H7	n6	H7	p6	H7	s6	Over	To
mm	mm	0,001 mm		0,001 mm		0,001 mm		0,001 mm		0,001 mm		0,001 mm		0,001 mm		0,001 mm		0,001 mm		0,001 mm		mm	mm
–	3	+60 0	−60 −120	+25 0	−20 −60	+25 0	−14 −39	+14 0	−6 −16	+10 0	−2 −8	+10 0	0 −6	+10 0	+6 0	+10 0	+10 +4	+10 0	+12 +6	+10 0	+20 +14	–	3
3	6	+75 0	−70 −145	+30 0	−30 −78	+30 0	−20 −50	+18 0	−10 −22	+12 0	−4 −12	+12 0	0 -8	+12 0	+9 +1	+12 0	+16 +18	+12 0	+20 +12	+12 0	+27 +19	3	6
6	10	+90 0	−80 −170	+36 0	−40 −98	+36 0	−25 −61	+22 0	−13 −28	+15 0	−5 −14	+15 0	0 -9	+15 0	+10 +1	+15 0	+19 +10	+15 0	+24 +15	+15 0	+32 +23	6	10
10	18	+110 0	−95 −205	+43 0	−50 −120	+43 0	−32 −75	+27 0	−16 −34	+18 0	−6 −17	+18 0	0 −11	+18 0	+12 +1	+18 0	+23 +12	+18 0	+29 +18	+18 0	+39 +28	10	18
18	30	+130 0	−110 −240	+52 0	−65 −149	+52 0	−40 −92	+33 0	−20 −41	+21 0	−7 −20	+21 0	0 −13	+21 0	+15 +2	+21 0	+28 +15	+21 0	+35 +22	+21 0	+48 +35	18	30
30	40	+160 0	−120 −280	+62 0	−80 −180	+62 0	−50 −112	+39 0	−25 −50	+25 0	−9 −25	+25 0	0 −16	+25 0	+18 +2	+25 0	+33 +17	+25 0	+42 +26	+25 0	+59 +43	30	40
40	50		−130 −290																			40	50
50	65	+190 0	−140 −330	+74 0	−100 −220	+74 0	−60 −134	+46 0	−30 −60	+30 0	−10 −29	+30 0	0 −19	+30 0	+21 +2	+30 0	+39 +20	+30 0	+51 +32	+30 0	+72 +53	50	65
65	80		−150 −340																		+78 +59	65	80
80	100	+220 0	−170 −390	+87 0	−120 −260	+87 0	−72 −159	+54 0	−36 −71	+35 0	−12 −34	+35 0	0 −22	+35 0	+25 +3	+35 0	+45 +23	+35 0	+59 +37	+35 0	+93 +71	80	100
100	120		−180 −400																		+101 +79	100	120

(Continues)

Table G.1 – *Selected ISO fits: Hole basis* *(Cont.)*

		Clearance fits												Transition fits				Interference fits					
Nominal sizes		**Tolerances**		**Tolerances**		**Tolerances**		**Tolerances**		**Tolerances**		**Tolerances**		**Tolerances**		**Tolerances**		**Tolerances**		**Tolerances**		**Nominal sizes**	
Over	To	H11	c11	H9	d10	H9	e9	H8	f7	H7	g6	H7	h6	H7	k6	H7	n6	H7	p6	H7	s6	Over	To
mm	mm	0,001 mm		0,001 mm		0,001 mm		0,001 mm		0,001 mm		0,001 mm		0,001 mm		0,001 mm		0,001 mm		0,001 mm		mm	mm
120	140	+250 0	−200 −450	+100 0	−145 −305	+100 0	−85 −185	+63 0	−43 −83	+40 0	−14 −39	+40 0	0 −25	+40 0	+28 +3	+40 0	+52 +27	+40 0	+68 +43	+40 0	+117 +92	120	140
140	160		−210 −460																		+125 +100	140	160
160	180		−230 −480																		+133 +108	160	180
180	200	+290 0	−240 −530	+115 0	−170 −355	+115 0	−100 −215	+72 0	−50 −96	+46 0	−15 −44	+46 0	0 −29	+46 0	+33 +4	+46 0	+60 +31	+46 0	+79 +50	+46 0	+151 +122	180	200
200	225		−260 −550																		+159 +130	200	225
225	250		−280 −570																		+169 +140	225	250
250	280	+320 0	−300 −620	+130 0	−190 −400	+130 0	−110 −240	+81 0	−56 −108	+52 0	−17 −49	+52 0	0 −32	+52 0	+36 +4	+52 0	+66 +34	+52 0	+88 +56	+52 0	+190 +158	250	280
280	315		−330 −650																		+202 +170	280	315
315	355	+360 0	−360 −720	+140 0	−210 −440	+140 0	−125 −265	+89 0	−62 −119	+57 0	−18 −54	+57 0	0 −36	+57 0	+40 +4	+57 0	+73 +37	+57 0	+98 +62	+57 0	+226 +190	315	355
355	400		−400 −760																		+244 +208	355	400
400	450	+400 0	−440 −840	+155 0	−230 −480	+155 0	−135 −290	+97 0	−68 −131	+63 0	−20 −60	+63 0	0 −40	+63 0	+45 +5	+63 0	+80 +40	+63 0	+108 +68	+63 0	+272 +232	400	450
450	500		−480 −880																		+292 +252	450	500

Figure G.2 – *Selected ISO fits: Shaft basis*

		Clearance fits												Transition fits				Interference fits					
Nominal sizes		Tolerances		Tolerances		Tolerances		Tolerances		Tolerances		Tolerances		Tolerances		Tolerances		Tolerances		Tolerances		Nominal sizes	
Over	To	h11	C11	h9	D10	h9	E9	h7	F8	h6	G7	h6	H7	h6	K7	h6	N7	h6	P7	h6	S7	Over	To
mm	mm	0,001 mm		0,001 mm		0,001 mm		0,001 mm		0,001 mm		0,001 mm		0,001 mm		0,001 mm		0,001 mm		0,001 mm		mm	mm
–	3	0 −60	+120 +60	0 −25	+60 +20	0 −25	+39 +14	0 −10	+20 +6	0 −6	+12 +2	0 −6	+10 0	0 −6	0 −10	0 −6	−4 -14	0 −6	−6 −16	0 −6	−14 −24	–	3
3	6	0 −75	+145 +70	0 −30	+78 +30	0 −30	+50 +20	0 −12	+28 +10	0 −8	+16 +4	0 −8	+12 0	0 −8	+3 −9	0 −8	−4 -16	0 −8	−8 −20	0 −8	−15 −27	3	6
6	10	0 −90	+170 +80	0 −36	+98 +40	0 −36	+61 +25	0 −15	+35 +13	0 −9	+20 +5	0 −9	+15 0	0 −9	+5 −10	0 −9	−4 -19	0 −9	−9 −24	0 −9	−17 −32	6	10
10	18	0 −110	+205 +95	0 −43	+120 +50	0 −43	+75 +32	0 −18	+43 +16	0 −11	+24 +6	0 −11	+18 0	0 −11	+6 −12	0 −11	−5 -23	0 −11	−11 −29	0 −11	−21 −39	10	18
18	30	0 −130	+240 +110	0 −52	+149 +65	0 −52	+92 +40	0 −21	+53 +20	0 −13	+28 +7	0 −13	+21 0	0 −13	+6 −15	0 −13	−7 -28	0 −13	−14 −35	0 −13	−27 −48	18	30
30	40	0 −160	+280 +120	0 −62	+180 +80	0 −62	+112 +50	0 −25	+64 +25	0 −16	+34 +9	0 −16	+25 0	0 −16	+7 −18	0 −16	-8 -33	0 −16	−17 −42	0 −16	−34 −59	30	40
40	50		+290 +130																			40	50
50	65	0 −190	+330 +140	0 −74	+220 +100	0 −74	+134 +60	0 −30	+76 +30	0 −19	+40 +10	0 −19	+30 0	0 −19	+9 −21	0 −19	−9 -39	0 −19	−21 −51	0 −19	−42 −72	50	65
65	80		+340 +150																		−48 −78	65	80
80	100	0 −220	+390 +170	0 −87	+260 +120	0 −87	+159 +72	0 −35	+90 +36	0 −22	+47 +12	0 −22	+35 0	0 −22	+10 −25	0 −22	−10 -45	0 −22	−24 −59	0 −22	−58 −93	80	100
100	120		+400 +180																		−66 −101	100	120

(Continues)

Figure G.2 – ***Selected ISO fits: Shaft basis*** ***(Cont.)***

		Clearance fits												Transition fits				Interference fits					
Nominal sizes		Tolerances		Tolerances		Tolerances		Tolerances		Tolerances		Tolerances		Tolerances		Tolerances		Tolerances		Tolerances		Nominal sizes	
Over	To	h11	C11	h9	D10	h9	E9	h7	F8	h6	G7	h6	H7	h6	K7	h6	N7	h6	P7	h6	S7	Over	To
mm	mm	0,001 mm		0,001 mm		0,001 mm		0,001 mm		0,001 mm		0,001 mm		0,001 mm		0,001 mm		0,001 mm		0,001 mm		mm	mm
120	140	0 −250	+450 +200	0 −100	+305 +145	0 −100	+185 +85	0 −40	+106 +43	0 −25	+54 +14	0 −25	+40 0	0 −25	+12 −28	0 −25	−12 -52	0 −25	−28 −68	0 −25	−77 −117	120	140
140	160		+460 +210																		−85 −125	140	160
160	180		+480 +230																		−93 −133	160	180
180	200	0 −290	+530 +240	0 −115	+355 +170	0 −115	+215 +100	0 −46	+122 +50	0 −29	+61 +15	0 −29	+46 0	0 −29	+13 −33	0 −29	−14 -60	0 −29	−33 −79	0 −29	−105 −151	180	200
200	225		+550 +260																		−113 −159	200	225
225	250		+570 +280																		−123 −169	225	250
250	280	0 −320	+620 +300	0 −130	+400 +190	0 −130	+240 +110	0 −52	+137 +56	0 −32	+62 +17	0 −32	+52 0	0 −32	+16 −36	0 −32	−14 -66	0 −32	−36 −88	0 −32	−138 −190	250	280
280	315		+650 +330																		−150 −202	280	315
315	355	0 −360	+720 +360	0 −140	+440 +210	0 −140	+265 +125	0 −57	+151 +62	0 −36	+75 +18	0 −36	+57 0	0 −36	+17 −40	0 −36	−16 -73	0 −36	−41 −98	0 −36	−169 −226	315	355
355	400		+760 +400																		−187 −244	355	400
400	450	0 −400	+840 +440	0 −155	+480 +230	0 −155	+290 +135	0 −63	+165 +68	0 −40	+83 +20	0 −40	+63 0	0 −40	+18 −45	0 −40	−17 -80	0 −40	−45 −108	0 −40	−209 −272	400	450
450	500		+880 +480																		−229 −292	450	500

Annex H (informative)
Filtration

H.1 Introduction

Filters can be used to isolate a specific range of structure on a feature to which a specification is applied. Filters are usually applied digitally, but in some cases may be applied mechanically. Filters can also be used to reject data which are not relevant to the properties under consideration

The practice of defining filters for surface texture definitions is well established, and default rules for filters for surface texture are given in BS EN ISO 4288. The possibility of defining filter properties for geometrical tolerances of form, such as flatness and roundness, has existed since 2003, when the ISO form standards BS EN ISO 12180, BS EN ISO 12181, BS EN ISO 12780 and BS EN ISO 12781 first appeared. BS EN ISO 1101 now provides the tools for applying filters to any geometrical specification.

Filters are usually applied to the frequency, or wavelength, of the variations in the feature in order to select a specific range, but may also be used to eliminate outliers or noise in data.

H.2 Filter types

Many different types of filters are available, and these are documented in the BS EN ISO 16610 series of standards, and listed in Annex C of BS EN ISO 1101:2017

Filers can be classified as long-wave pass, short-wave pass and band pass:

1) **Long-wave pass filters**

   Long-wave pass filters allow longer wavelength properties to pass through and filter out the shorter wavelength properties. This is probably the most common type of filter.

2) **Short-wave pass filters**

   Short-wave pass filters allow shorter wavelength properties to pass through and filter out the longer wavelength properties.

3) **Band pass filters**

   Band pass filters have longer and shorter wavelength limits, allowing a specified range of wavelengths through.

Another way of classifying filters is as profile and areal filters:

a) **Profile filters**

   Profile filters are applied to a two-dimensional profile.

b) **Areal filters**

   Areal filters are applied to three-dimensional areas.

H.3 Filters and nesting indices

When a filter is defined, there are two components: one is the type of filter, and the other is the parameter (or parameters) required for that type of filter.

An analogy sometimes used is that of a sieve, used to filter out objects larger than the holes in the sieve. The type of filter is the type of sieve; some have round holes, some have square holes, and some may have slots arranged in various ways. The parameters associated with the filter define the properties of the aperture (such as the size of the hole or the width of the slot).

The parameter associated with a filter (in the analogy, the size of the aperture in the sieve) is referred to as the “nesting index”. As the nesting index approaches zero, the size of the aperture approaches zero and nothing can get through, so all data representing the surface are captured, and nothing is filtered out.

When a filter is applied to an “open” feature, a feature which terminates at an edge or a point, such as a line element or plane, the nesting index is defined in millimetres.

When a filter is applied to a “closed” surface, a feature which is continuous, such as a circular line element, or a cylinder in the circumferential direction, the nesting index is defined in “undulations per revolution”, or UPR.

In the case of a Morphological filter, the nesting index is the size of the structuring element.

H.4 Default filters

No default filters are currently defined for geometrical specifications. In the absence of a specific filter being indicated, it can be assumed that the tolerance applies to the feature without any filtration.

In the next revision of BS EN ISO 5459 (datums and datum systems), a default filter will be specified for use when defining datums.

H.5 Examples

The use of filters with geometrical specifications is new, and guidance and examples are relatively sparse. However, a range of examples is given in Annex D of BS EN ISO 1101:2017.

Annex I (informative) Dimensioning of slots

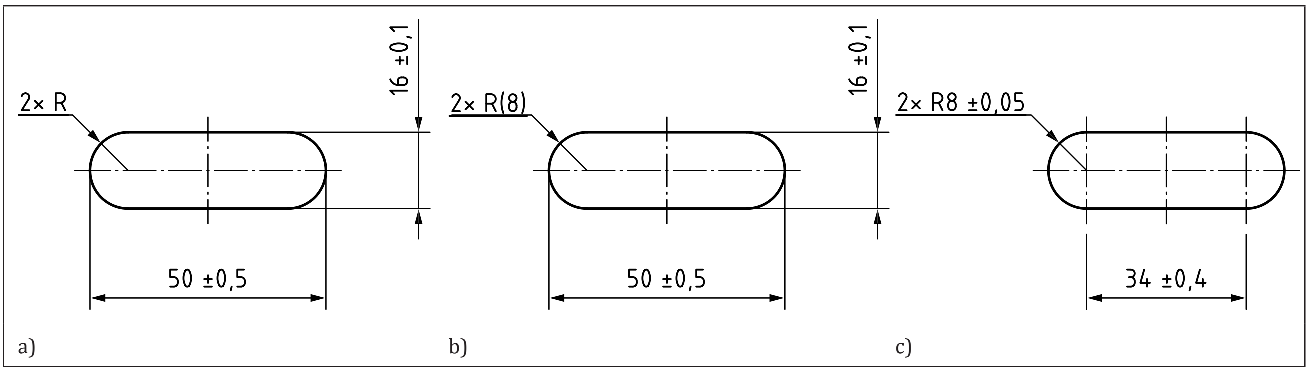
*COMMENTARY ON **ANNEX I***

This annex describes how slots can be dimensioned and toleranced. The guidance is applicable in the case of a protrusion with the same profile.

I.1 Defining the size of the slot

When dimensioning slots with rounded ends, it is acceptable to define the length of the slot with a dimension tangent to the ends [see Figure I.1a) and b)], or with a dimension between the centres of the two end radii [see Figure I.1c)].

Figure I.1 – *Dimensioning slots with rounded ends*



The width of the slot can be defined with a dimension defining the distance between the parallel sides of the slot, or with a dimension defining the radius of the ends.

When the width of the slot is defined with a dimension between the parallel sides of the slot, the radius of the end can be indicated with a radius arrow and the symbol R without a dimension value, or alternatively as an auxiliary dimensions, e.g. R(8) [see Figure I.1b)].

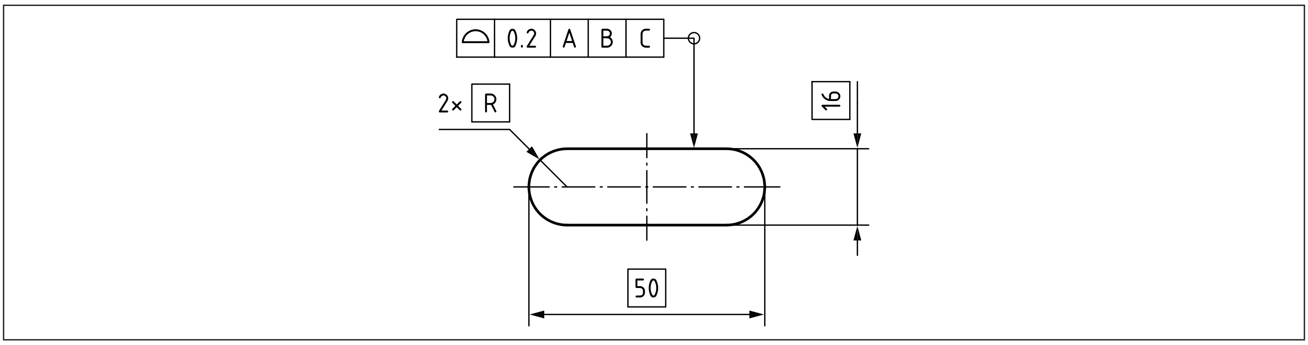
When the width of the slot is defined with a dimension defining the radius of the ends of the slot [see Figure I.1a)], the tolerance on the width of the slot will be double the tolerance on the radius.

The dimensioning scheme selected should be the one that best represents the functional requirements for the feature.

In Figure I.1a) and b), the radius has no dimension or tolerance defined. It is assumed that the radius adjusts to match the width of the slot, but there is no direct control over the radius geometry. If the geometrical properties of the radii are functionally significant, TEDs and a profile tolerance should be used (see Figure I.2).

NOTE In this example, the surface profile tolerance is controlling the size of the slot and the location of the slot at the same time.

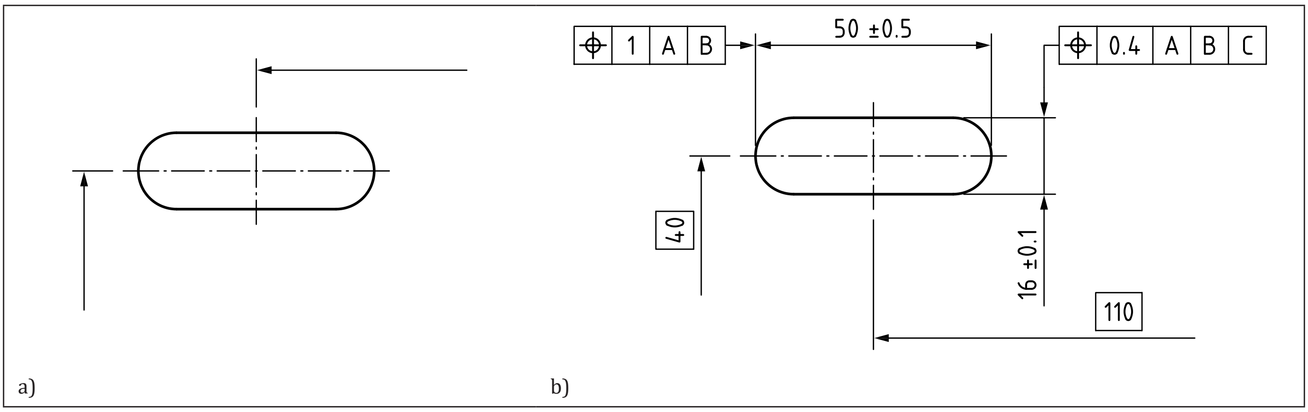
Figure I.2 – ***Profile tolerance***



I.2 Defining the location of the slot

When dimensions are used to locate the slot, they can be applied to the centre of the slot as shown in Figure I.3a) and b). Dimensions with +/- tolerances can be used to locate the slot [see Figure I.3a)], but the shortcomings of this approach are described in BS EN ISO 14405–2. The use of TEDs and surface profile tolerances [see Figure I.2], or TEDs and position tolerances [see Figure 3b)] provides a less ambiguous definition.

Figure I.3 – ***Defining the location of a slot***



a) b)

British Standards Institution (BSI)

BSI is the national body responsible for preparing British Standards and other standards-related publications, information and services.

BSI is incorporated by Royal Charter. British Standards and other standardization products are published by BSI Standards Limited.

About us

We bring together business, industry, government, consumers, innovators and others to shape their combined experience and expertise into standards -based solutions.

The knowledge embodied in our standards has been carefully assembled in a dependable format and refined through our open consultation process. Organizations of all sizes and across all sectors choose standards to help them achieve their goals.

Information on standards

We can provide you with the knowledge that your organization needs to succeed. Find out more about British Standards by visiting our website at bsigroup.com/standards or contacting our Customer Services team or Knowledge Centre.

Buying standards

You can buy and download PDF versions of BSI publications, including British and adopted European and international standards, through our website at bsigroup.com/shop, where hard copies can also be purchased.

If you need international and foreign standards from other Standards Development Organizations, hard copies can be ordered from our Customer Services team.

Copyright in BSI publications

All the content in BSI publications, including British Standards, is the property of and copyrighted by BSI or some person or entity that owns copyright in the information used (such as the international standardization bodies) and has formally licensed such information to BSI for commercial publication and use.

Save for the provisions below, you may not transfer, share or disseminate any portion of the standard to any other person. You may not adapt, distribute, commercially exploit or publicly display the standard or any portion thereof in any manner whatsoever without BSI's prior written consent.

Storing and using standards

Standards purchased in soft copy format:

- A British Standard purchased in soft copy format is licensed to a sole named user for personal or internal company use only.
- The standard may be stored on more than one device provided that it is accessible by the sole named user only and that only one copy is accessed at any one time.
- A single paper copy may be printed for personal or internal company use only.

Standards purchased in hard copy format:

- A British Standard purchased in hard copy format is for personal or internal company use only.
- It may not be further reproduced – in any format – to create an additional copy. This includes scanning of the document.

If you need more than one copy of the document, or if you wish to share the document on an internal network, you can save money by choosing a subscription product (see 'Subscriptions').

Reproducing extracts

For permission to reproduce content from BSI publications contact the BSI Copyright and Licensing team.

Subscriptions

Our range of subscription services are designed to make using standards easier for you. For further information on our subscription products go to bsigroup.com/subscriptions.

With **British Standards Online (BSOL)** you'll have instant access to over 55,000 British and adopted European and international standards from your desktop. It's available 24/7 and is refreshed daily so you'll always be up to date.

You can keep in touch with standards developments and receive substantial discounts on the purchase price of standards, both in single copy and subscription format, by becoming a **BSI Subscribing Member.**

PLUS is an updating service exclusive to BSI Subscribing Members. You will automatically receive the latest hard copy of your standards when they're revised or replaced.

To find out more about becoming a BSI Subscribing Member and the benefits of membership, please visit bsigroup.com/shop.

With a **Multi-User Network Licence (MUNL)** you are able to host standards publications on your intranet. Licences can cover as few or as many users as you wish. With updates supplied as soon as they're available, you can be sure your documentation is current. For further information, email subscriptions@bsigroup.com.

Revisions

Our British Standards and other publications are updated by amendment or revision.

We continually improve the quality of our products and services to benefit your business. If you find an inaccuracy or ambiguity within a British Standard or other BSI publication please inform the Knowledge Centre.

Useful Contacts

Customer Services
Tel: +44 345 086 9001
Email (orders): orders@bsigroup.com
Email (enquiries): cservices@bsigroup.com

Subscriptions
Tel: +44 345 086 9001
Email: subscriptions@bsigroup.com

Knowledge Centre
Tel: +44 20 8996 7004
Email: knowledgecentre@bsigroup.com

Copyright and Licensing
Tel: +44 20 8996 7070
Email: copyright@bsigroup.com

BSI Group Headquarters

389 Chiswick High Road London W4 4AL UK