COURSES

Course

Geometrical Dimensioning and Tolerancing

cedeo*erkend Bedrijfsopleidingen Maatwerk
cedeo*erkend Bedrijfsopleidingen Open
TÜV NORD ISO 9001
DIENSTVERLENER VOOR DE KMO-PORTEFEUILLE
DV.O 105511
MIKRO CENTRUM at the heart of hightech

Januari 2023 edition
Release: GDT_adv_2.01

No part of this work may be reproduced in any form by print, photoprint, microfilm or any other means without written permission from Mikrocentrum.

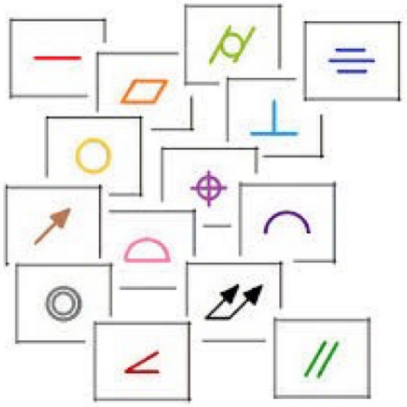
© 2023 Stichting Mikrocentrum Nederland
GMT-05-17-1117 ENGELS

PREFACE

The intend of the syllabus is being a book for self study or as a book used on Geometric Dimensioning and Tolerancing trainings. Knowledge and best practice in this field are constantly changing. Therefore regular updates will become necessary.

There are two major standards when it gets to “Geometrical Dimensioning and Tolerancing” (GD&T). There are the American ASME Y14.5 standards ‘Dimensioning and Tolerancing’ and the ISO series of standards. The relation between the several ISO standards is defined in ISO 14638 which is a matrix of the Geometrical Product Specification Standards (GPS).

ASME and ISO have different philosophies and can have different interpretation of a geometric characteristic.



The syllabus is updated to the GD&T standards as released up to 2022.

Drawings in the syllabus are for illustration purposes only.

All drawings are dimensioned and tolerated with the metric system of measurement.

Hans van Kimmenade

info@tolerantiemanagement.nl

TABLE OF CONTENT

Preface II
Table of content III
Icons used IV
Abbreviations V
Glossery of terms VI

Module I

1 Introduction 1
2 Concepts, Terms and Symbols 7
3 Modifying Symbols 30

Module II

4 Tolerances of Form 56
5 Datum System 71
6 Tolerances of Orientation 88

Module III

7 Tolerances of Location 104
8 Tolerances of Run-out 137
9 Tolerances of Profile 145
10 3D PMI MBD 155
11 References 168

ICONS USED



This icon refers to important information or definitions that are worthwhile to remember.



Exercises.



The information symbol indicates more complex examples and broadened information. Subjects will result in more in-depth knowledge.



The information can be interesting but is not necessarily required for understanding the subject.

ABBREVIATIONS

A	Derived feature
ACS	Any Cross Section
CZ	Combined Zone
E	Envelope requirement
F	Free state
GPS	Geometrical Product Specification
L	Least material requirement
LD	Least (minimum) Diameter
M	Maximum material requirement
P	Projected tolerance zone
PD	Pitch Diameter
R	Reciprocity requirement
RFS	Regardless of Feature Size
SIM	Simultaneous requirement
SZ	Seperate zone
TED	Theoretical Exact Dimension
UF	United feature
UZ	Unequal Zone

GLOSSERY OF TERMS

Datum - A theoretical exact point, axis or plane derived from a true geometric counter part of the specified datum feature

Datum axis - The theoretical axis derived from the true geometric counter part of a specified datum feature

Datum plane - A datum established from the datum feature simulator of a nominal flat datum feature

Datum system - A set of symbols and rules on how to constrain a part to establish a relation between the part and geometric tolerance zones

Datum targets - A set of symbols that describe the shape, size and location of datum feature simulators that are used to establish datum planes, axis or points.

Feature - The physical portion of the workpiece, such as a point, hole, pin 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). Definitions of features are taken from ISO 14660-1 and ISO 14660-2

Feature of size - The shape defined by a linear or angular dimension that is a size. Typically, a feature of size is a cylinder, a sphere, two opposite parallel surfaces, a cone or wedge. (ISO 14660-1).

Gage - A device to measure a part characteristics.

Geometric tolerance - The general term applied to the category of tolerances used to control form, profile, orientation, location and run-out.

Least Material Condition (LMC) - The condition in which a feature of size contains the least amount of material within the stated limits of size

Maximal Material Condition (MMC) - The condition in which a feature of size contains the maximum amount of material within the stated limits of size.

Modifiers - Symbols or keywords that communicate additional information about the tolerancing of a part.

Simulated datum - A datum established from a physical datum feature simulator.

Theoretical Exact Dimension (TED) - A dimension which is not affected by an individual or general tolerance.

Tolerance zone - the area (zone) that represents the total amount that part features are allowed to vary from their specified dimension.

True position - The theoretical exact location of a feature of size as established by its theoretical exact dimension (TED).

Virtual condition - A fixed size boundary generated by the collective effects of a feature of size's specified MMC or LMC and the geometric tolerance for that material condition.

MODULE I

1 INTRODUCTION

1.1 GEOMETRICAL TOLERANCING OBJECTIVES

Geometric Tolerancing is an international language that is used on engineering drawings to accurately describe the size, form, orientation, and location of part features. It is also a design dimensioning philosophy to define a part based on its functions.

GD&T will improve product designs and result in lower cost. Machine operators and manufacturing engineers use the language to interpret the design intent and to determine the best manufacturing approach. Quality control and inspection use the GD&T language to determine proper set-up and part verification. GPS reduces controversy, guesswork, and assumptions throughout the design, manufacturing and inspection process.

1.2 HISTORY

Rene Descartes a French mathematician, philosopher & scientist, developed the Cartesian coordinate system. During an illness, as he lay in bed sick, Descartes saw a fly buzzing around on the ceiling, which was made of square tiles. As he watched he realized that he could describe the position of the fly by the ceiling tile it was on. After this experience he developed the coordinate plane to make it easier to describe the position of objects. This coordinate system is the basis of the coordinate tolerancing method using ± tolerances.



Figuur 1.1a Rene Descartes (1596 - 1650)



Figuur 1.1b Frederick Taylor (1856 -1915)

Frederick Winslow Taylor 1856 in Germantown, Pennsylvania. He was born into a wealthy family. Taylor became an apprentice pattern maker and machinist. He finished his four-year apprenticeship and became a machine-shop laborer. He was quickly promoted to chief engineer.

He joined Bethlehem Steel in 1893 in order to solve an expensive machine shop capacity problem.

After leaving the company, Taylor focused the rest of his career on promoting his management and machining methods. Frederick Winslow Taylor caught pneumonia and died on March 21, 1915.

Taylor is one of the pioneers on Scientific management a theory of management that analyzes and synthesizes workflows. Its main objective is improving economic efficiency, especially labor productivity. It was one of the earliest attempts to apply science to the engineering of processes to management.

Taylor also influenced Henri Ford where the Fordism was developed. Fordism is a manufacturing technology that serves as the basis of modern economic and social systems in industrialized, standardized mass production and mass consumption.

Related to GD&T Taylor developed limit gauges. A 'GO' gauge should check all related dimensions (form and size) simultaneously the 'NOGO' gauge should check only one dimension at a time. We know this GO requirement as envelope requirement.



Figure 1.2 Taylor gauge 24H7

Driven by the demands of the 2nd World War the British innovated and standardized. Stanley Parker of the Royal Torpedo Factory created in 1940 a positional tolerancing system. He found that coordinate tolerances result in a square tolerance zone, but that parts outside the square might be actually be good, as long as they are within a circle that encompasses the square's corners.

Parker came up with a concept on defining allowed deviations on a workpiece. This concept developed into what know as Geometrical Dimensioning and Tolerancing. From here a concept symbol language was developed. In the US as first ANSI GD&T standard.

The British set up the first standard on position tolerancing published in 1948 "Dimensional Analysis of Engineering Design". The American National Standards Institute publication in 1982 of ANSI Y14.5M-1982 was a turning point in standardization of the methodology.

Today we know two major dimensioning and tolerancing standards the ASME Y14.5 published by the American Society of Mechanical Engineers and the ISO GPS standards defined in ISO matrix 14638 published by International Organization of Standards. A few examples of GD&T standards are ISO 1101; ISO 5458; ISO 5459 and ISO 1660.

The ASME and ISO standards look similar but are very different in application and interpretation.

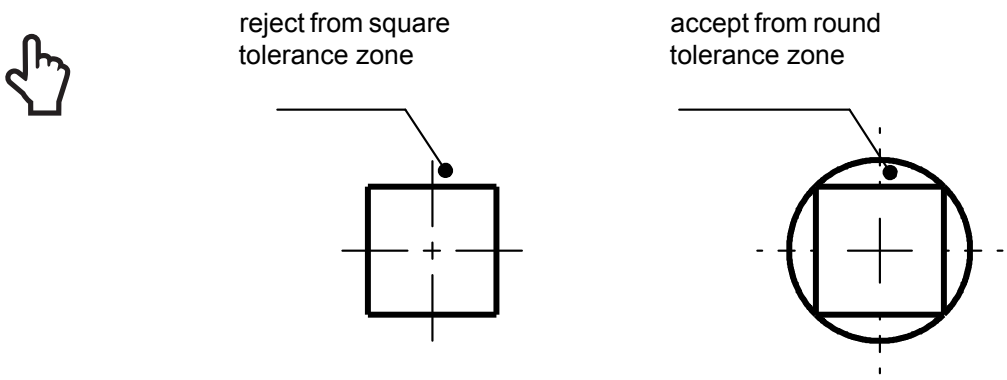


Figure 1.3 Square and round tolerance zones

1.3 SURFACE CONDITION

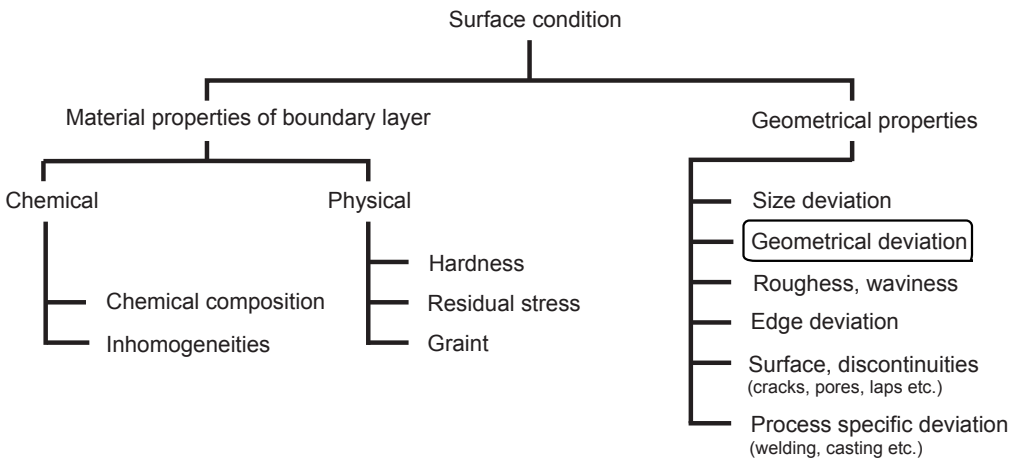


FIGURE 1.4 Workpiece surface conditions

The geometrical characteristics are defined as deviations from geometrical ideal integral or derived features of a workpiece. Integral features are surfaces which have geometrical, unique and nominal form like planes, cylinders, spheres, and cones. Derived are axes, midpoints media planes. In general features of size (FOS).

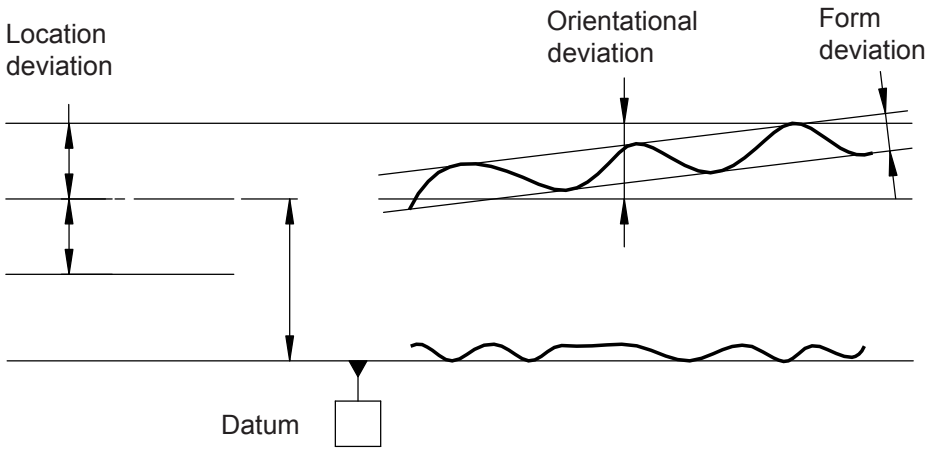


FIGURE 1.5 Form, locational and orientational deviation

Geometrical deviations are:

- size deviations
- form deviations
- orientation deviations
- locational deviations
- run out
- waviness
- roughness
- surface discontinuities
- edge deviations

Size deviation is the difference between actual size and nominal size.

Form deviation is the deviation of a feature (geometrical element, surface or line) from its nominal form.

Orientational deviation is the deviation of a feature from its nominal form and orientation. The orientation is related to one or more datum features. The orientation deviation includes indirect the form deviation.

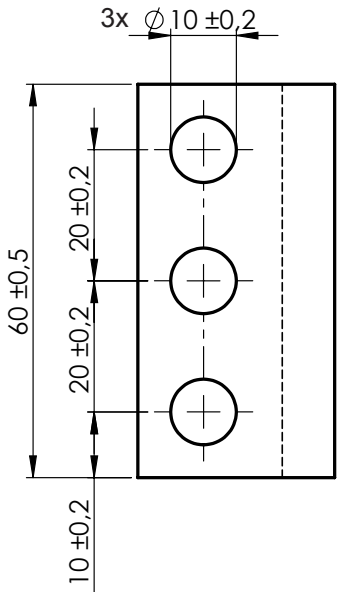
Locational deviation is the deviation of a feature (surface, line, point) from its nominal location. The location is related to none, one or more (other) datum feature(s). The locational deviation includes indirect also the form deviation and the orientation deviation (of the surface, axis, or median face).

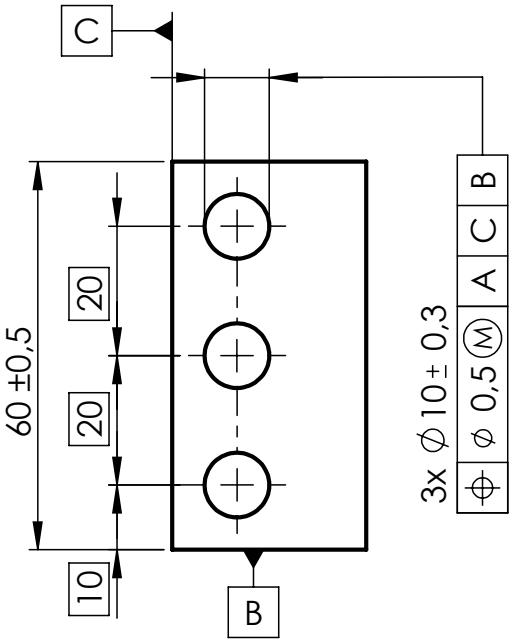
Waviness, roughness, surface discontinuities and edge deviations are not part of this document.

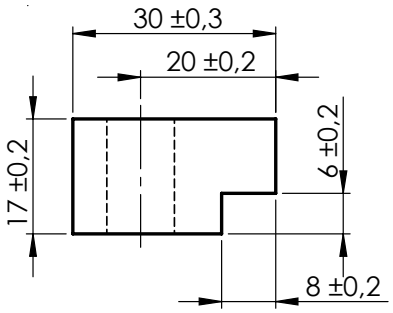
Only complete and correct tolerated drawings or 3D models enable the production of workpieces to be as precise as necessary and as economic as possible. When all tolerances necessary to define a workpiece are indicated individually the drawing or model becomes overloaded with indications and will be hard to read. Therefore general tolerances can be applied according ISO 22081. General tolerances shall be applied by an indication in or near the title block of the drawing or annotation plane in a CAD model.

When there is no appropriate ISO Standard or National Standard available a company standard should be elaborated.

1.4 DIMENSIONING METHOD







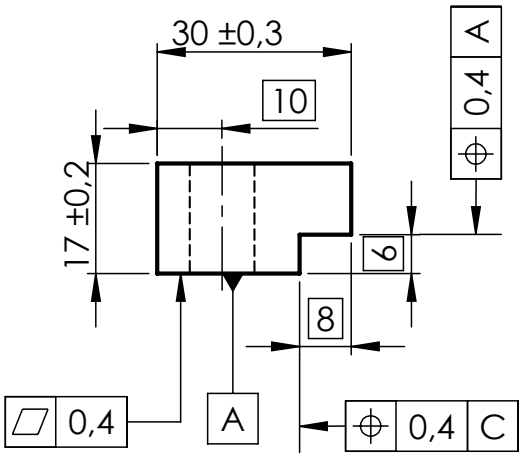


FIGURE 1.6 Coordinate dimensioning

FIGURE 1.7 Geometrical dimensioning

Coordinate tolerancing is an over simplification of part definition leaving out important pieces of information. Major shortcomings are:

- Undefined measurement setup
- No indication of measurement origin
- Rectangular tolerances zones
- Accumulation of tolerances
- Fixed size tolerance zones
- Exact start point of dimensions undefined

Avoiding these shortcomings GPS gives the several benefits:

- Improved communication
- Better ‘functional’ product design
- Increased tolerances
- Lower costs;

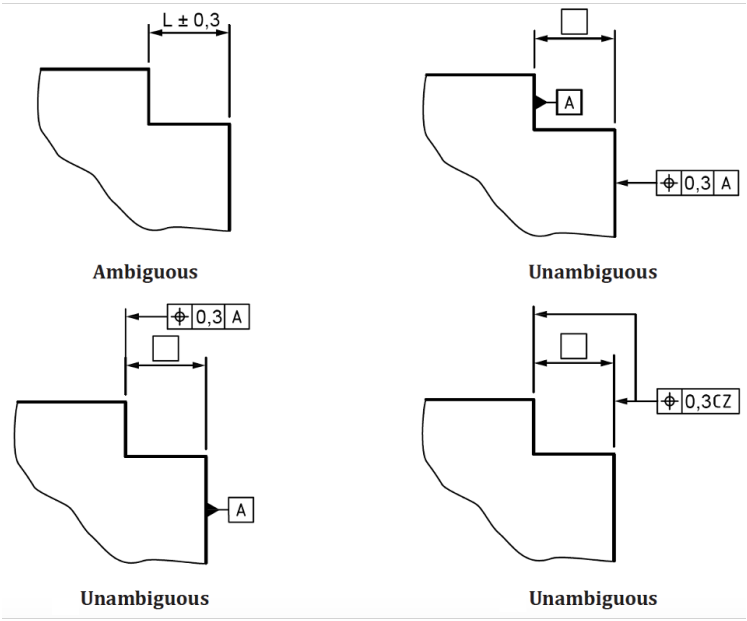


FIGURE 1.8 Step dimensions vs GD&T on a surface

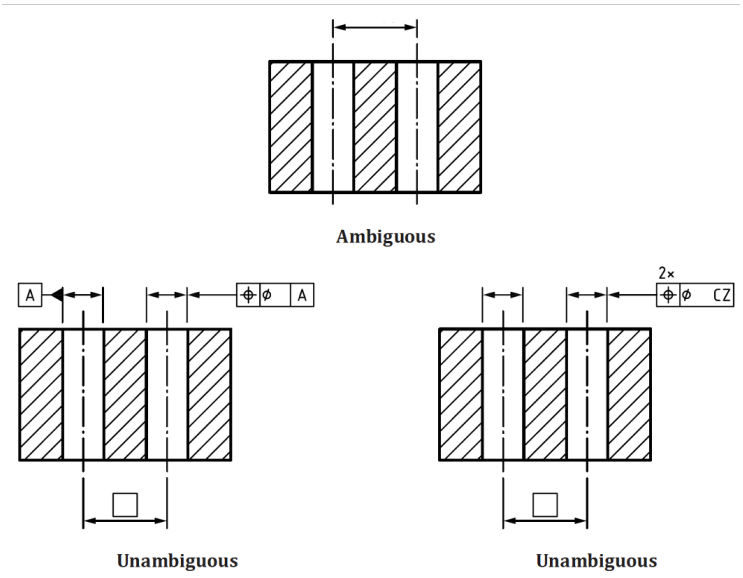


FIGURE 1.8 Step dimensions vs GD&T on a derived feature

Plus/minus dimensions, which are two point local sizes, and tolerances are still used but their use should be limited to defining features of size and the depth or length of features such as holes and pins. For many reasons, ± dimensions and tolerances should not be used to locate features.

2 CONCEPTS, TERMS AND SYMBOLS

2.1. CONCEPTS AND TERMS

The fundamentals, concepts, principles and rules of the Geometric Dimensioning and Tolerancing (GD&T) are specified in ISO 8015.

Geometric tolerances shall be specified in accordance with functional and economical requirements. Manufacturing and inspection can also influence the geometrical tolerancing.

A geometrical tolerance applied to a feature defines the allowed deviation from nominal surface or position.

2.1.1 INVOCATION

Once a portion of the ISO GPS system is invoked in a mechanical engineering product documentation, the entire ISO GPS system is invoked, unless otherwise indicated on the documentation.

The ISO GPS system is defined in a hierarchy of standards including the following types Symbols of standards:

Fundamental standards e.g. ISO 8015
General standards e.g.ISO 1101
Additional standards e.g. ISO 22081

Normative standards applicable for Geometric Tolerancing are summarized in ISO 1101.

2.1.2 FEATURES AND FEATURE PRINCIPLE

FEATURE

A feature is a physical portion of the workpiece, such as a point, hole, pin or a surface. These features can be integral features being surfaces or surfaces lines or derived features (e.g. a median line or median plane). Definitions of features are taken from ISO 14660-1 and ISO 14660-2
ww

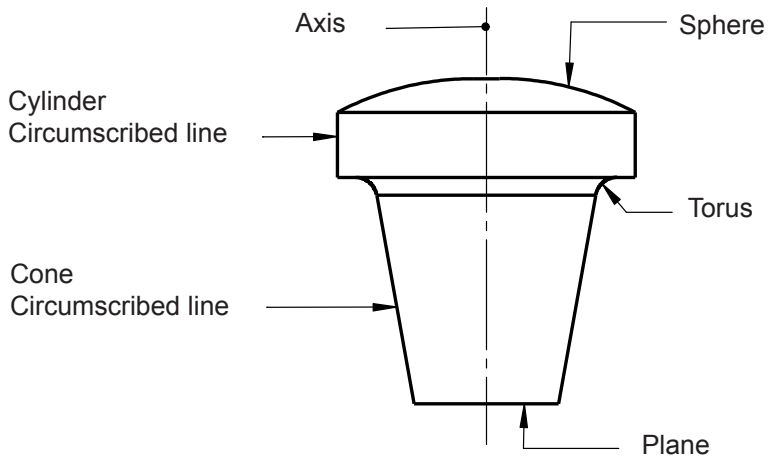


FIGURE 2.1 Workpiece features

Feature of size is a geometrical shape defined by a linear or angular dimension that is a size. The size of a future of size is always obtained by a two point measurement. Typically, a feature of size is a cylinder, a sphere, two opposite parallel surfaces, a cone or wedge. (ISO 14660-1)

Integral feature is a surface or a surface line

Derived feature is a mid point, center line, or media plane

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

2.1.3 THEORETICAL EXACT DIMENSION

Theoretically Exact Dimension (TED) also known true dimension, is a theoretical exact location of a feature of size defined by nominal dimensions.

The TED is shown in a rectangular frame and has no tolerance. Theoretically exact dimensions may only vary by the theoretical tolerance that is stated in the associated tolerance frame.

TED are used to dimension the theoretical exact location of position, angularity, line profile and surface profile.

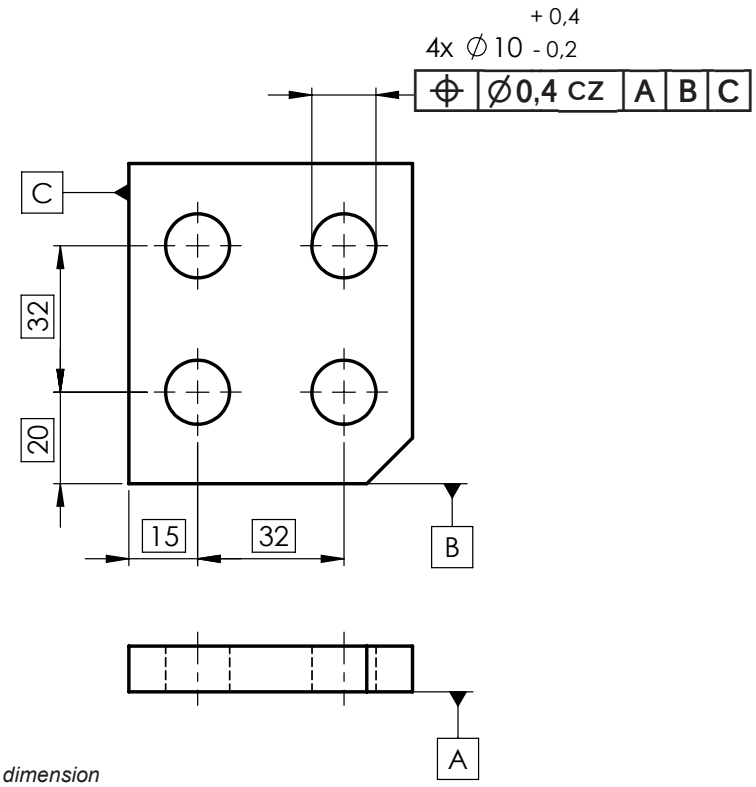


Figure 2.2 Theoretical exact dimension

2.1.4 RIGID WORK PIECE PRINCIPLE

By default, a work piece shall be considered as having infinite stiffness and all GPS specifications apply in the free state, undeformed by any external forces including the force of gravity. Requirements that apply to non rigid work pieces shall be defined in the drawing according to ISO 10579.

2.1.5 PRINCIPLE OF INDEPENDENCY

By default, every GPS specification for a feature or relation between features shall be fulfilled independent of other specifications except when it is stated in a standard or by special indication (e.g. Ⓜ modifiers according to ISO 2692, CZ according to ISO 1101 or Ⓔ modifiers according to ISO 14405-1) as part of the actual specification.

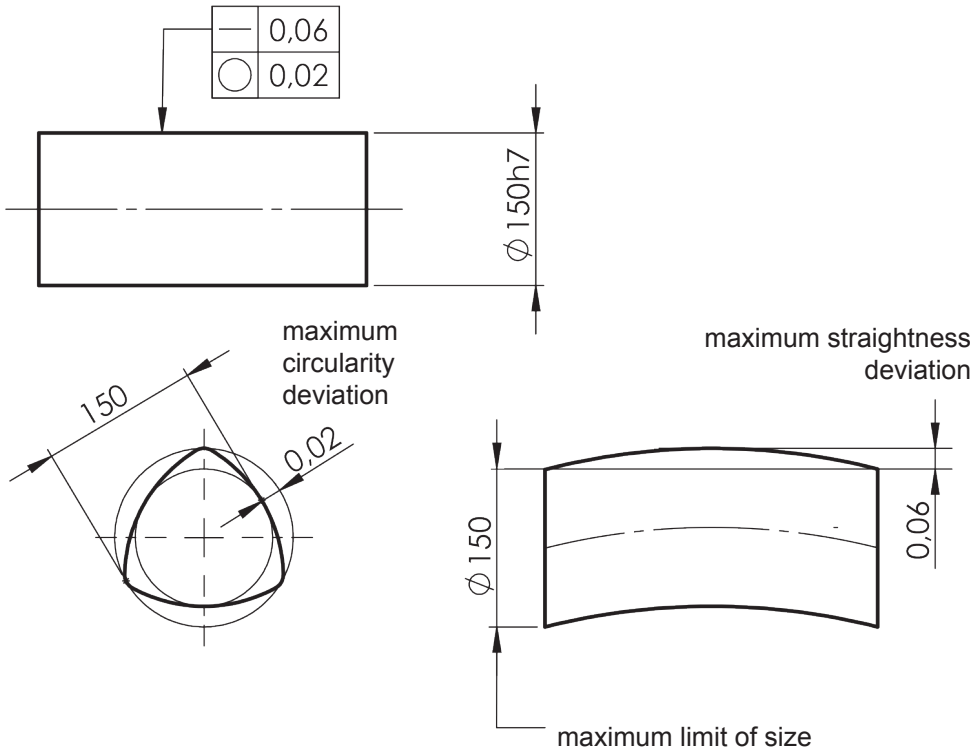


FIGURE 2.3 Principle of indendency

An altered default specification operator is applied when non ISO standards are used. and shall be defined in a relevant document.
The altered default specification operator shall be thorough, unambiguous and completely defined in order to be regarded as a complete specification operator.

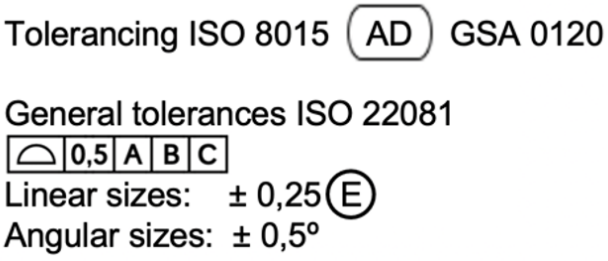


FIGURE 2.4 Drawing footer or annotation plane

2.1.6. ENVELOPE REQUIREMENT

The envelope requirement according to ISO 14405 specifies that the surface of a single feature of size should not violate the imaginary envelope of perfect form at maximum material size.

An envelope requirement makes sense with e.g. fittings. In most cases an envelope requirement may not be needed. A survey in several countries showed that an envelope requirement makes sense in only in 10% of the cases.

The envelope requirement may be specified either:

- by indication of the symbol 'E' placed after the linear (size) tolerance
- by indication in the drawing title box "ISO 2768 .... **E** "
- by a company standard

The envelope requirement cannot be applied to features for which a straightness or flatness tolerance is specified that is larger than the size tolerance.

The envelope may also be indicated as a form tolerance with '0 Ⓜ' in the feature frame.

Envelope requirement over a restricted lenght is indicated as: ø 20 ± 0,03 Ⓔ / 10

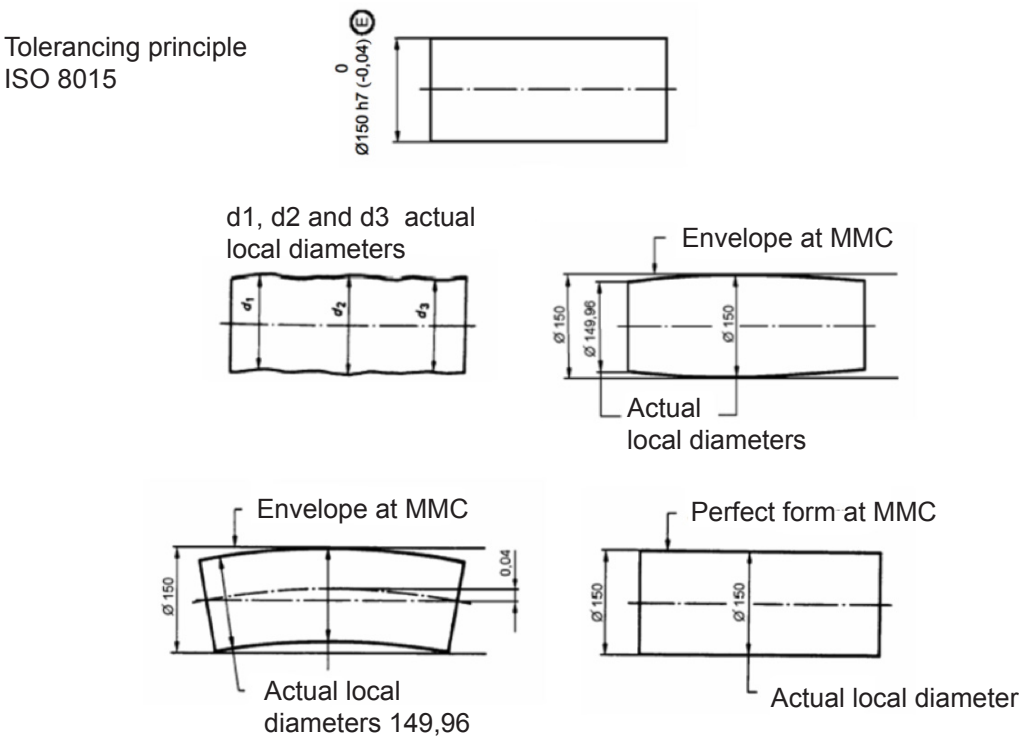


Figure 2.5 Envelope requirement

2.2 GD&T SYMBOLS

The symbols on indications of geometrical tolerances according to ISO 1101

Geometric tolerances can be applied to three kinds of features:

- Integral feature, surfaces or surface lines
- Derived features, midpoints,center lines, media planes, features of size
- Patterns

2.2.1 GEOMETRIC CHARACTERISTIC SYMBOLS

The geometric characteristic symbols are 14 symbols used to describe the geometry attribute of a part. The symbols are divided into four types: form, orientation, location and run-out. Notice that line profile and surface profile can define form, orientation and location.

Based on the type they may have never, always or sometimes reference to a datum.

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
Location	Position	⌖	yes or no
	Concentricity (for centre points)	◎	yes
	Coaxiality (for axes)	◎	yes
	Symmetry	⌯	yes
	Line profile	⌒	yes
	Surface profile	⌓	yes
Run-out	Circular run-out	↗	yes
	Total run-out	⌰	yes

TABLE 2.1 Geometric characteristics symbols

Form tolerance

Form tolerances limit the deviations of a feature from its geometrical ideal line or surface form.

Orientation tolerance

Orientation tolerances limit the deviations of a feature from its geometrical ideal orientation with respect to the datum(s).

Location tolerance

The location tolerance zone is the geometrical ideal location, orientation and distance, with respect to the datum(s). Location also limits the orientation and the form tolerance of the toleranced feature.

Run-out

Run-out tolerances are partly orientation tolerances (axial circular run-out, axial total runout) and partly location tolerances (radial cirular run-out, radial total run-out).

Geometrical tolerances and tolerance symbols				
Unrelated geometrical tolerances (form tolerances)				
	Profile (form) of lines (line profile)		⌒	
				⏤
				○
	Profile (form) of surfaces (surface profile)		⌓	
		Flatness		▱
		Cylindricity		⌭
Related geometrical tolerances				
			∠	
				//
				⊥
			⌖	
				◎
		Symmetry		⌯
	Run-out	Circular run-out	Circular radial run-out	↗	
			Circular axial run-out		
		Total run-out	Total radial run-out	⌰	
			Total axial run-out		

Table 2.2 Geometric tolerances and tolerance symbols

2.3 THE TOLERANCE FRAME

2.3.1

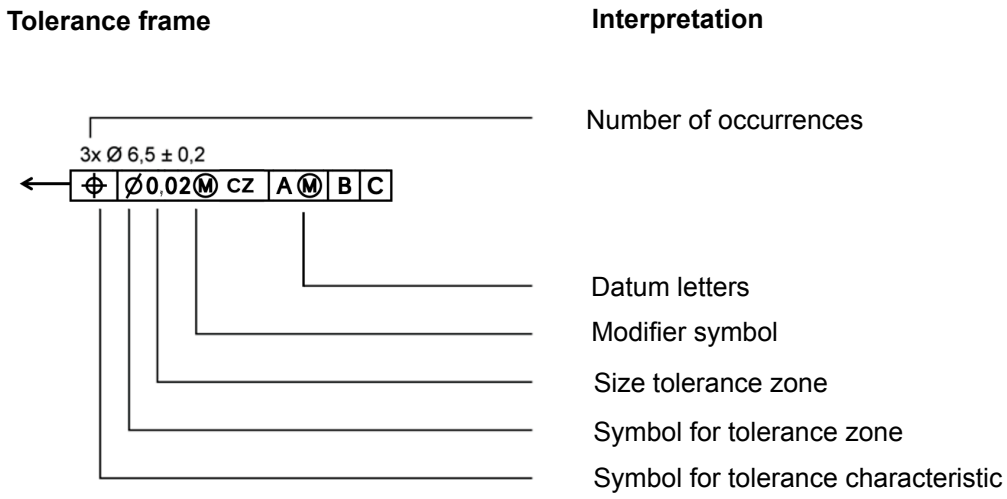


FIGURE 2.6 Tolerance frame composition

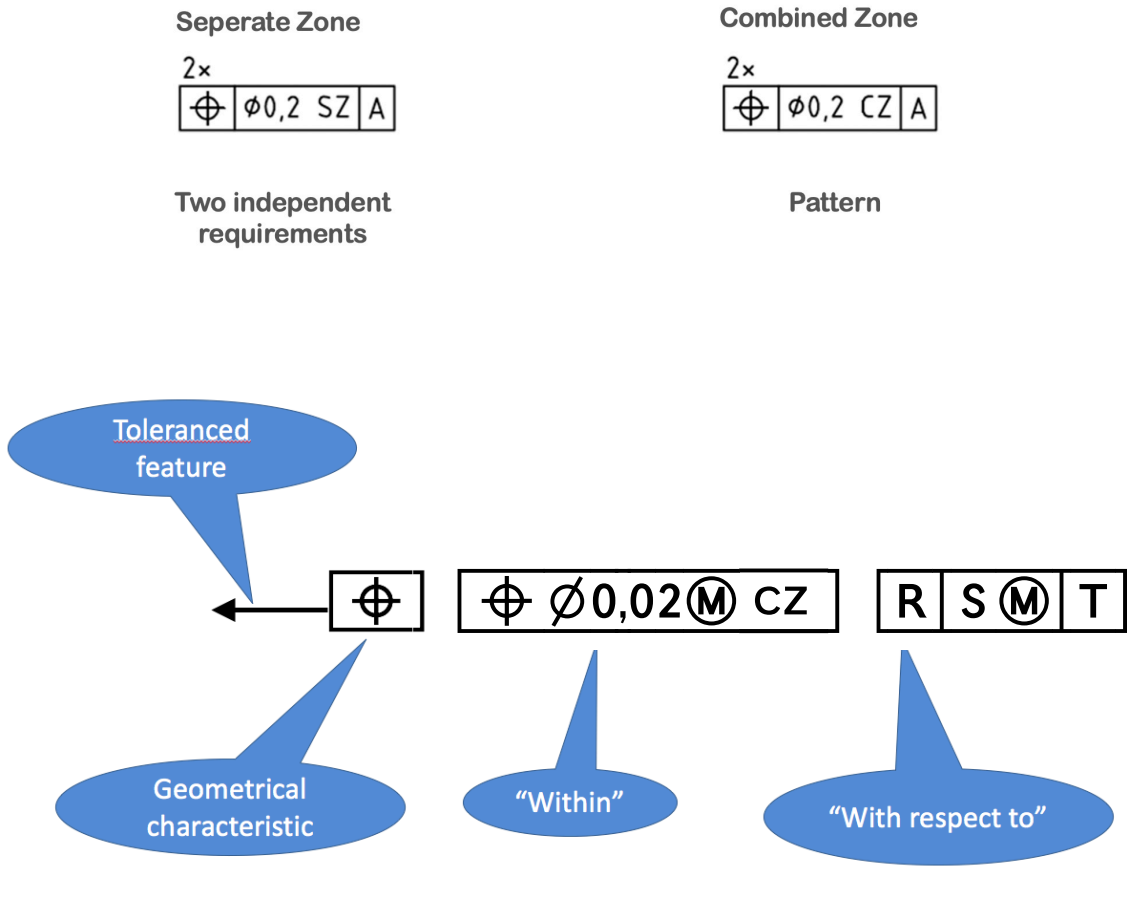
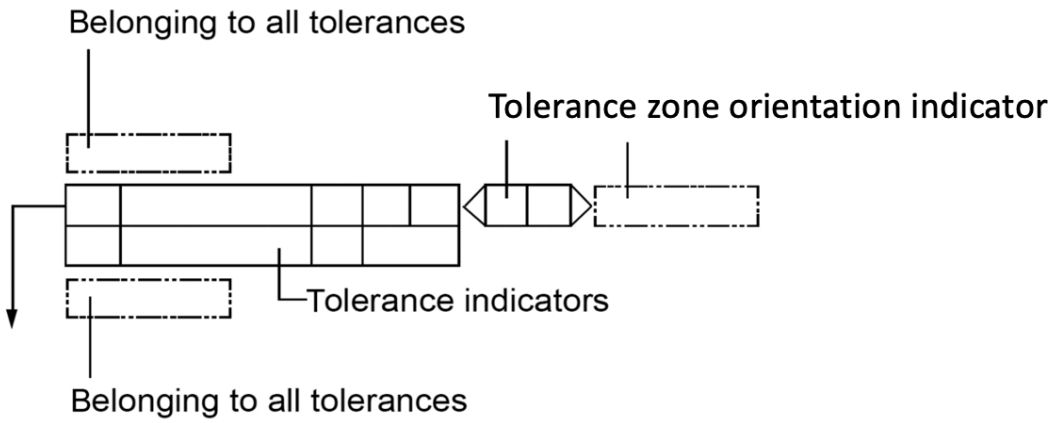


FIGURE 2.7 Tolerance and datum frames



Examples:

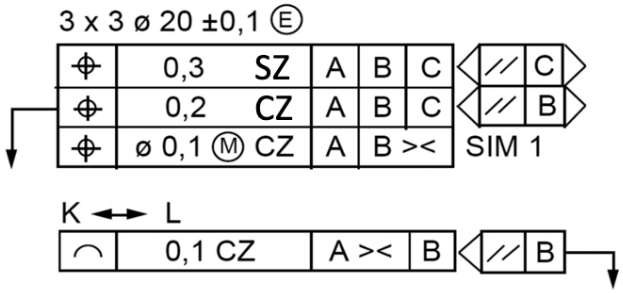
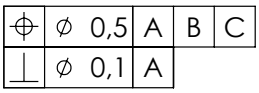


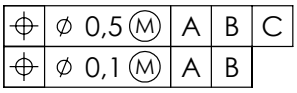
FIGURE 2.8 Complete tolerance frame

◎ ∅ 0,2 A-B	Datum established by two features.
⌖ Ø0,2 CZ R S T	Cylindrical tolerance zone, three datums.
⌖ S Ø0,3 CZ R S T	Spherical tolerance zone, three datums.
⊥ 0,2 A	Tolerance zone space between two parallel planes, one datum.
▱ 0,2	Form requirement, no datum.
6x ▱ 0,1	Requirement applies to more features.
8x ∅12 ± 0,05 ⌖ Ø0,1 CZ R S T	Requirement applies to more features, dimension over tolerance frame.
▱ 0,1 NC	Indication qualifying the form “Non Convex” (obsolete since ISO 1101:2017).

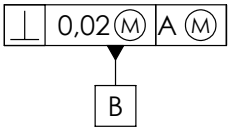
Figure 2.9 *Tolerance frames*



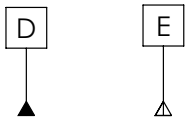
Superposition,
different type geometric requirements



Superposition,
same type geometric requirements



Datum reference attached to tolerance frame



Open or closed datum reference traingle

FIGURE 2.10 Tolerance frame and datum reference

2.3.2 FEATURE INDICATION

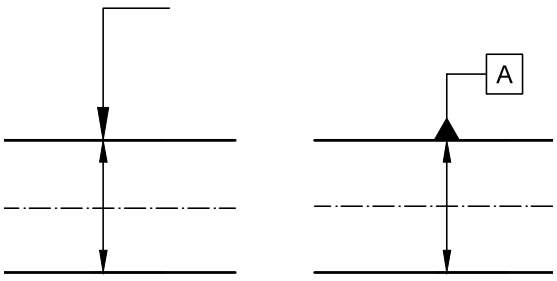
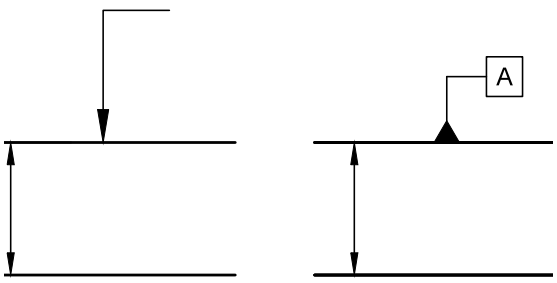
Symbols	Interpretation
	Axis or media plane as toleranced feature or datum feature
	Surface or section line as toleranced feature or datum feature

FIGURE 2.11 Derived feature

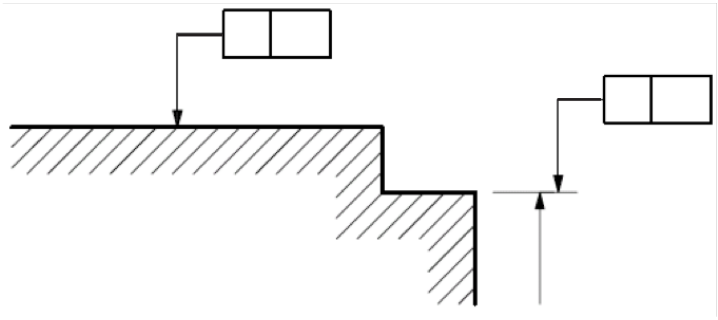
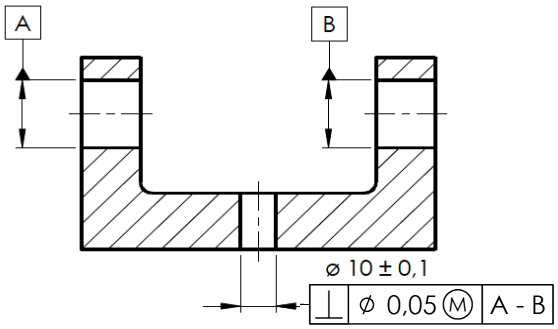
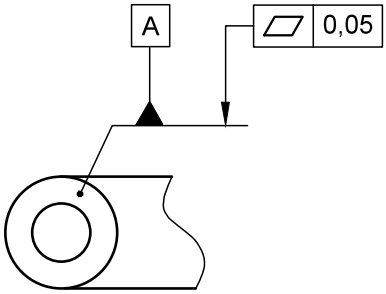


Figure 2.10 Indication on surface or extension line



Axis or media plane common datum established by two features

Figure 2.12 Common datum from A and B



Top side as datum feature with form tolerance

Figure 2.13 Position of the arrow and datum indicator

Depreciated and former practices

On deprecated and former practices consult ISO 1101. For detailed information on datums see ISO 5459

2.3 WIDTH AND DIRECTION OF THE TOLERANCE ZONE

Tolerance zones have limiting lines or surfaces that are equidistant from the nominal (geometrical ideal) form. The shape of the tolerance zone is independent of the size of the feature(s). Exceptions are tolerance zones of line profile or surface profile where the nominal (geometrical ideal) form is defined by theoretical exact dimensions (TEDs).

The width of the tolerance zone is equal to the tolerance value, and is directed according to ISO 1101 in the direction of the leader line arrow:

- with tolerances of axes and median faces perpendicular to the toleranced axis or median face;
- with tolerances of surfaces or lines perpendicular to the surface.

Drawing

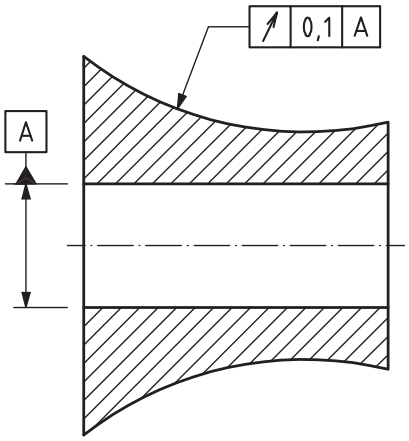


FIGURE 2.14 Arrow perpendicular to surface

Tolerance zone

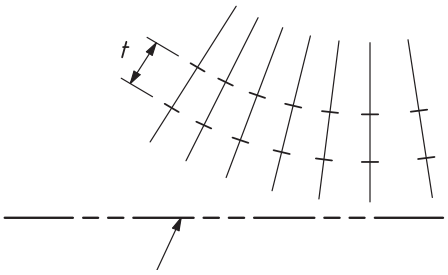


FIGURE 2.15 Arrow perpendicular to surface

In the figure the width of the tolerance zone is in the direction of the leader line arrow perpendicular to the surface.

Below when the width of the tolerance zone is not perpendicular to the surface or the axis or the median face, the direction of the width of the tolerance zone shall be indicated as shown

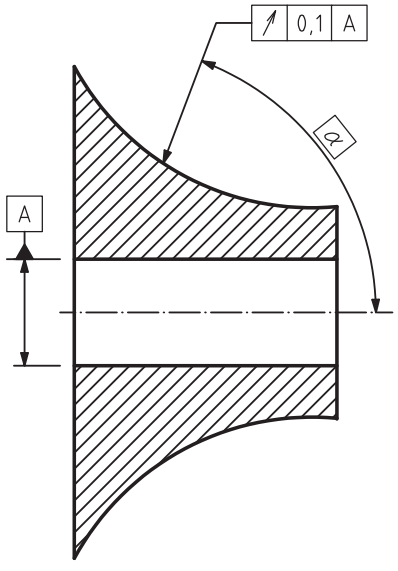


FIGURE 2.16 Arrow not perpendicular to surface but under given angle

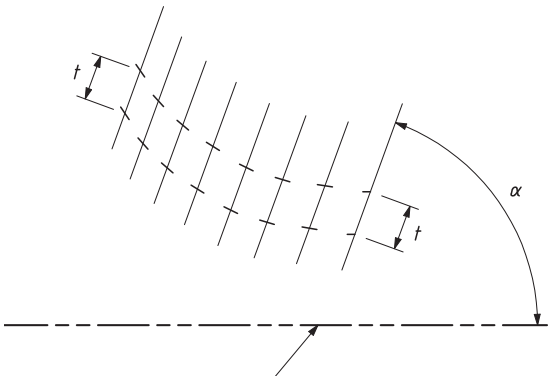


FIGURE 2.17 Tolerance zone not perpendicular to surface

2.4 SHAPE OF THE TOLERANCE ZONE

Depending on the toleranced characteristic and depending on the drawing indication the tolerance zone is one of the following.

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

Drawing **Tolerance zone**

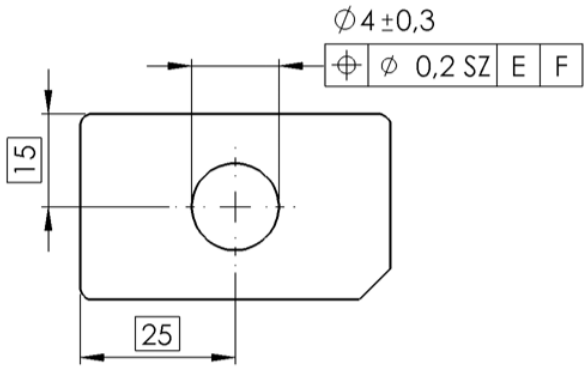


FIGURE 2.18 Position tolerance

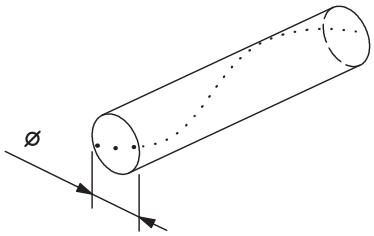


FIGURE 2.19 Cylindrical tolerance zone

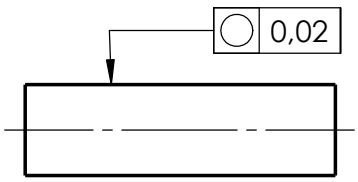


FIGURE 2.20 Roundness

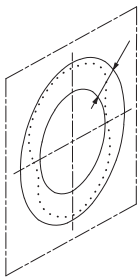


FIGURE 2.21 Tolerance zone of two concentric circles

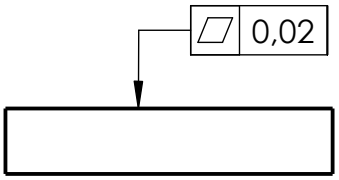


FIGURE 2.22 Flatness

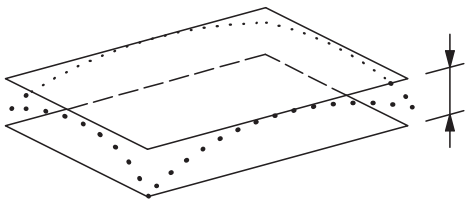


FIGURE 2.23 Space between two equidistant planes

Drawing

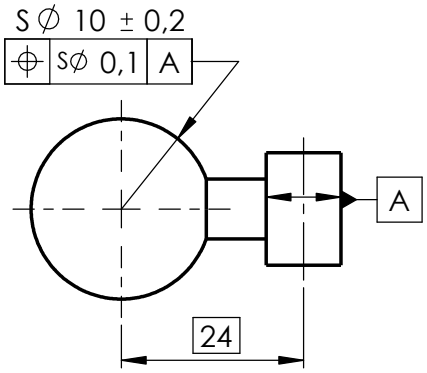


Figure 2.24 Position of a sphere

Tolerance zone

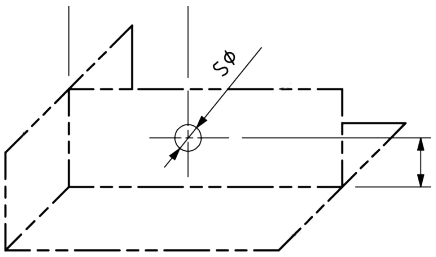


Figure 2.25 Spherical tolerance zone

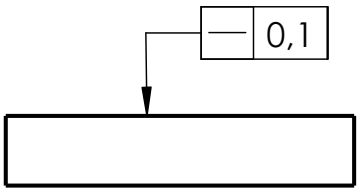


Figure 2.26 Straightness

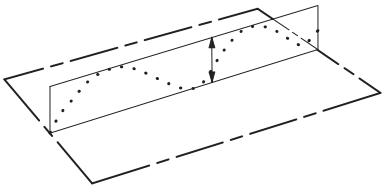


Figure 2.27 Tolerance zone of wo parallel lines

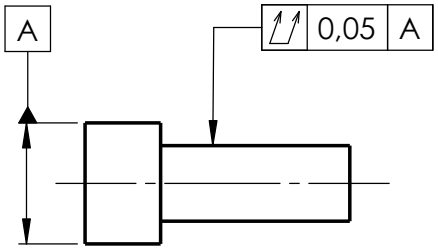


Figure 2.28 Total run-out

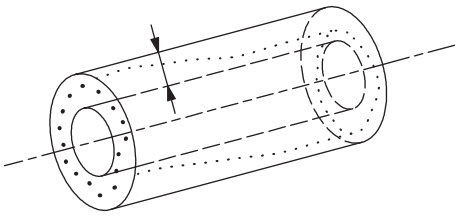


Figure 2.29 Tolerance zone of two coaxial cylinders with respect to a datum axis

2.4.1 LIMITED TOLERANCE ZONE

If not otherwise indicated, the geometrical tolerance (tolerance zone) applies to the entire length or surface of the feature. When the leader line arrow points to a thick chain line or area within a thick chain line, the tolerance applies to this region.

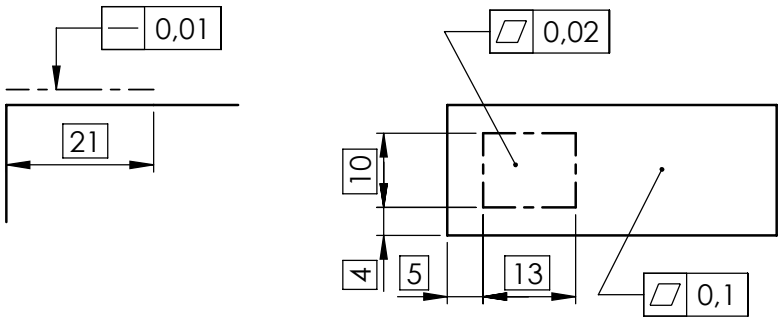


FIGURE 2.30 Requirement over a restricted length or surface

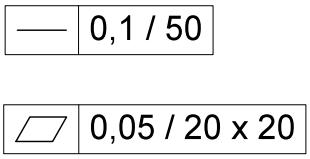


FIGURE 2.31 Requirement over a restricted length or surface

When the tolerance value is followed by an oblique stroke and another value (e.g. 0,1/50), the second value (50 here for the straightness requirement) indicates the length within which the tolerance (0,1) applies. The tolerance zone (of 0,1 width) of the specified length (50) applies to all possible locations on the feature.

2.5 AUXILIARY PLANES

2.5.1 INTERSECTION PLANE

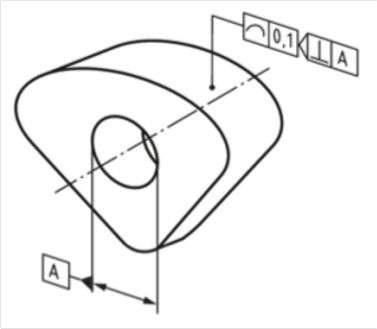


FIGURE 2.32 Line profile, lines perpendicular to axis A

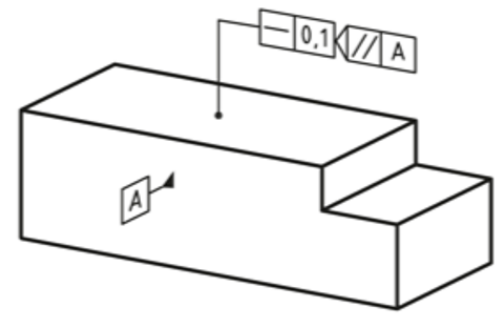


FIGURE 2.33 Straightness, lines parallel to datum A

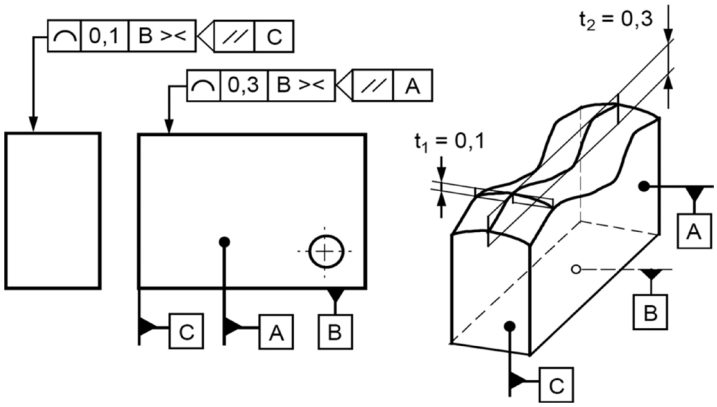


FIGURE 2.34 Line profile, 0,3 parallel to datum A and 0,1 parallel to datum C. Datum B for orientation only.

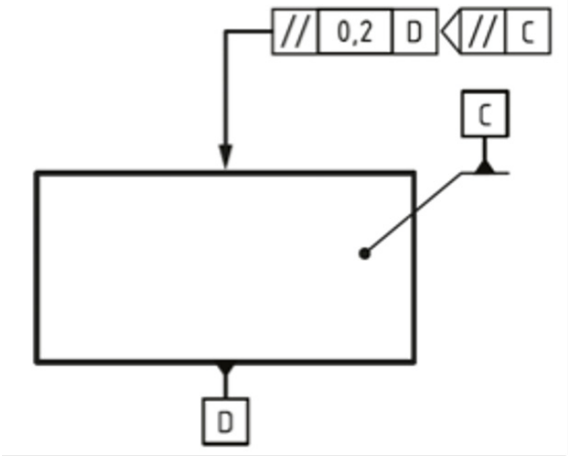
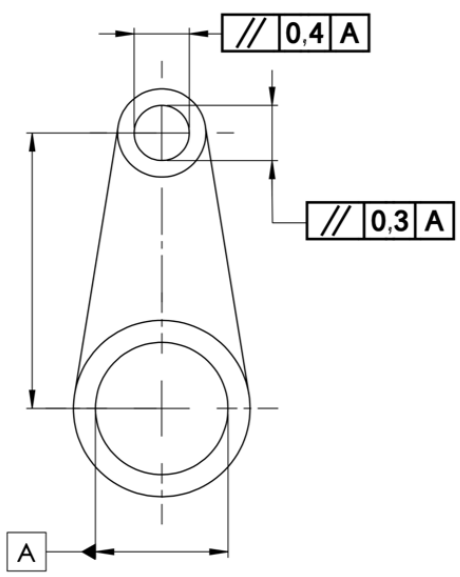


FIGURE 2.35 Lines parallel to D orientation parallel to datum C

2.5.2 ORIENTATION PLANE

In the case of a median feature (center point, median line, median surface) tolerated in one direction: In 2D view, when the direction of the width of a tolerance zone is at 0° or 90° relative to the datum or relative to the pattern of the theoretically exact dimensions, the arrow of the leader line gives this direction.



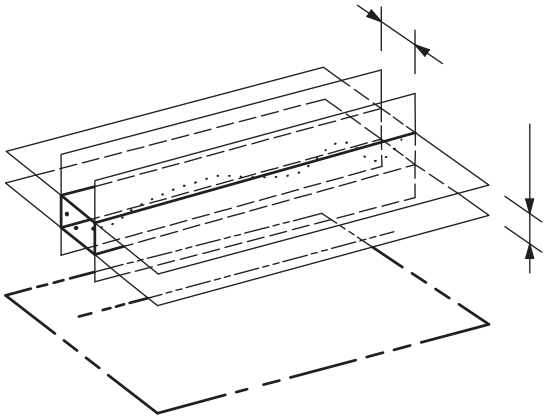
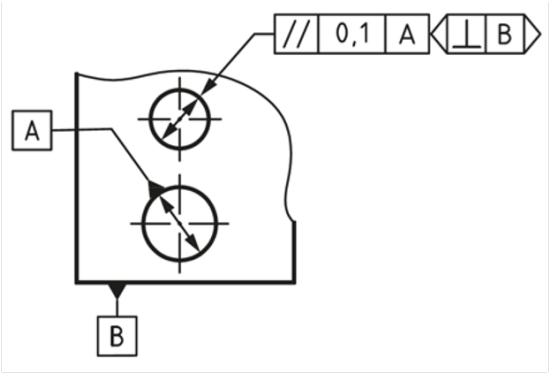


FIGURE 2.36 The direction of the arrows make up the planes for the tolerance zone

The way having the leader lines defining the direction of the tolerance zone plane is deprecated.



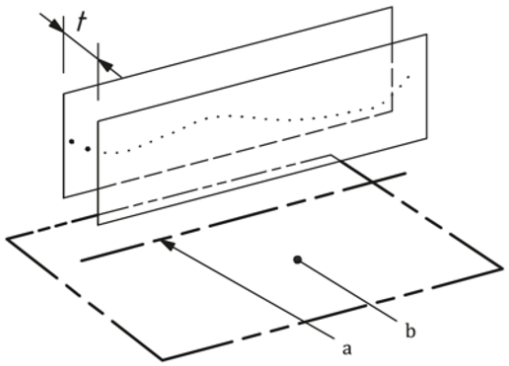


FIGURE 2.37 Centerline between two planes perpendicular to datum B

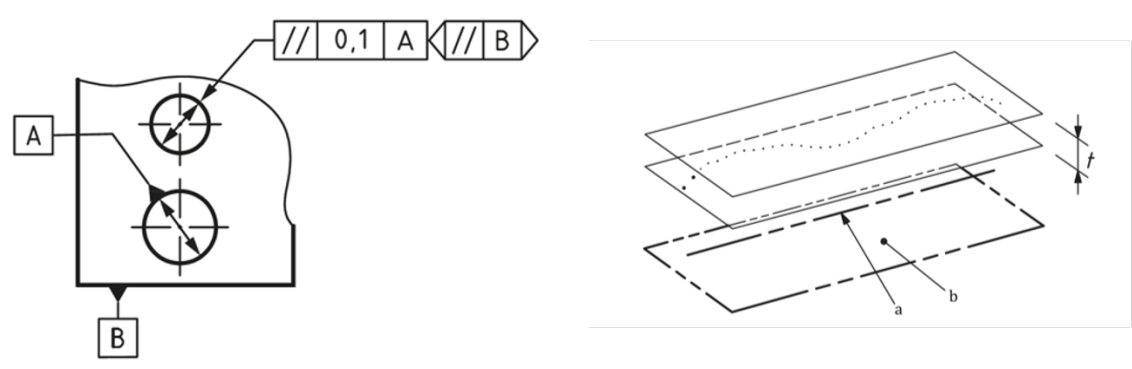


Figure 2.38 Centerline between two planes parallel to datum B

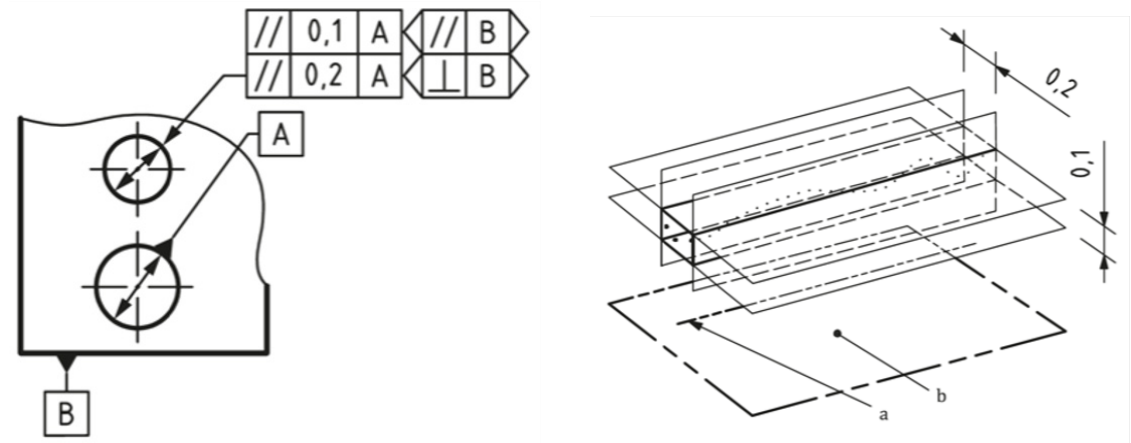


Figure 2.39 Centerline between four planes.

2.5.3 DIRECTION FEATURE

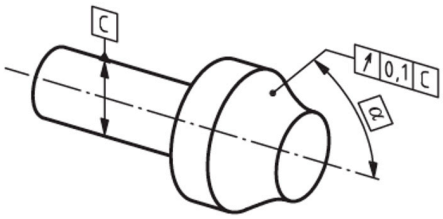


FIGURE 2.40 Circular run-out, orientation of the tolerance zone direct implied

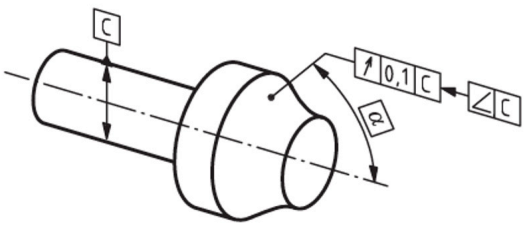


FIGURE 2.41 Circular run-out, orientation of the tolerance zone indicated by direction feature

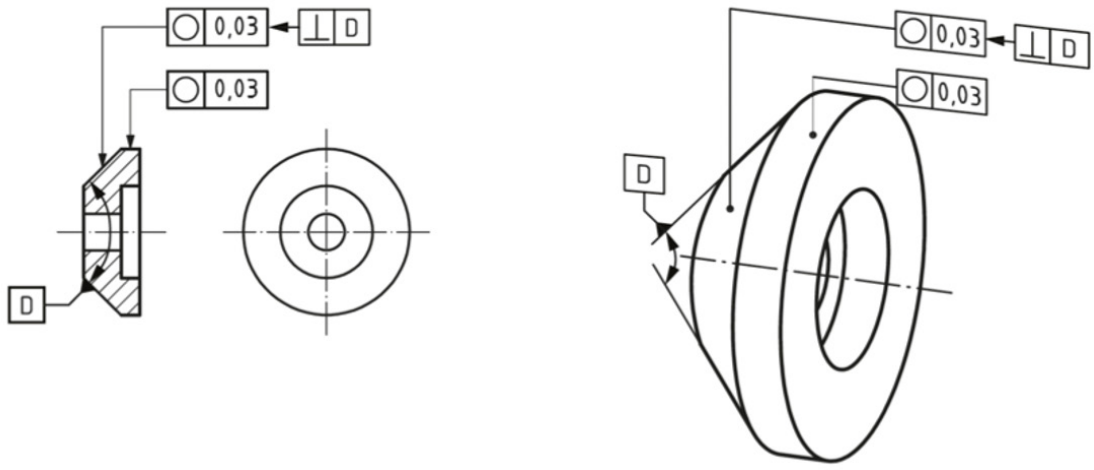


FIGURE 2.42 Tolerance zone for roundness of the cone perpendicular to the cone axis

2.5.4 COLLECTION PLANE

A collection plane shall be indicated when the “around” symbol is used to identify that a specification applies to a collection of features.

A collection plane identifies a set of single features whose intersection with any plane parallel to the collection plane is a line or a point

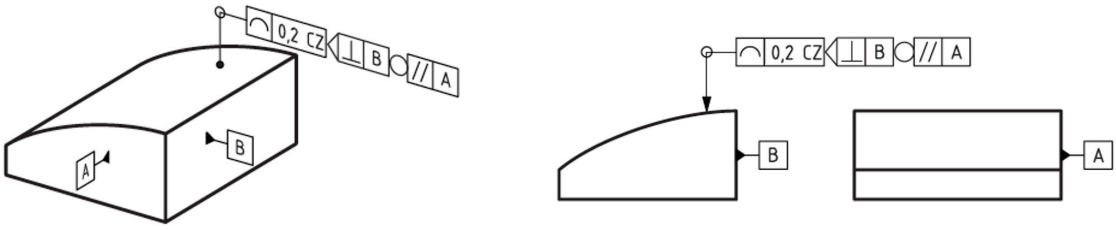


FIGURE 2.43 Tolerance zone for a set of lines on the closed compound continuous surface

2.6 GENERAL TOLERANCES

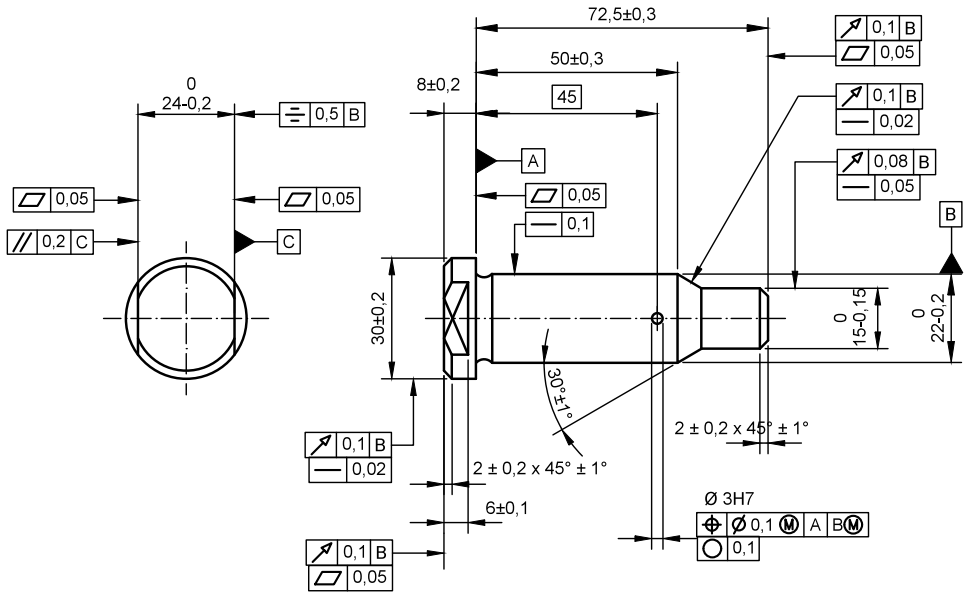


FIGURE 2.45 All tolerances stated

All features of a workspiece have a form and dimension. Therefore all elements need to be dimensioned and need a geometric tolerance. The use of ISO 2768 simplifies the drawing or model and makes sure all elements are tolerated.

ISO 2768-1 Tolerances for linear dimensions and angles
ISO 2768-2 Geometric tolerances (obsolete from the introduction of ISO 22081 in 2021)

The general tolerances use classes. When this standard is used the classes need to be stated. For example ISO 2768 mK

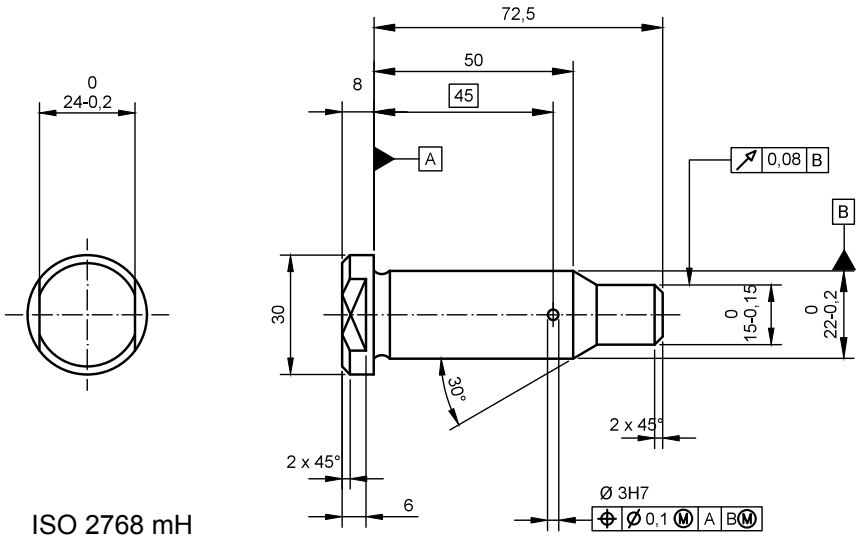


FIGURE 2.46W General tolerances according ISO 2768

Limits in mm for nominal sizes in mm								
Tolerance class	0,5 to 3	above 3 to 6	above 6 to 30	above 30 to 120	above 120 to 400	above 400 to 1000	above 1000 to 2000	above 2000 to 4000
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)	± 0,15	± 0,2	± 0,5	± 0,8	± 1,2	± 2	± 3	± 4
v (very coarse)	-	± 0,5	± 1	± 1,5	± 2,5	± 4	± 6	± 8

TABLE 2.4 General linear tolerances

The table is an example of the limits for nominal sizes as stated in ISO 2768-1. Tolerances for dimensions smaller the 0,5 mm need to be stated on the drawing. This standard includes linear standard tolerances, Tolerances for fillets and chamfers and angular tolerances. The angular tolerances define the global orientation of the line of line piece in degrees. Form deviation is not included in angular general tolerance.

ISO 2768-2, obselote for new design from 2021, is on the geometrical deviations. On form there are general tolerances on straightness and flatness. There are no general standards on roundness and cylindricity. On the related tolerances there are symmetry, parallelism, perpendicularity and circular run-out. There are no general tolerances on cylindricity, total run-out and profile.

straightness and flatness tolerances for nominal lenghts in mm						
Tolerance class	t/m 10	larger than 10 to 30	larger than 30 to 100	larger than 100 to 300	larger than 300 to 1000	larger than 1000 to 3000
H	0,02	0,05	0,1	0,2	0,3	0,4
K	0,05	0,1	0,2	0,4	0,6	0,8
L	0,1	0,2	0,4	0,8	1,2	1,6

TABEL 2.5 General tolerances for straightness and flatness

Unless otherwise stated workpieces exceeding the general tolerances can not merely rejected when the workpieces is meeting it's function.

General tolerances apply to all form tolerances except:

- Cylindricity (fittings might use the envelope requirement)
- Parallelism (
- Angularity
- Coaxiality
- Line profile and Surface profile
- Position
- Total run-out

OTHER GENERAL TOLERANCES

- ISO 3302: Rubber dimensional tolerances
- ISO 8062: Dimensional en geometrical tolerances for molded part
- ISO 13920: General tolerances for welded constructions
- ISO 20457: Plastic molded parts
- ……..

3 MODIFYING SYMBOLS

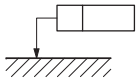
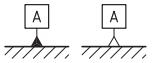
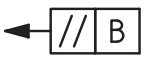
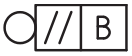
Description	Symbol
Toleranced feature indication	
Datum feature indication	
Datum target indication	
Theoretically exact dimension	
Median feature	Ⓐ
Unequal tolerance zone (offset)	UZ
Between	
From ... to	(obsolete 2017)
Projected tolerance zone	Ⓟ
Maximum material requirement	Ⓜ
Least material requirement	Ⓛ
Free state condition (non-rigid parts)	Ⓕ
All around (profile)	
Envelope requirement	Ⓔ
Combined zone	CZ
Minor diameter	LD
Major diameter	MD
Pitch diameter	PD
Line element	LE (obsolete 2017)
Not convex	NC (obsolete 2017)
Any cross-section	ACS
Direction feature	
Collection plane	
Intersection plane	
Orientation Plane	

TABLE 3.1 Geometric modifying symbols

3.1 COMBINED ZONE

Modifying symbols are symbols that communicate additional information about the tolerancing of a part.or datum.

Individual tolerance zones having the same value applied to different features may be indicated as show.

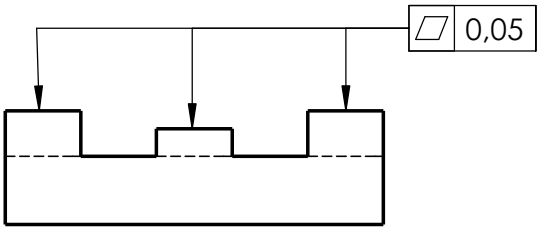


FIGURE 3.1 Individual tolerance zones

When the features are to be contained in a combined tolerance zone, this is to be indicated by “CZ” after the tolerance value in the tolerance frame.

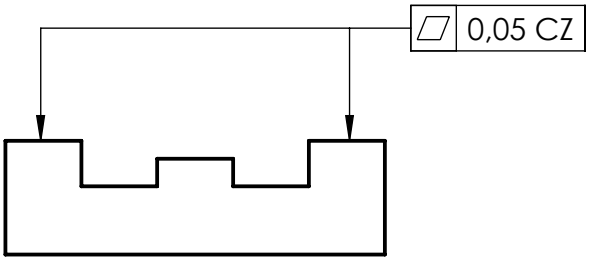


FIGURE 3.2 Combined zone tolerance

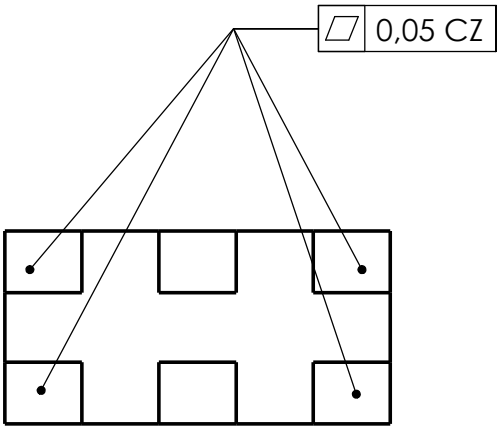


FIGURE 3.3 Combined zone tolerance

Where CZ is indicated in the tolerance frame, all the related individual tolerance zones shall be constrained in location and in orientation. Surfaces are considered one surface and axises as combined axis.

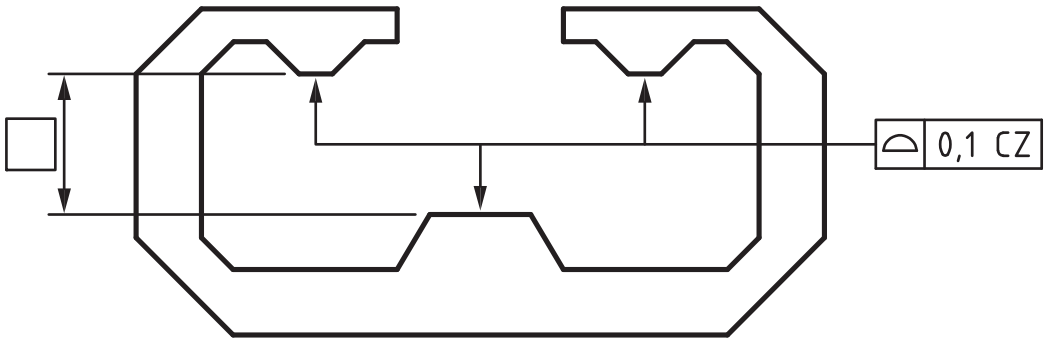


Figure 3.4 Combined zone tolerance

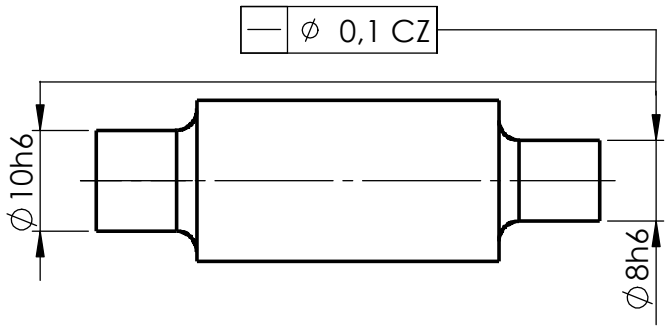


Figure 3.5 Combined zone tolerance

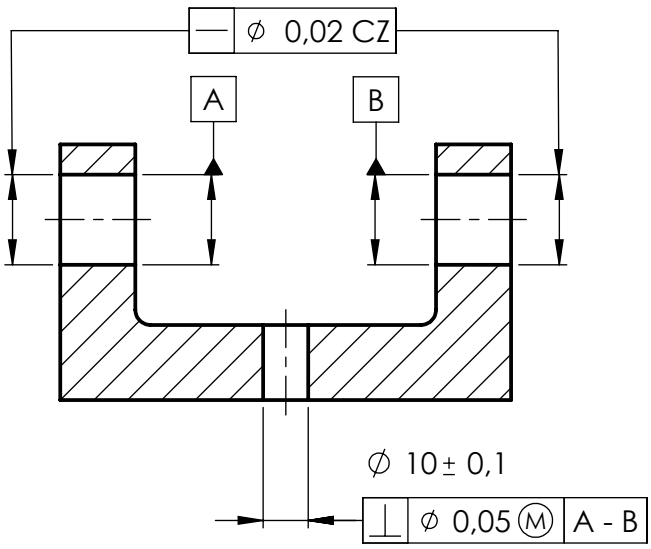


Figure 3.6 Datum with combined zone straightness tolerance

3.2 PROJECTED TOLERANCE FEATURE

Projected zone tolerance is used to assure that there will be no interference between a screw, stud or dowel pin and the mating part. The projected zone, an orientation (angularity) and position tolerance, applies to the virtual external projection of the feature indicated by the symbol Ⓟ.

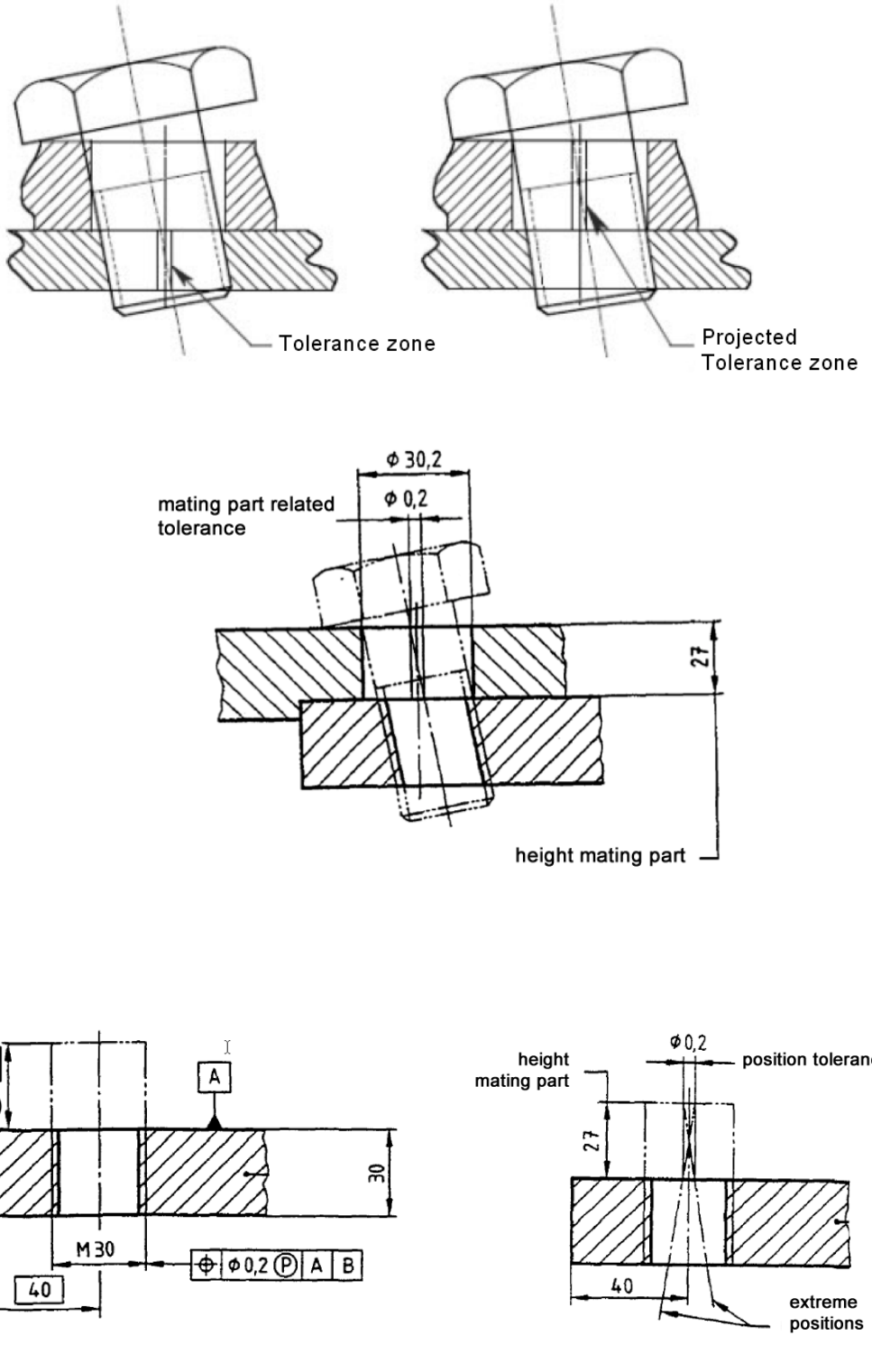


FIGURE 3.7 Projected tolerance zone

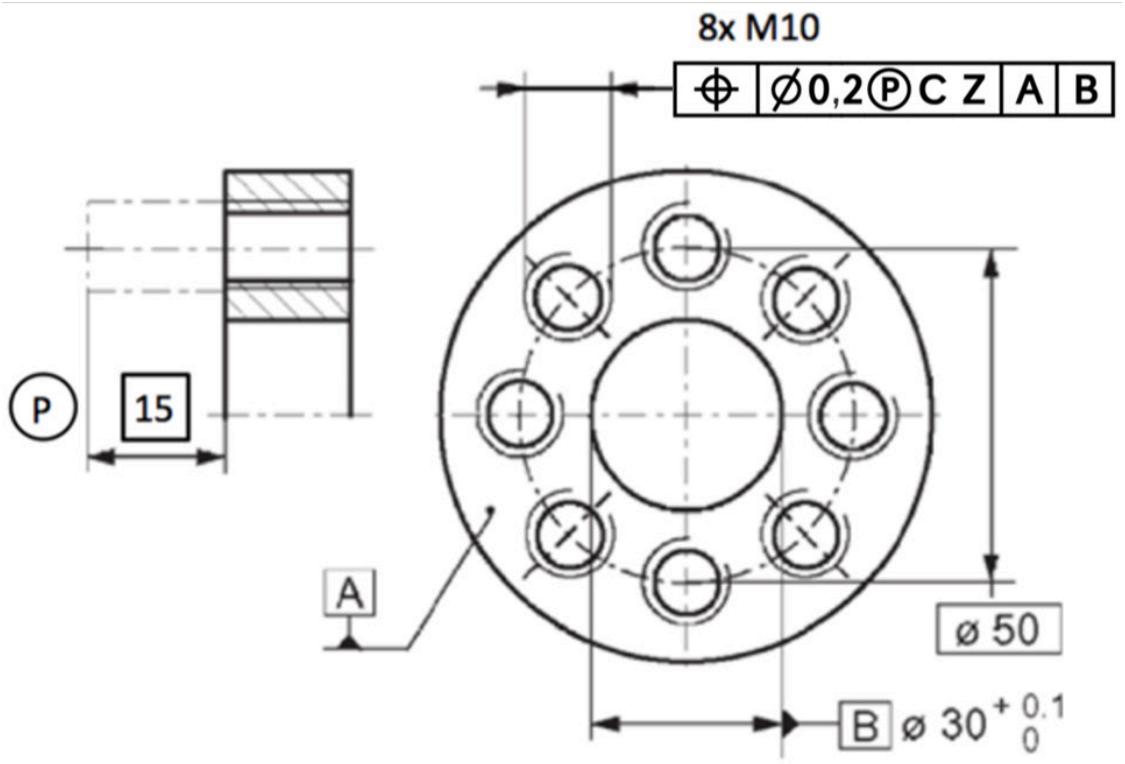


Figure 3.8 Projected tolerance zone

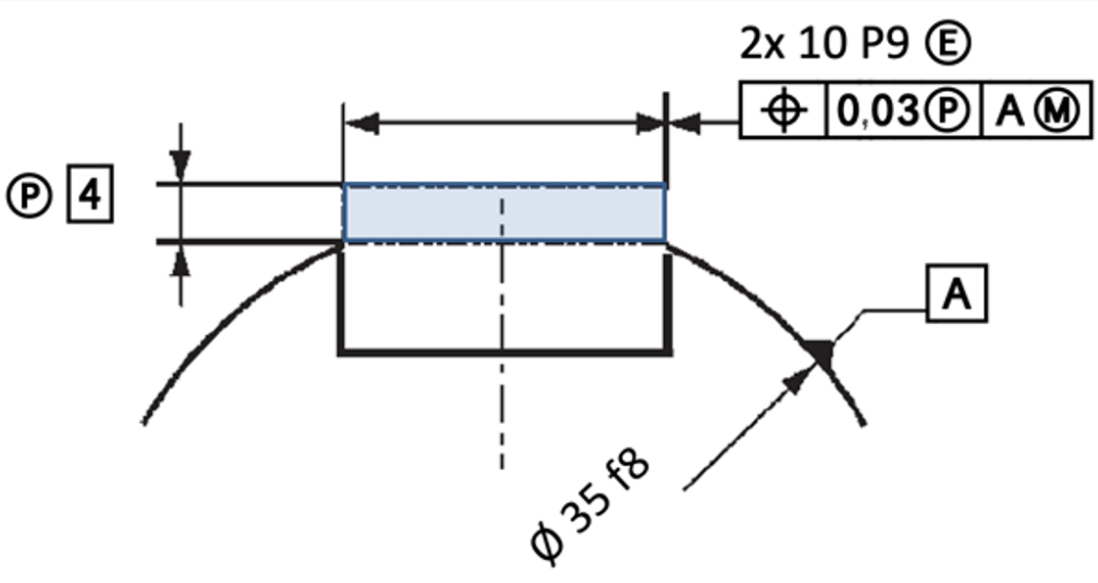


Figure 3.9 Projected tolerance zone

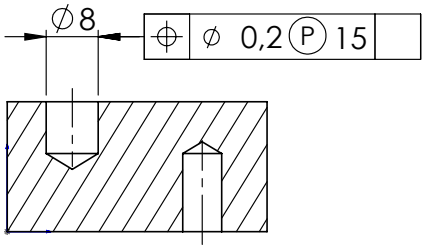


FIGURE 3.10 Indirect identification of the tolerance zone

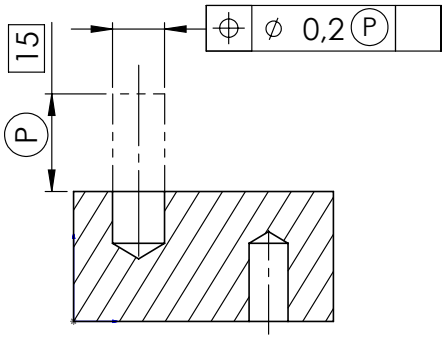


FIGURE 3.11 Direct identification of the tolerance zone

The external projection is to be indicated by a long-dashed double dotted narrow line. The length of the external projection shall be indicated by a Theoretical Exact Dimension (TED) or might be placed in the second compartment of the tolerance control frame after the symbol Ⓟ.

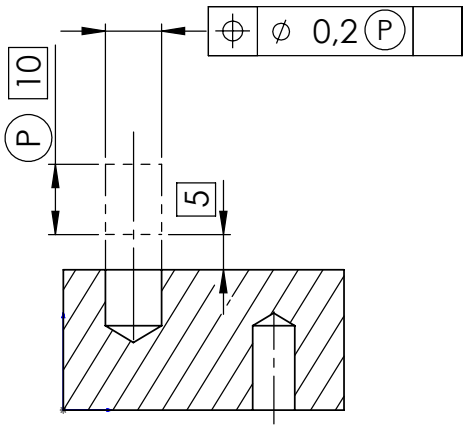


FIGURE 3.12 Tolerance zone with offset

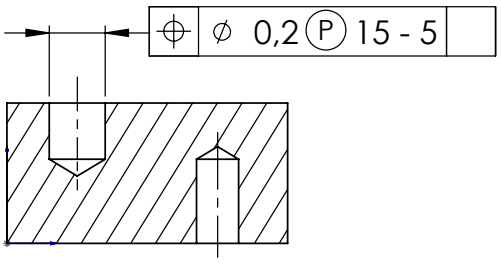
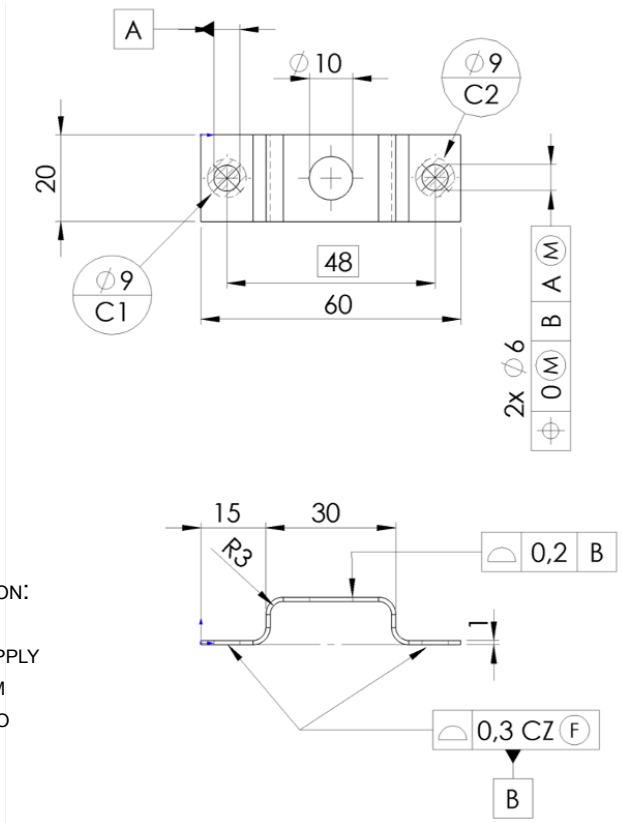


FIGURE 3.13 Tolerance zone with offset

When indirectly indicated the first value after the modifier indicates to the farest limit of the extended feature the second value indicates the nearest limit. The lenght of the extended feature is the difference between the values.

3.3 FREE STATE CONDITION

Tolerancing a non-rigd part needs to be done according ISO 10579. All dimensions and tolerances are to be met in restrained condition. A note is required stating how the part is to be restrained. Only tolerances modified with the modifier Ⓕ apply when the parts is in the free state condition, free of applied forces.



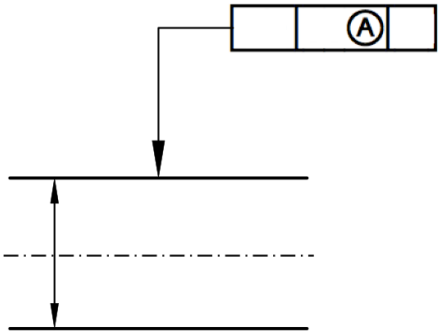
ISO 10579 RESTRAINED CONDITION:

DIMENSIONS AND TOLERANCES APPLY WITH PART RESTRAINED TO DATUM FEATURE B WITH 10N APPLIES TO EACH DATUM TARGET C

FIGURE 3.14 Free state

Here the only tolerance that applies in the free state is the profile of a surface tolerance on the two surfaces established by datum B. All other tolerances apply in the restrained condition.

3.4 MEDIAN FEATURE



Axis or media plane as toleranced feature or datum feature

FIGURE 3.15 Media plane or axis

3.5 UNEQUAL ZONE

The profile surface shall be contained between two equidistant surfaces enveloping spheres of defined diameter equal to the tolerance value. The centres of the spheres have an offset from the theoretical surface. The direction and value of the offset is given between brackets. A "+" sign indicates"out of the material" and the "-" sign "into the material".

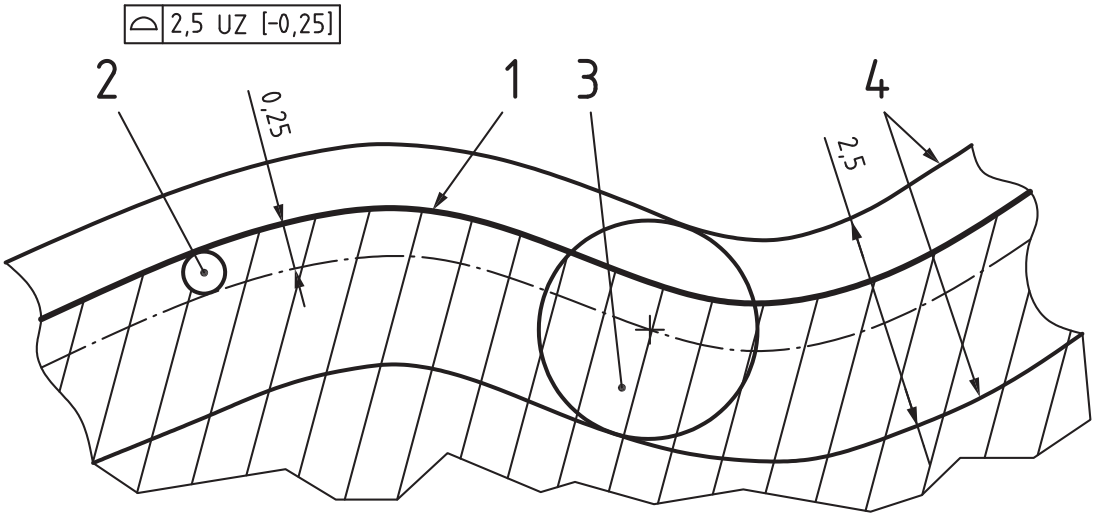


FIGURE 3.16 Unequal zone

1. theoretical profile in this example, the material is below the profile
2. sphere to define the offset theoretical profile
3. sphere to define the tolerance zone
4. limits of the tolerance zone

3.6 ALL AROUND

The ***all around*** symbol in figure 3.16 is used to extend the tolerance zone to include all the features in the view shown.

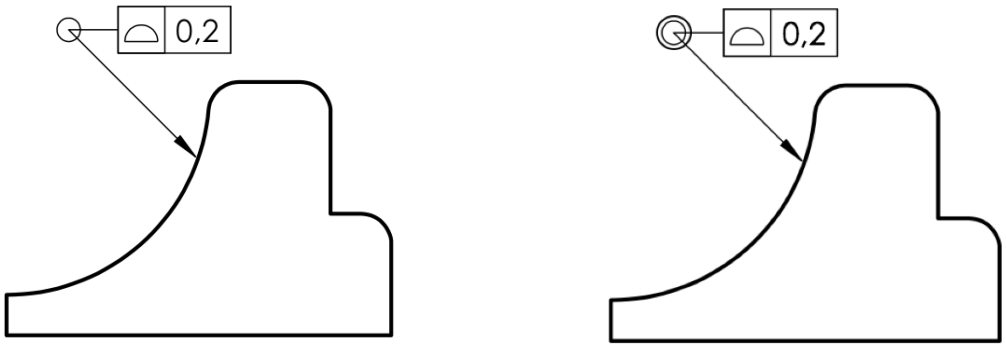


FIGURE 3.17 All around and all over symbol

3.7 BETWEEN

Figure 3.17 shows the ***between symbol*** used to indicate that the tolerance applies to the features between two points. Indicated by the TED dimension is the minimal surface. Minimal surface results that the TED of 8 can only get smaller.

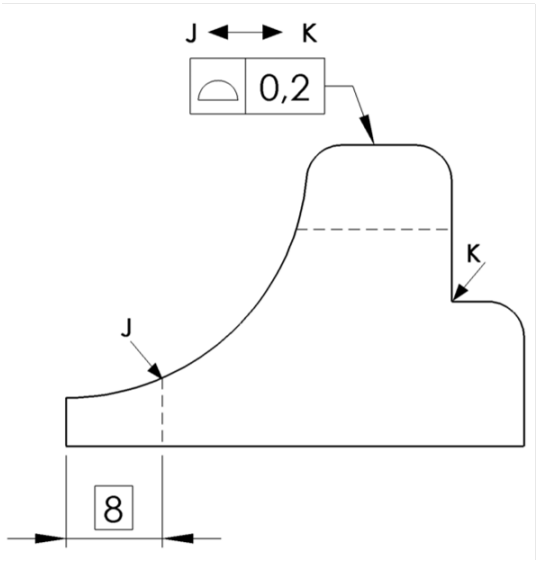


FIGURE 3.18 Between symbol

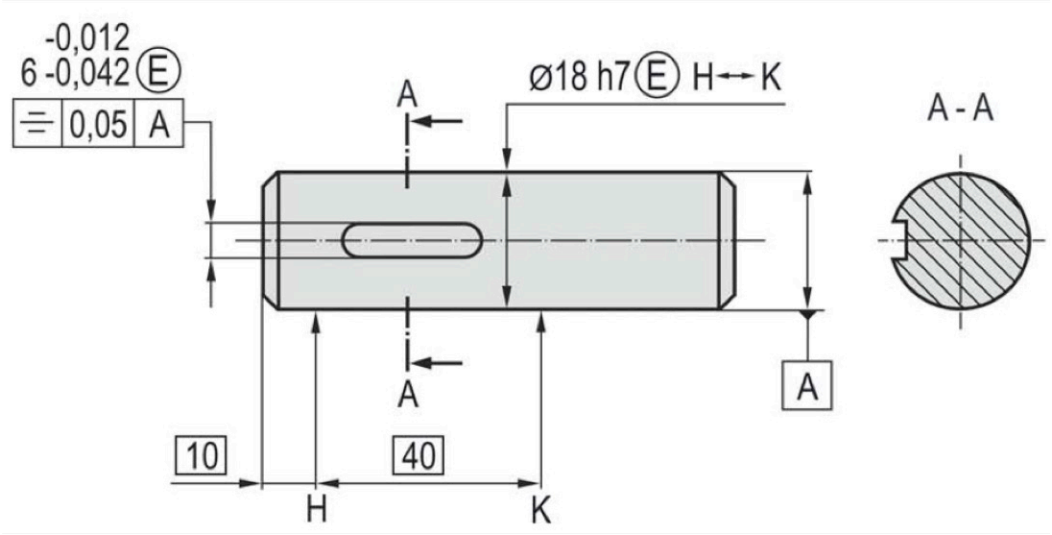


Figure 3.19 Between symbol

The diameter of the shaft is ø 18h7. This is size only. Between H and K is the additional envelope requirement. The keyway also needs an envelope requiremnet to avoid it to be a banana shape. The keyway is positioned symmytical to the shaft centerline.

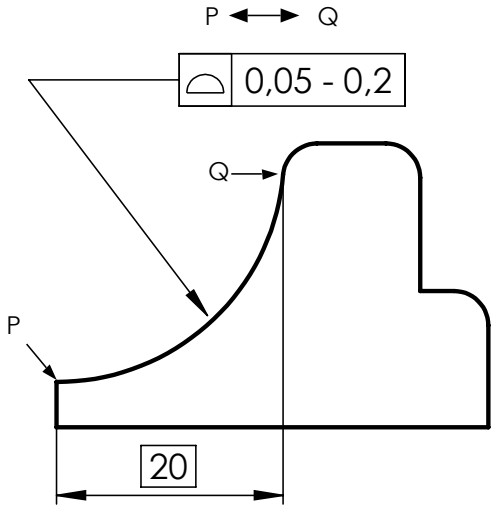


Figure 3.20 Variable tolerance zone

The tolerance zone value is variable along the considered feature.
Here at 'P' the tolerance is 0,05 and increases proportional till 0,2 at 'Q'.

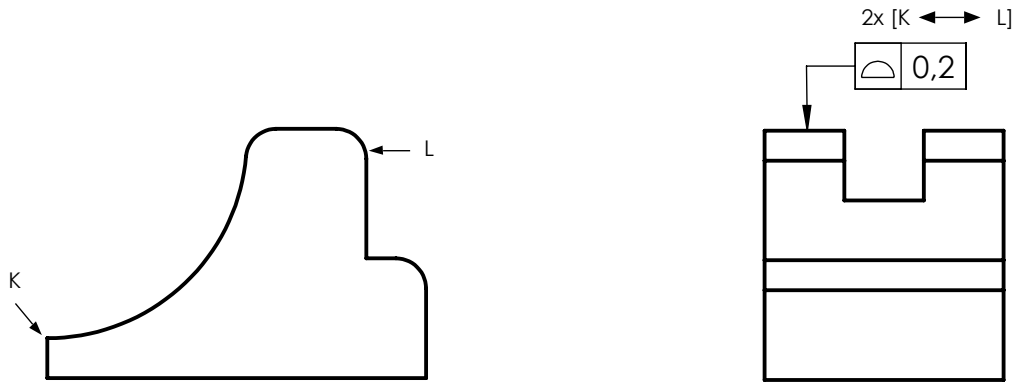


Figure 3.21 More surfaces

To extend the profile tolerance to include more surfaces in this case the notation ‘2x’ is used. The letters indentifying the start and end need to be placed between brackets.

3.8 SCREW THREADS, MULTIPLE SPLINES AND GEARS

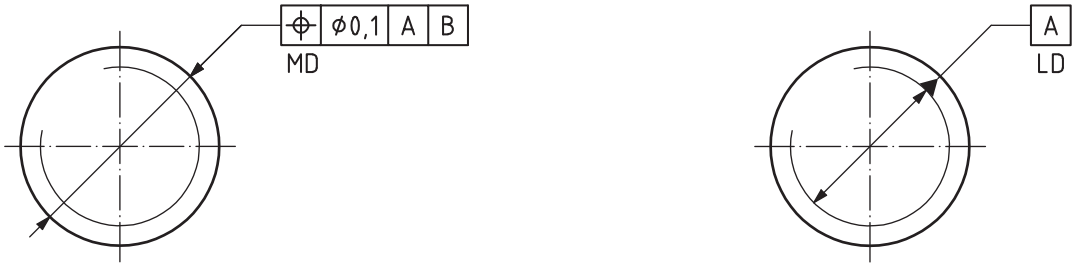


Figure 3.22 Screw threads, multiple splines and gears

Tolerances and datums specified for screw threads apply to the axis derived from the pitch cylinder, unless otherwise specified, e.g. “MD” for major diameter and “LD” for minor diameter. Tolerances and datums specified for muliple splines and gears shall designate the specific feature to which they apply, i.e. “PD ” for pitch diameter, “MD ” for major diameter or “LD ” for minor diameter.

3.9 MAXIMAL MATERIAL REQUIREMENT

The Maximum Material Requirement (MMR) and the Least Material Requirement (LMR), indicated by Ⓜ or Ⓛ in the feature control frame datum frame or both, take into account the mutual relationship of the size and the geometrical tolerance of interrelated features. When MMR or LMR is specified size and geometrical tolerance are transformed into one collective requirement. A to the Ⓜ or Ⓛ additional modifier is the Reciprocity Requirement (RPR) modifier indicated by Ⓡ. MMR, LMR and RPR are standardized in ISO 2692.

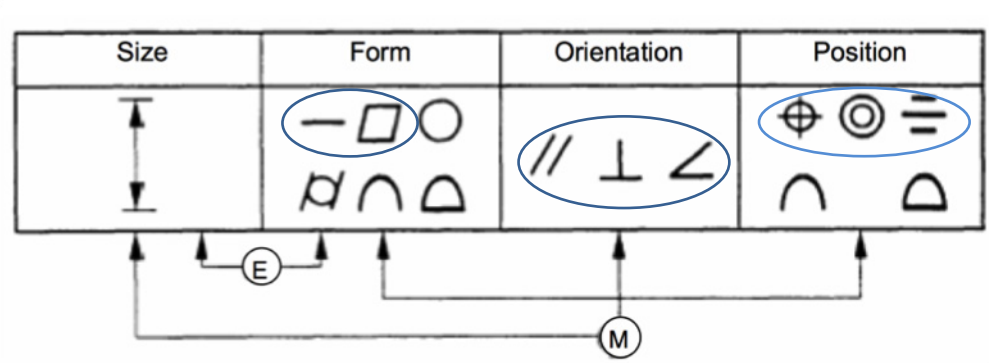


FIGURE 3.23 Relations between size and GD&T characteristics

The MMR can be applied when there is a functional relation between size and form or between size and GD&T characteristics.

Definition	Abbreviation	Meaning
Maximum Material Condition	MMC	The state of a feature where the feature is everywhere at its maximum of material • For a shaft this is the maximum diameter • For a hole this is the minimum diameter
Maximum Material Size	MMS	The limit of size where the material of a feature is at its maximum material condition • Shaft: maximum limit of size • Hole: minimum limit of size
Maximum Material Virtual Size	MMVS	The collective effect of the maximal material size and the geometrical tolerance followed by Ⓜ • Shaft: MMVS = MMS + geometrical tolerance • Hole: MMVS = MMS - geometrical tolerance
Maximum Material Vitual Condition	MMVC	A feature limiting boundery of perfect form and of MMVS.

TABLE 3.2 MMR definitions

As the size of a feature departs from Maximum Materiaal Condition (MMC) towards Least Material Condition (LMC) a bonus tolerance is gained. The bonus tolerance is the difference between the actual feature size and the MMC of the feature.

The bonus tolerance is added to the geometric tolerance specified in the feature control frame without violating the Maximal Material Virtual Condition (MMVC) not vice versa !.

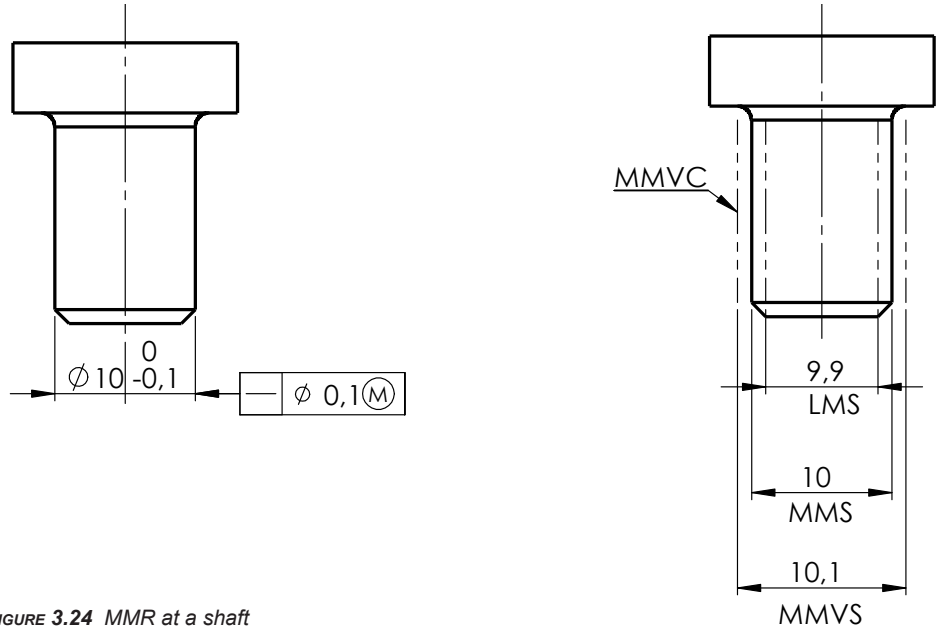


FIGURE 3.24 MMR at a shaft

In figure 3.24 the bonus tolerance is maximal 0,1 the difference between the actual feature size at LMC which is 9,9 and the MMC of the feature being 10. The bonus tolerance of 0,1 is added to the geometrical tolerance of 0,1 resulting in a maximal allowed straightness deviation of 0,2 at LMC.

DIAMETER SHAFT	STRAIGHTNESS TOLERANCE	BONUS TOLERANCE	TOTAL TOLERANCE
10	0,1	0	0,1
9,95	0,1	0,05	0,15
9,9	0,1	0,1	0,2

TABLE 3.3 Straightness applied to a shaft at MMC

The MMVS which is not to be violated:

- For the shaft : MMVS = MMS + geometrical tolerance
  MMVS = 10 + 0,1 = 10,1
  (the shaft should fit in a functional gauge of 10,1 mm)

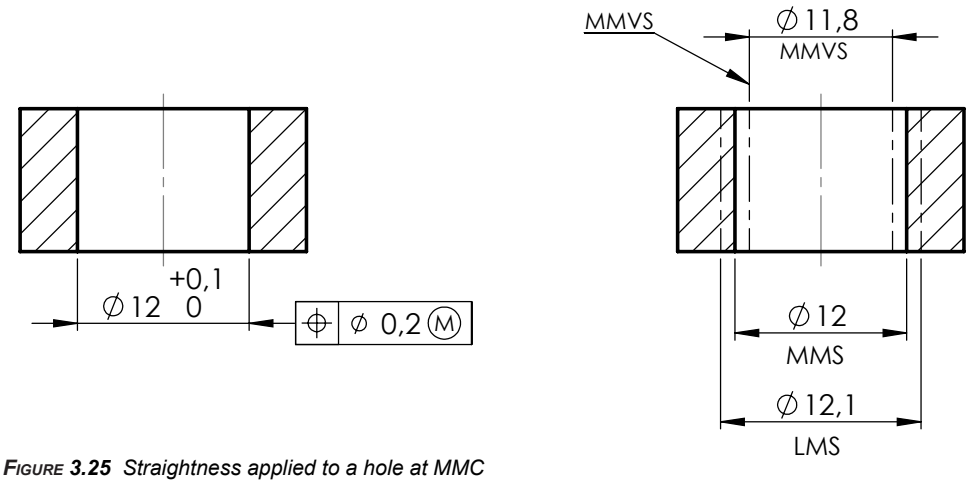


FIGURE 3.25 Straightness applied to a hole at MMC

The bonus tolerance is maximal 0,1 the difference between the actual feature size at LMC which is 12,1 and the MMC of the feature being 12. The bonus tolerance of 0,1 is added to the geometrical tolerance of 0,2 resulting in a maximal straightness deviation of 0,3 at LMC.

DIAMETER HOLE	STRAIGHTNESS TOLERANCE	BONUS TOLERANCE	TOTAL TOLERANCE
12	0,2	0	0,2
12,05	0,2	0,05	0,25
12,1	0,2	0,1	0,3

TABLE 3.4 Straightness applied to a shaft at MMC

The MMVS which is not to be violated:

- For the hole: MMVS = MMS - geometrical tolerance
  MMVS = 12 - 0,2 = 11,8
  (the holes should take a functional gauge pin of 11,8 mm)

3.9.1 MMR TRANSFERING MMVS

The intended function of the parts could be an assembly where the functional requirement is that at the same time the two planar faces shall be in contact AND the pin shall fit over the total length into the hole.

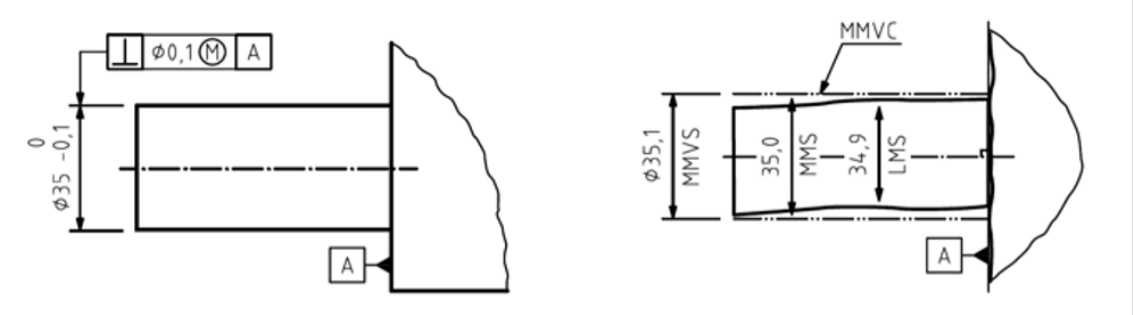


FIGURE 3.26 Perpendicularity applied to a shaft at MMC and resulting MMVS

The MMVS of the shaft is MMS + GD&T equals 35 + 0,1 = 35,1mm
To meet the requirement the MMVS from the shaft is transferred to the hole.

For the hole we need a size and a perpendicular requirement making up a MMVS of 35,1mm.
Choosing 0,1 for the perpendicularity results in a minimum size of the hole of ø 35,2mm.

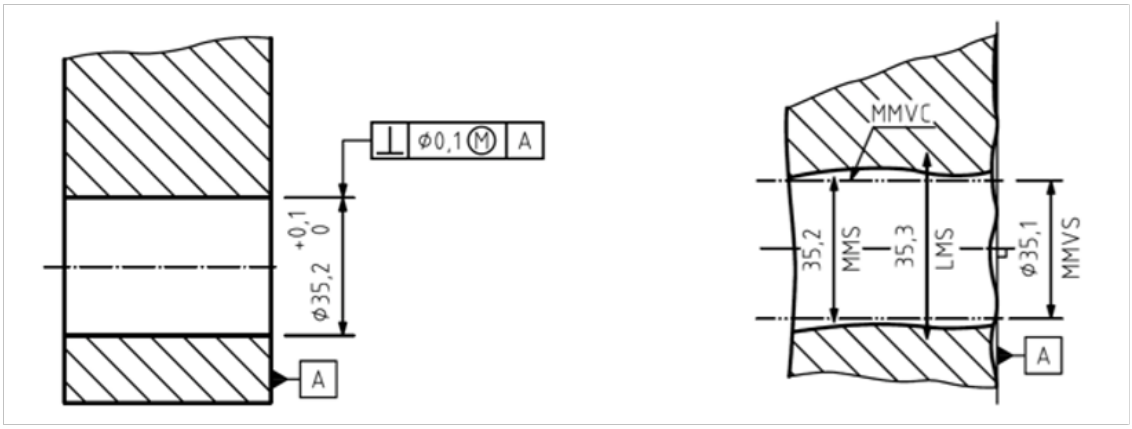


FIGURE 3.27 Transferring MMVS to define perpendicularity and size of a mating hole

The intended function of the part could be an assymbly where the functional requirement is that the two planar faces A shall be in contact and the two planar faces B sahll both simultaneously be in contact with a plane (plane not shown) and the total pin fits.

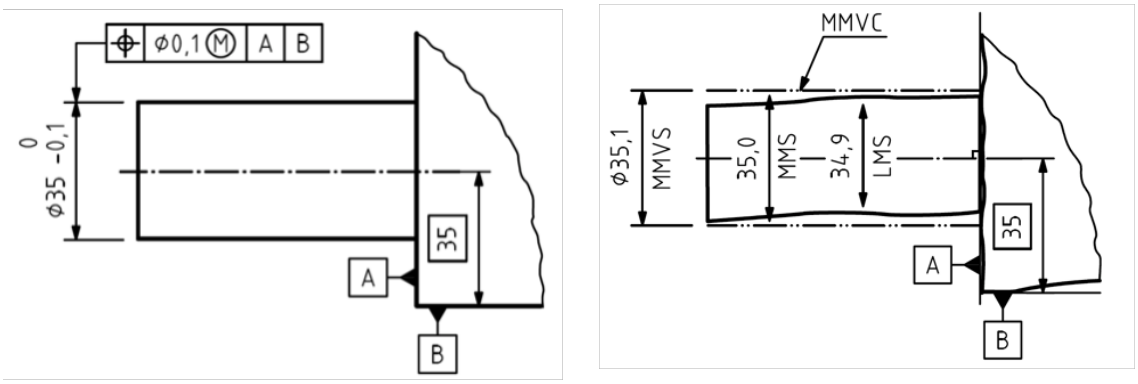


FIGURE 3.28 Location applied to a shaft at MMC and resulting MMVS

The MMVS of the shaft is MMS + GD&T equals 35 + 0,1 = 35,1mm.
To meet the requirement the MMVS from the shaft is transferred to the hole.

For the hole we need a size and a location requirement making up a MMVS of 35,1mm.
Choosing 0,1 for the location results in a minimum size of the hole of ø 35,2mm.

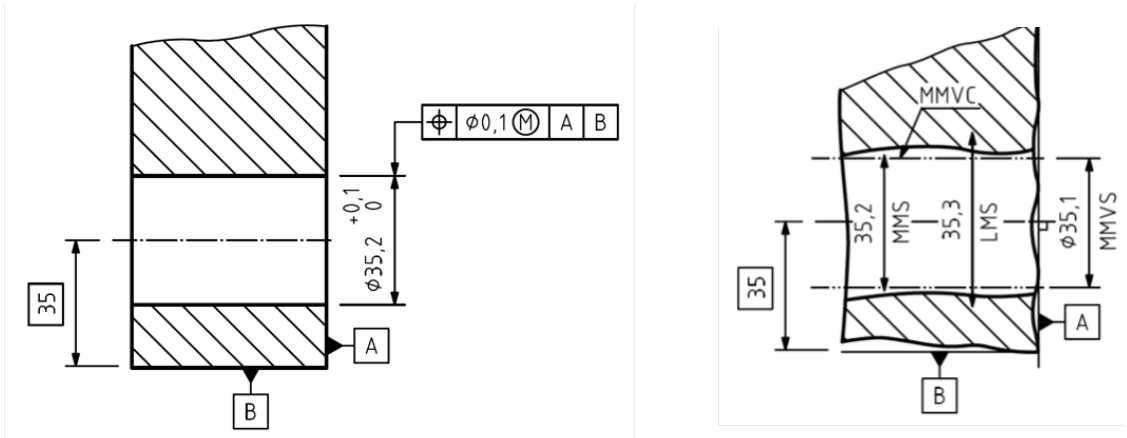


FIGURE 3.29 Transferring MMVS to define location and size of a mating hole

3.9.2 MMR ON A DATUM FEATURE

For the pattern of the 6 holes, indicated as CZ, a MMR applies. The allowed position deviation is the range between 0,2mm to 0,4mm depending on the size of the holes of ø 5mm.

The MMVS for the 6 holes: MMVS = MMS - GD&T = 4,9 - 0,2 = 4,7.

Since datum A constrains two rotations datum feature B is used to constrain the translations only. There is no MMR for the datum in the position tolerance frame. Therefore the requirement for the 6 holes is Regardless Feature Size (RFS) of datum B. Independent of the size of B.

Datum feature B has a straightness requirement at MMC.

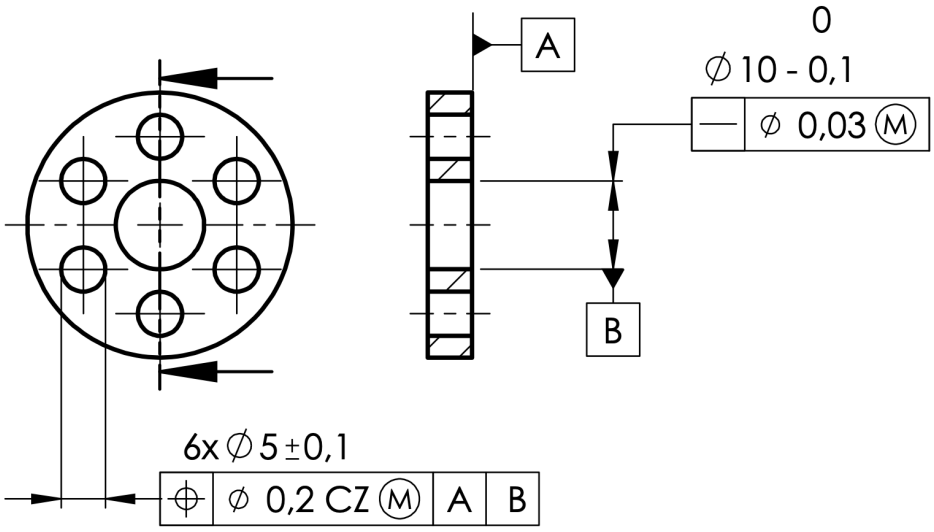


FIGURE 3.30 Position tolerance for the 6 holes RFS of datum B.

In the tolerance frame for the 6 holes in figure 3.31 also the datum feature B is at MMR. This allows a float of the pattern 6 holes (CZ) with respect to the theoretical center of datum B. The float is 0,1mm from the datum size. The straightness is not included for the float.

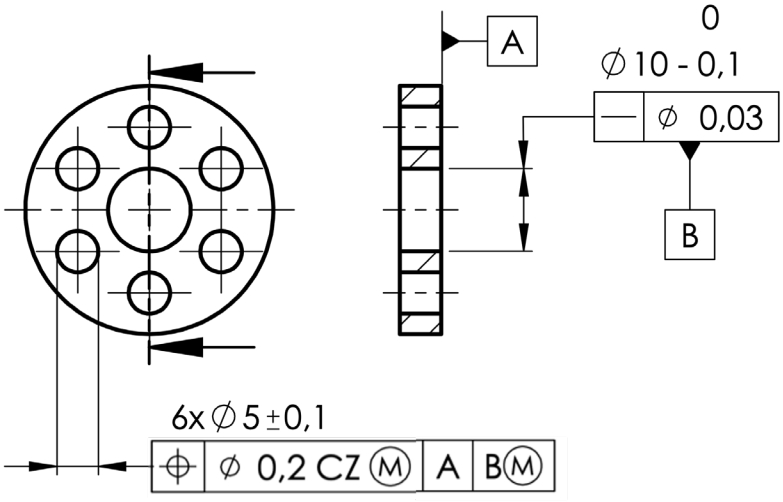


FIGURE 3.31 Straightness to be disregarded.

The straightness tolerance on datum feature B has a MMR the straightness needs to be included in the float. The maximal allowed float is 0,1 + 0,03 = 0,13mm.

The MMVS which is not to be violated for the datum is :

MMVS = MMS - geometrical tolerance
MMVS = 9,9 - 0,03 = 9,87

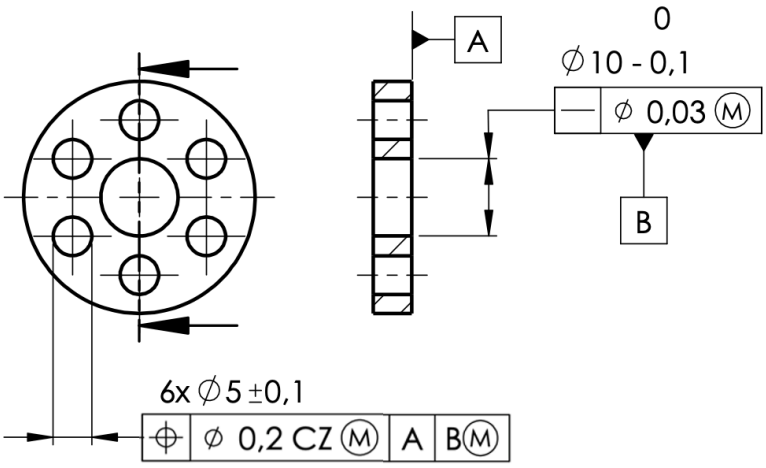
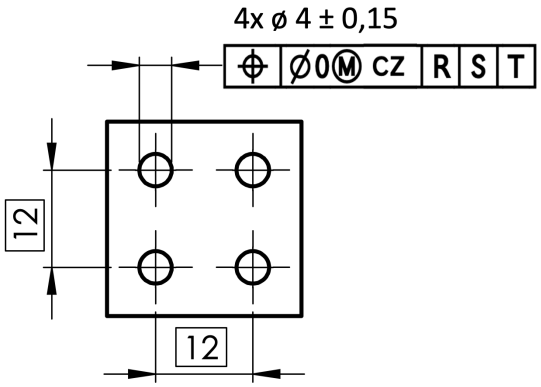


Figure 3.32 Straightness to be regarded, pattern of holes seven pin caliber.

3.9.3 MAXIMAL MATERIAL REQUIREMENT 0

When the tolerance is not distributed on size and position but is provided for both for random distribution this is indicated by 0 Ⓜ. Here also the MMVS must not be violated.

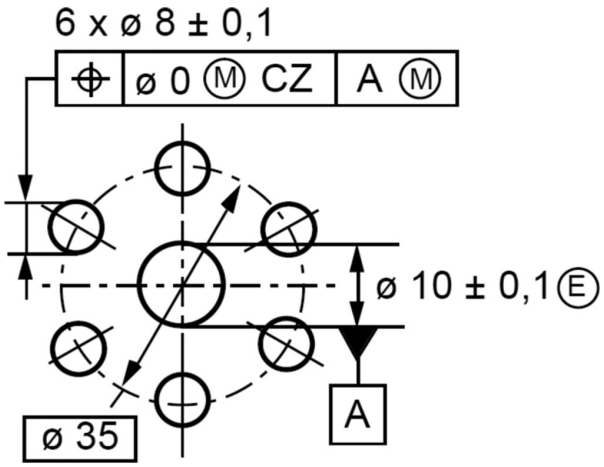


Position tolerance is ø 0,3 at LMS and is ø 0 at MMS. MMS equals MMVS.

The use of 0 Ⓜ allows to freely choose were the tolerance is applied to either size, geometrical tolerance or any ratio between these tolrances.

Figure 3.33 MMC with 0 tolerance

3.9.4 GAUGING CZ AND SZ



On the left the 6 holes make up a pattern size de modifier CZ is used. A functional gauge includes 6 pins. of :
MMVS = MMS - GD&T = 7,9 - 0 = 7,9mm.

Datum A is defined at MMC. A center pin of 9,9mm completes the gauge.

FIGURE 3.34 Pattern of holes and datum feature at MMC

The lower illustration states the 6 independent, holes. The gauge is an indexing gauge one pin of 7,9mm and one pin of 9.9mm.

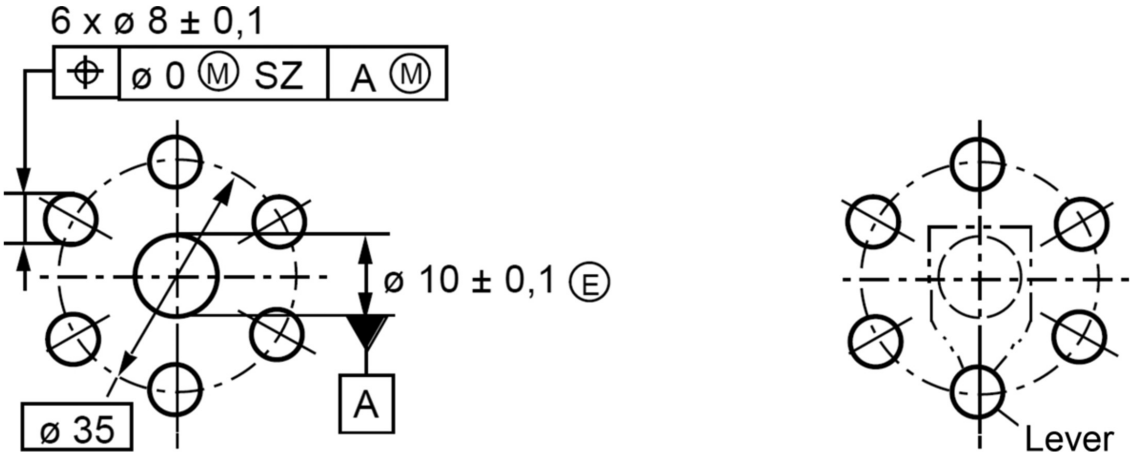


FIGURE 3.35 Requirement on the individual holes (SZ), holes and datum feature at MMC

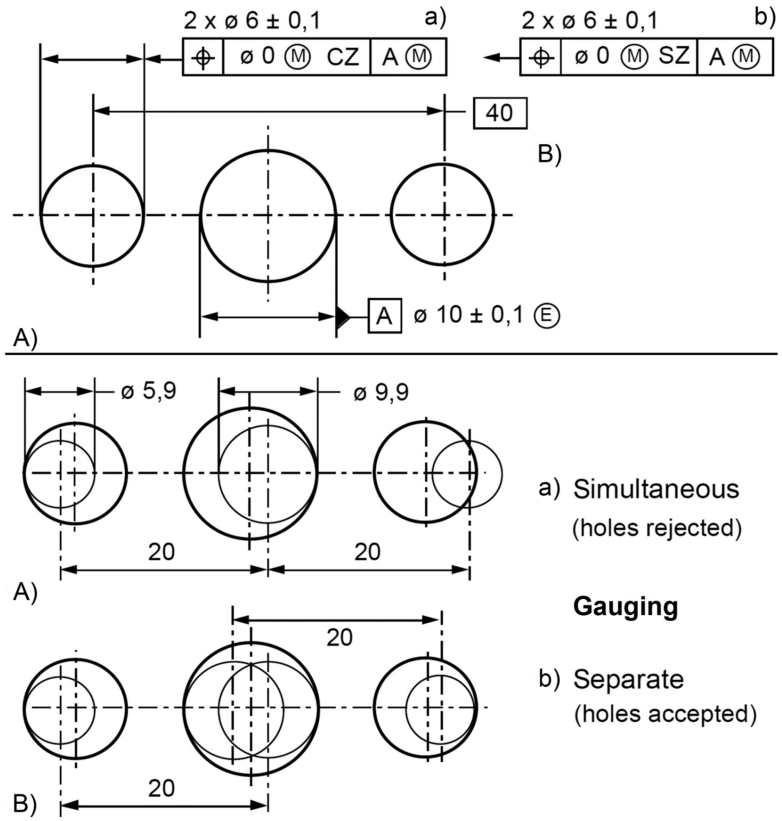


Figure 3.36 Gauging CZ and SZ

The top illustration concerns the relation of two features, 2 holes of ø 6mm, to each other and to a single datum.

Tolerance frame a) has a CZ requirement. Both holes of ø 6 need to meet the requirement with respect to the datum simultaneously.
Tolerance frame b) has a SZ requirement Both holes of ø 6 need to meet the requirement with respect to the datum independently.

The lower two illustrations show realised positions of the holes. When gauging simultaneously, a gauge with three pins, the two holes have a position to the datum simultaneously then the gauging results in a reject.
When the gauging is from the datum to each hole separate, a gauge with two pins gauging from the datum to one hole and a second gauging to the other hole, then the holes are accepted.

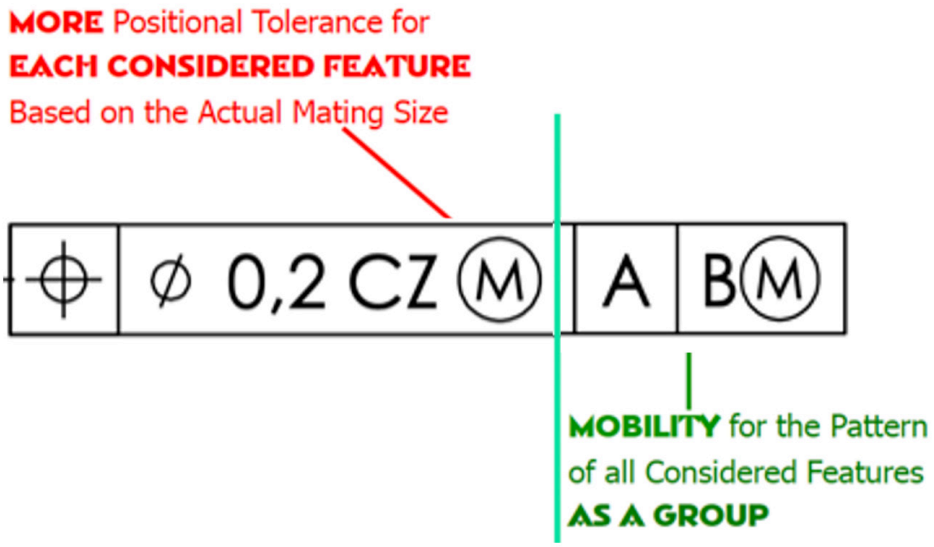


FIGURE 3.37 MMR for location and the datum feature creating mobility / float

- **Allows BONUS TOLERANCE**
  Max. allowable tolerances promotes LOWEST MANUFACTURING COSTS
- **Allows FUNCTIONAL GAUGING**
- **Allows DATUM FEATURE SHIFT (float / mobility)**
- **Common Usage: 100% INTERCHANGEABILILTY**
  Parts assembled with clearance fit
- **Must apply to FEATURE OF SIZE**

FIGURE 3.38 Benefits using MMR

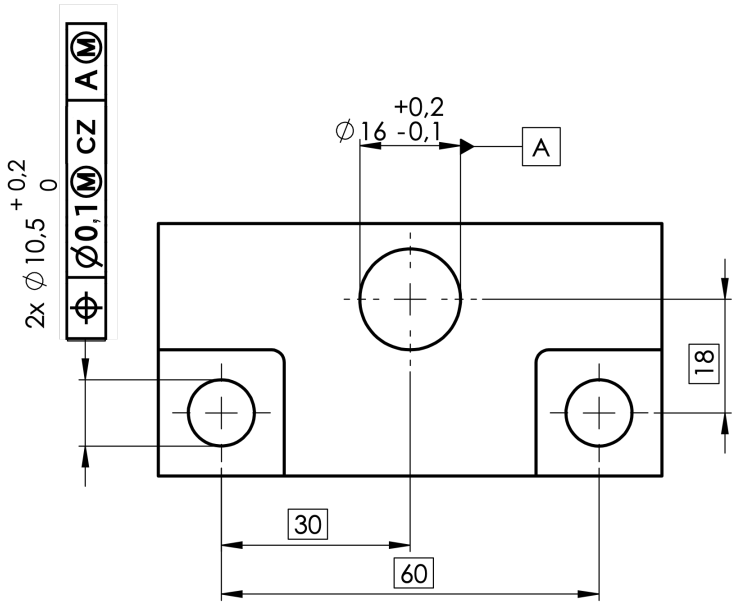


FIGURE 3.39 Block with MMR on toleranced features and datum feature

3.9.5 APPLICATIONS OF MAXIMAL MATERIAL REQUIREMENT

	Tolerance	Symbol	Toleranced feature						Datum feature			
					Axis	Media face	Surface			Axis	Media face	Surface
Unrelated tolerance	Line profile	⌒										
	Straightness	⏤			x							
	Roundness	○										
	Surface profile	⌓										
	Flatness	▱				x						
	Cylindricity	⌭										
Related tolerance	Line profile, datum	⌒										
	Surface profile, datum	⌓										
	Angularity	∠			x	x				x	x	
	Parallelism	//			x	x				x	x	
	Perpendicularity	⊥			x	x				x	x	
	Position	⌖			x	x				x	x	
	Coaxiality / Concentricity	◎			x					x		
	Symmetry	⌯			x	x				x	x	
	Circular run-out	↗										
	Total run-out	⌰										
Ⓜ	Max. Mat. Requirement		Ⓛ	Least Mat. Requirement								

TABLE 3.5 Possible applications of MMR and LMR



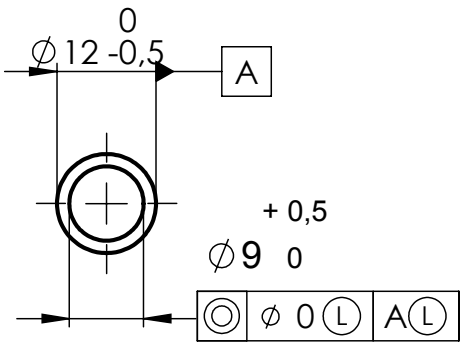
THE PRECISE FUNCTIONAL REQUIREMENT CAN OFTEN ONLY BE INDICATED USING THE MAXIMAL MATERIAL REQUIREMENT. ONLY THEN THE LARGEST POSSIBLE TOLERANCES APPEAR RESULTING IN THE MOST ECONOMIC PRODUCED PART.

3.10 LEAST MATERIAL CONDITION

The least material condition (LMC) is the condition where the feature of size has the least amount of material.

Definition	Abbreviation	Meaning
wLeast Material Condition	LMC	The state of a feature where the feature is everywhere at its minimum material • For a shaft the minimum diameter • For a hole the maximal diameter
Least Material Size	LMS	The limit of size where the material of a feature is at its minimum. • Shaft: minimum limit of size • Hole: maximum limit.
Least Material Virtual Size	LMVS	The minimum material size plus or minus the geometrical tolerance. • Shaft: LMVS = LMS - geometrical tolerance • Hole: LMVS = LMS + geometrical tolerance
Least Material Virtual Condition	LMVC	A feature limiting boundary of form and LMVS

TABLE 3.6 Least Material Condition



The LMVC of the cylinders may not be abused. The result is a minimal wall thickness.

FIGURE 3.46 Coaxiality with LMR 0 Ⓜ

In the example above, the minimum wall thickness of 1 mm has been achieved when the workpiece has been manufactured to LMC at 0 Ⓛ.

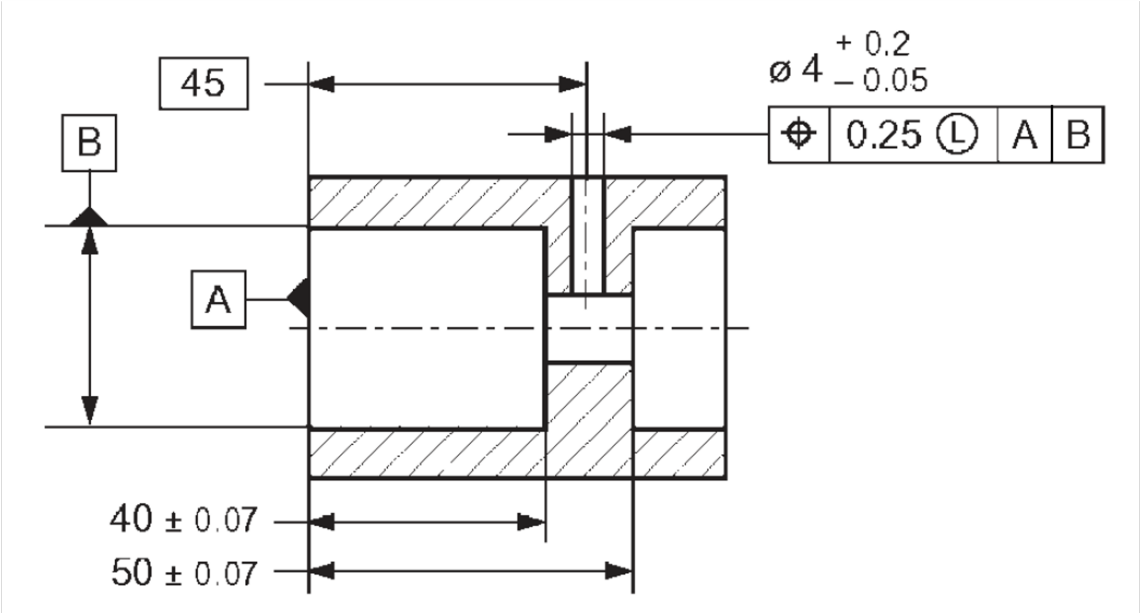


Figure 3.47 Controlling minimum wall thickness between the hole ø 4 and the bottom of datum hole B

3.11 RECIPROCITY REQUIREMENT

When MMR, Ⓜ, or LMR, Ⓛ, is specified, size and geometrical tolerance are transformed into one collective requirement. Where the geometric tolerance will be enlarged by the size tolerance not vice versa.

Added to the Ⓜ or Ⓛ additional modifier is the reciprocity requirement indicated by Ⓡ to indicate that the size tolerance is allowed to increase by the difference between the geometrical tolerance and the geometrical deviation.

Reciprocity requirement is indicated on drawings as an additional requirement to MMR or LMR by the symbol Ⓡ placed after the symbol Ⓜ, or the symbol Ⓡ placed after the symbol Ⓛ.

Reciprocity requirement is only applicable for tolerated features not for datums.

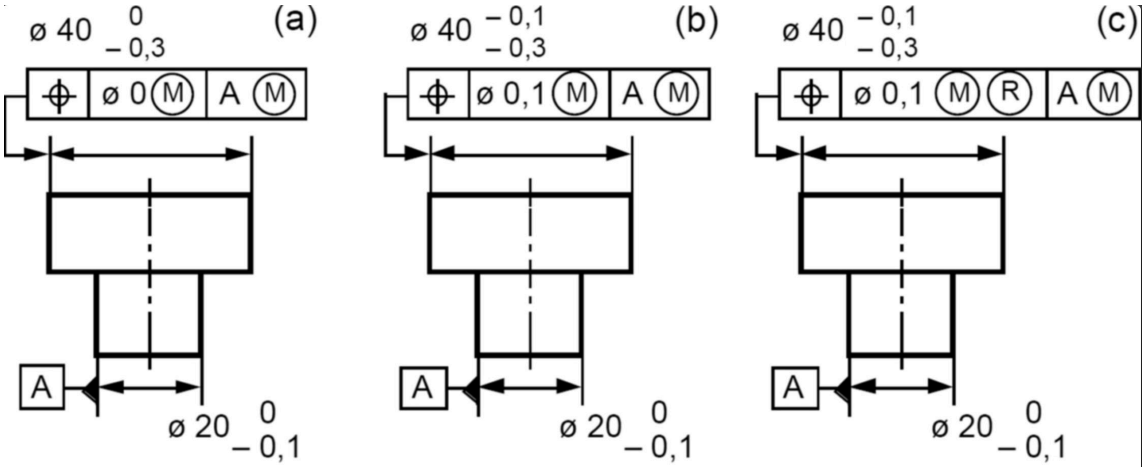


FIGURE 3.48 Reprocity requirement with MMR

The same MMVC applies to the requirements a) and b). In both cases MMVS is ø 40. The difference is the size tolerance. In case b) the size tolerance can not be enlarged by the non utilized coaxiality tolerance.

In case c) the RPR has the same effect as 0 Ⓜ. The total tolerance may be used for deviations of size, form, orientation and location in an arbitrary way. However the RPR requirement is a recommendation for the tolerance distribution.

MODULE II

4 TOLERANCES OF FORM

4.1 FLATNESS

Flatness is one of the four form tolerances. A flatness tolerance in general limits the flatness of a planar surface.

Flatness tolerances are often used to ensure a good joint. Applications are sealings, load disdribution, appearance, assembly mating surfaces.

Flatness tolerance can also be used to define the flatness of a media plane of a feature of size. It is important to be able to determine whether a flatness applies to a surface or to a feature of size dimension because the interpretation for each is different.

4.1.1 FLATNESS APPLIED TO A SURFACE

SYMBOL	TYPE OF TOLERANCE	DATUM REFERENCE	ALLOWABLE TOLERANCE MODIFIERS	ALLOWABLE DATUM MODIFIERS
▱	Form	None	Ⓕ	N/A
Flatness tolerance zone Space between parallel planes				

TABLE 4.1 Flatness applied to a surface

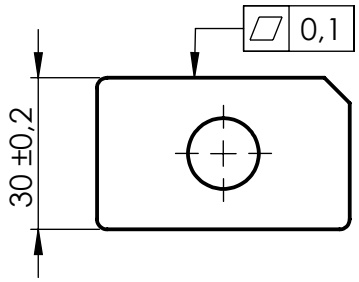


FIGURE 4.1 Flatness applied to a surface

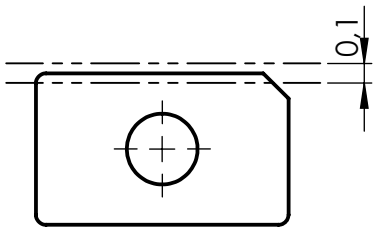
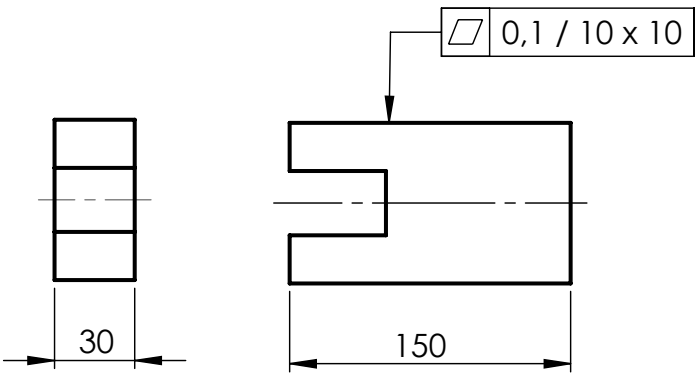


FIGURE 4.2 Interpretation

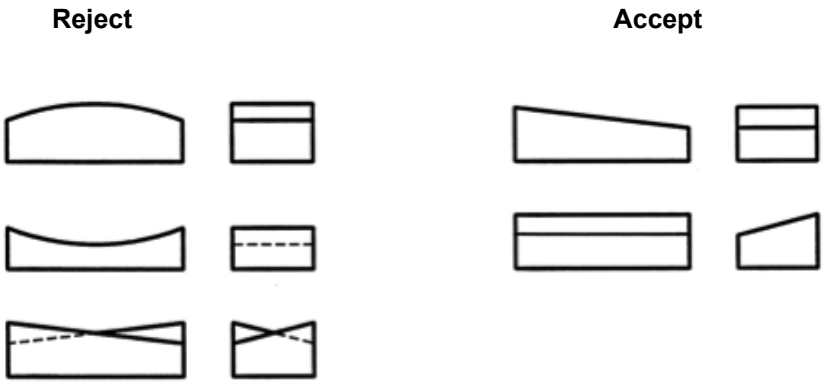


The requirement is that every random area of 10 x10 mm. has to be flat within 0,1 mm.

FIGURE 4.3 Flatness of restricted area

The requirement is that the top surface has to be flat within a tolerance of 0,1 mm. The tolerance zone for flatness tolerance is the space between two parallel planes. The distance between the planes is equal to the flatness tolerance value.

Possible form deviations and their acceptance:



Figuur 4.4 Possible deviations

4.1.2 ASSESMENT OF FLATNESS DEVIATIONS OF A SURFACE

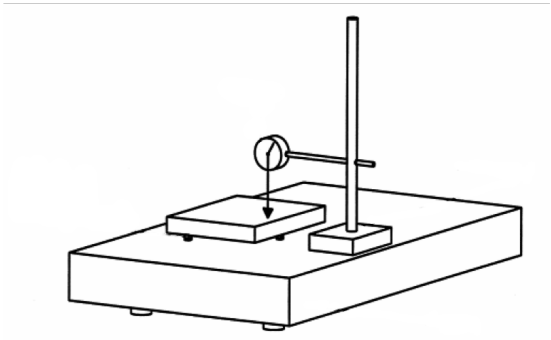


Figure 4.5 Verifying flatness

The deviations of form of a surface are measured with respect to an ideal plane. The flatness deviation is difference between the largest and smallest value.

Flatness is a form requirement and has no location or orientation. Therefor three adjustable support are used to correct for orienational effects.

In the example the requirement is a flatness of 1 mm. The left measurements would give a reject. After an orientational correction the part is accepted.

Reject			Accept		
(- 0,8)	- 0,5	0	(- 0,4)	- 0,5	0
0	(+ 0,5)	+ 0,2	0	(+ 0,5)	+ 0,2
+ 0,2	+ 0,1	0	+ 0,2	+ 0,1	+ 0,4

Figure 4.6 Rejects and accepts

4.1.3 FLATNESS APPLIED TO A FEATURE OF SIZE AT MMC

SYMBOL	TYPE OF TOLERANCE	DATUM REFERENCE	ALLOWABLE TOLERANCE MODIFIERS	ALLOWABLE DATUM MODIFIERS
▱	Form	None	Ⓕ Ⓜ Ⓛ	N/A
Flatness tolerance zone Space between parallel planes				

FIGURE 4.2 Flatness applied to a feature of size

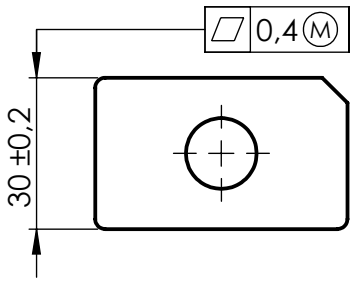


FIGURE 4.7 Flatness to a feature of sizes

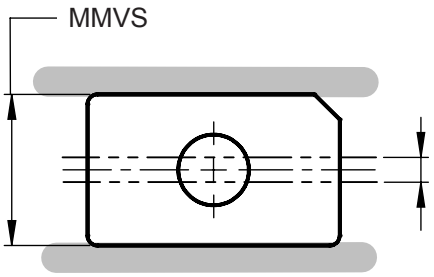


FIGURE 4.8 Interpretation

If the flatness is applied to a feature of size the media plane must be within the tolerance zone. The part has to be within the limit of size but using the Ⓜ (or Ⓛ) modifier allows a bonus tolerenace. The part has to meet the MMVC. The tolerance may be larger than the size tolerance.

SIZE	TOLERANCE ZONE WIDTH
30,2	0,4
30,0	0,6
29,8	0,8

TABLE 4.3 Virtual condition at MMC

Assesment of the flatness of a feature of size at MMR can be done using a caliber.

Exercise



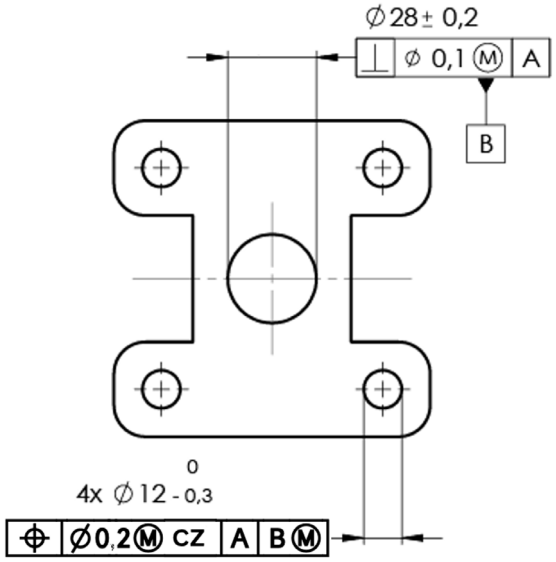


Figure 4.9 Exercise

- Where does the flatness tolerance apply to?
- What is the shape and size of the flatness tolerance zone?
- Are there more form requirements?

4.2 STRAIGHTNESS

Straightness tolerance is one of the four form tolerances. Straightness tolerances are important to ensure good joint design by defining the allowable straightness deviation of a surface line element. Straightness can also be used to define the allowed straightness deviation of an axis to ensure assembly. It is important to be able to determine whether a straightness applies to a surface or to a feature of size dimension because the interpretation for each is different.

4.2.1 STRAIGHTNESS APPLIED TO A SURFACE

SYMBOL	TYPE OF TOLERANCE	DATUM REFERENCE	ALLOWABLE TOLERANCE MODIFIERS	ALLOWABLE DATUM MODIFIERS
—	Form	No	Ⓕ	None
Straightness tolerance zone Two parallel lines.				

TABLE 4.4 Straightness

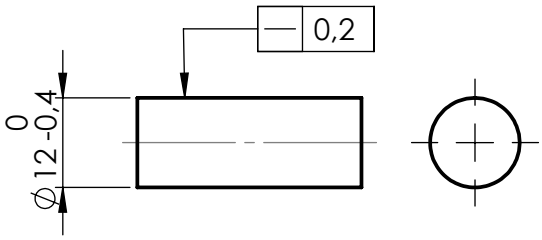


FIGURE 4.10 Straightness of a line element

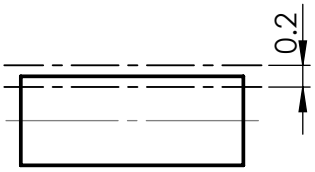


FIGURE 4.11 Interpretation

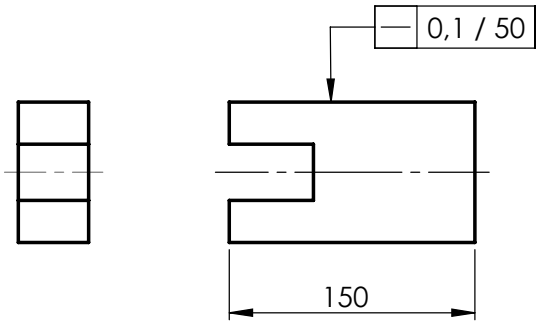


FIGURE 4.12 Straightness over a restricted length

The requirement is that every random length of 50 mm has to be straight within 0,1 mm.

The requirement is that each line element has to be within a tolerance of 0,2 mm. The tolerance zone for the straightness tolerance is the space between two parallel lines. The distance between the lines is equal to the straightness tolerance value.

Possible form deviations and their acceptance:

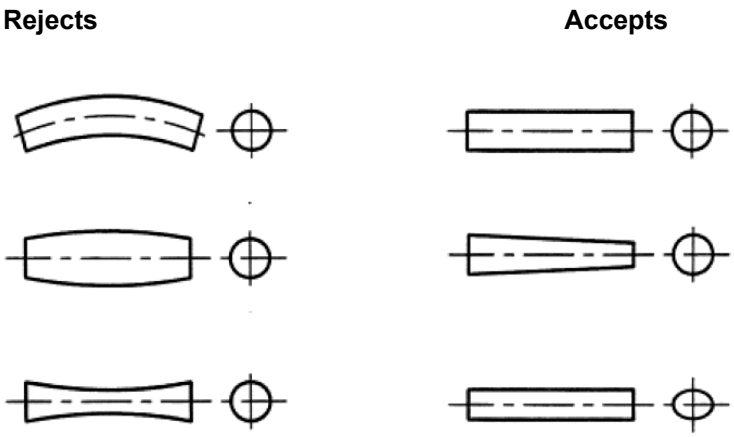


FIGURE 4.13 Possible deviations

4.2.2 ASSESEMENT OF STRAIGHTNESS OF A SURFACE

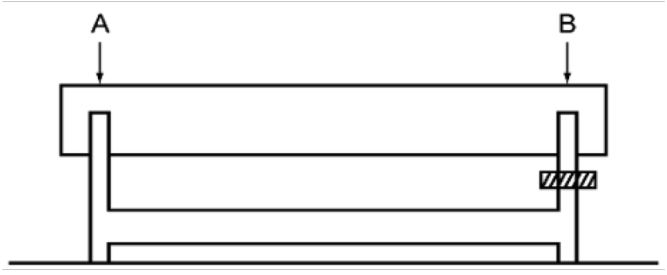


FIGURE 4.14 Assesment of straightness of a surface

The deviations of the line of the workpiece surface from a geometrical ideal reference are measured. The workpiece is leveled using the adjustment at B creating a reference line. The difference between the largest and smallest measurement is the straightness deviation.

4.2.3 STRAIGHTNESS APPLIED TO A FEATUSE OF SIZE

SYMBOL	TYPE OF TOLERANCE	DATUM REFERENCE	ALLOWABLE TOLERANCE MODIFIERS	ALLOWABLE DATUM MODIFIERS
—	Form	No	Ⓕ Ⓛ Ⓜ	None
Straightness tolerance zone Cylinder				

TABLE 4.5 Straighness of a feature of size

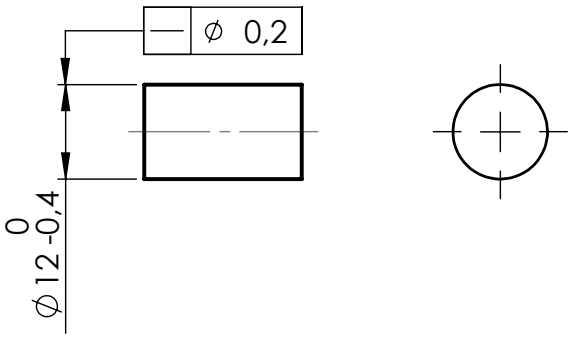


FIGURE 4.15 Straightness of a feature of size

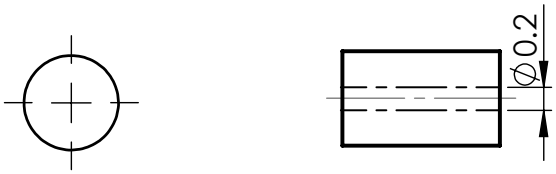


FIGURE 4.16 Interpretation

The requirement is that the straightness tolerance of the axis has to be within a cylindrical tolerancezone of ø 0,2 mm.

Possible form deviations and their acceptance:

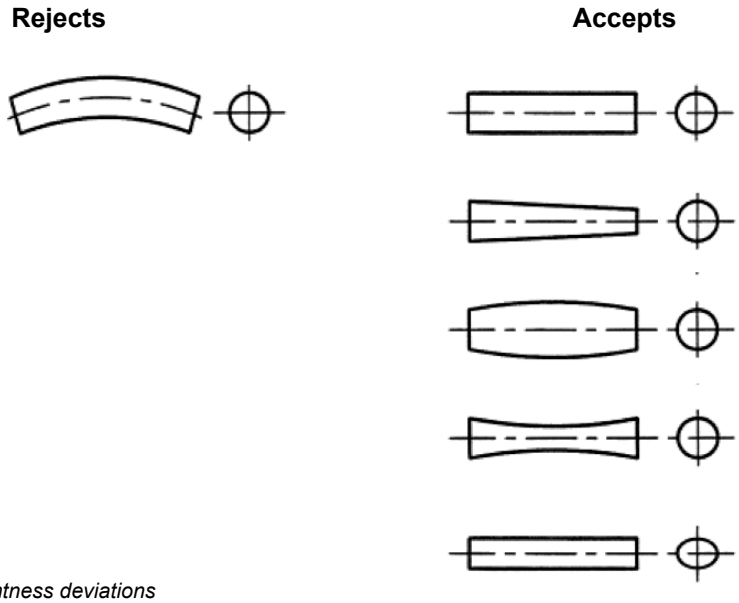


FIGURE 4.17 Straightness deviations

4.2.4 STRAIGHTNESS AT MMC

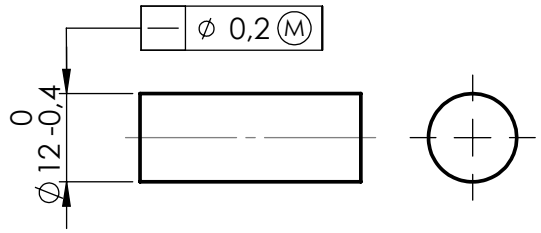


FIGURE 4.18 Straightness of a feature of size

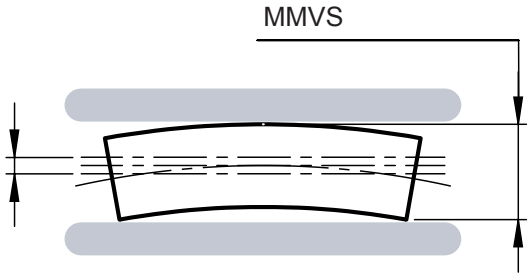


FIGURE 4.19 Virtual condition at MMC

When straightness is applied to a feature of size the axis must be within the tolerance zone. The part has to be within the limit of size but using the Ⓜ (or Ⓛ) modifier allows a bonus tolerance. The part has to meet the virtual condition of ø 12,2. The geometric tolerance may be larger than the size tolerance.

SIZE	TOLERANCE ZONE DIAMETER
12	0,2
11,8	0,4
11,6	0,6

TABLE 4.6 Virtual condition at MMC

4.2.5 STRAIGHTNESS AND COMBINED ZONE REQUIREMENT

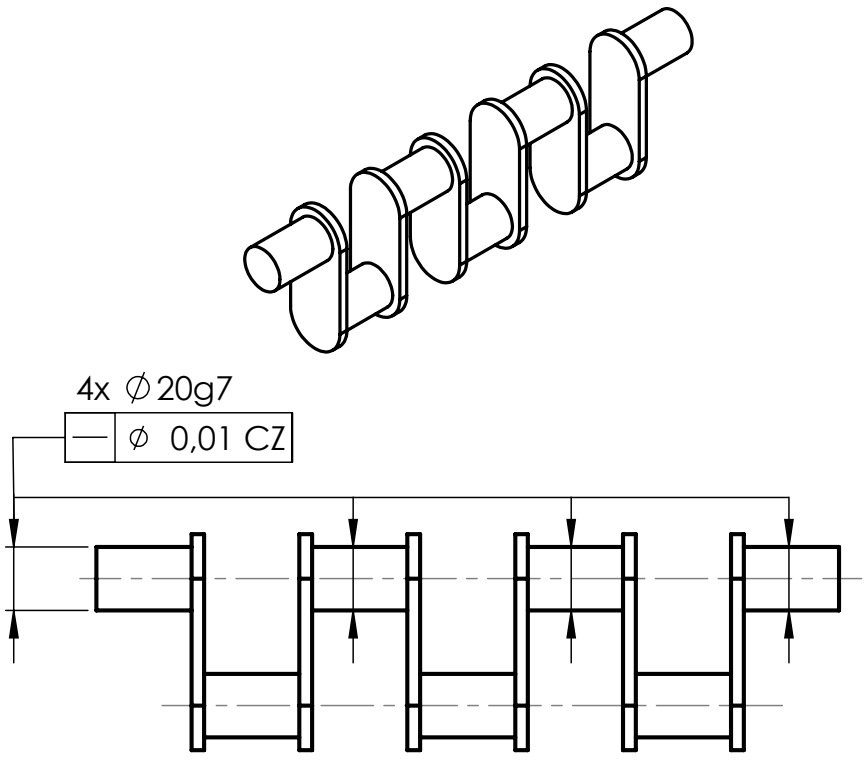


FIGURE 4.20 Straightness and CZ

4.2.6 ASSESSMENT OF STRAIGHTNESS DEVIATION OF A FEATURE OF SIZE

A simple approximation method for assing the straightness deviation of an axis is using two dial indicators. The values have to be determinated at several, at least three, measuring positions.

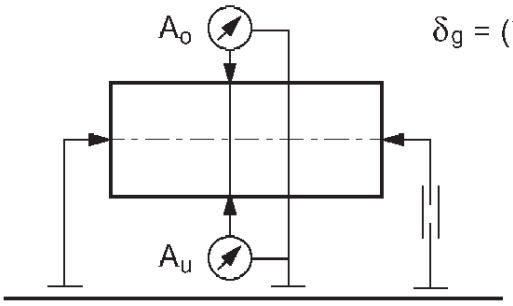


FIGURE 4.21 Assesment of straightness deviation of an axis

EXERCISE

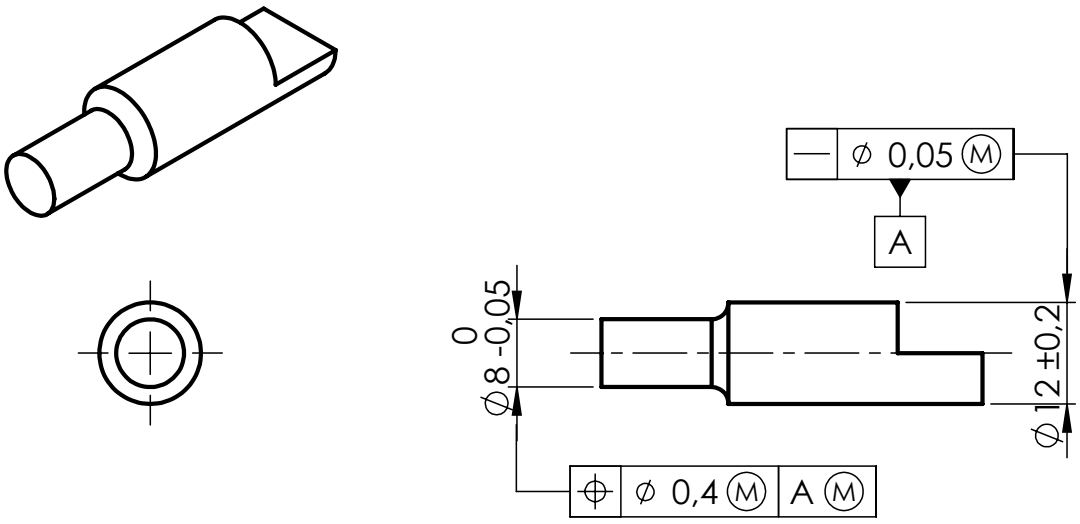


FIGURE 4.22 Exercise

4.3 ROUNDNESS

Roundness is one of the four form tolerances. The roundness tolerance is used to limit deviations of circular elements.

A roundness tolerance is often used to ensure a good seal, rolling characteristics, bearing support or appearance.

4.3.1 ROUNDNESS APPLIED TO A SURFACE

SYMBOL	TYPE OF TOLERANCE	DATUM REFERENCE	ALLOWABLE TOLERANCE MODIFIERS	ALLOWABLE DATUM MODIFIERS
○	Form	None	Ⓕ	None
Roundness tolerance zone Two concentric circles				

TABLE 4.7 Roundness

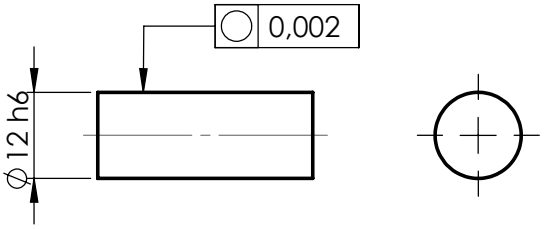


FIGURE 4.23 Roundness

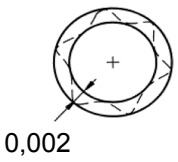


FIGURE 4.24 Interpretation

The requirement is that the roundness has to be within a tolerance of 0,002 mm at any cross section. The tolerance zone for roundness is de distance between two concentric circles. The difference between the radii is equal to the roundness tolerance.

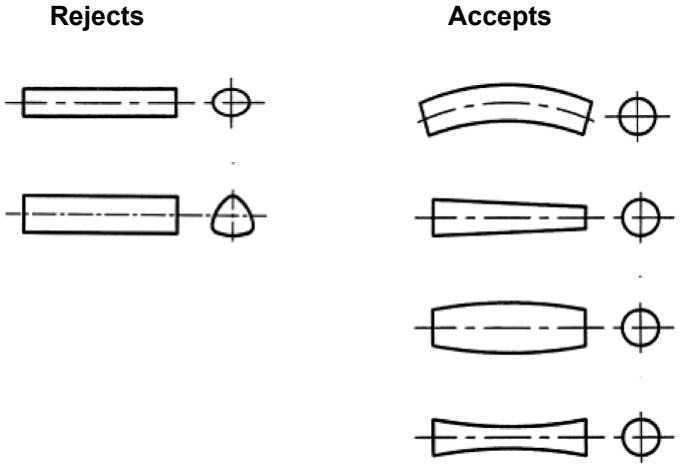


Figure 4.25 Roundness deviations

4.3.2 ASSESMENT OF ROUNDNESS DEVIATIONS

A diameter measurement does just what the word implies; it measures the diameter. It does not check the shape of the surface which is what roundness and cylindricity control. Since the roundness or cylindricity tolerance is a radial distance between concentric boundaries, a radial method of checking the surface w.r.t. an ideal reference circle is necessary.
.

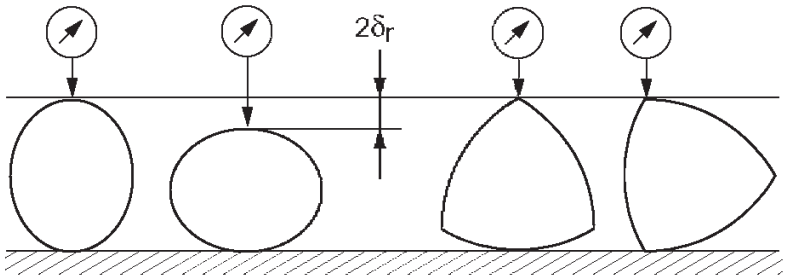


FIGURE 4.26 Oval and lobbed forms

However, rotating a part between centers is not an acceptable method since it relates the part surface to an axis, which technically is a check of another geometric tolerance called runout. With three point measurements (V-blocks) is is possible to detect different types of lobbed forms. (e.g. centreless grinding, reaming).

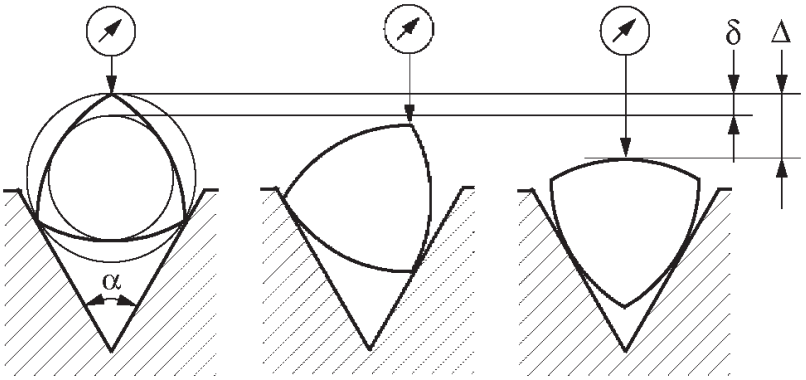


FIGURE 4.27 Assessment of lobed forms in V-blocks

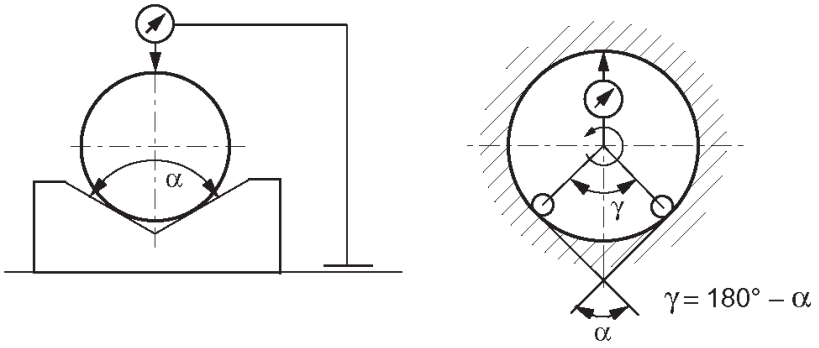


FIGURE 4.28 Three point measurement shaft and hole

	V-block angle						
	30º	60º	90º	108º	120º	150º	180º
2 lobs	2,3	0	1	1,4	1,6	1,9	2
3 lobs	3,7	3	2	1,4	1	0,3	0
4 lobs	2,9	0	0,4	0,4	0,4	1,5	2
5 lobs	2,7	0	2	2,2	2	0,7	0

Figure 4.8 *Correction values k for measurement of roundness deviations*

The number of lobs may be detected by counting the maximums during one revolution of the workpiece in the V-block. With regular lobs the unroudness tolerance can be found by dividing the reading by the corrections in the table.

$$\delta_r = \Delta / k = (A_{max} - A_{min}) / k$$

When there is a superposition of harmonics the selection of the proper correction is practically impossible.

To truly check for the roundness or cylindricity of a surface without regard to the axis of the part, the part must be rotated about the ultra-precision spindle of a specialized roundness measuring machine. A probe contacts the surface and transcribes an enlarged profile of the surface onto a polar graph. The profile is then checked against a clear overlay of concentric circles to determine if it falls within the allowable tolerance zone.

4.4 CYLINDRICITY

Cylindricity is one of the form tolerances and is one of the most common shapes across all industries.

A few examples where cylindricity tolerances are used are seals around shafts, lip seal applications and bearing applications.

4.4.1 CYLINDRICITY APPLIED TO A FEATURE

SYMBOL	TYPE OF TOLERANCE	DATUM REFERENCE	ALLOWABLE TOLERANCE MODIFIERS	ALLOWABLE DATUM MODIFIERS
⌭	Form	None	Ⓕ	None
Cylindricity tolerance zone Space between two coaxial cilinders				

TABLE 4.9 Cylindricity

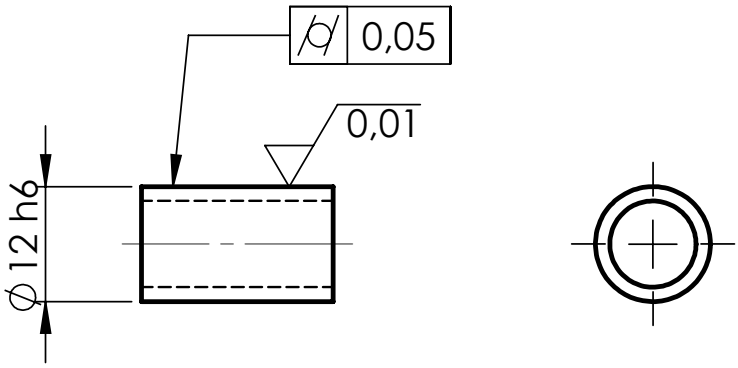
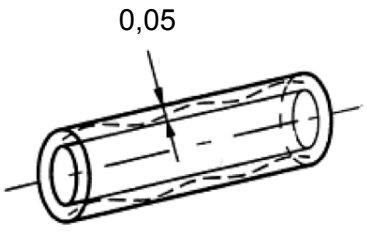


FIGURE 4.29 Cylindricity

FIGURE 4.30 Interpretation



Possible form deviations and their acceptance:

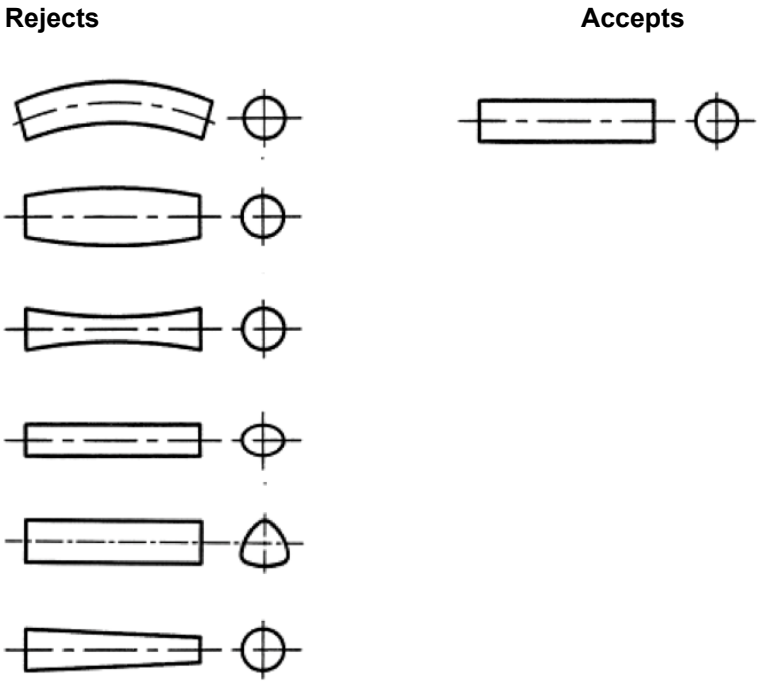


FIGURE 4.31 Possible deviations

4.4.2 ASSESMENT OF CYLINDRICITY DEVIATIONS

Deviations of the workpiece surface from an almost ideal reference cylinder are measured. There are several strategies to verify cylindrical tolerance. With the radial selection method profile lines of several cross-sections perpendicular to the axis of measurement are plotted and evaluated according to the requirement. The maximum measuring difference of all measurement divided by two gives the deviation δ_r for cylindricity.

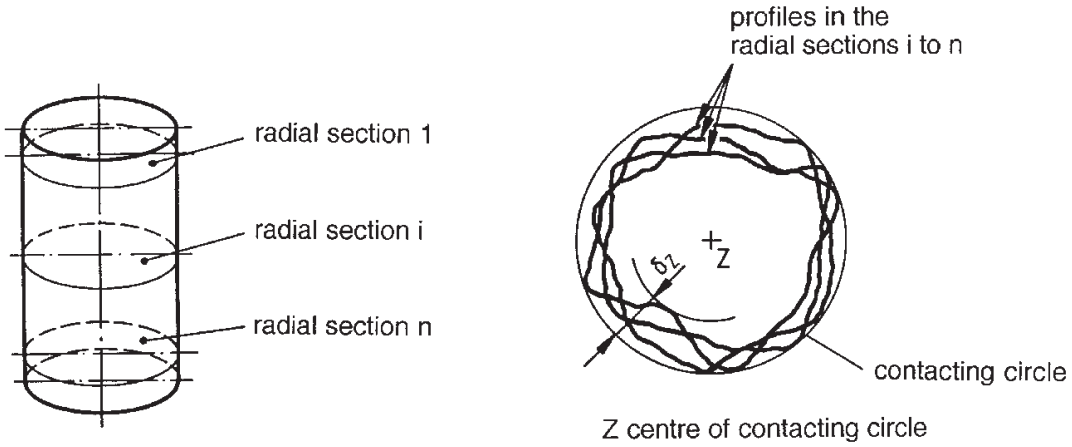


FIGURE 4.32 Verifying a cylindricity tolerance

5 DATUM SYSTEM

A datum is theoretical exact point, axis or plane derived from a true geometric counter part of the specified datum feature.

The datum system defines functional relationships between part features. Datums define the orientation and/or location of the tolerance zone and provide the origin for part measurement.

The datums can be seen as a means to lock degrees of freedom (DOF's) of a tolerance zone. The number of degrees of freedom of the tolerance zone which are locked depends on the nominal shape of the datum features and the degrees of freedom which have been locked by preceding datums.

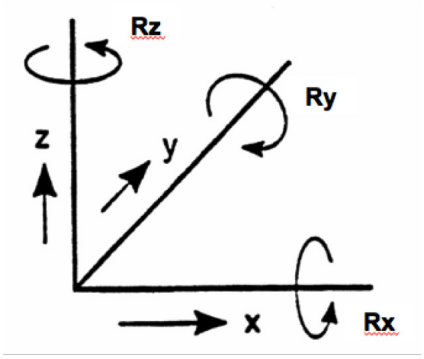


FIGURE 5.1 Degrees Of Freedam (DOF)

FIXED DEGREES OF FREEDOM:

Surface, 3 DOF's
one translation, two rotations

Cylinder, 4 DOF's
two translations, two rotations

Sphere, 3 DOF's
three translations

5.1 SIMULATED DATUM FEATURE

Datums are based on datum features.

DATUM: A theoretical exact point, axis or plane derived from a true geometric counter part of the specified datum feature.

DATUM FEATURE: Real feature of a workpiece such as an edge, a plane or hole, used to establish the location of the datum.

SIMULATED DATUM FEATURE: Real surface of sufficiently precise form contacting the datum features and used to establish the datum.

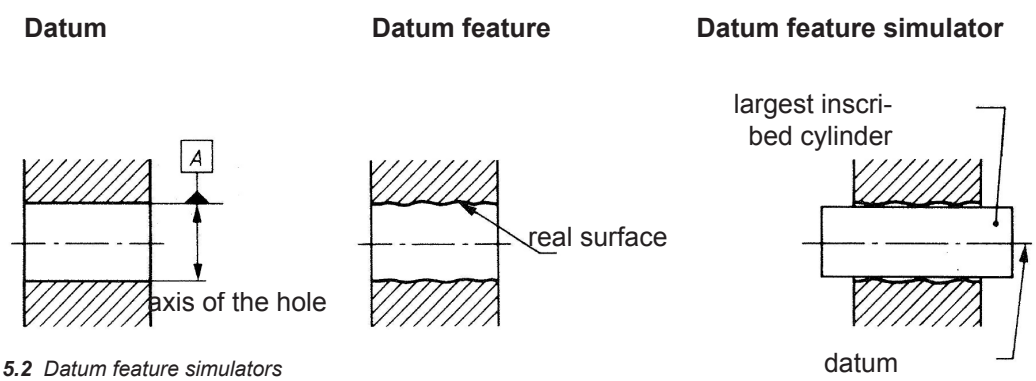


FIGURE 5.2 Datum feature simulators

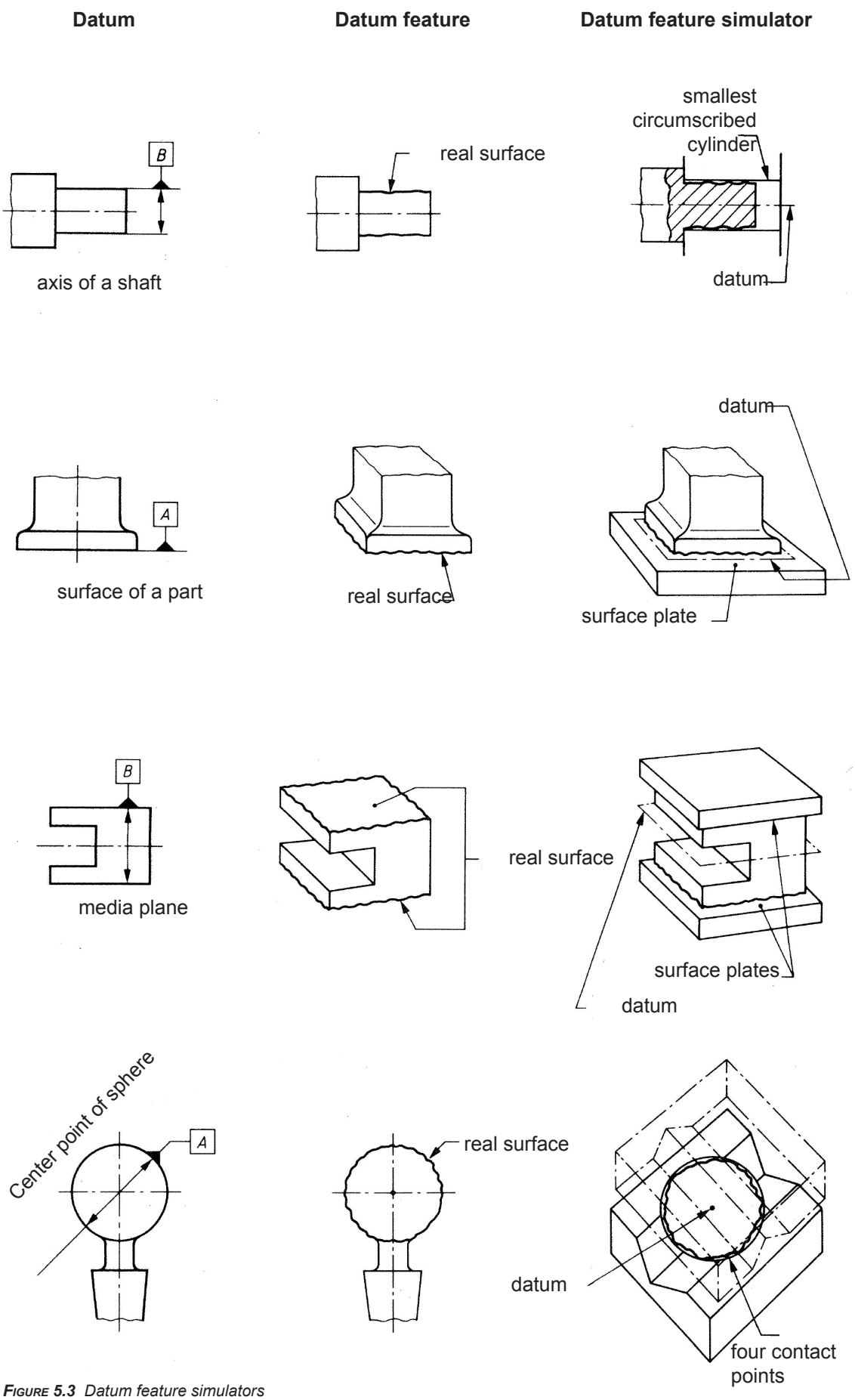


Figure 5.3 Datum feature simulators

5.2 MINIMUM ROCK REQUIREMENT

When a datum feature is not stable relative to the simulated datum feature it shall be arranged so that the possible movement in any direction is equalized. In other words the datum feature shall be aligned relative to the simulated datum feature into a median position.

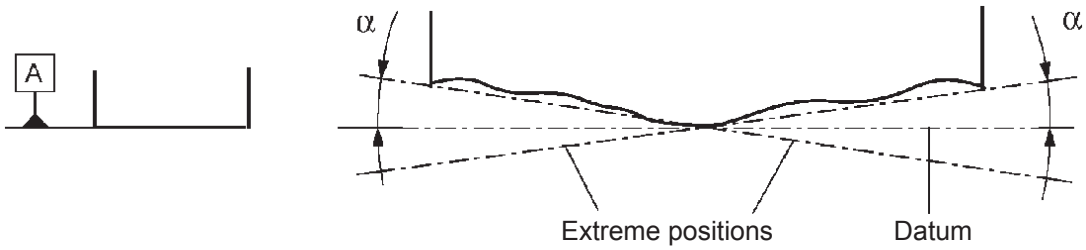


FIGURE 5.4 Minimum rock requirement for a surface

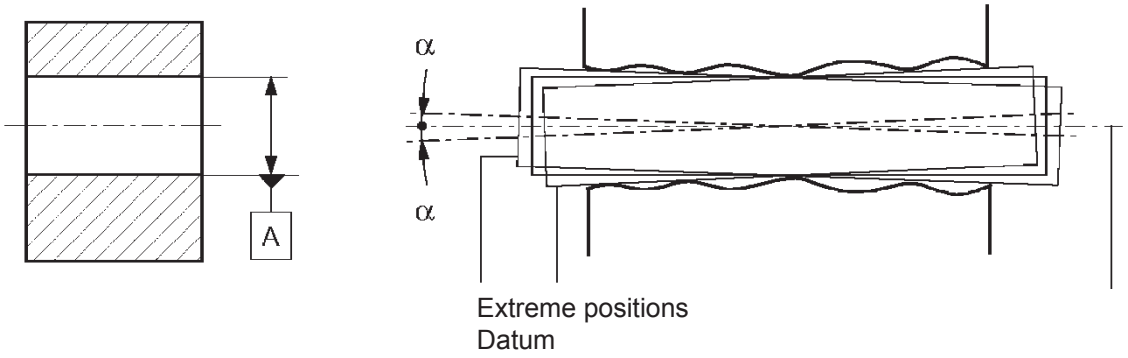


FIGURE 5.5 Minimum rock requirement for a shaft

When the datum feature is the common axis of two coaxial cylinders with different diameters the datum is the common axis established by the smallest circumscribed cylinders.

Datum	Datum feature and datum feature simulator

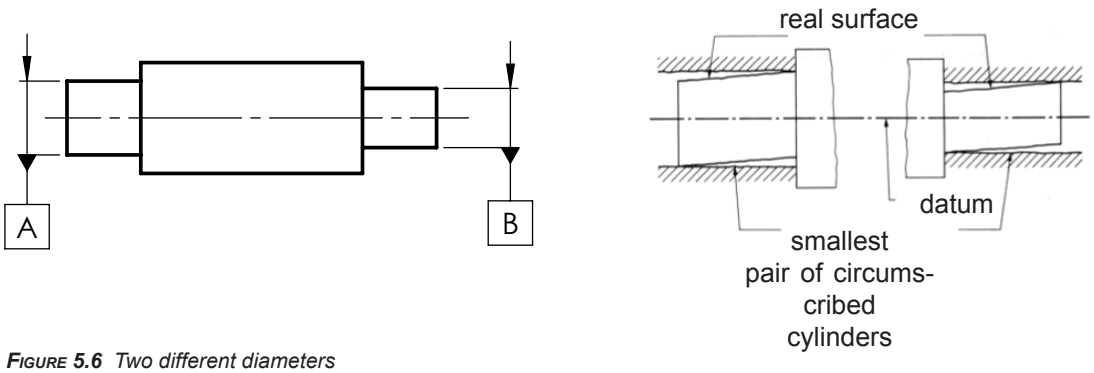


FIGURE 5.6 Two different diameters



It is good pratice to have form requirements on datum features. This by tolerancing the feature or by using a general standard like ISO 22081.

5.3 INDICATION OF DATUMS ON THE DRAWING

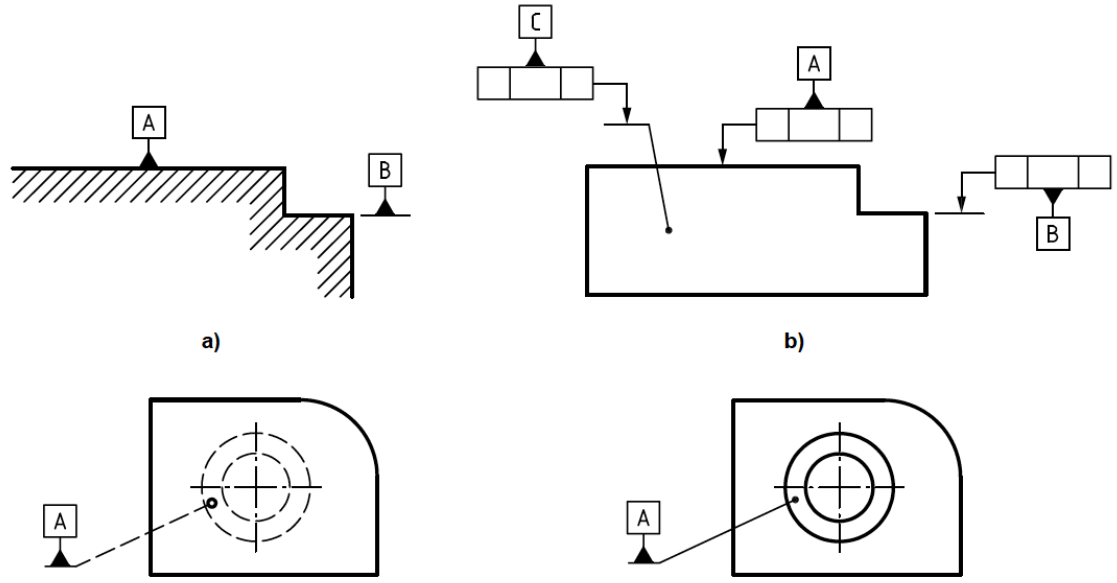
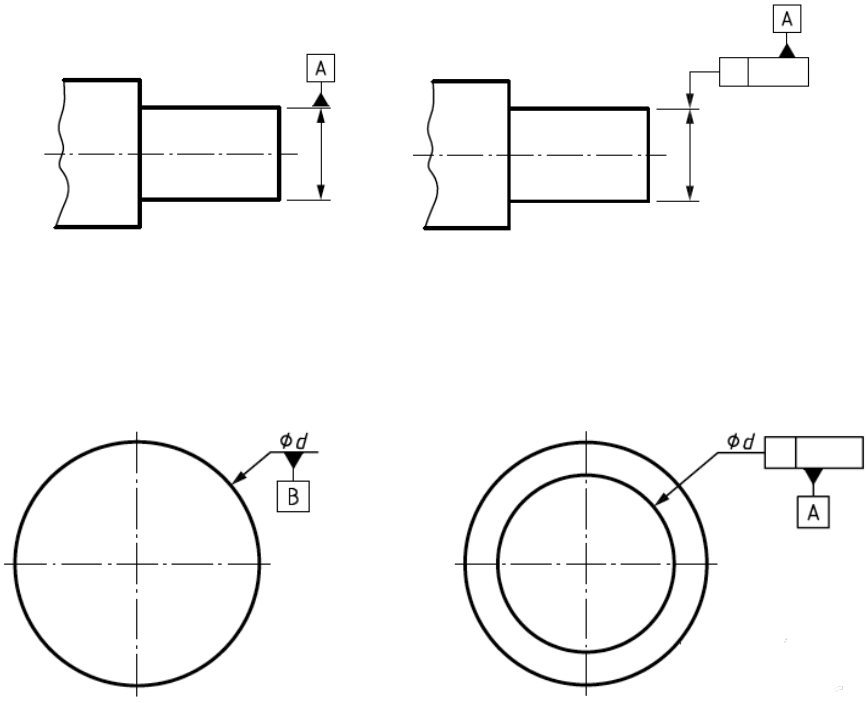
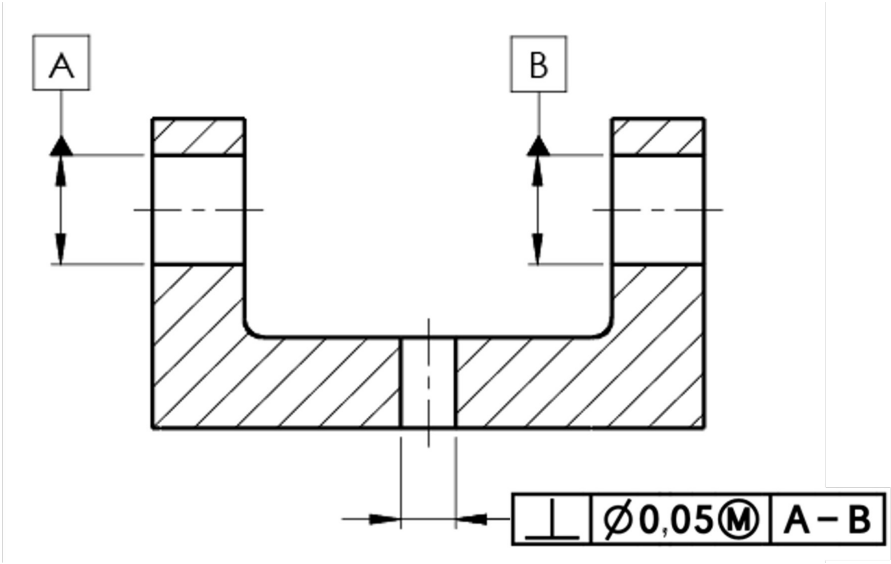


FIGURE 5.7 Indication of a datum reference plane



FIGUUR 5.8 Indication of a datum axis



Figuur 5.9 One datum feature from more (two) elements

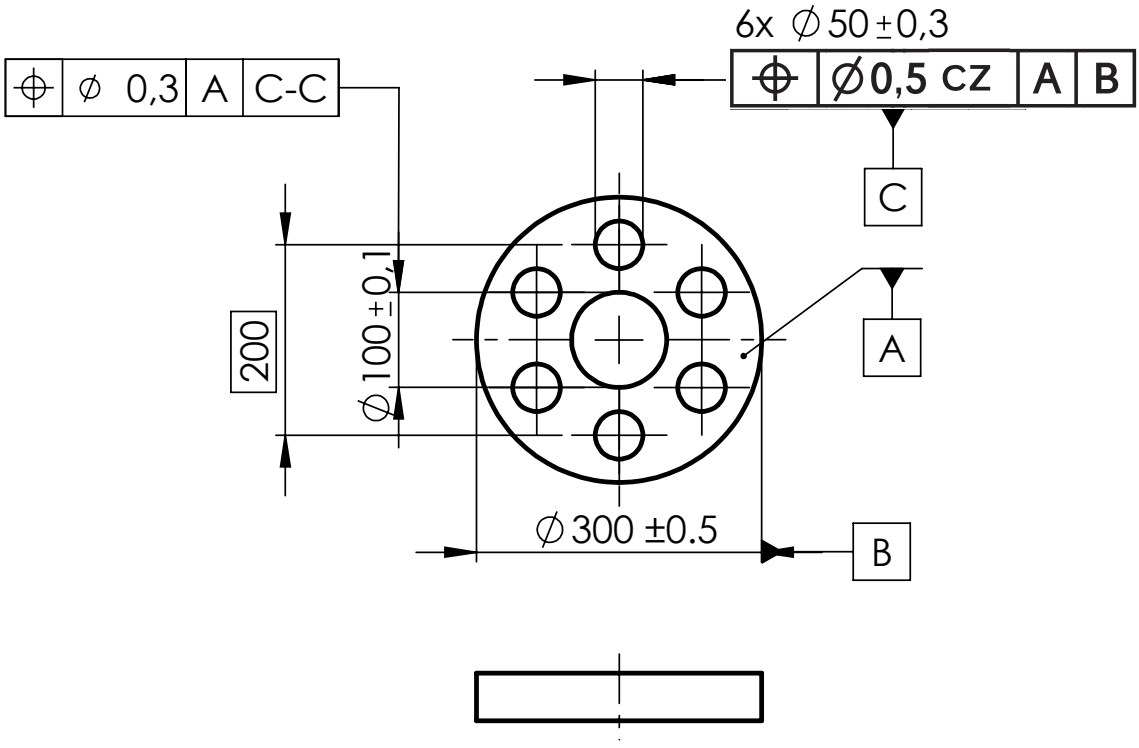


Figure 5.10 Pattern of holes used as datum

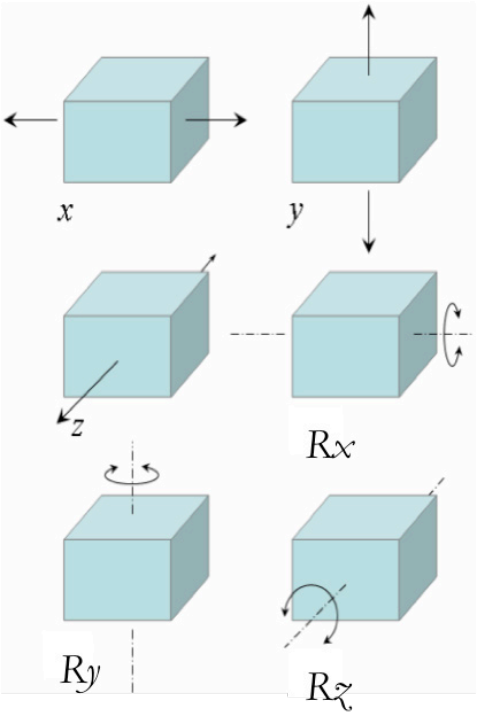


FIGURE 5.11 Degrees Of Freedom

Datums limit the Degrees Of Freedom.

The number of DOF's restricted depends on the feature type and the DOF's restricted by preceding datums.

Surface limits 3 DOF's	One translation, two rotations.
Cylinder limits 4 DOF's	Two translations, two rotations.
Bol limits 3 DOF's	Three translations.

5.4 DATUM SYSTEM

5.4.1 NECESSITY FOR DATUMS AND DATUMS SYSTEMS

Where one or more datum systems are established each by two or more datum features only those DOF's can be locked which not have been locked by preceding datums.

The sequence of the datums influences the result obtained.

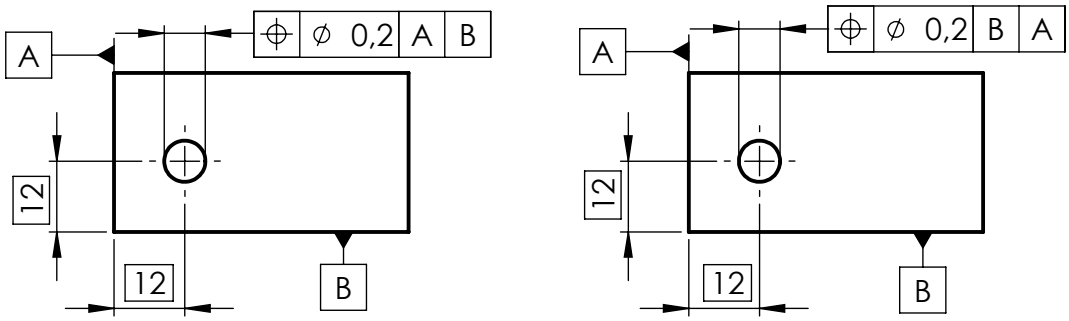


FIGURE 5.12 Datum system of two datums

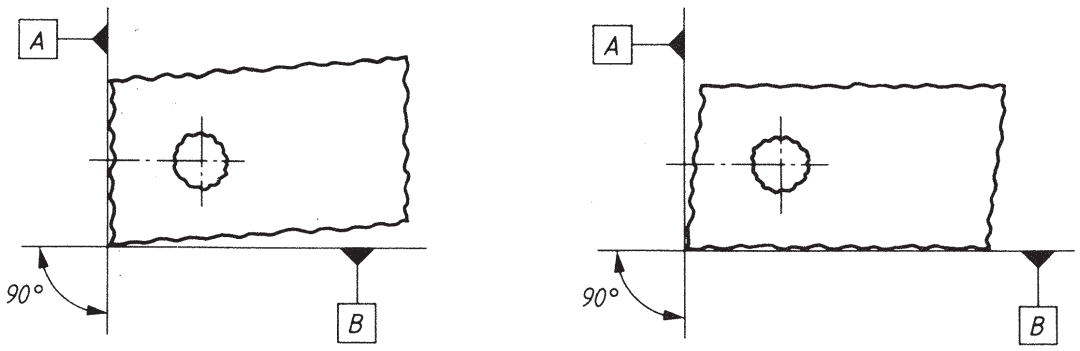
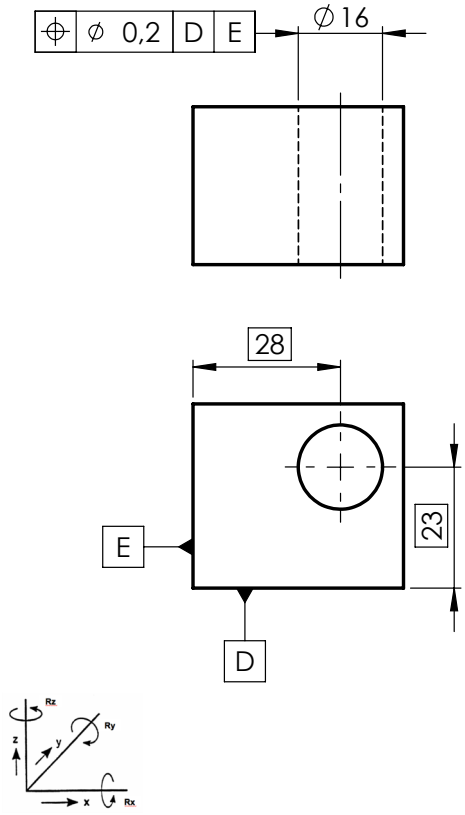


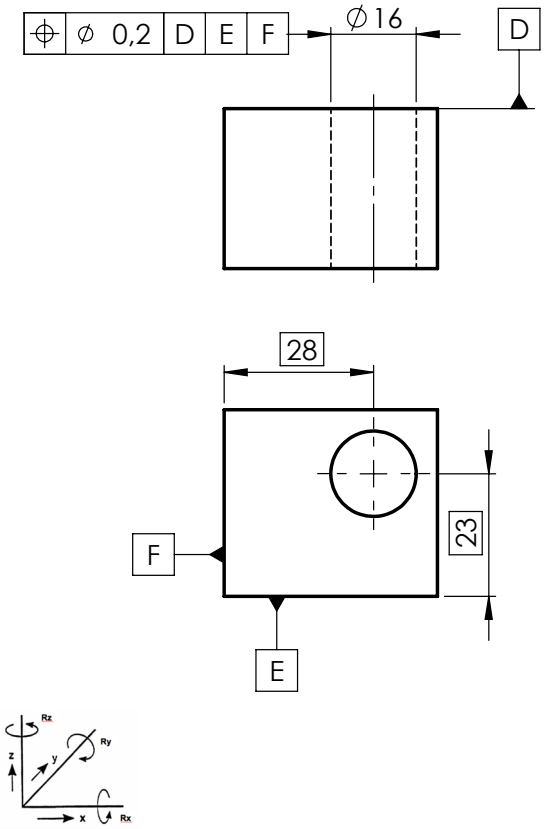
FIGURE 5.13 Effects of the sequence of the datums

5.4.2 Datum system and DOF



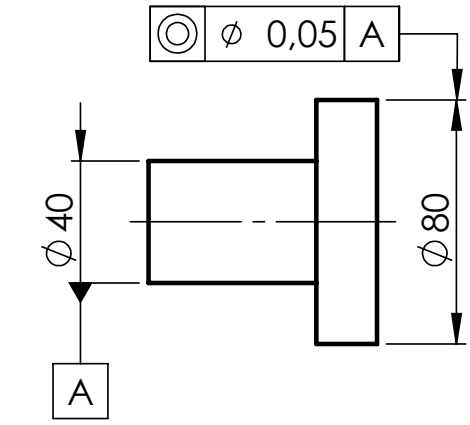
	X	Y	Z	Rx	Ry	Rz
D	-	-	x	x	x	-
E	x	-	-	-	o	x

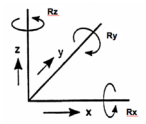
Figure 5.14 Effect of choosing datums



	X	Y	Z	Rx	Ry	Rz
D	-	x	-	x	-	x
E	-	-	x	o	x	-
F	x	-	-	-	o	o

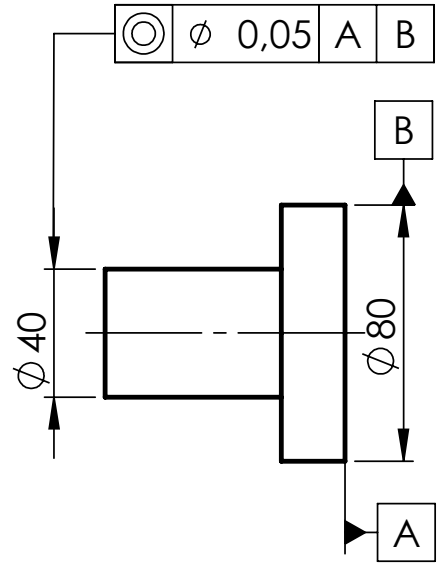
Figure 5.15 Effect of choosing datums

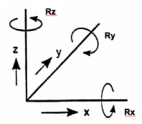




	X	Y	Z	Rx	Ry	Rz
A	-	x	x	-	x	x

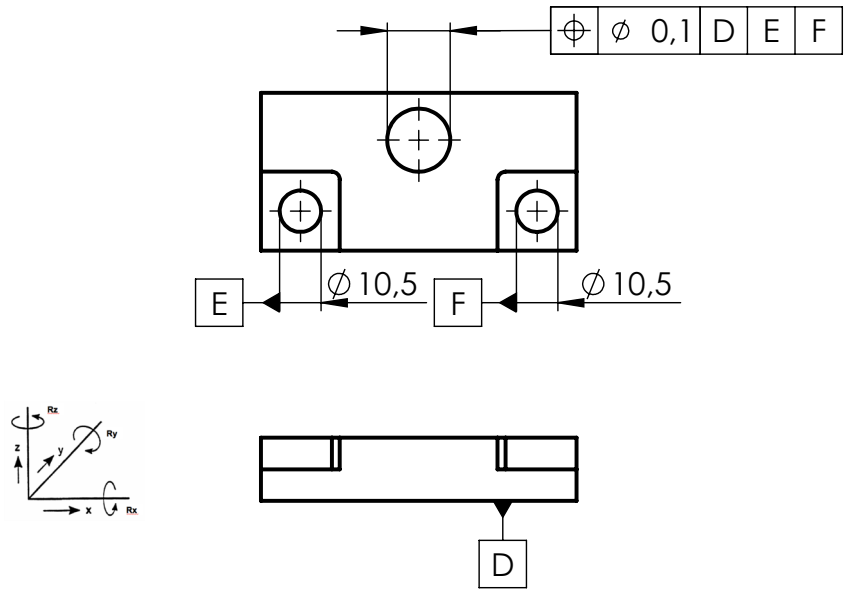
Figure 5.16 Effect of choosing datums





	X	Y	Z	Rx	Ry	Rz
A	x	-	-	-	x	x
B	-	x	x	-	o	o

Figure 5.17 Effect of choosing datums



	X	Y	Z	Rx	Ry	Rz
D	-	-	x	x	x	-
E	x	x	-	o	o	-
F	o	o	o	o	o	x!

FIGURE 5.18 Effect of choosing datums

The degree of freedom Rz is contrained by using both datum E and datum F.

5.4.3 Datum at MMC

When a datum feature is applied on a MMC bases a fixed sized datum simulator (gage) at MMVC must be used.The size of the MMVC depends on the MMS and the geometrical tolerance applied to the datum feature.

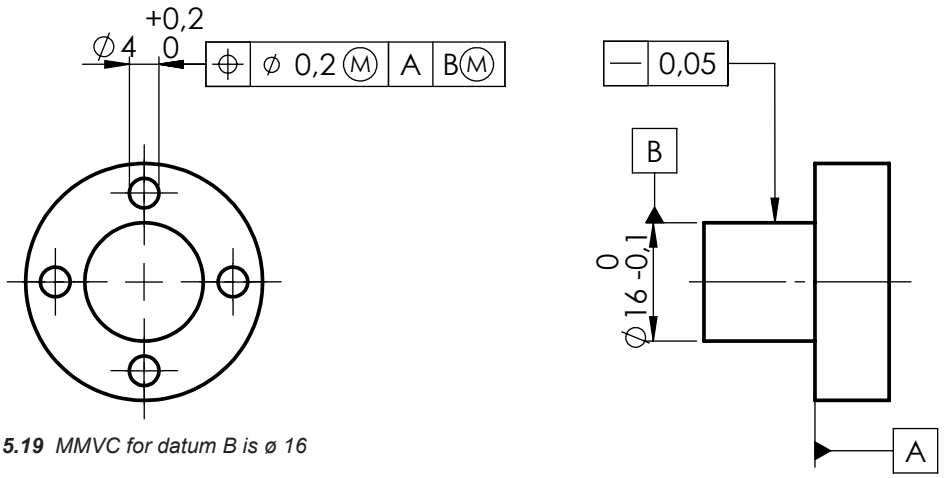


Figure 5.19 MMVC for datum B is ø 16

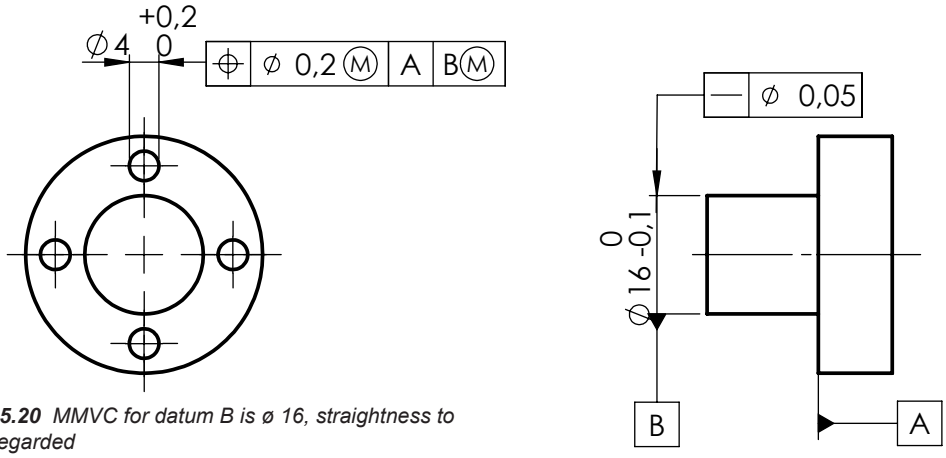


Figure 5.20 MMVC for datum B is ø 16, straightness to be disregarded

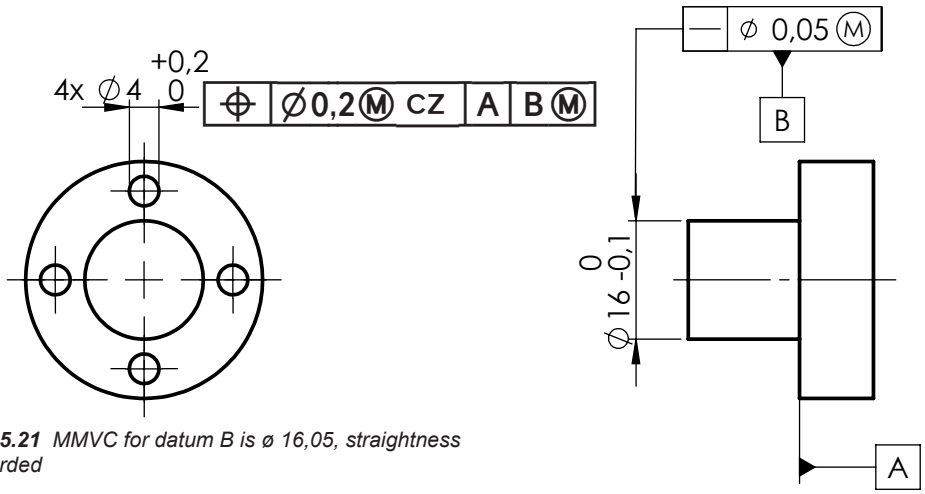


Figure 5.21 MMVC for datum B is ø 16,05, straightness to regarded

5.5 DATUM TARGETS

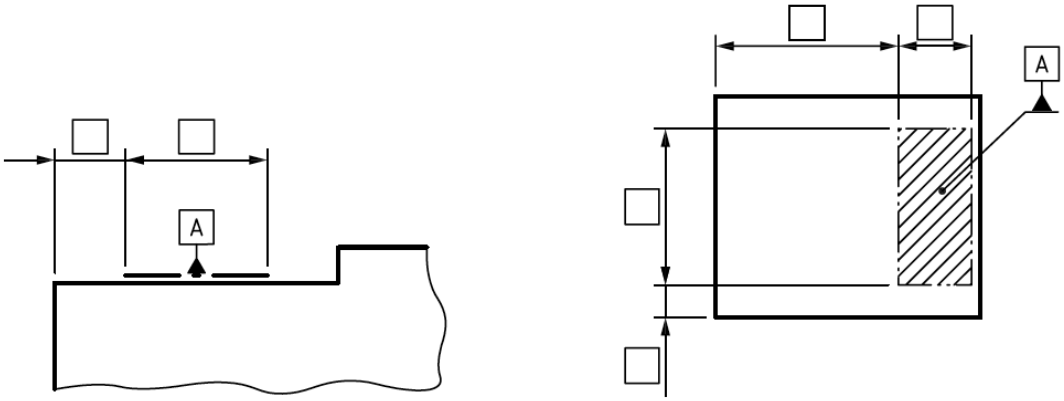


FIGURE 5.22 Only a restricted are of the element is used as datum

Datum A is not the total surface but only a single restricted area is used as datum.

A datum target frame is a circle, divided into two compartments by a horizontal line. The lower compartment is reserved for the datum feature identifier followed by a digit corresponding to the datum target number. The upper compartment is reserved for additional information, such as dimensions of the target area. The target dimension can be in or next to the circle.

Datum targets enable repeatable targets to be established from uneven or warped surface and from partial surfaces.

Using datum targets can make the workpiece cheaper.

Datum D is determined by three areas. Two rectangular datum targets and a circular datum taget. The target dimensions of the rectangles are places next to the datum target circles.

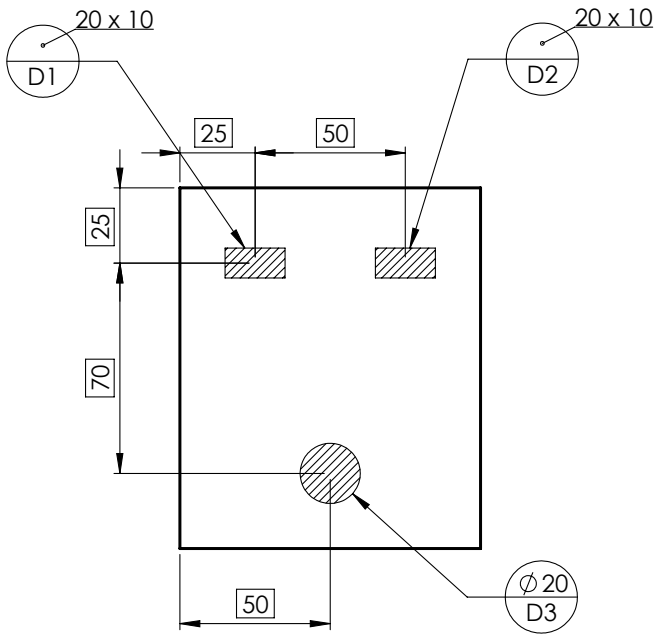


FIGURE 5.23 Three surfaces as datum targets

Datum A is determined by three points. The TED dimensions indication the location of the points have no tolerance.

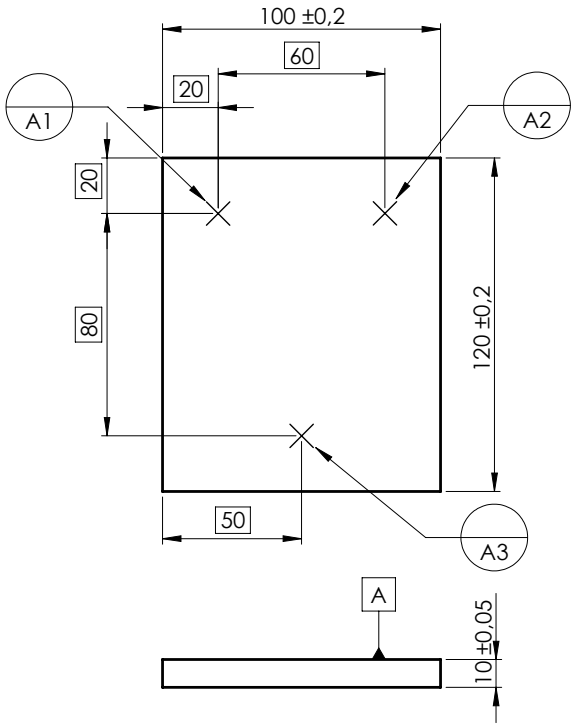


Figure 5.24 Datum target A is determined by three points

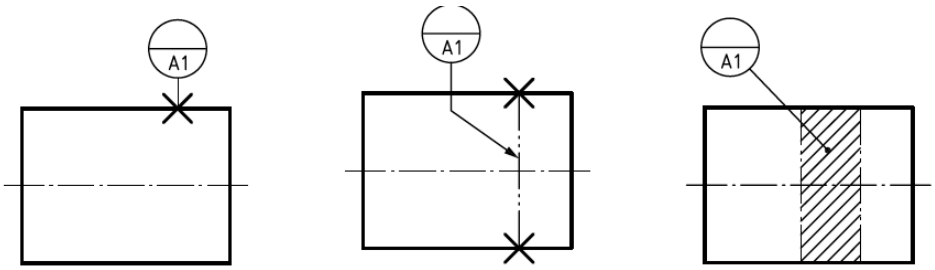


Figure 5.25 Datum targets as point, line and surface

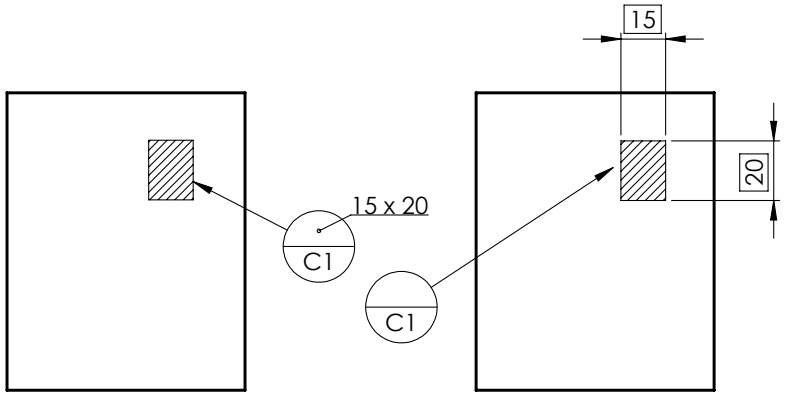
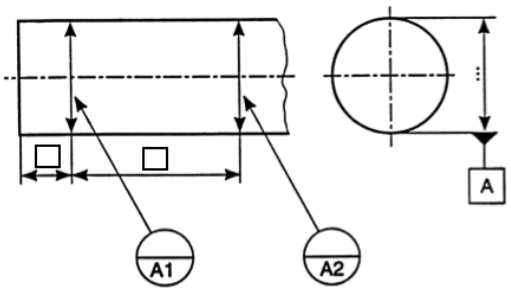
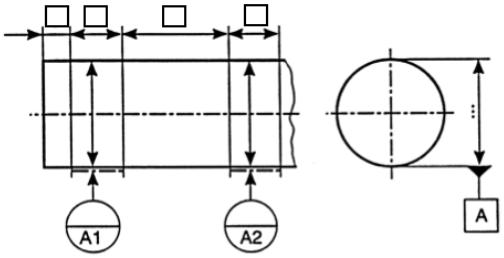


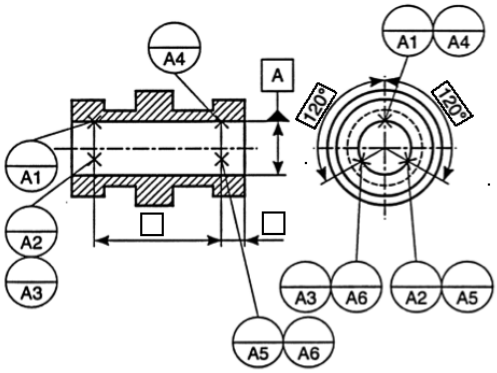
Figure 5.26 Datum target size in circle and target dimensioned using TED's



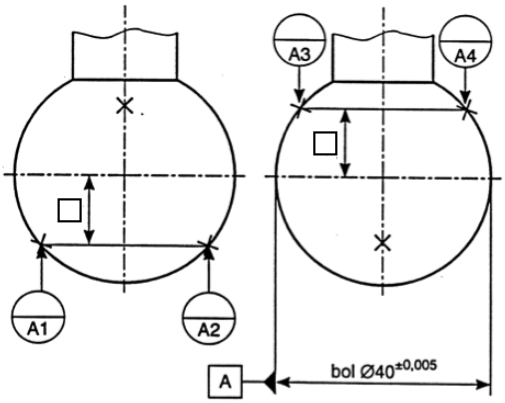
Datum A is determined by the line connecting the centers of the cross section A1 and A2.



Datum A is determined by the shared center lines of the sections A1 and A2.



Datum A is determined by the centerline connecting the center of the circle created by A1, A2, en A3 and the center of the circle created by A4, A5 and A6.



Datum A is determined by the point A1 to A4 defining the center of the sphere. The points need to be equidistant.

FIGURE 5.27 Datum targets

EXAMPLES DATUM TARGETS

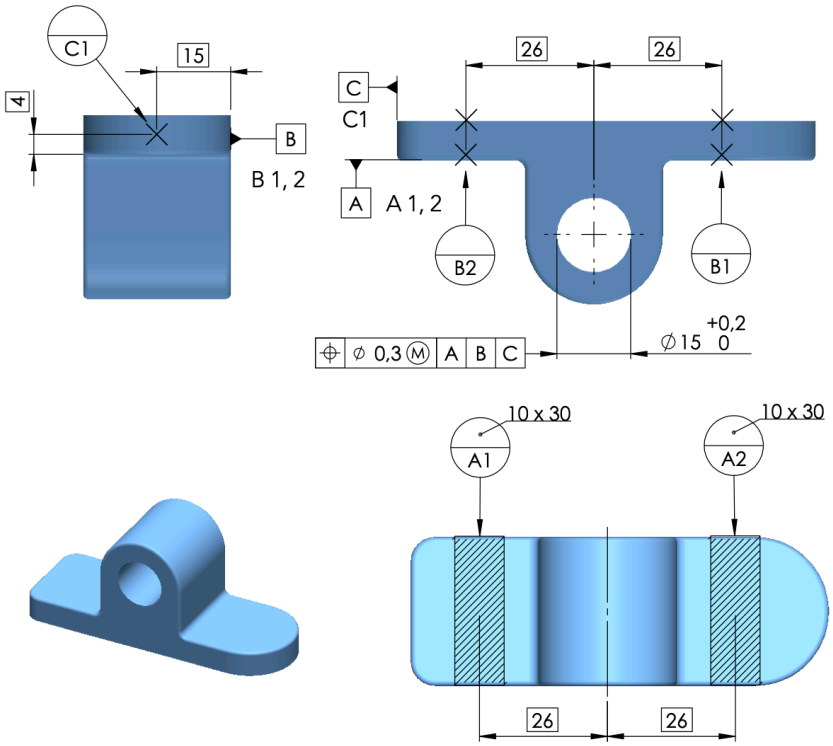


FIGURE 5.28 Datum targets

- Area's as datum targets : A1, A2 Primary datum A
- Lines as datum targets : B1, B2 Secondary datum B
- Point as datum target : C1 Tertiary datum C

There is no over constraining when using only points as datum targets. Each of the six DOF is only constrained once.

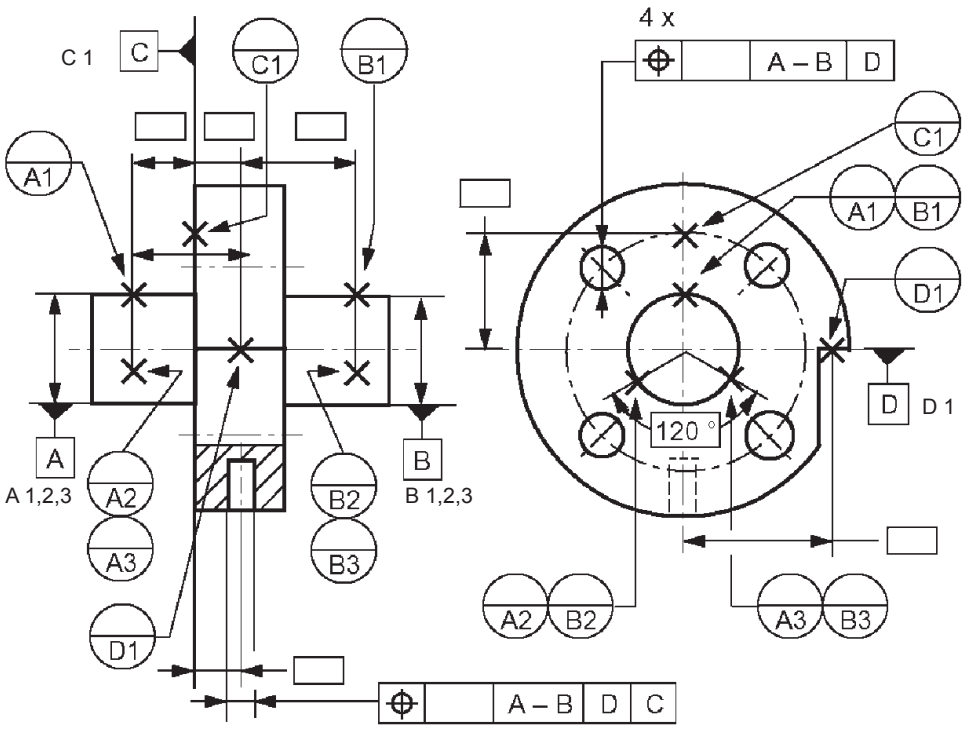


FIGURE 5.29 Datum targets on cylindrical surface

A surface constrains 3 DOF's being one translation an two rotations. By using the symbol ><, a surface only contrains the two orientations.

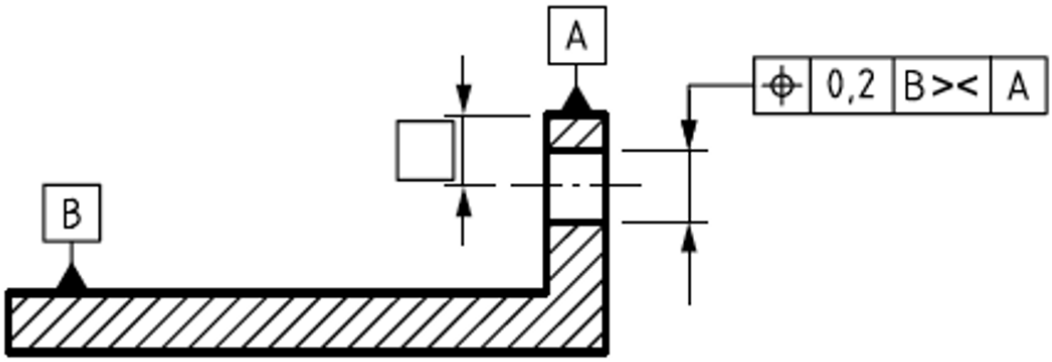


Figure 5.30 Rotation only

Number of degrees of freedom contrained:

1. **Point**
2. **Orientation only**
3. **Surface**
4. **Cylinder**
5. **Cone**

6. All DOF's

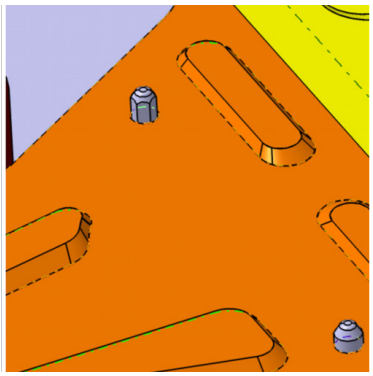


Figure 5.31 6 DOF constrained

6 TOLERANCES OF ORIENTATION

Orientation tolerances define angular deviations of surfaces or features of size but cannot control location.

6.1 PERPENDICULARITY

A perpendicularity tolerance limits the perpendicular deviation of a planar surface or feature of size relative to a datum reference. A perpendicularity tolerance can only constrain rotational deviations.

A perpendicular tolerance always uses a datum. It can control form and orientation.

Perpendicular tolerances are used in assemblies, appearance, establish the relation between datum features or supports (guides and stops).

6.1.1 PERPENDICULARITY APPLIED TO A SURFACE

SYMBOL	TYPE OF TOLERANCE	DATUM REFERENCE	ALLOWABLE TOLERANCE MODIFIERS	ALLOWABLE DATUM MODIFIERS
⊥	Orientation	Required	Ⓕ	Ⓕ Ⓛ Ⓜ
Perpendicularity tolerance zone Space between parallel planes Space between two parallel lines				

TABLE 6.1 Perpendicularity applied to a surface

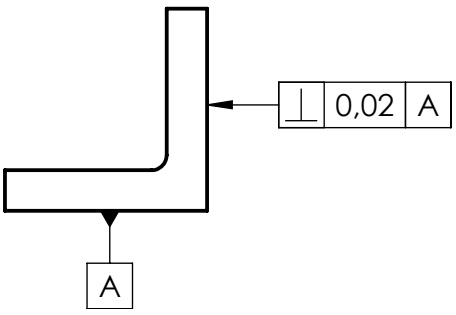


FIGURE 6.1 Perpendicularity applied to a surface

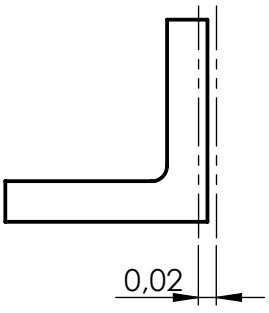


FIGURE 6.2 Interpretation

The functional requirement is that the righthand surface of the workpiece has to be within a tolerance zone of 0,02 mm perpendicular to datum A.

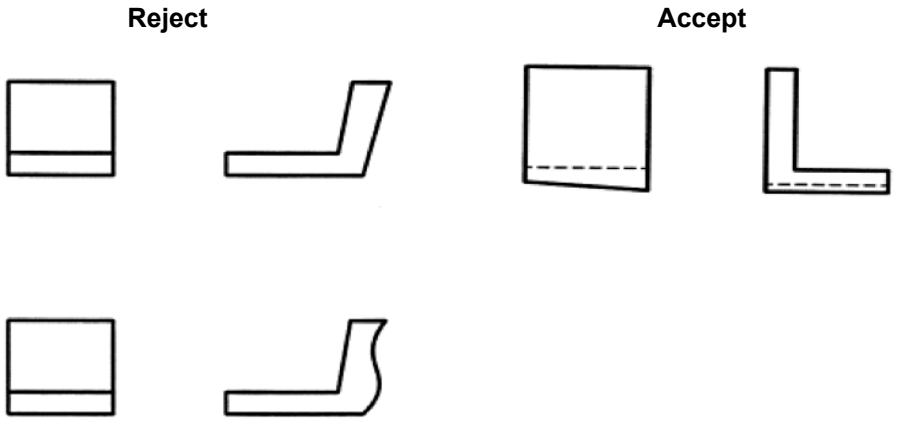


Figure 6.3 Possible deviations

6.1.2 ASSESSMENT OF PERPENDICULARITY APPLIED TO A SURFACE

he deviations of the workpiece surface from a reference element are measured. The reference element needs to be aligned according to the datum (perpendicular, parallel or under a specified angel). The orientation deviation δ_d is the difference between the largest distance A_{max} and the smallest distance A_{min}.

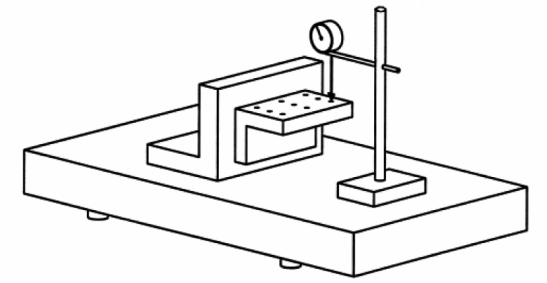


Figure 6.4 Assessment of perpendicularity

$$\delta_p = A_{max} - A_{min} \leq t_p$$

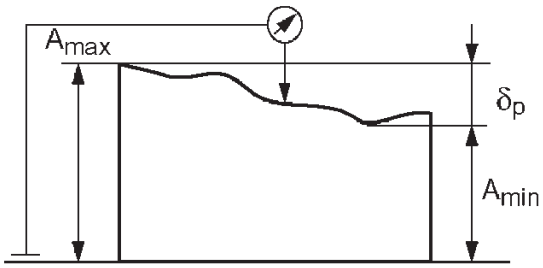


Figure 6.5 Assessment of the orientation deviation δ_d, here perpendicularity, of a surface

The dial indicator is set to zero in the lefthand site corner in the front. The orientation deviation δ_d is :

0,003 + 0,005 = 0,008 mm.

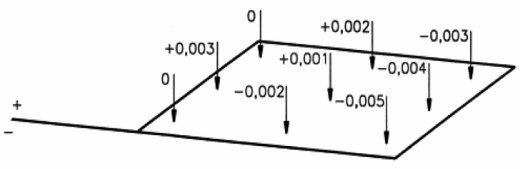


Figure 6.6 Assessment of orientation

6.1.3 PERPENDICULARITY OF A MEDIA PLANE AT MMC

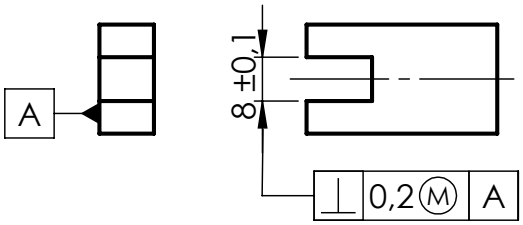


FIGURE 6.7 Perpendicularity of a media plane at MMC

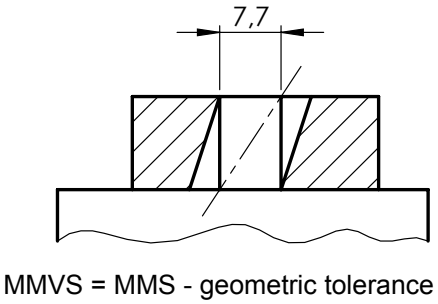


FIGURE 6.8 Virtual condition boundary

WIDTH SLOT	PERPENDICULARITY TOLERANCE	BONUS TOLERANCE	TOTAL TOLERANCE
7,9	0,2	0	0,2
8,0	0,2	0,1	0,3
8,1	0,2	0,2	0,4

TABLE 6.2 Bonus tolerances

There are several ways a perpendicularity tolerance at MMC can be verified. A common method is the use of a gage. The gage verifies that the feature of size fits into its MMVC. The part must rest on the datum and the feature must fit into or over its MMVC (acceptance boundary). The size and location needs to be verified separately.

6.1.4 PERPENDICULARITY APPLIED TO A CYLINDRICAL FEATURE OF SIZE

SYMBOL	TYPE OF TOLERANCE	DATUM REFERENCE	ALLOWABLE TOLERANCE MODIFIERS	ALLOWABLE DATUM MODIFIERS
⊥	Orientation	Required	Ⓕ Ⓛ Ⓜ Ⓟ	Ⓕ Ⓛ Ⓜ
Perpendicularity tolerance zone Space between two parallel planes Space within a cylinder (when ø is used)				

TABLE 6.3 Perpendicularity applied to a feature of size

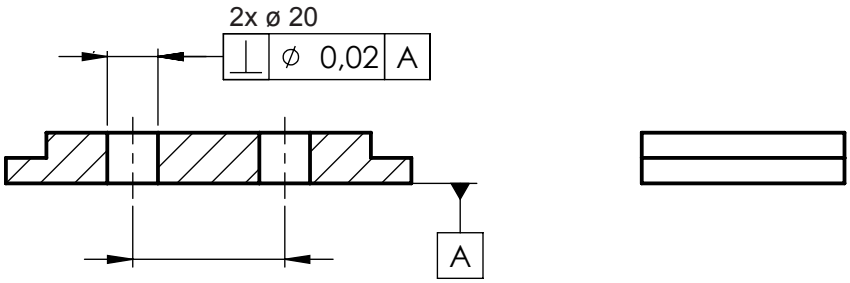


FIGURE 6.9 Perpendicularity applied to a feature of size

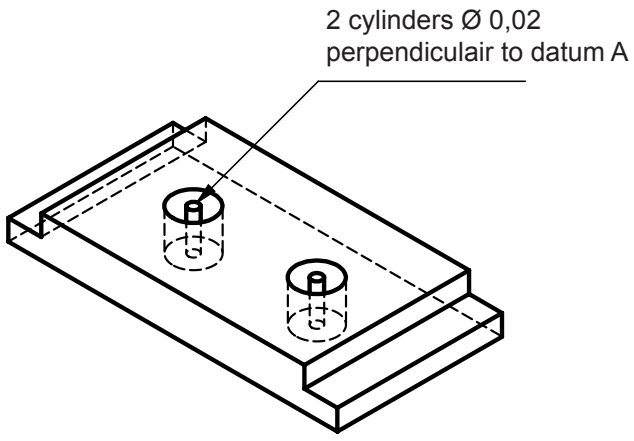


FIGURE 6.10 Interpretation

The requirement is that the axis of the two holes are perpendicular to the datum.

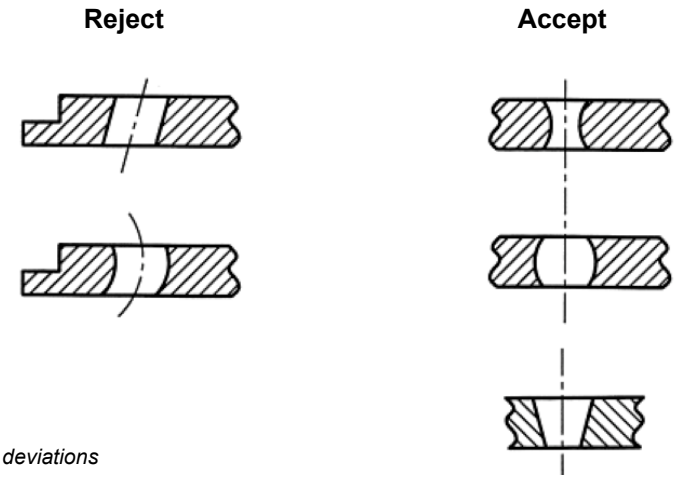


FIGURE 6.11 Possible deviations

6.1.5 ASSESSMENT OF ORIENTATION APPLIED TO A FEATURE OF SIZE

The deviations of the workpiece feature of size, either being hole, slot, parallel planes, shaft or bar, from a reference element are measured.The reference element needs to be aligned according to the datum (perpendicular, parallel or under a specified angle). For the measuring method below the distances from a measuring plate are measured. The orientation deviation δ_{nx} is calculated from the deviation in x being δ_{nx} and the deviation in y being δ_{ny}..

$$\delta_n = \sqrt{(\delta_{nx}^2 + \delta_{ny}^2)} \leq t_n \qquad \delta_{nx} \approx \frac{(A_{ro} - A_{ru}) - (A_{lo} - A_{lu})}{2} \frac{l}{l_m} \leq t_n$$

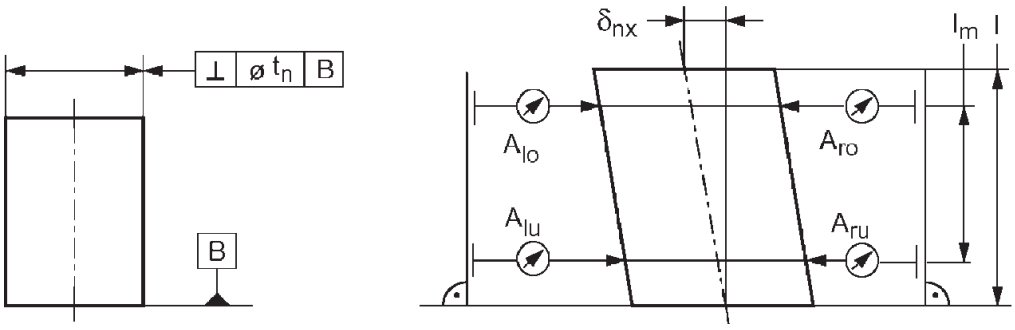


FIGURE 6.12 Assesment of perpendicularity of a feature of size

6.1.6 PERPENDICULARITY TO A CYLINDRICAL FEATURE OF SIZE AT MMC

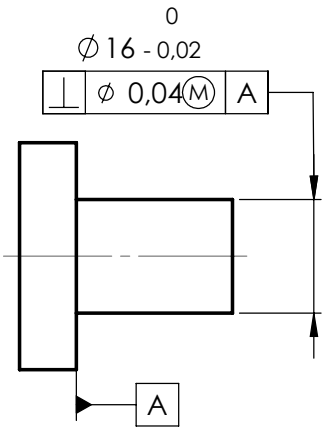


FIGURE 6.13 Penpedicularity at MMC

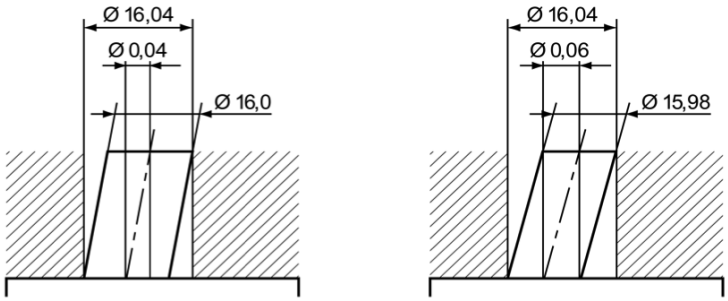


FIGURE 6.14 Interpretation

MMVS = MMS + geometric tolerance

MMVS = ø 16 + 0,04 = ø 16,04

DIAMETER BORE	PERPENDICULARITY TOLERANCE	BONUS TOLERANCE	TOTAL TOLERANCE
16,00	0,04	0	0,04
15,99	0,04	0,01	0,05
15,98	0,04	0,02	0,06

TABLE 6.4 Bonus tolerances

There are several ways a perpendicularity tolerance at MMC can be verified. A common method is the use of a gage. The gage verifies that the feature of size fits into its virtual condition. The part must rest on the datum and the feature must fit into or over its virtual condition (acceptance boundary). In the example the maximal material virtual size equals 16,04 mm. The size and location needs to be verified separately.

6.2 PARALLELISM

A parallelism tolerance limits the amount a surface, axis or center plane is allowed to deviate from parallelism relative to a datum reference. Parallelism is an orientation tolerance and does not control the location of the feature. Only rotational degrees of freedom are constrained. An orientation tolerance can translate relative to the datum but not rotate.

A parallelism tolerance always uses a datum. It can control form and orientation.

Parallelism tolerances are used in assemblies (maintain uniform gap), supports (distribute load and reduce wear) and performance (lineair motion).

6.2.1 PARALLELISM APPLIED TO A SURFACE

SYMBOL	TYPE OF TOLERANCE	DATUM REFERENCE	ALLOWABLE TOLERANCE MODIFIERS	ALLOWABLE DATUM MODIFIERS
//	Orientation	Required	Ⓕ	Ⓕ Ⓛ Ⓜ
Parallelism tolerance zone Space between parallel planes Space between two parallel lines				

TABLE 6.5 Parallelism applied to a surface

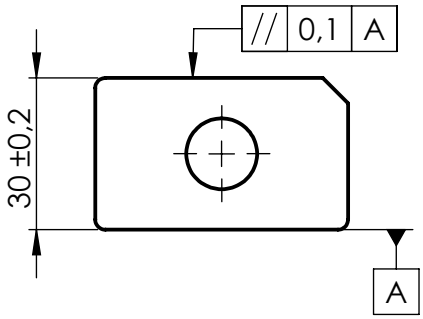


FIGURE 6.15 Parallelism applied to a surface

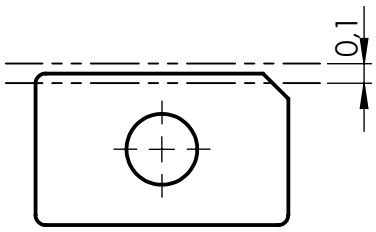


FIGURE 6.16 Interpretation

The requirement is that the top surface is parallel to the datum.

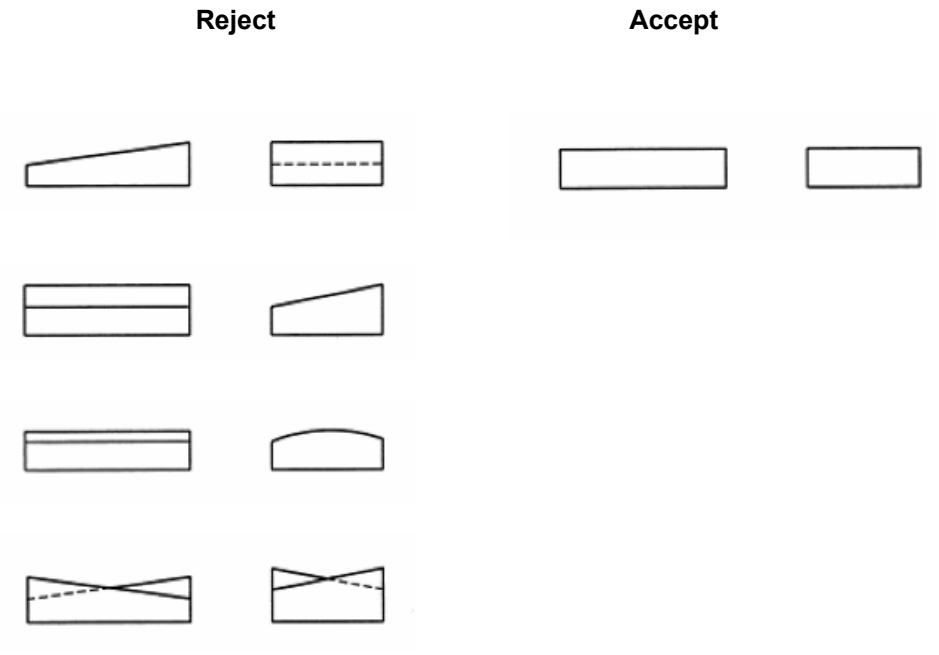


FIGURE 6.17 Possible deviations

6.2.2 ASSESSMENT OF PARALLELISM APPLIED TO A SURFACE

$$\delta_p = A_{max} - A_{min} \leq t_p$$

The deviations of the workpiece surface from a reference element are measured. The reference element needs to be aligned according to the datum (perpendicular, parallel or under a specified angle). The orientation deviation δ_d is the difference between the largest distance A_{max} and the smallest distance A_{min}.

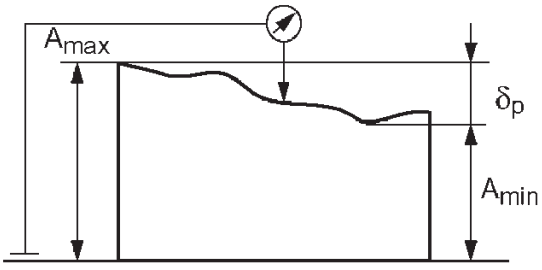


FIGURE 6.18 Assessment of the orientation deviation δ_d, here parallelism, of a surface

6.2.3 PARALLELISM APPLIED TO A CYLINDRICAL FEATURE OF SIZE

SYMBOL	TYPE OF TOLERANCE	DATUM REFERENCE	ALLOWABLE TOLERANCE MODIFIERS	ALLOWABLE DATUM MODIFIERS
//	Orientation	Required	Ⓕ Ⓛ Ⓜ Ⓟ ⌀	Ⓕ Ⓛ Ⓜ
Parallelism tolerance zone Space between parallel planes Space within a cylinder (when ø is used)				

TABLE 6.6 Parallelism applied to a feature of size

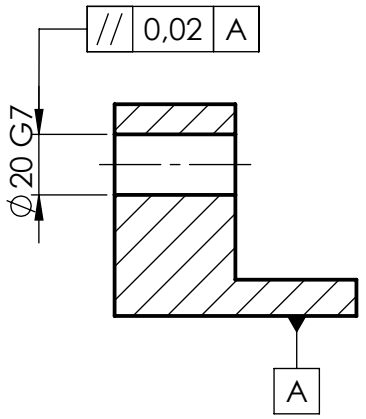


FIGURE 6.19 Parallelism applied to a feature of size

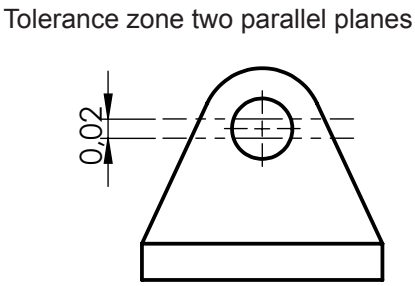


FIGURE 6.20 Interpretation

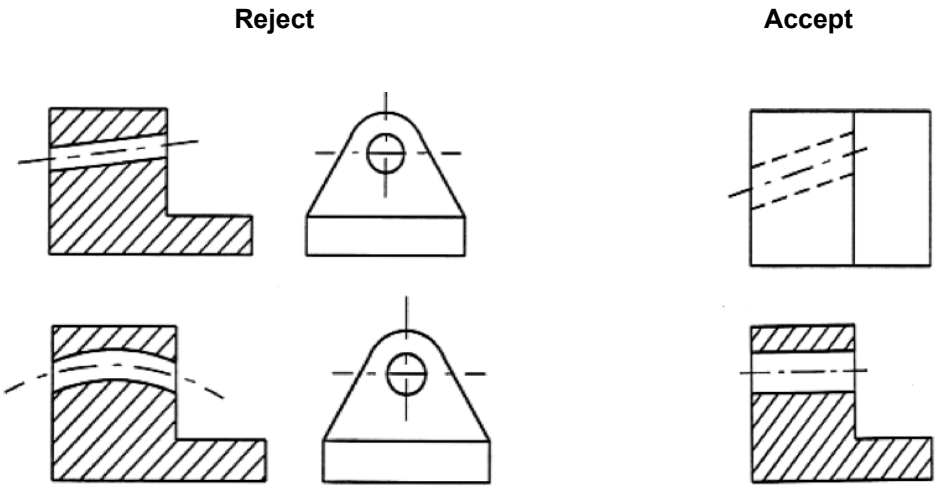


FIGURE 6.21 Possible deviations

6.2.4 ASSESSMENT OF PARALLELISM APPLIED TO A FEATURE OF SIZE

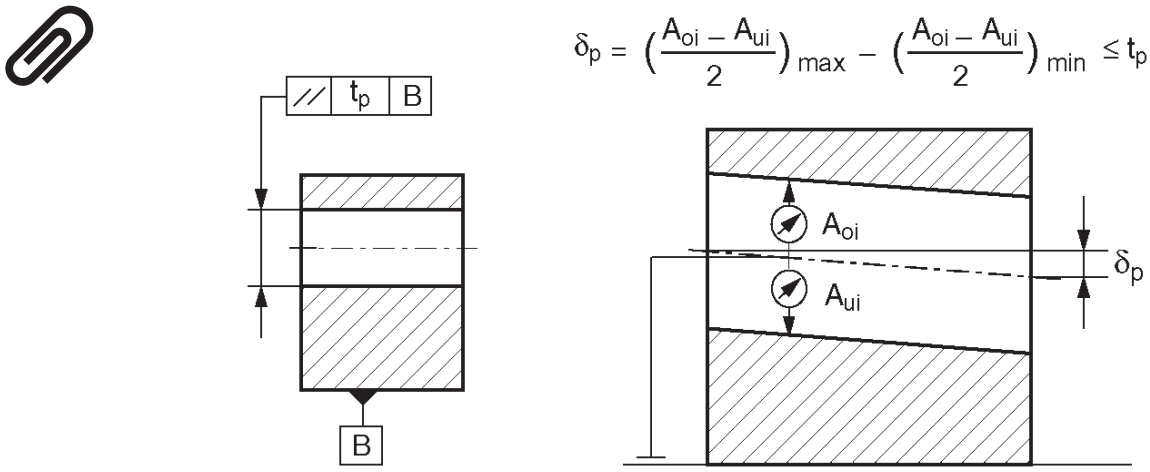


FIGURE 6.22 Assesment of the parallelism deviation δ_p, of a feature of size

The differences of the distance from a measuring plate are to be measured.

6.2.5 PARALLELISM APPLIED TO A FEATURE OF SIZE AT MMC

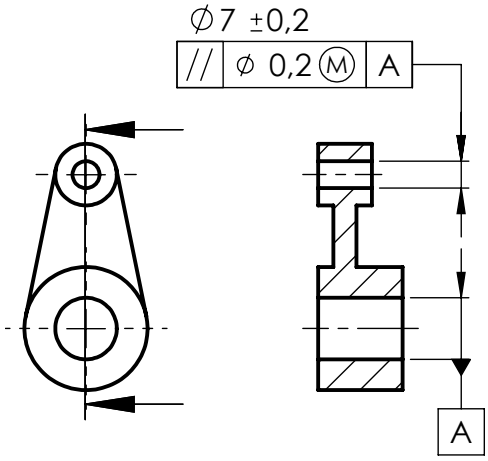


FIGURE 6.23 Parallelism applied to a feature of size

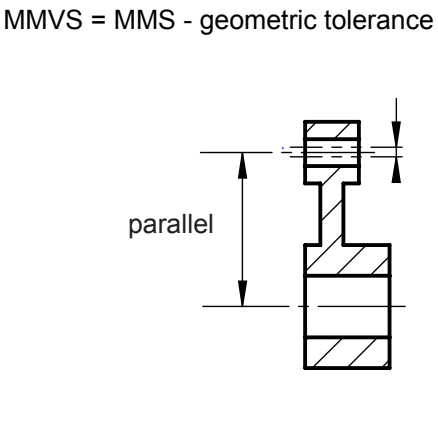


FIGURE 6.24 Interpretation

DIAMETER HOLE	PARALLELISM TOLERANCE	BONUS TOLERANCE	TOTAL TOLERANCE
6,8	0,2	0	0,2
7,0	0,2	0,2	0,4
7,2	0,2	0,4	0,6

TABLE 6.7 Parallelism applied to a feature of size at MMC

A method for verifying parallelism at MMC is the use of a gage. The part must rest on the datum and the feature must fit over its virtual condition (acceptance boundary). In the example the virtual condition equals 6,6 mm. The size and location need to be verified separately.

6.3 ANGULARITY

An angularity tolerance is a geometric tolerance that limits the amount a surface, axis or center plane is allowed to deviate from a theoretical angle relative to a datum. Since angularity is a orientation tolerance it does not control location.

6.3.1 ANGULARITY APPLIED TO A SURFACE

SYMBOL	TYPE OF TOLERANCE	DATUM REFERENCE	ALLOWABLE TOLERANCE MODIFIERS	ALLOWABLE DATUM MODIFIERS
∠	Orientation	Required	Ⓕ	Ⓕ Ⓛ Ⓜ
Angularity tolerance zone Space between two parallel planes Space between two parallel lines				

TABLE 6.8 Angularity applied to a surface

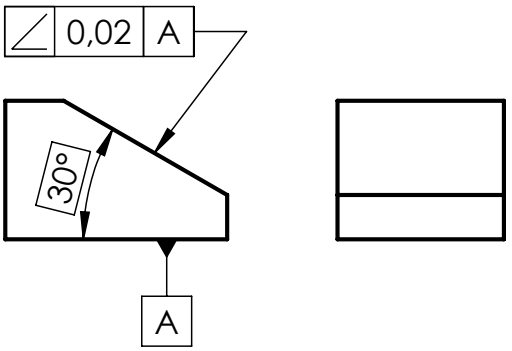


FIGURE 6.25 Angularity applied to a surface

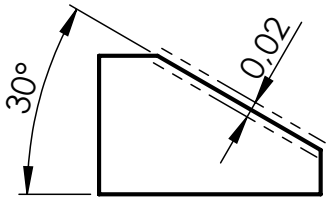


FIGURE 6.26 Interpretation

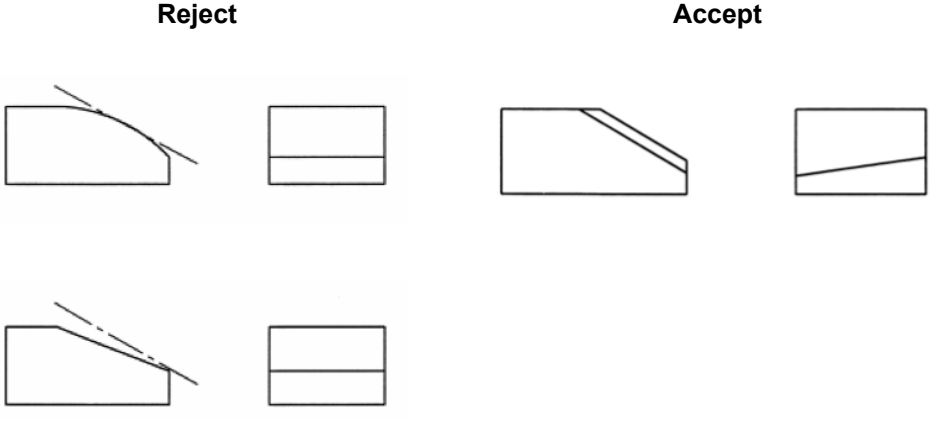


FIGURE 6.27 Possible deviations

6.3.2 ASSESSMENT OF ANGULARITY APPLIED TO A SURFACE

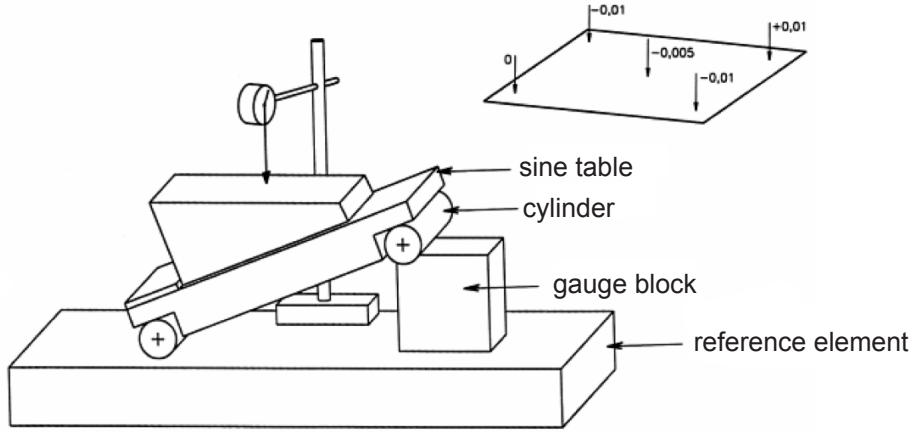


FIGURE 6.28 Assessment of agularity, of a surface

$$\delta_p = A_{max} - A_{min} \leq t_p$$

The deviations of the workpiece surface from a reference element are measured. The reference element needs to be aligned according to the datum (perpendicular, parallel or under a specified angle). The orientation deviation δ_d is the difference between the largest distance A_{max} and the smallest distance A_{min}.

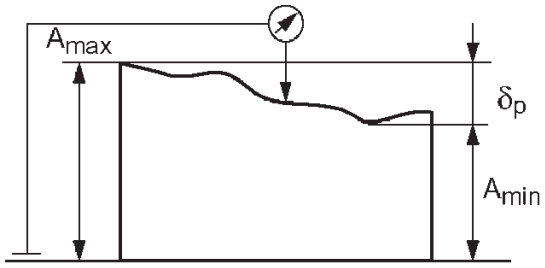


FIGURE 6.29 Assessment of the orientation deviation δ_d, here agularity, of a surface

6.3.3 ANGULARITY APPLIED TO A CYLINDRICAL FEATURE OF SIZE

SYMBOL	TYPE OF TOLERANCE	DATUM REFERENCE	ALLOWABLE TOLERANCE MODIFIERS	ALLOWABLE DATUM MODIFIERS
∠	Orientation	Required	Ⓕ Ⓛ Ⓜ Ⓟ ⌀	Ⓕ Ⓛ Ⓜ
Angularity tolerance zone Space between parallel planes Space within a cylinder (when ø is used)				

TABLE 6.9 Angularity applied to a feature of size

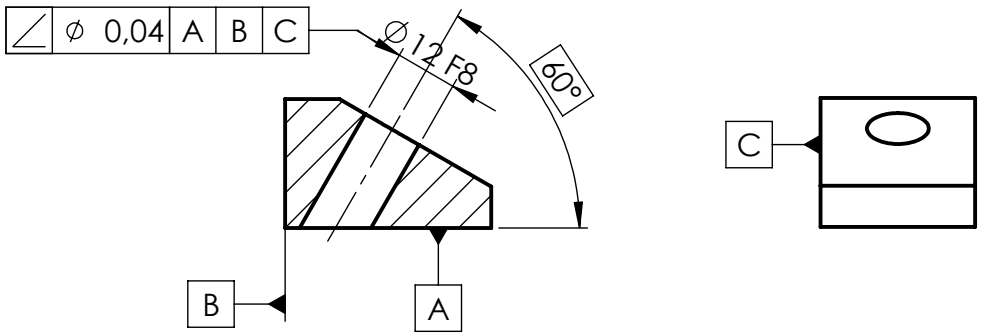


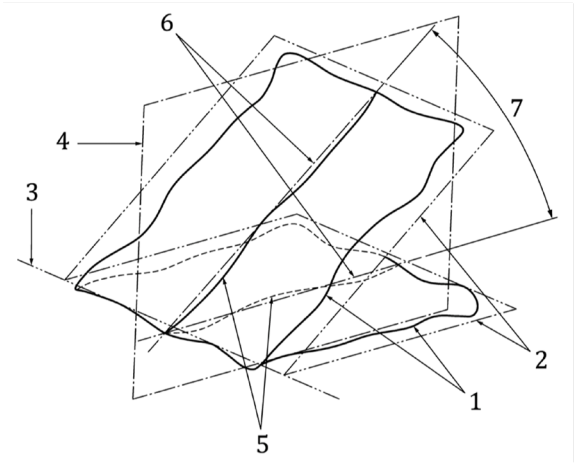
FIGURE 6.30 Angularity applied to a feature of size



FIGURE 6.31 Interpretation

6.3.4 ANGULAR SIZE

The ISO default for an dimensional angular specfication is a two line criterium with minmax association. The default can be changed using modifyers,



1. Reel feature
2. Associated feature
3. Intersection line
4. Plane perpendicular to 3
5. Two extracted lines
6. Two associated straight lines
7. Two point angular size

FIGURE 6.31 Dimensional specification

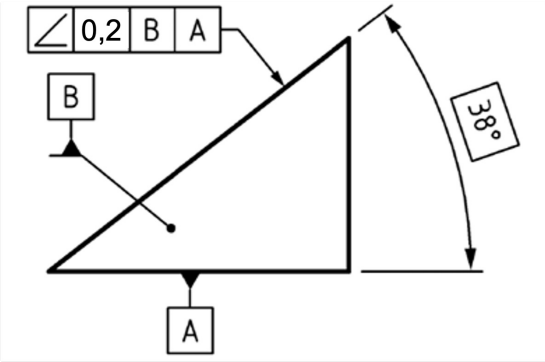


FIGURE 6.31 Angularity, two datums to constrain

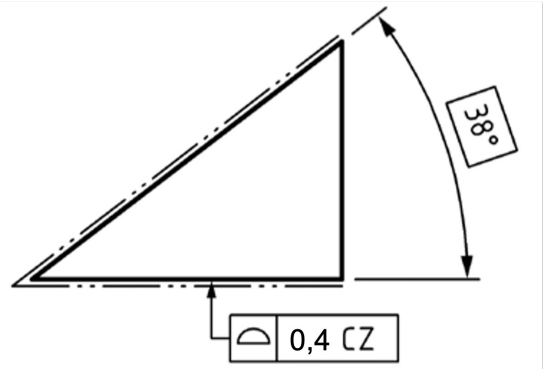


FIGURE 6.31 Angle tolerated using profile

6.3.5 ASSESSMENT OF ORIENTATION APPLIED TO A FEATURE OF SIZE

The deviations of the workpiece feature of size either being hole, slot, shaft or bar, from a reference element are measured. The reference element needs to be aligned according to the datum (perpendicular, parallel or under a specified angle). For the measuring method the distances from a measuring plate are measured. The orientation deviation δ_{nx} is calculated from the deviation in x being δ_{nx} and the deviation in y being $\delta_{ny.}$

$$\delta_n = \sqrt{(\delta_{nx}^2 + \delta_{ny}^2)} \leq t_n \qquad \delta_{nx} \approx \frac{(A_{ro} - A_{ru}) - (A_{lo} - A_{lu})}{2} \cdot \frac{l}{l_m} \leq t_n$$

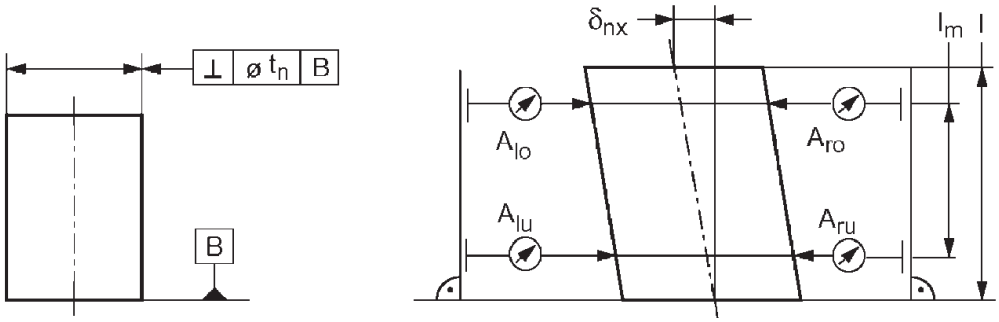


FIGURE 6.32 Assessment of orientation applied to a feature of size

MODULE III

7 TOLERANCES OF LOCATION

In a previous chapter position was discussed being the most common tolerance for location. Coaxiality / Concentricity and symmetry are two more geometric tolerances controlling location. All location tolerances also indirect control orientation and form.

7.1 COAXIALITY / CONCENTRICITY

7.1.1 COAXIALITY

The coaxiality tolerance is a geometric tolerance that defines the permissible deviation of the axis of a surface of revolution from a datum axis.

SYMBOL	TYPE OF TOLERANCE	DATUM REFERENCE	ALLOWABLE TOLERANCE MODIFIERS	ALLOWABLE DATUM MODIFIERS
◎	Location	Required	Ⓕ Ⓜ Ⓛ ⌀*	Ⓕ Ⓜ Ⓛ
Coaxiality tolerance zone Space within a cylinder			* Required	

TABLE 7.1 Coaxiality

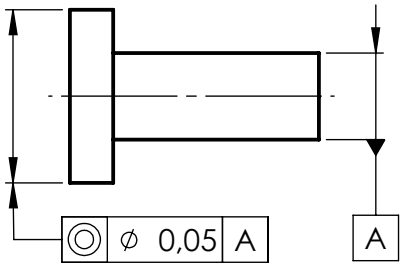


FIGURE 7.1 Coaxiality

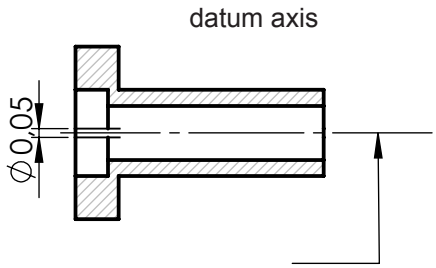


FIGURE 7.2 Interpretation

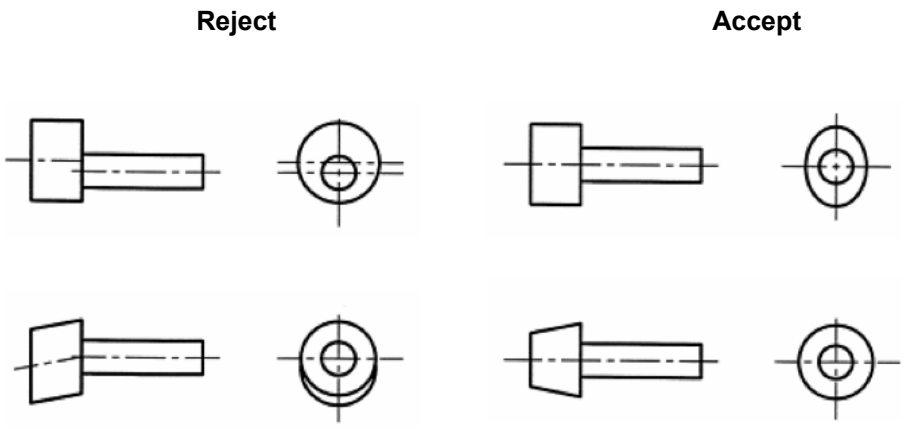


Figure 7.3 Possible deviations

Possibilities for indicating coaxiality

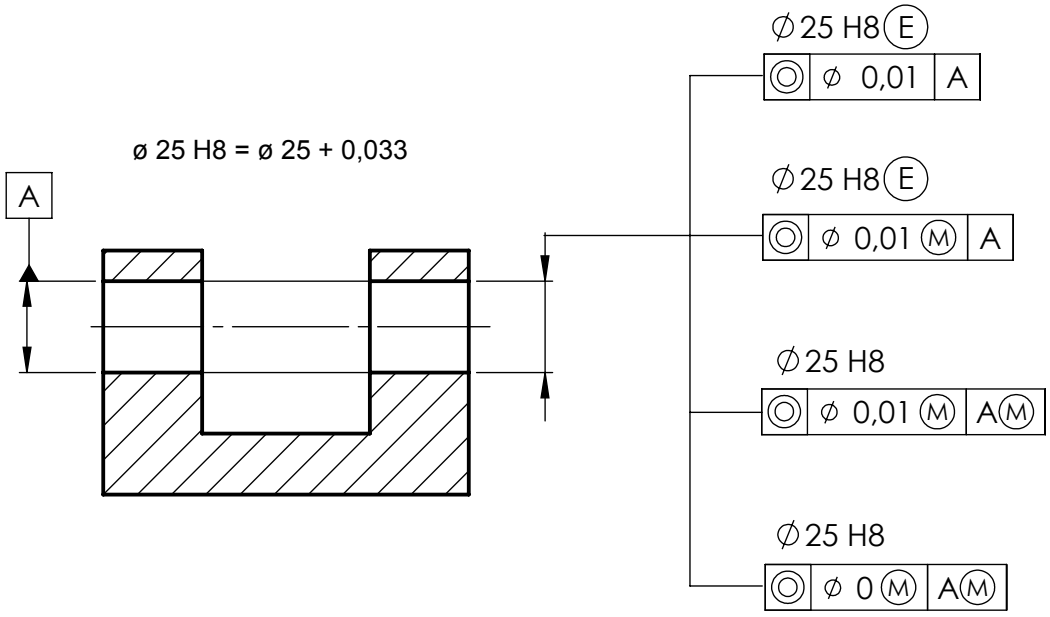


Figure 7.4 Possibilities of indicating coaxiality

From top to bottom:

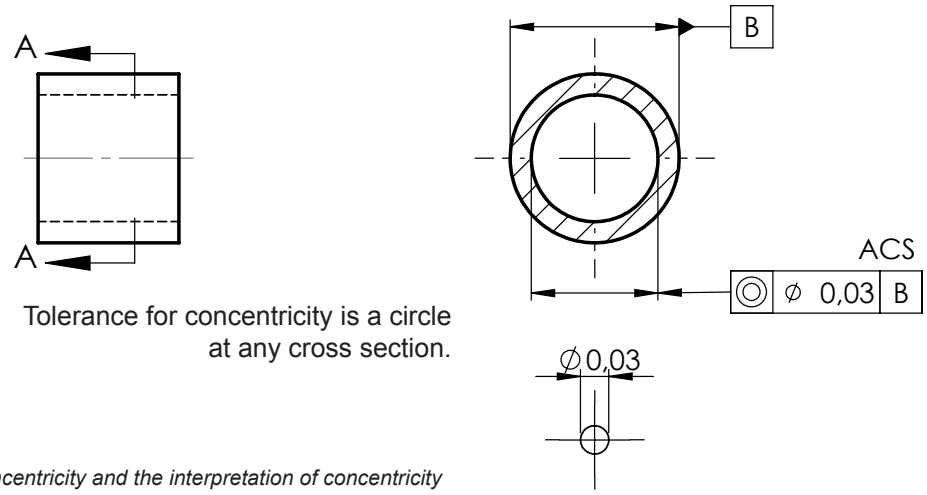
1. Each hole must respect the envelope requirement separately. The axis of the right hole must be contained in a cylinder of ø 0,01 coaxial to datum A.
2. Each hole must respect the envelope requirement separately The right hole must respect the MMVS (= ø 25 - 0,01 =ø 24,99) that is coaxial to the datum
3. A stepped caliber must fit in which has on the right the MMVS (ø 24,99) and the left the MMS (ø 25)
4. A caliber at MMS must fit both holes

7.1.2 CONCENTRICITY

The concentricity tolerance is a geometric tolerance that defines the permissible deviation of the media points of a surface of revolution from a datum center point.

SYMBOL	TYPE OF TOLERANCE	DATUM REFERENCE	ALLOWABLE TOLERANCE MODIFIERS	ALLOWABLE DATUM MODIFIERS
◎	Location	Required	Ⓕ Ⓜ Ⓛ ⌀*	Ⓕ Ⓜ Ⓛ
Concentricity tolerance zone Space within a circle			* Required	

TABLE 7.2 Concentricity



Tolerance for concentricity is a circle at any cross section.

FIGURE 7.5 Concentricity and the interpretation of concentricity

7.1.3 MMR AND COAXIALITY

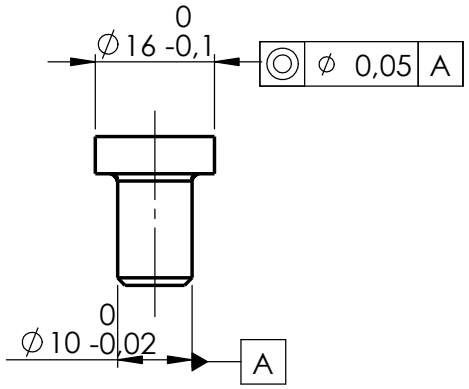


FIGURE 7.6 Coaxiality without MMR

Tolerance zone for coaxiality is a cylinder of Ø 0,05 regardless feature size.

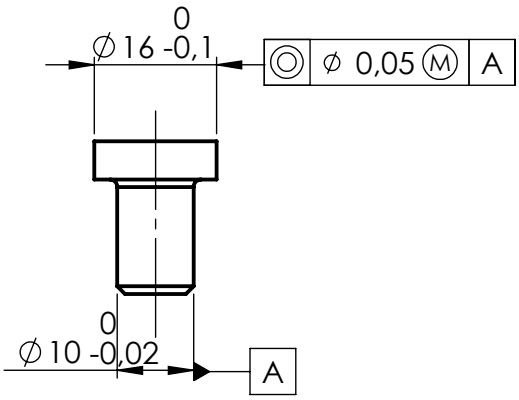


***Figure 7.7** MMR on the toleranced feature*

MMR applied to the toleranced feature

The tolerance is Ø 0,05 and as much more as the MMS of the toleranced feature of Ø 16 falls short.

Examples for the coaxiality tolerance:
Ø 16,00 >> 0,05 + 0 = Ø 0,05
Ø 15,95 >> 0,05 + 0,05 = Ø 0,10
Ø 15,90 >> 0,05 + 0,10 = Ø 0,15

When the MMR is applied to a feature the MMVC of the toleranced feature shall not be violated when applied to a datum the related MMVC of the datum feature shall not be violated.

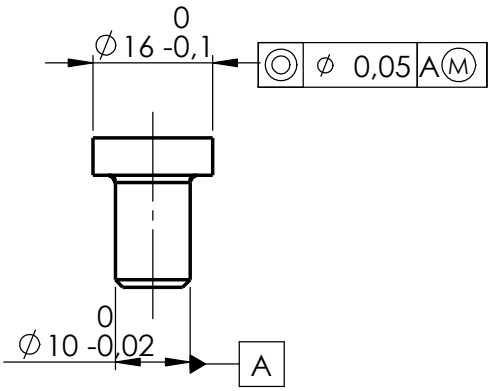


FIGURE 7.8 MMR on the datum feature

The modifier Ⓜ at the datum makes the tolerance for coaxiality dependent on the dimension of the datum.

The coaxiality tolerance is Ø 0,05 and as much more as the MMS of Ø 10 falls short.

Examples for the coaxiality tolerance:
Ø 10,00 >> 0,05 + 0 = Ø 0,05
Ø 9,99 >> 0,05 + 0,01 = Ø 0,06
Ø 9,98 >> 0,05 + 0,02 = Ø 0,07

.

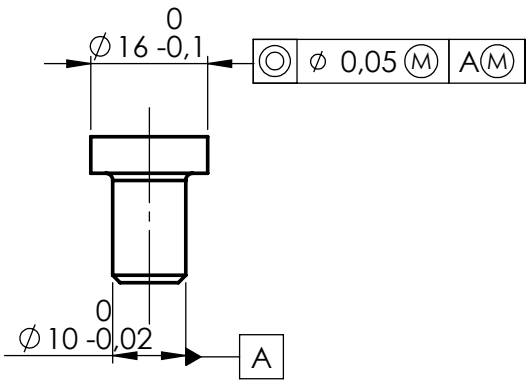


FIGURE 7.9 MMR both on toleranced and datum feature

The modifier Ⓜ at the toleranced feature and the datum feature makes the tolerance for coaxiality dependent on both the dimension of the toleranced feature and the datum feature.

The coaxiality tolerance is Ø 0,05 and as much more as the dimensions Ø 16 of the toleranced feature and the dimension of the datum feature are smaller then the MMS.

Example for the maximal coaxiality.tolerance

Ø 15,90 and Ø 9,98 >>
0,05 + 0,10 + 0,02 = Ø 0,17

7.2 SYMMETRY

7.2.1 SYMMMETRY TOLERANCE

Symmetry defines the symmetry deviation of a with feature of size to a datum center plane or datum axis. One of the features can be symmetrical and the other cylindrical.

When dimensioning symmetrical relationships several geometrical tolerances may be used. The choices are position, profile and symmetry. The use of symmetry is rare. Both position and symmetry have the same shape of tolerance zone and both limit deviations in location and orientation.

SYMBOL	TYPE OF TOLERANCE	DATUM REFERENCE	ALLOWABLE TOLERANCE MODIFIERS	ALLOWABLE DATUM MODIFIERS
⌯	Location	Required	Ⓕ Ⓜ Ⓛ	Ⓕ Ⓜ Ⓛ
Symmetry tolerance zone Space between parallel planes				

TABLE 7.3 Symmetry

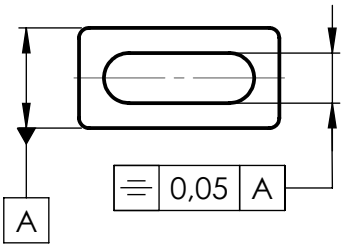


FIGURE 7.10 Symmetry

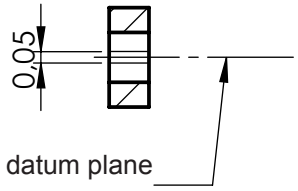


FIGURE 7.11 Interpretation

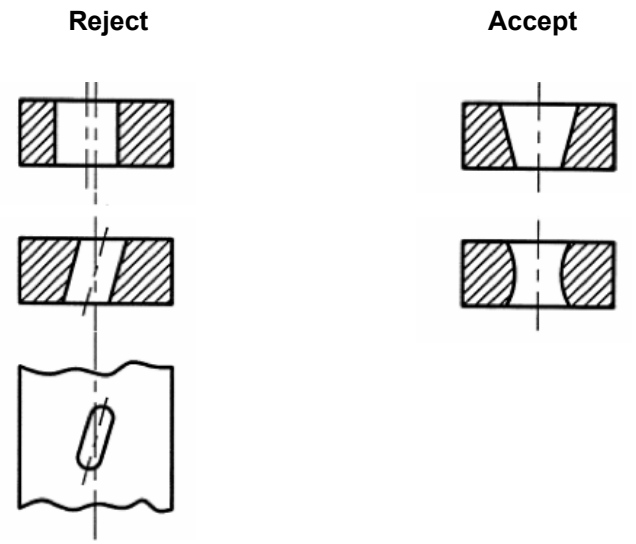
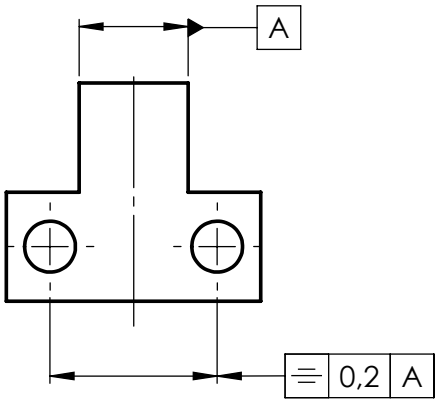
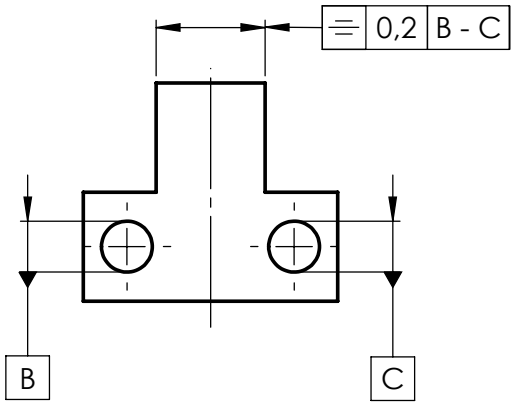


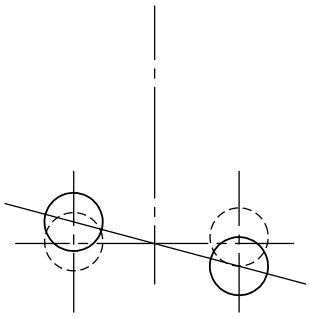
Figure 7.12 Possible deviations

7.2.2 INTERCHANGING TOLERANCED FEATURE AND DATUM FEATURE









In the example to the left the perpendicularity deviation of the plane through the holes is not limited in relation to the datum media plane.

Making the holes the datum features the perpendicularity is also limited.

Figure 7.13 Interchanging toleranced feature and datum feature

Exercise

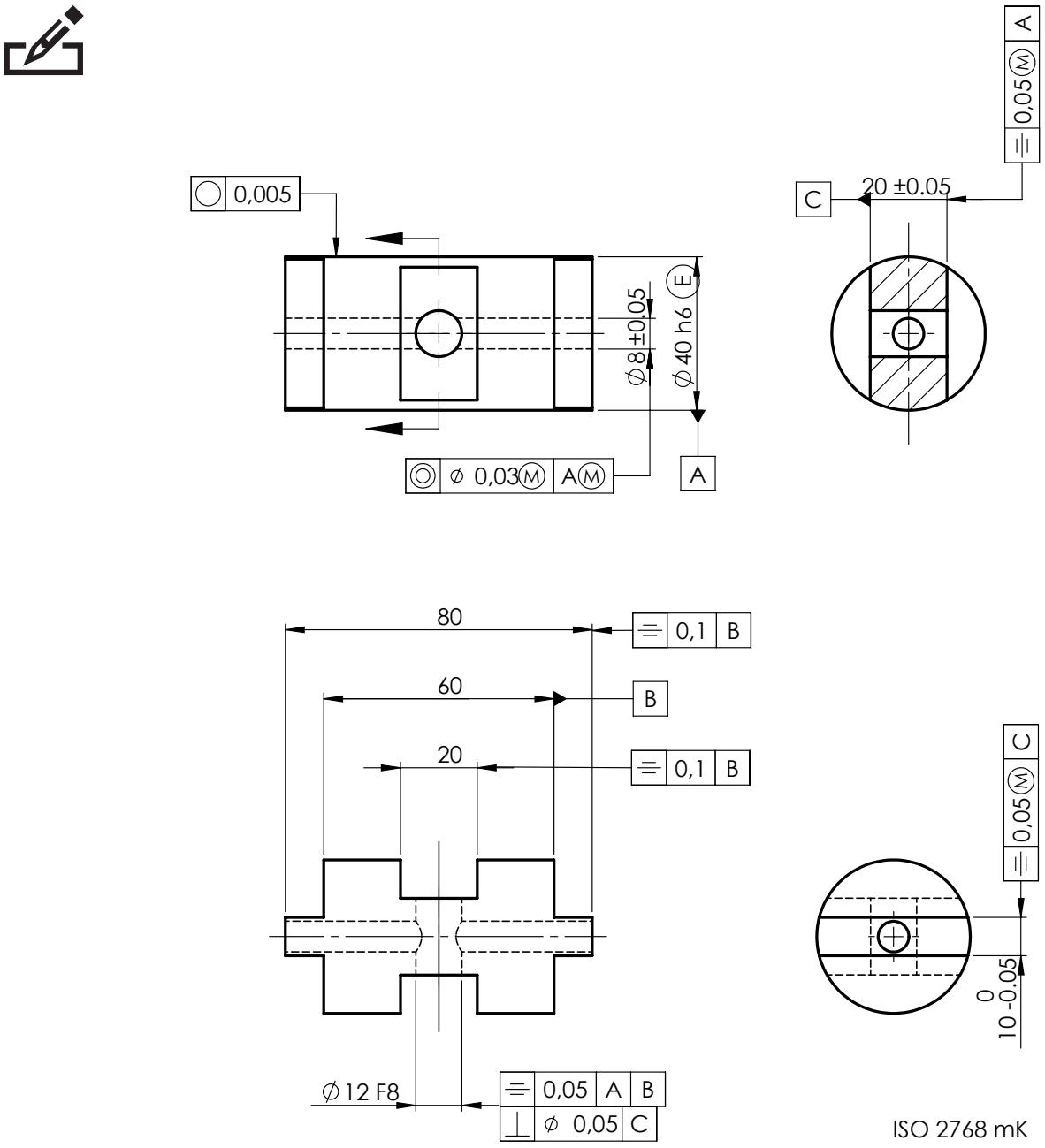


Figure 7.14 Exercise

- What is the shape and size of the form tolerance zone?
- Where is the envelope requirement used for?
- Which orientation tolerances are stated and what are the shape and size of the tolerance zones?
- Which location tolerances are stated and what are the shape and size of the tolerance zones?

7.3 POSITION TOLERANCE

A position tolerances is a tolerance of location that directly controls location and indirectly controls form and orientation of a feature of size. It's among the most common used tolerances.

A position tolerance limits the amount a center point, axis or center plane of a feature is permitted to deviate from nominal position defined by theoretical exact dimensions.

A datum may or may not be used.

In comparison to coordinate tolerancing position tolerances offer several advantages.

- Provides 57% larger tolerance zone
- Permits bonus tolerances (MMC and LMC)
- Prevents tolerance accumulation
- Permits use of function gage (MMC and LMC)
- Lowers manufacturing and inspection costs

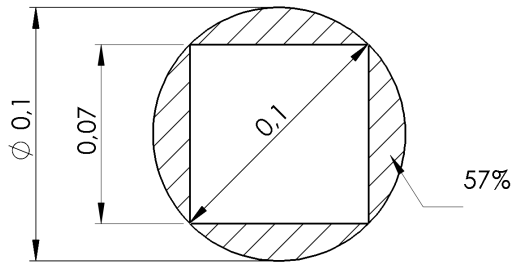


FIGURE 7.15 Comparison of zones with round and square cross-section

7.3.1 THEORETICAL EXACT DIMENSION

A Theoretical Exact Dimension (TED) is the theoretical exact location of a feature of size as established by nominal dimensions.

TED's also known as boxed dimensions must not be toleranced. The dimension is shown in a rectangular frame.

Theoretically exact dimensions may only vary by the geometric tolerance that is stated in the tolerance frame associated with them.

Theoretically exact dimensions should be used when dimensioning the theoretically exact location of features for tolerances of Angularity, Position, Line Profile and Surface Profile

Position applied to a surface

The requirement is that the toleranced feature is positioned with a tolerance of 0,03 mm with respect to datum feature A. The tolerance zone location is located and oriented by the TED. The deviations of form, orientation and location are controlled.

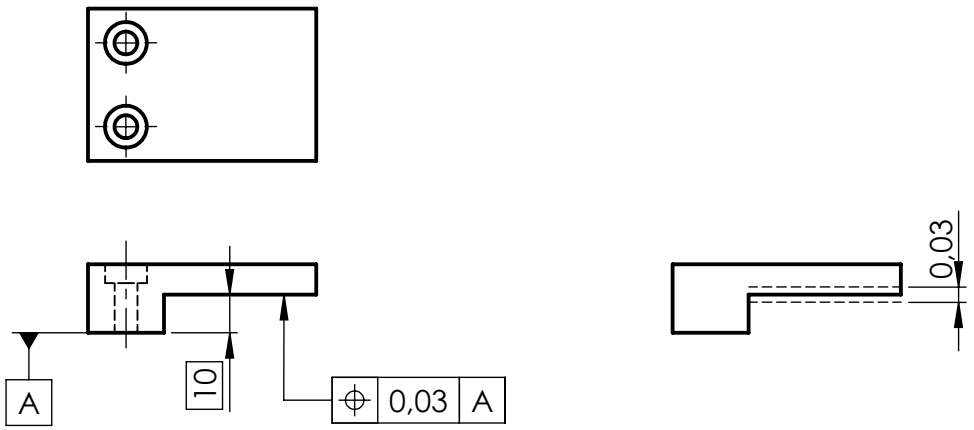


Figure 7.16 Position applied to a surface

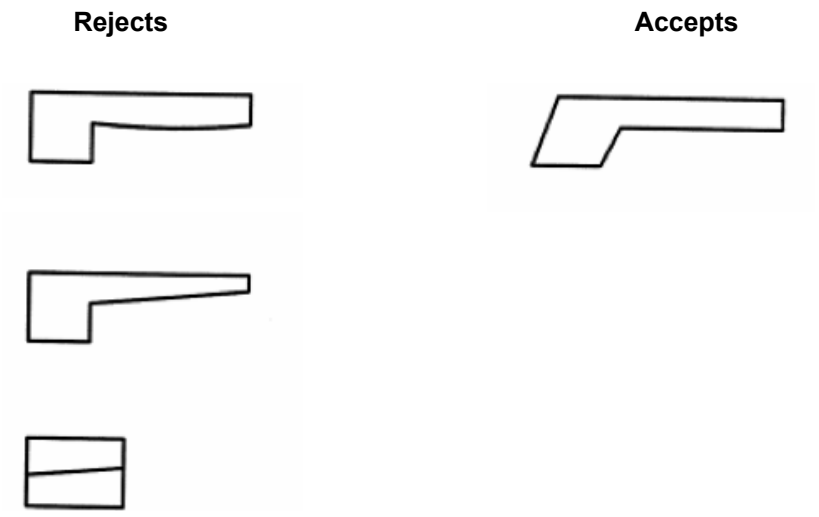


Figure 7.17 Position deviations

Position applied to a hole

Position tolerances are commonly used to control :

- Distance between features of size
- The location of a pattern of features of size
- The coaxiality between features of size
- A symmetrical relation between features of size

SYMBOL	TYPE OF TOLERANCE	DATUM REFERENCE	ALLOWABLE TOLERANCE MODIFIERS	ALLOWABLE DATUM MODIFIERS
⌖	Location	Required	Ⓜ Ⓛ Ⓟ Ⓕ ∅	Ⓜ Ⓛ Ⓕ
Position tolerance zone Spaced between two parallel planes Space within a cylinder (when ø is shown)				

Table 7.4 Interpretation

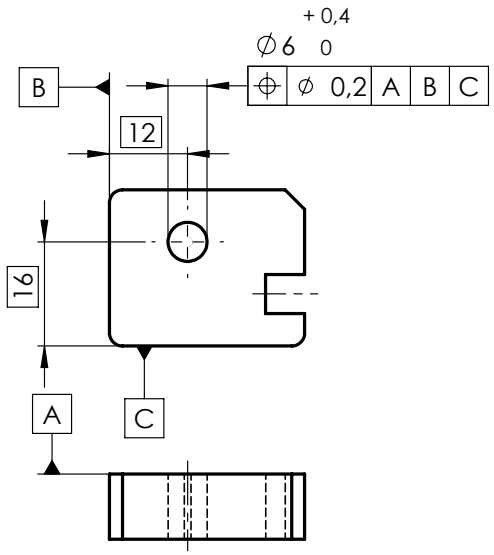


Figure 7.18 Position applied to a hole

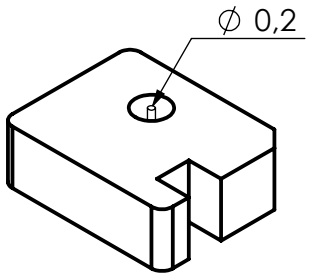


Figure 7.19 Interpretation

The default tolerance zone is two parallel planes. Where the diameter symbol is specified in the feature control frame the tolerance zone shape is a cylinder.

A position tolerance zone is located and oriented by the specified TED's. In most cases datum reference features are used in the feature control frame. Here the tolerance zone is perpendicular to datum A.

Position, Best Fit

The requirement is that the centerlines of the 4 holes have to be within a cylindrical tolerance zones. The position of the tolerance zones is determinted by the TED's. When there are no datums then the requirement is 'Best fit'

'Best fit': The workpiece is to be rotated and translated in all directions for smallest deviations of the position tolerance related features.

Orientation and location of the side, top and bottom surfaces is random.

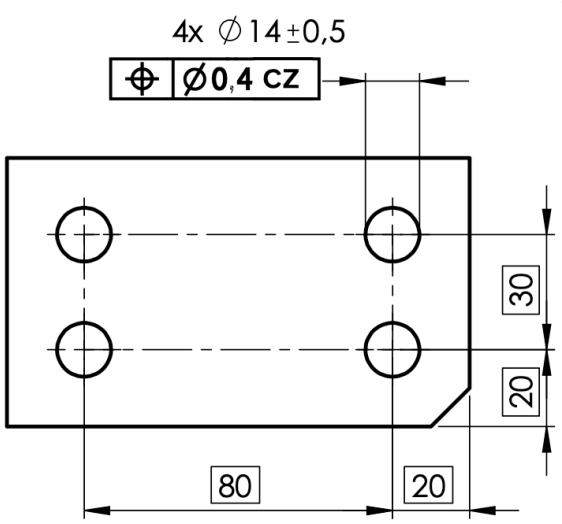


Figure 7.20 Positioning of a pattern of holes

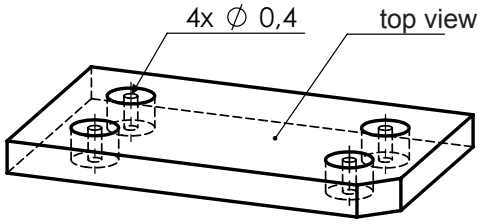


Figure 7.21 Interpretation

Coordinate dimensions are used to position the pattern of the with respect to the workpiece surfaces. Position tolerances are used to define the pattern.

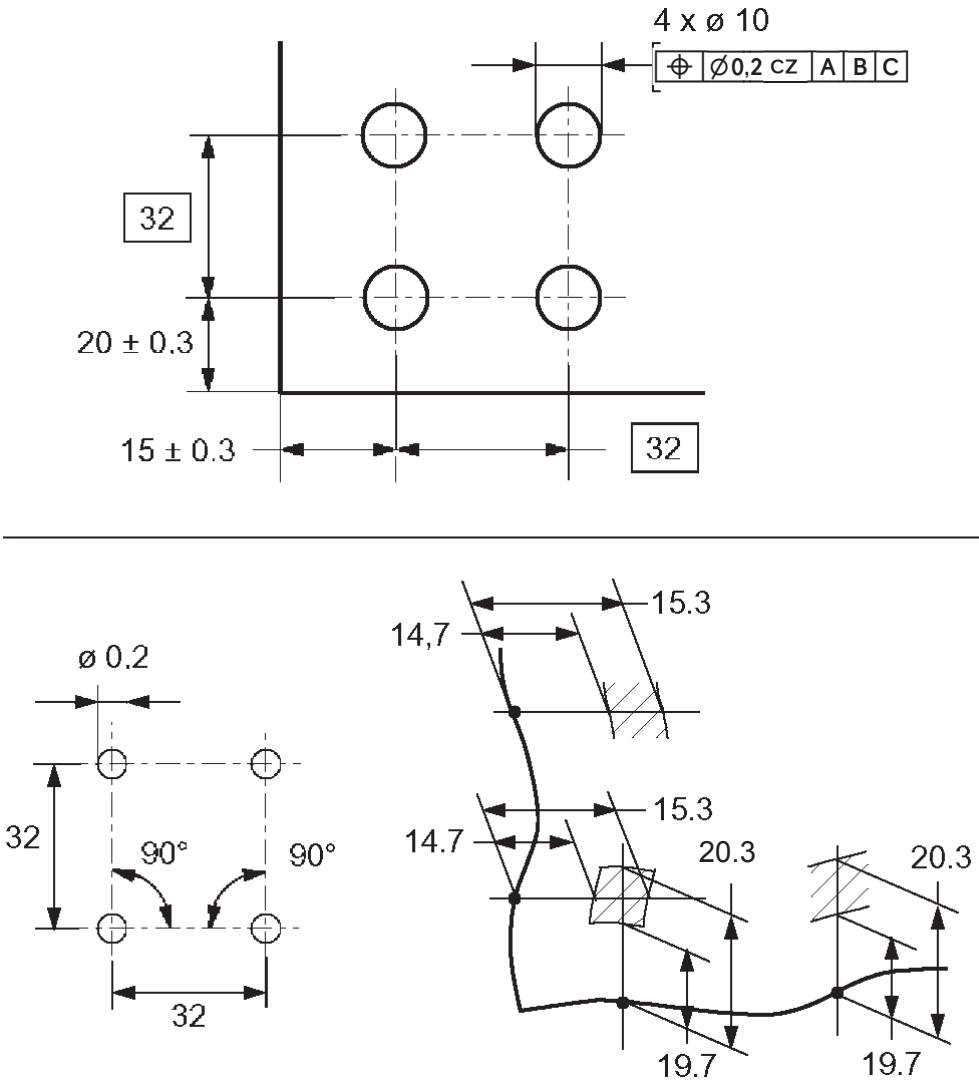


FIGURE 7.22 Positioning a pattern of holes

Superposition of positional tolerances

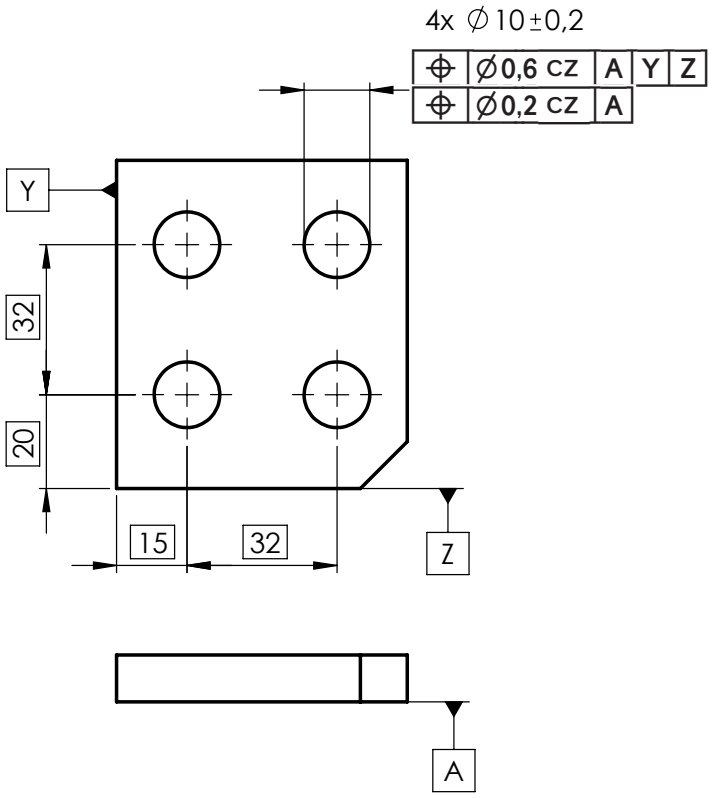


Figure 7.23 Superposition of position tolerances

The axis of the four holes shall be within a cylindrical tolerance zone of ø 0,2 perpendicular to datum A. The actual axis of each hole shall be within a cylindrical tolerance zone of ø 0,6. The positional tolerance zones are located to their theoretical exact positions in relation to the datums A, Y and Z.

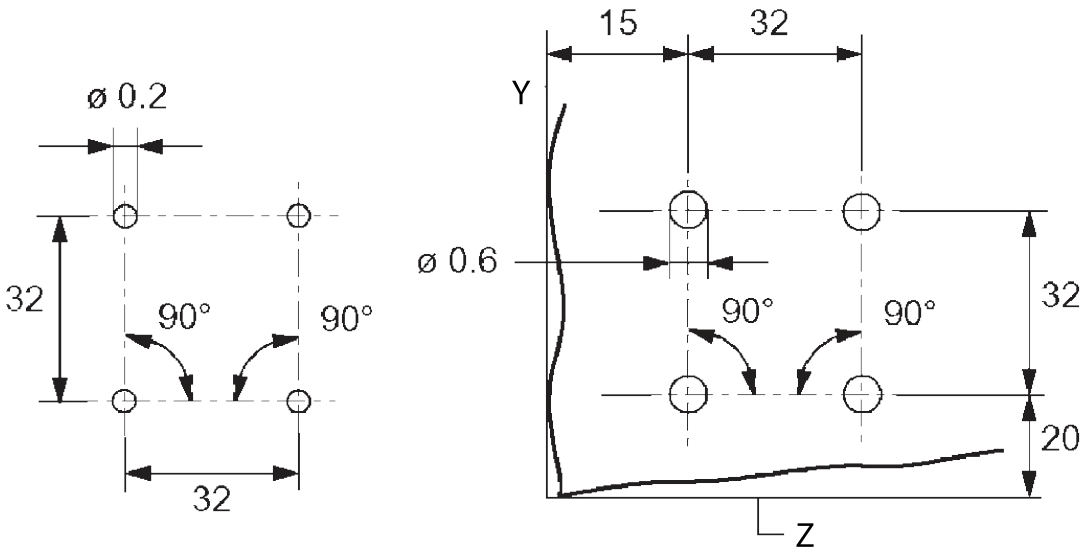


Figure 7.24 Superposition tolerances explained

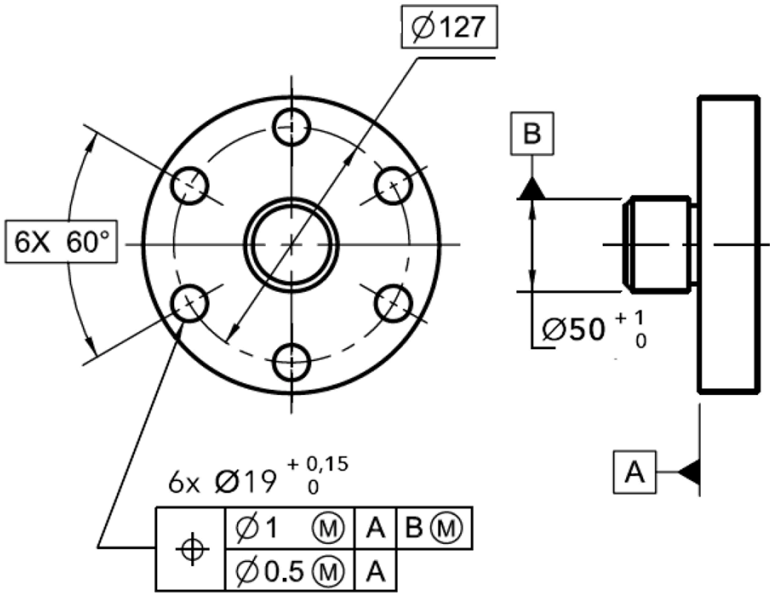


FIGURE 7.25 Superposition, example from ASME

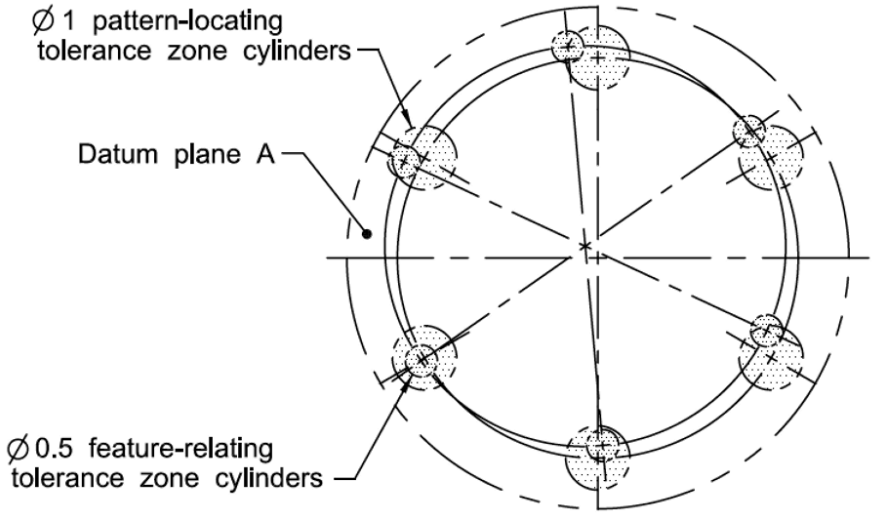


FIGURE 7.26 Superposition tolerances explained

7.4 ASSESSMENT OF TRUE POSITION OF HOLES

The center line of a hole to the center line of another hole or with respect to a surface can be assessed using a center line caliper. What is measured however are coordinate dimensions. These coordinate dimensions can be translated towards position tolerances.

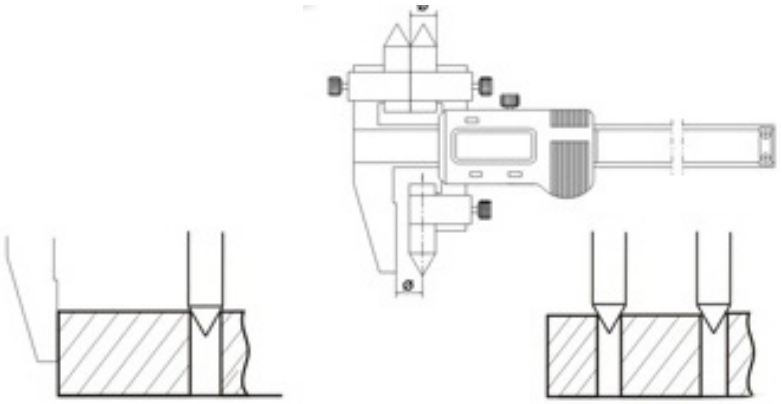


FIGURE 7.27 Centerline caliber

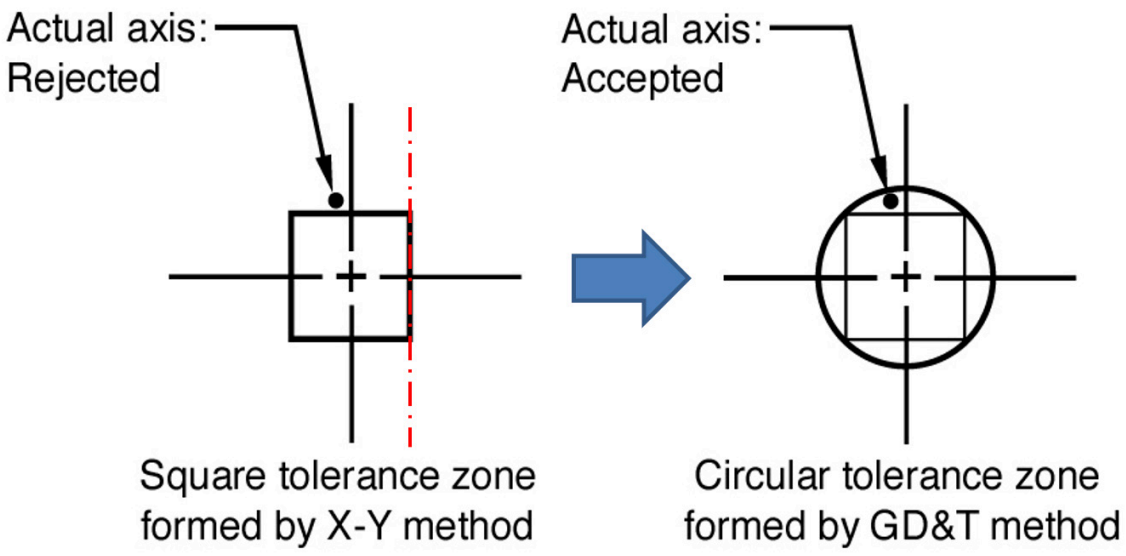


FIGURE 7.28 True position

7.5 POSITION AT MMC

Where a position tolerance applies at MMC the tolerance zone is a virtual condition acceptance boundary located (VBC = MMVC) at true position which the surface of the tolerance feature of size must not violate.

MMC for a hole is when the hole is at its smallest size.

MMVC = MMS - geometrical tolerance = 6 - 0,4 = 5,6

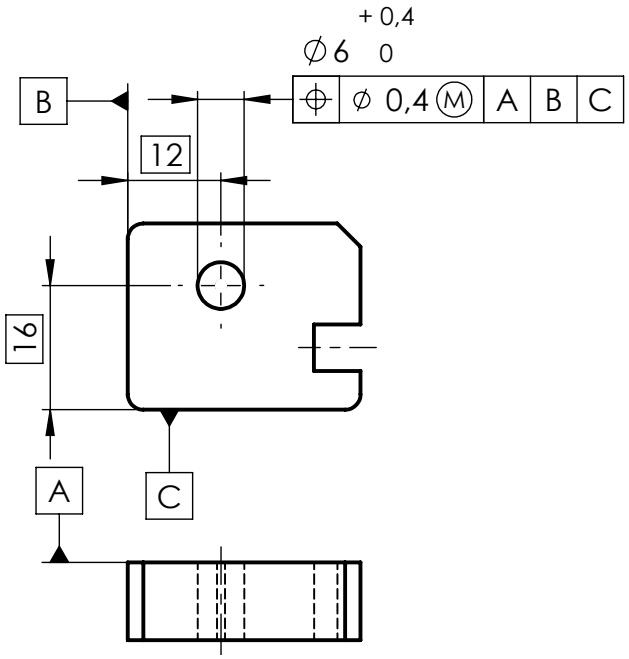


FIGURE 7.29 Position MMC applied to a hole

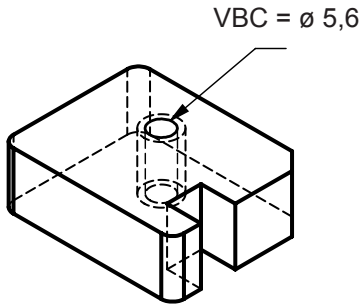


FIGURE 7.30 Interpretation

DIAMETER HOLE	POSTION Ø TOLERANCE	BONUS TOLERANCE	TOTAL LOCATION TOLERANCE Ø
6,0 MMC	0,4	0	0,4
6,2	0,4	0,2	0,6
6,4 LMC	0,4	0,4	0,8

TABLE 7.5 Position MMC applied to a hole

7.6 ASSESSMENT OF TRUE POSITION OF PATTERN OF HOLES

When true position is applied to a pattern of holes the tolerance is the virtual boundary or MMVC shall not be violated.

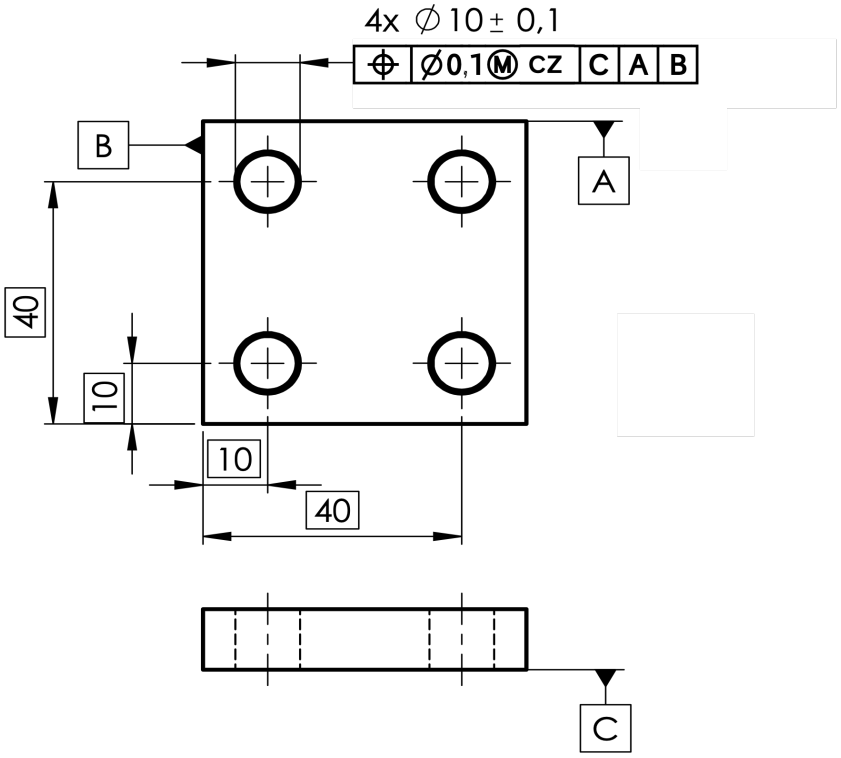


FIGURE 7.31 Pattern of holes in MMC

The MMS for the holes is 9,9

MMVC = MMS - geometric tolerance = 9,9 - 0,1 = 9,8

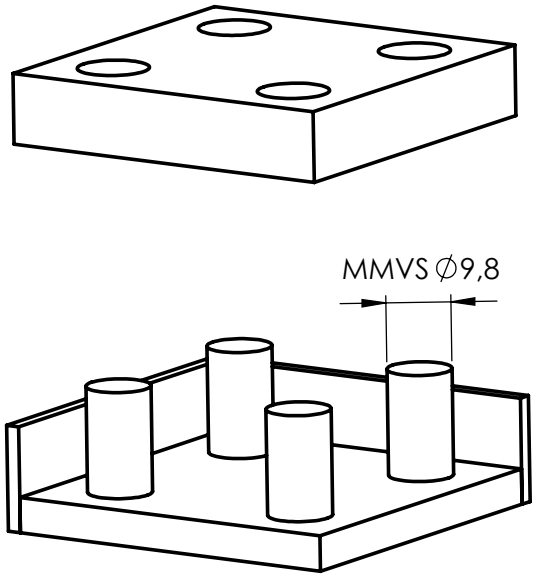


Figure 7.32 Caliber for the pattern of holes

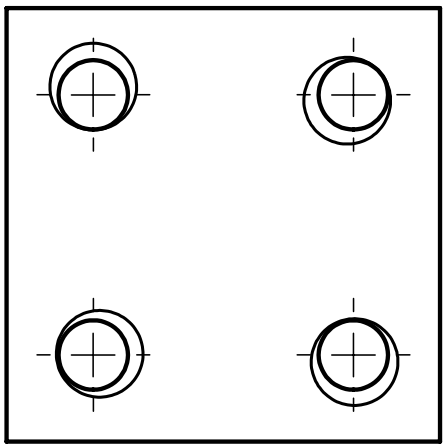


Figure 7.33 Possible position of the holes

7.7 GROUP OF HOLES AS DATUM

The pattern of four holes is related to datum A, B and C. The MMVS (ø 11,85) of this group together establishes the datum D-D for the positional tolerance ø 0,15 of the three hole group.

The MMVS of the four holes (ø 11,85) together with the MMVS of the three holes (ø 7,75) must fit into the holes. In addition perpendicular to D the MMVS (ø7,85) for the positional tolerance ø 0,05 of the three holes must fit together into the three holes.

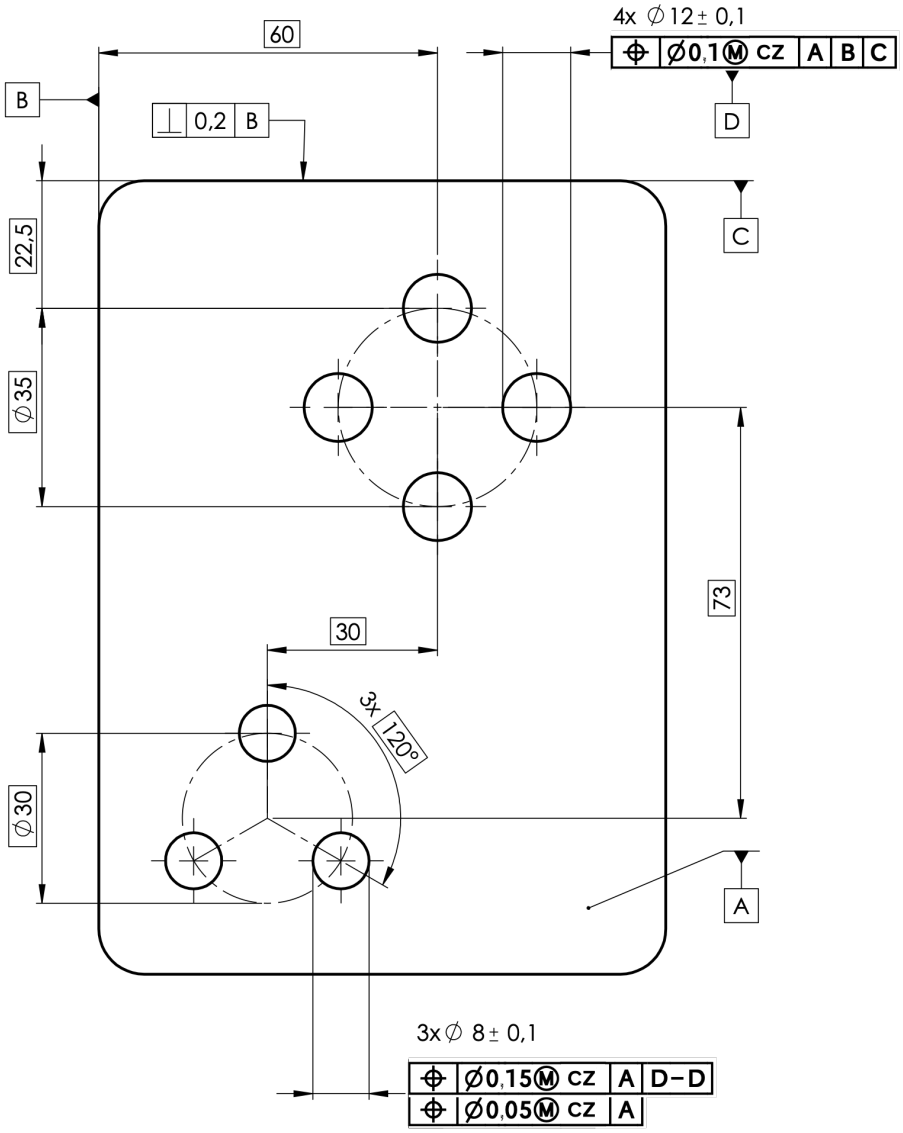


FIGURE 7.34 Super position and a group of features as datum

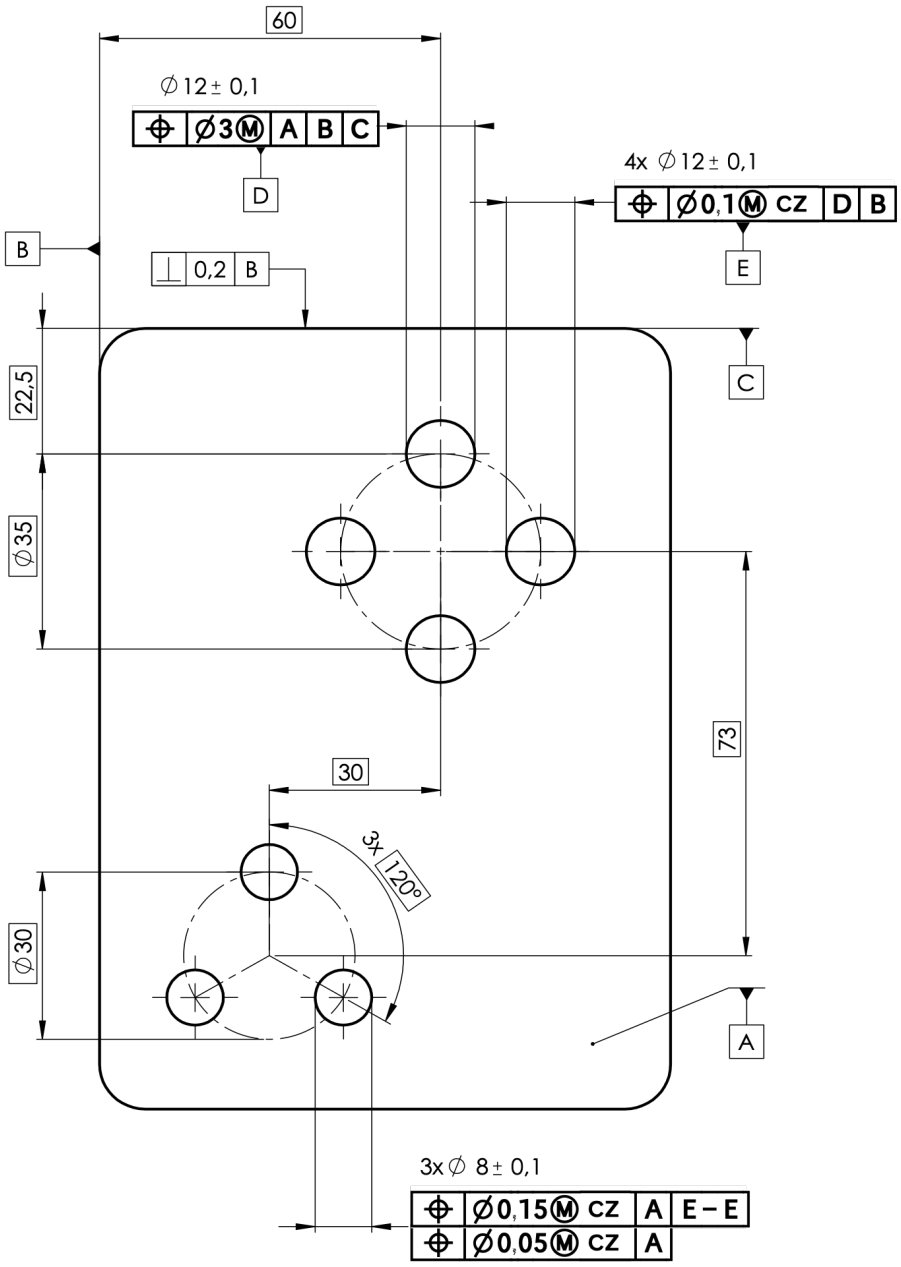


FIGURE 7.35 Super position and a group of features as datum

7.8 MAXIMAL MATERIAL REQUIREMENT AT 0

How tolerances are specified affects the part function and manufacturing costs. Tight tolerances do not guarantee a quality part only an expensive one.

"Zero tolerance at MMC" protects the parts function and reduces costs by offering maximal flexibility for manufacturing.

"Zero tolerance at MMC" is indicated by 0 Ⓜ and states that the functional tolerance is not distributed on size and position but provided for both.

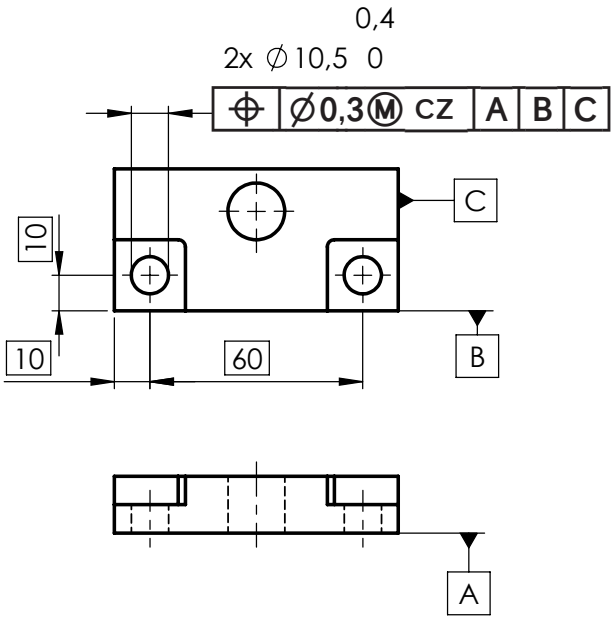


FIGURE 7.36 Conventional tolerancing

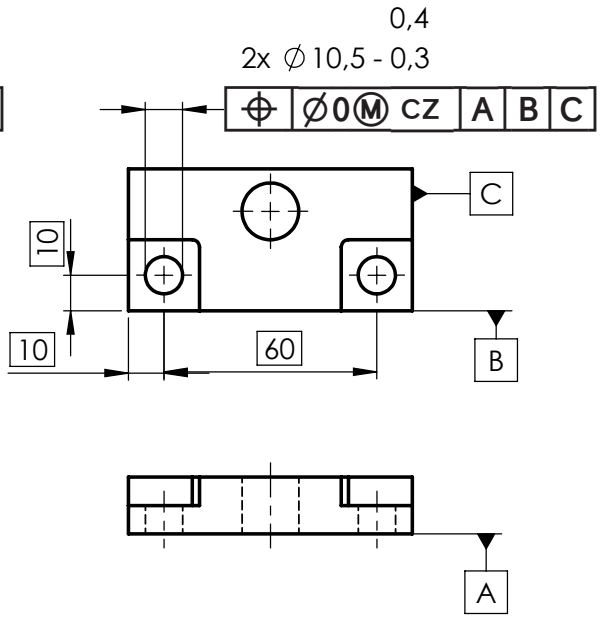


FIGURE 7.37 Zero tolerance at MMC

Position tolerance:	0,3
Bonus tolerance	0,4
Total tolerance	0,7

MMC hole	10,5
Position tolerance	- 0,3
Virtual condition	10,2

Position tolerance:	0
Bonus tolerance	0,7
Total tolerance	0,7

MMC hole	10,2
Position tolerance	- 0,0
Virtual condition	10,2

7.9 FIXED AND FLOATING FASTENERS

When in an assembly two or more parts are held together using fasteners the position tolerance 'T' for the holes can be calculated

One part can be threaded then this is called "fixed fasteners". When both holes have clearance and bolts are used this is called "floating fasteners"

T = Total position tolerance
H = MMC of the clearance hole
F = MMC of the fastener

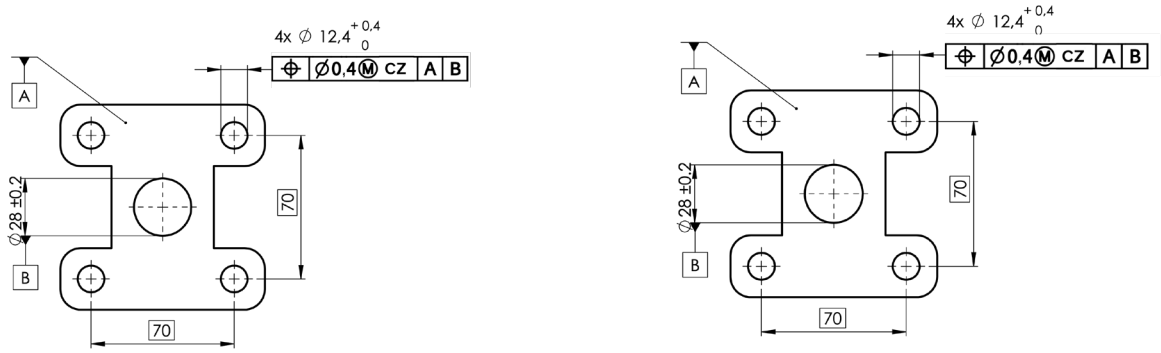


FIGURE 7.38 "Floating fasteners"

T =H - F

MMS of a M12 bolt is considered 12

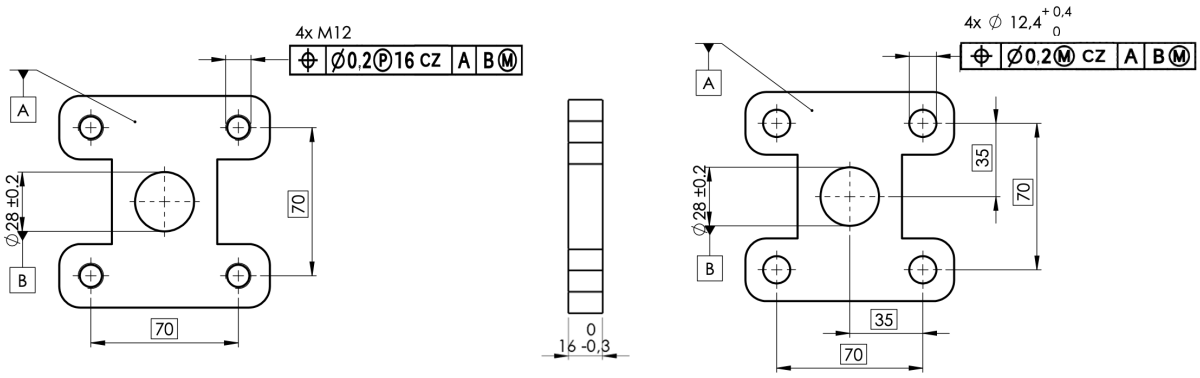


FIGURE 7.39 "Fixed fasteners"

Position tolerance: T = (H - F)/2

MMS of a M12 bolt is considered 12

The height of the part is 16mm

7.9 ASSESSMENT OF LOCATION TOLERANCES

$$A_i = \frac{A_{oi} + A_{ui}}{2} - A_{th}$$

$$\delta_o = \sqrt{(A_{ix}^2 + A_{iy}^2)} \leq t_o/2$$

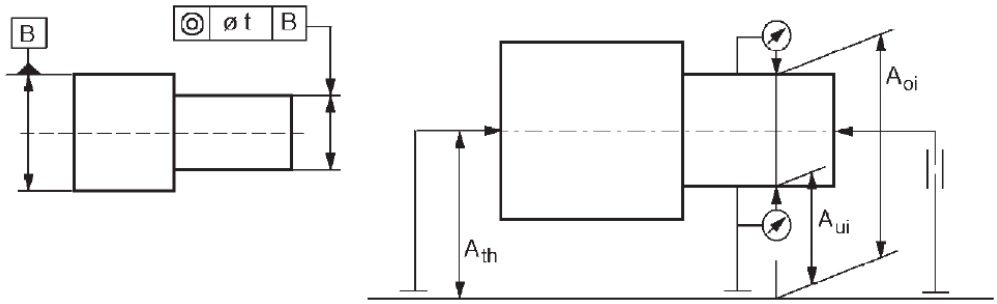


FIGURE 7.40 Assessment of the location deviation δ_o

On the cylindrical surface the values A_i and A_u must be accessed in each axial location in sections perpendicular to each other Ai_x and Ai_y δ_o is calculated using Pythagoras. δ_o is half the location tolerance.

Location deviations of a surface is verified w.r.t. a surface like the flatness assessment.

$$\delta_s = \frac{|A_{i1} - A_{i2}|_{max}}{2} \leq t/2$$

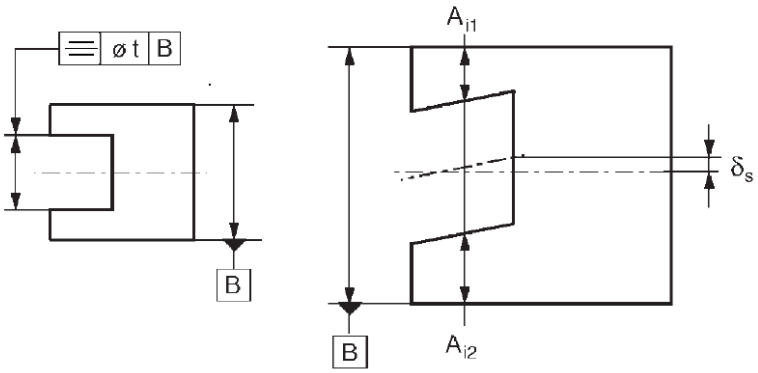


FIGURE 7.41 Assessment of location deviation

7.10 PATTERNS AND COMBINED SPECIFICATIONS

The left hand side examples are equal in ISO 5458:1998 and ISO 1101: 2017. However in 1998 standard the two small holes are considered one feature with respect to datum hole A. The holes were on one 180º line.

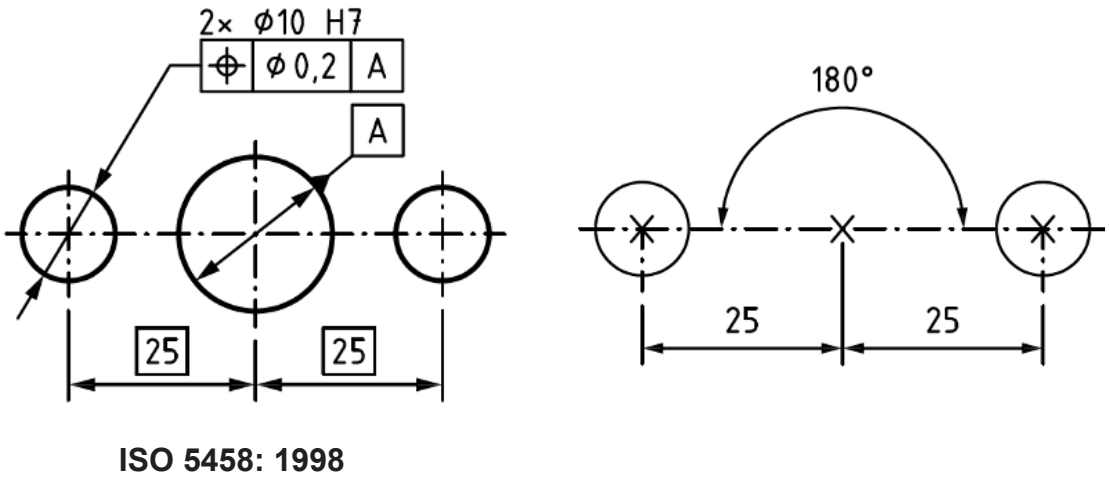


FIGURE 7.42 Ambigeous indication for patterns

In 2017 the same example had a different interpretation due to the focus on the indendency principle. The position requirement only states the location of the holes as indicated y the TED's. The holes are no longer a group of holes with respect to the datum

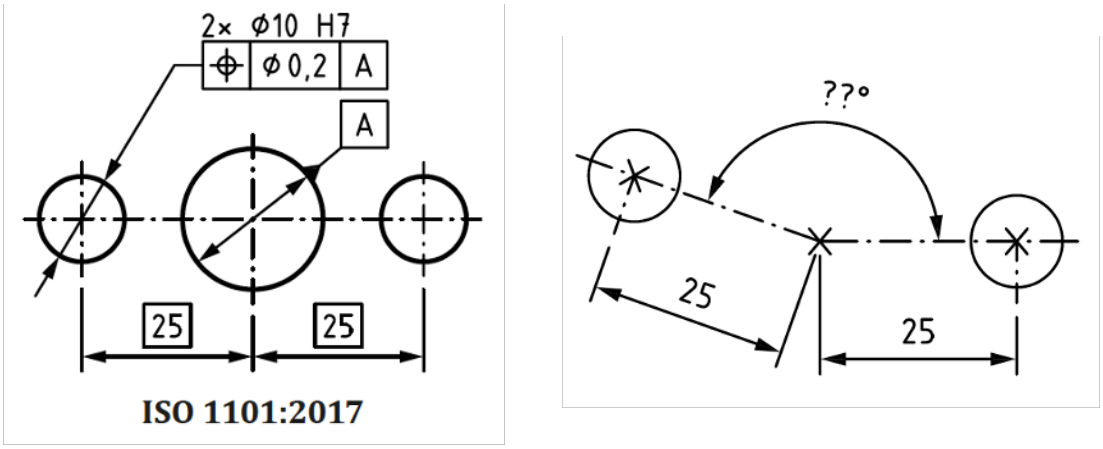


FIGURE 7.43 Ambigeous indication for patterns

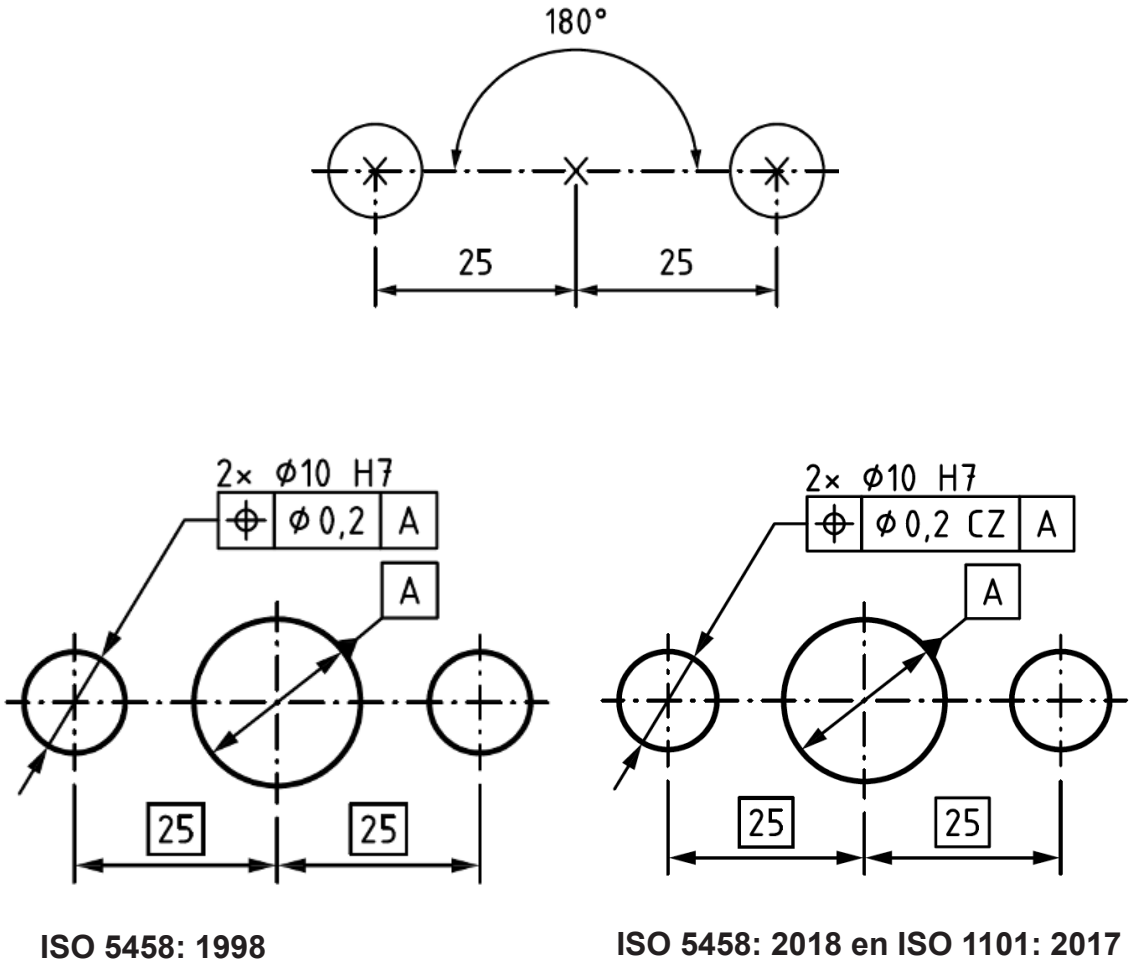
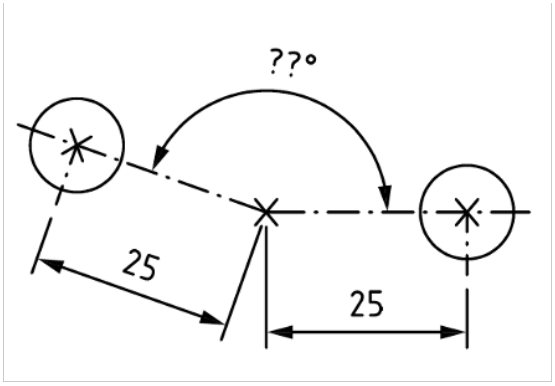
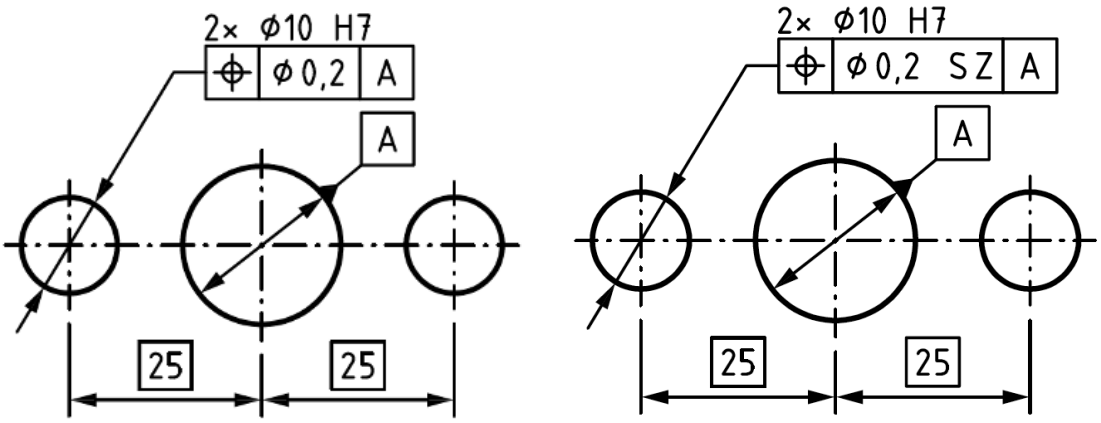


Figure 7.44 Ambigeous indication for patterns





ISO 1101: 2017 **ISO 5458: 2018**

Figure 7.45 Ambiguous indication for patterns

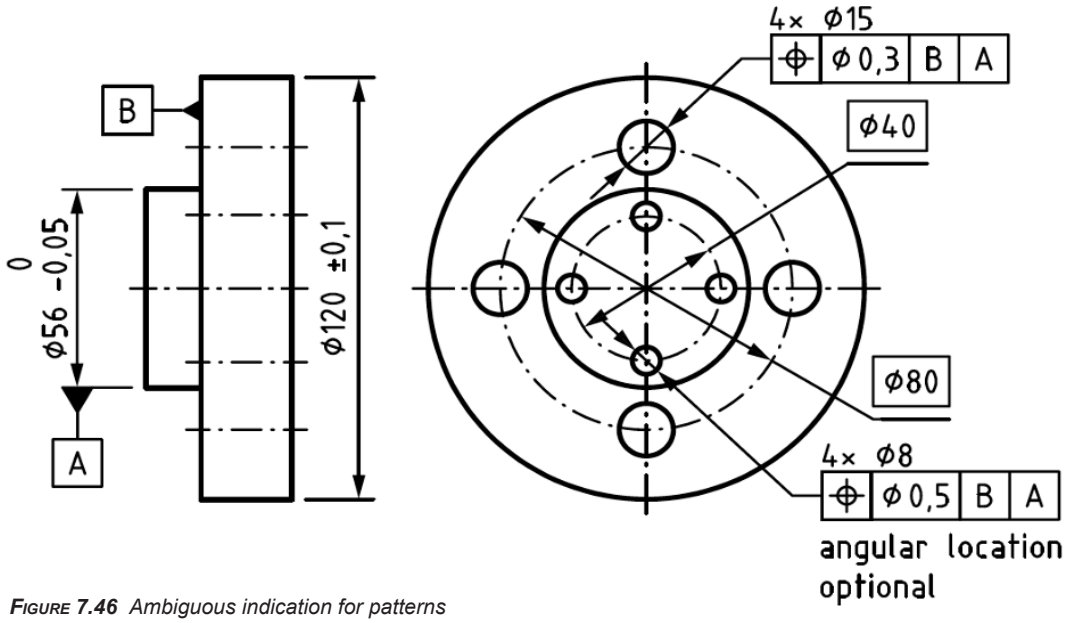


Figure 7.46 Ambiguous indication for patterns

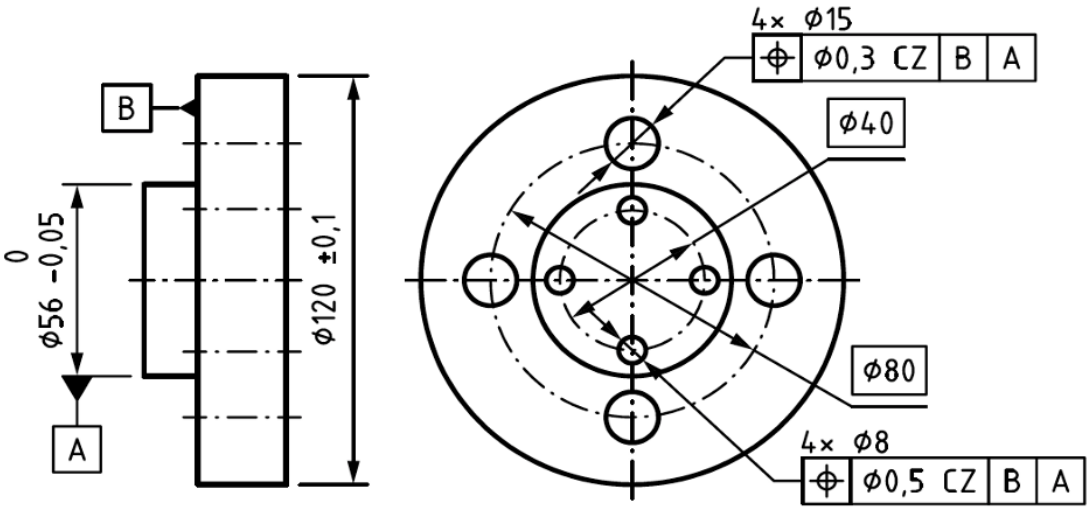


Figure 7.47 Unambiguous indication for patterns

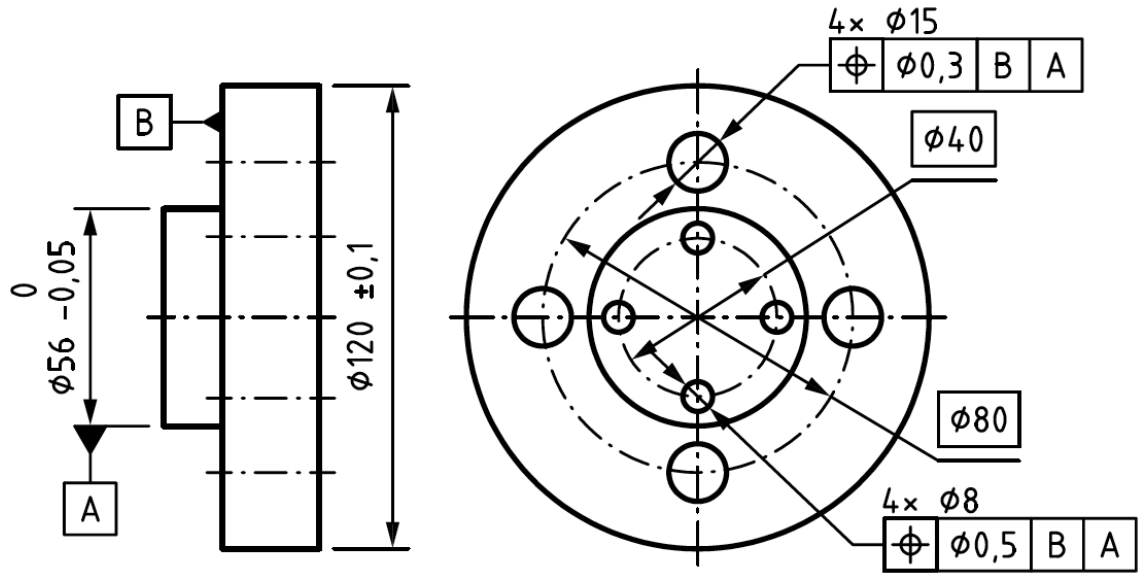


Figure 7.48 Ambigeous indication for patterns

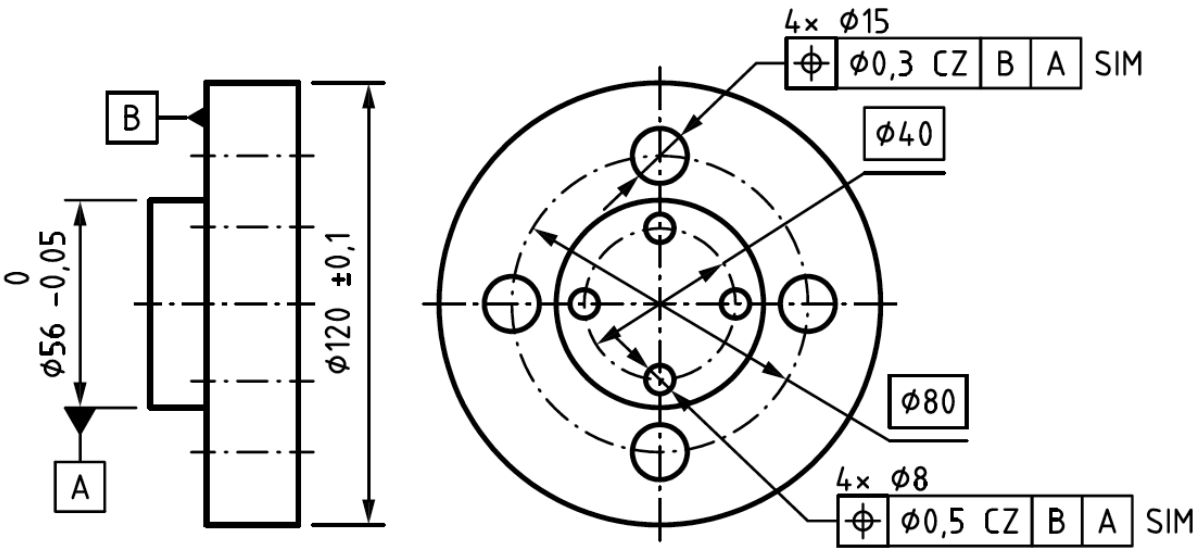


Figure 7.49 Unambigeous indication for patterns

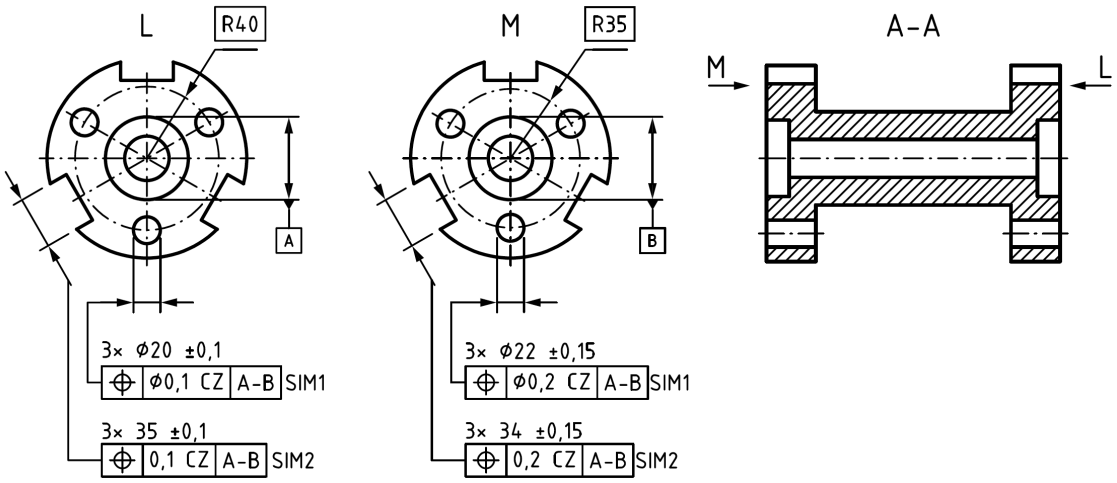


FIGURE 7.50 Unambiguous indication for patterns

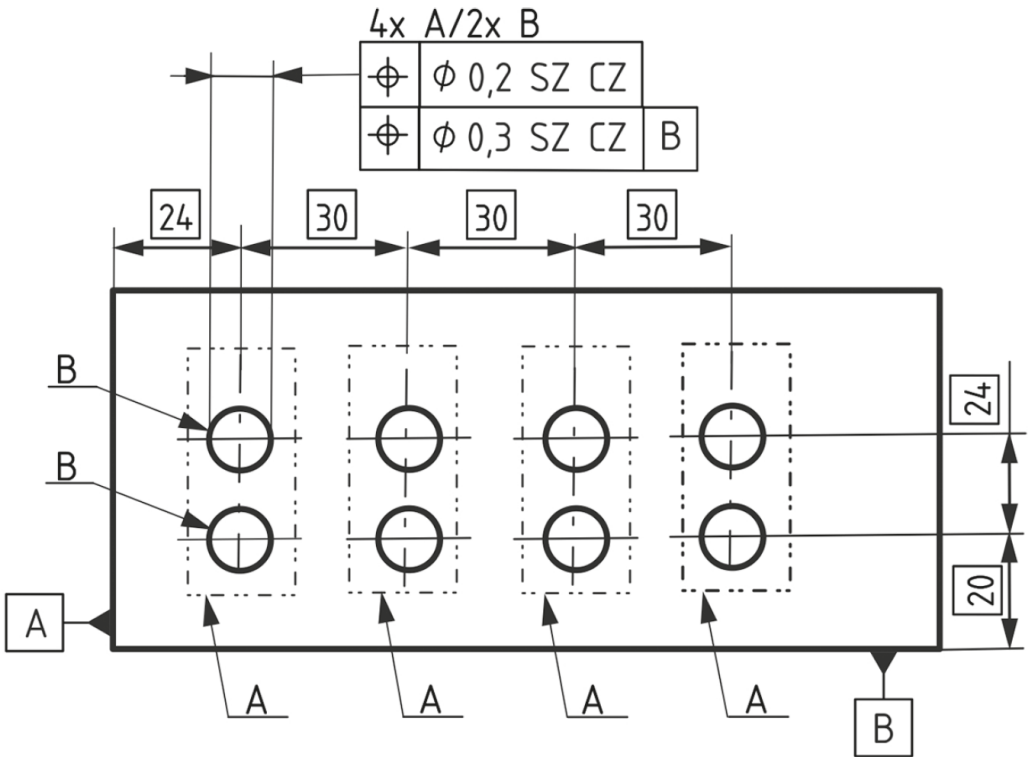


FIGURE 7.51 Ambiguous indication for patterns

7.11 MMR AND GROUPED ELEMENTS

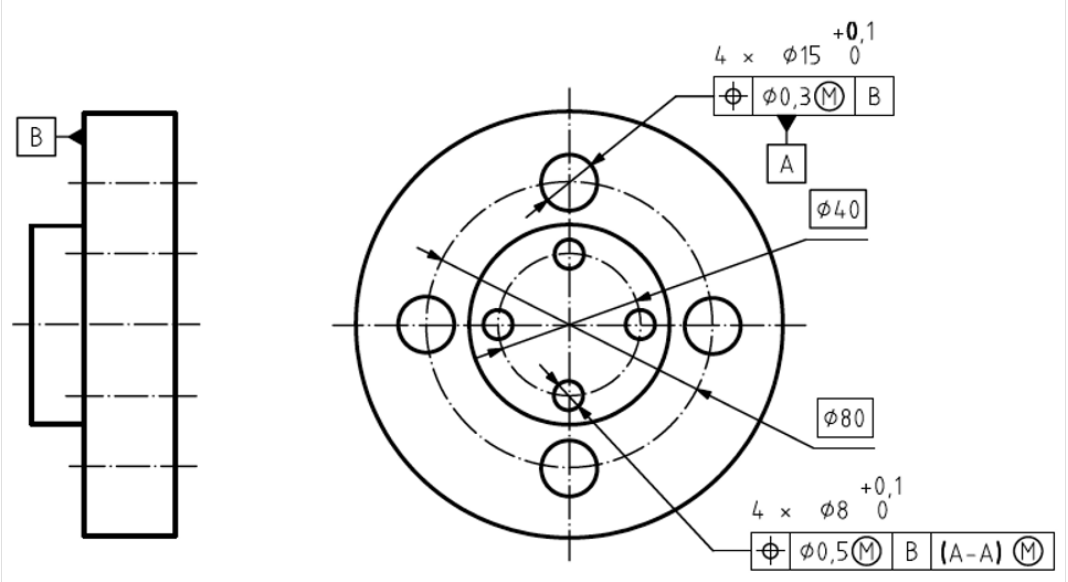


FIGURE 7.52 Till 2021 considered as definition of a pattern

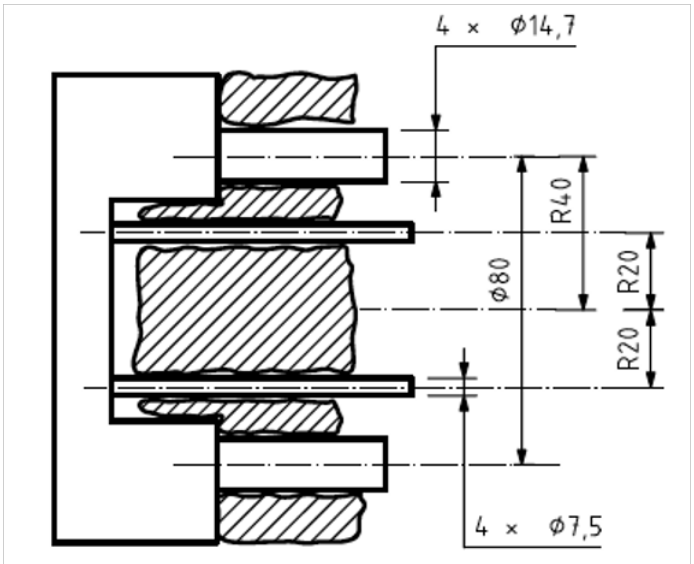


FIGURE 7.53 Interpretation for the pattern

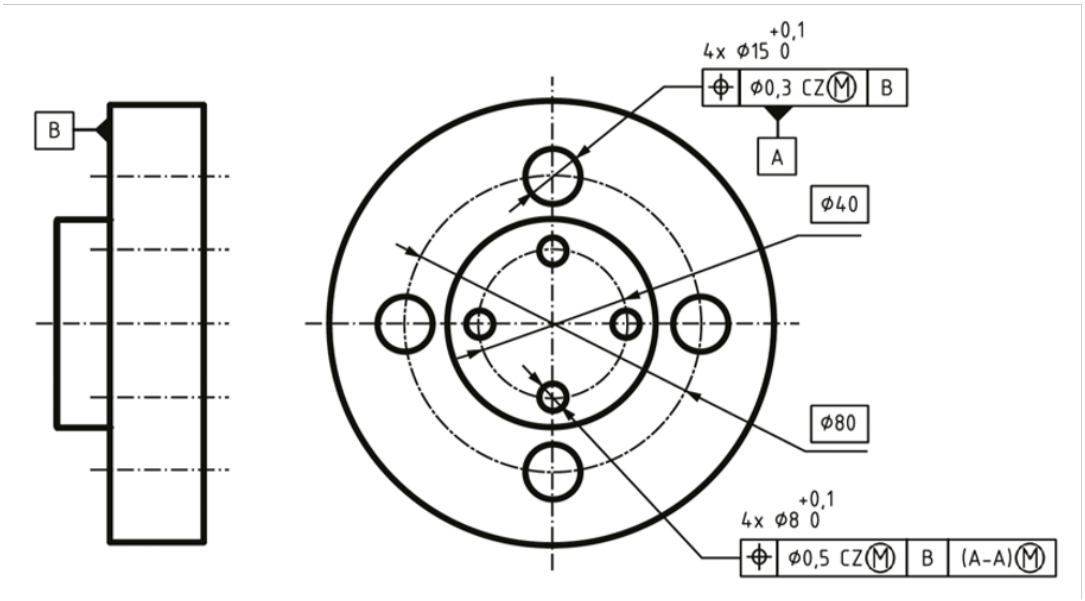


Figure 7.54 Two patterns defined by the modifier CZ

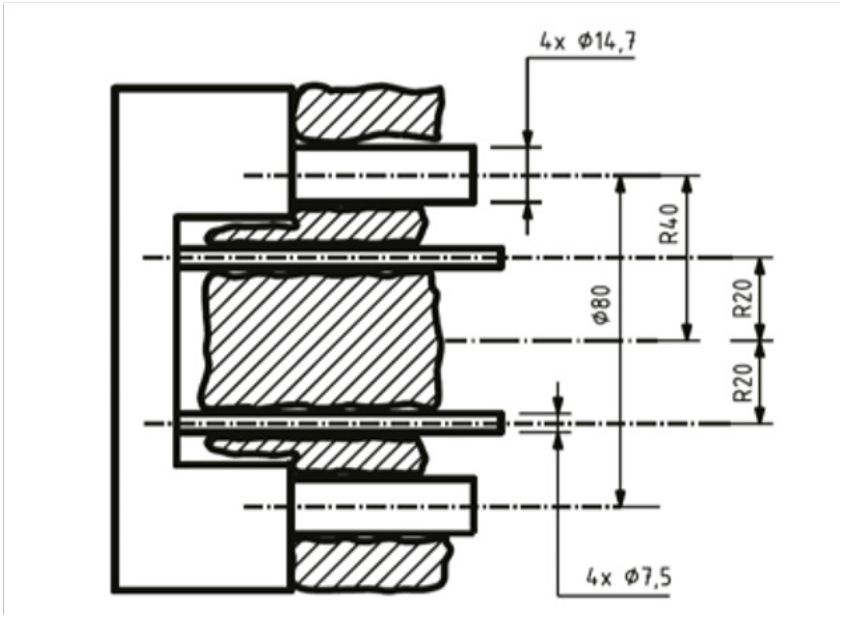


Figure 7.55 Interpretation for the two patterns

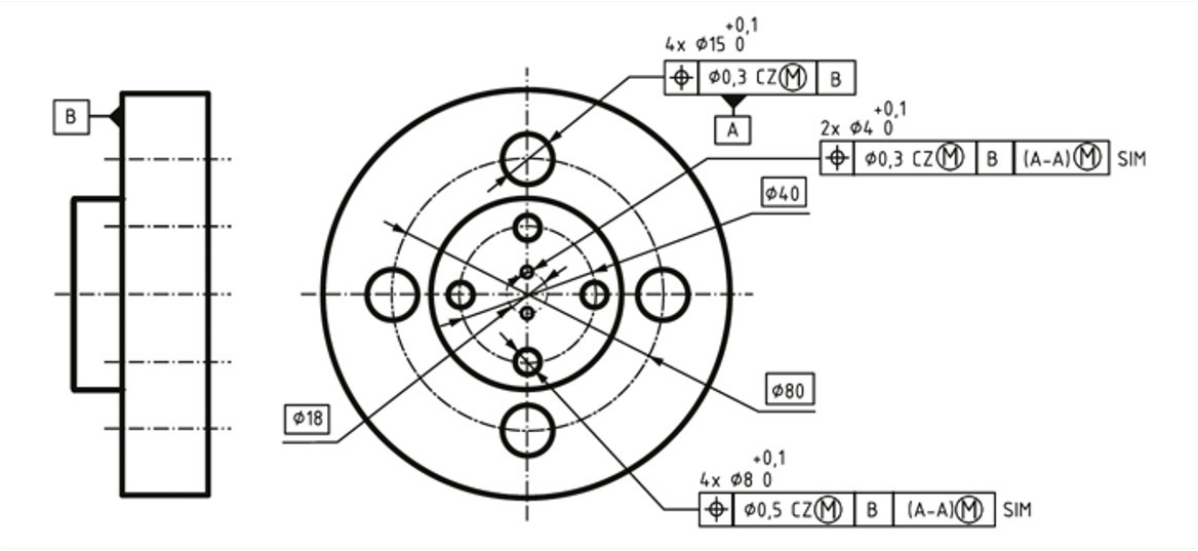


Figure 7.56 Three patterns defined by the modifier CZ and the simultaneous requirement for the two middle,patterns

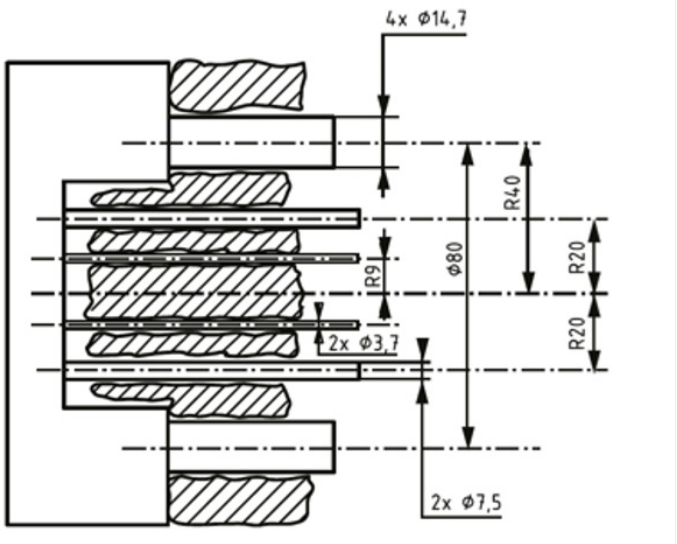


Figure 7.57 Interpretation for the three patterns

8 TOLERANCES OF RUN-OUT

Run-out tolerances are used to control the functional relation of a surface or surface element relative to a datum axis. Run-out is the high to low point deviation of the surface elements realative to the datum axis.

A few applications for circular and total run-out are clearance between rotating parts, gear mesh, balance rotating parts and reduce vibrations.

8.1 CIRCULAR RUNOUT

Circular run-out limits the high to low point deviation of circular elements of any surface of revolution. The tolerance zone applies independently at each circular element of the toleranced surface.

Circular run-out controls roundness and axis offset

8.1.1 RADIAL CIRCULAR RUN-OUT

SYMBOL	TYPE OF TOLERANCE	DATUM REFERENCE	ALLOWABLE TOLERANCE MODIFIERS	ALLOWABLE DATUM MODIFIERS
↗	Location	Required	Ⓕ	Ⓕ
Radial circular run-out tolerance zone Radial space between two coaxial circles				

TABLE 8.1 Radial circular run-out

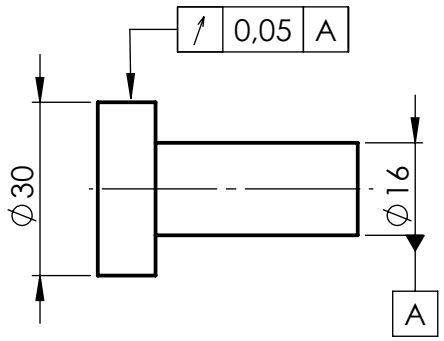


FIGURE 8.1 Radial circular run-out

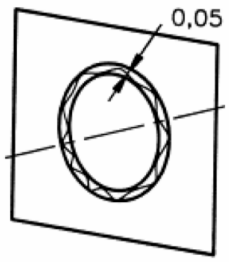


FIGURE 8.2 Interpretation

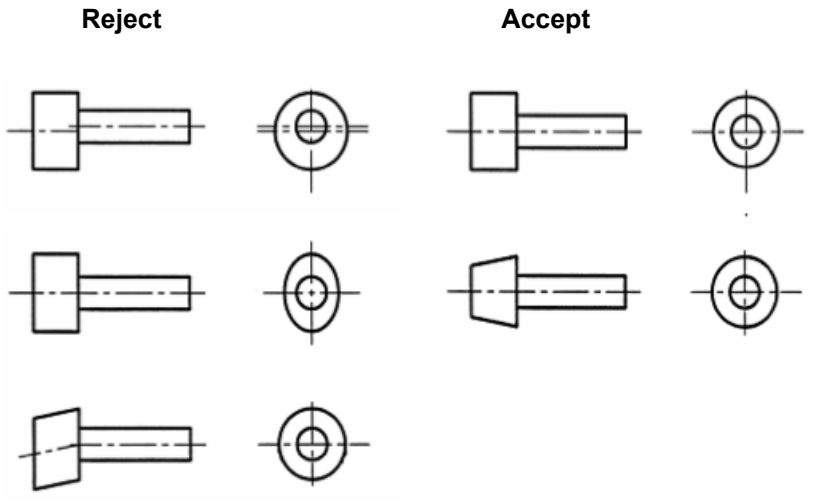


Figure 8.3 Possible deviations

8.1.2 AXIAL CIRCULAR RUN-OUT

SYMBOL	TYPE OF TOLERANCE	DATUM REFERENCE	ALLOWABLE TOLERANCE MODIFIERS	ALLOWABLE DATUM MODIFIERS
↗	Location	Required	Ⓕ	Ⓕ
Axial circular runout tolerance zone Axial space between two coaxial circles (offset axially)				

TABLE 8.2 Axial circular run-out

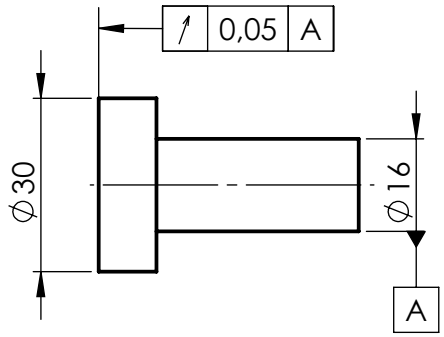


FIGURE 8.4 Axial circular run-out

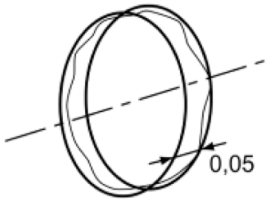


FIGURE 8.5 Interpretation

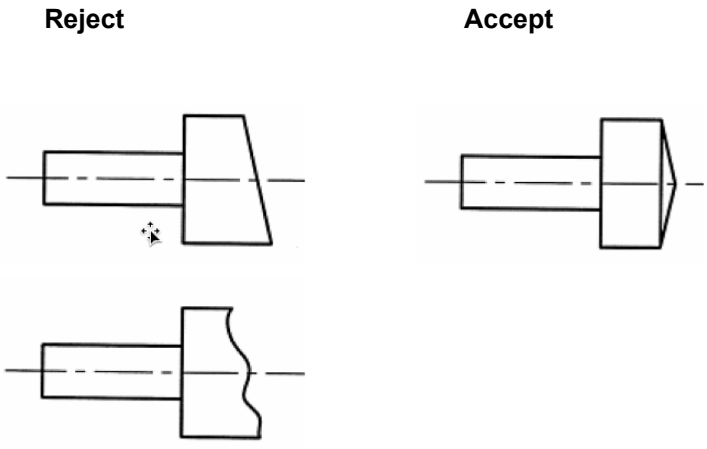


FIGURE 8.6 Possible deviations

8.2 TOTAL RUNOUT

Total run-out limits the high to low point deviation of all surface elements of any surface of revolution.

Total run-out controls straightness, roundness, cylindricity, taper and axis offset.

8.2.1 RADIAL TOTAL RUNOUT

SYMBOL	TYPE OF TOLERANCE	DATUM REFERENCE	ALLOWABLE TOLERANCE MODIFIERS	ALLOWABLE DATUM MODIFIERS
↗	Location	Required	Ⓕ	Ⓕ
Radial total run-out tolerance zone Radial space between two coaxial cylinders				

TABLE 8.3 Radial total runout

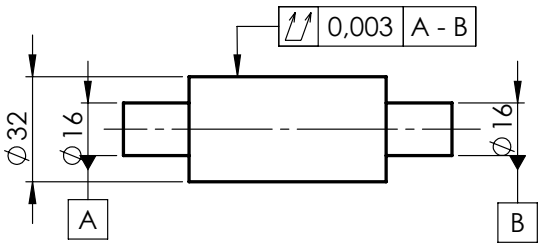


FIGURE 8.7 Radial total run-out

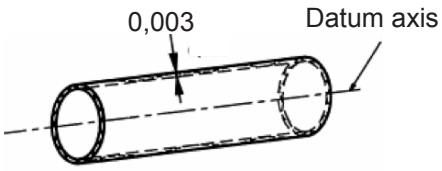


FIGURE 8.8 Interpretation

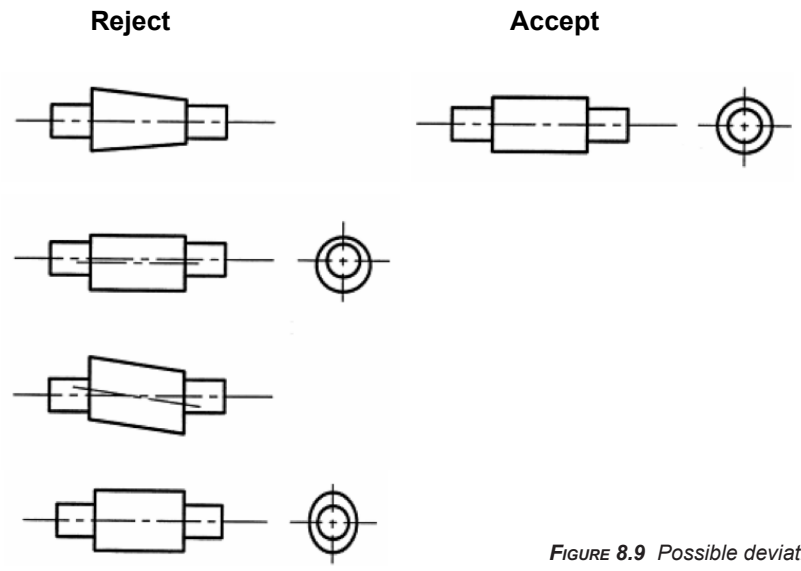


FIGURE 8.9 Possible deviations

8.2.2 AXIAL TOTAL RUN-OUT

SYMBOL	TYPE OF TOLERANCE	DATUM REFERENCE	ALLOWABLE TOLERANCE MODIFIERS	ALLOWABLE DATUM MODIFIERS
↗↗	Location	Required	Ⓕ	Ⓕ
Axial total run-out tolerance zone Axial space between two planes (offset axially)				

TABLE 8.4 Axial total runout

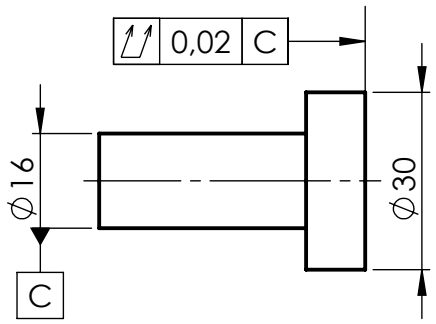


FIGURE 8.10 Axial total run-out

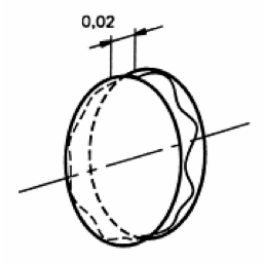


FIGURE 8.11 Interpretation

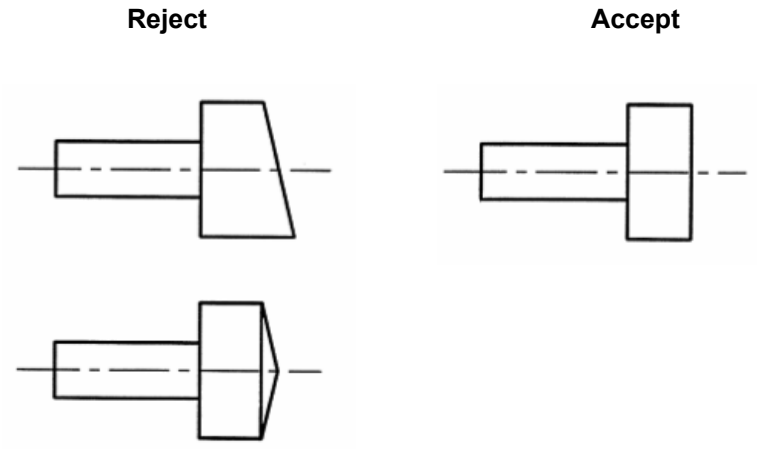


FIGURE 8.12 Possible deviations

Examples of run-out

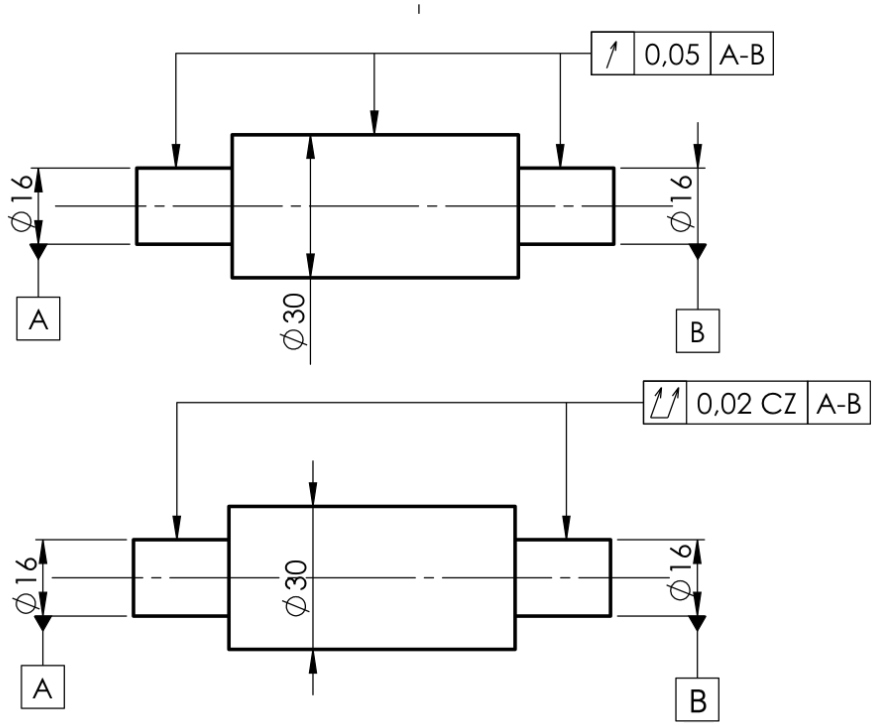


Figure 8.13 Examples of run-out

Exercise

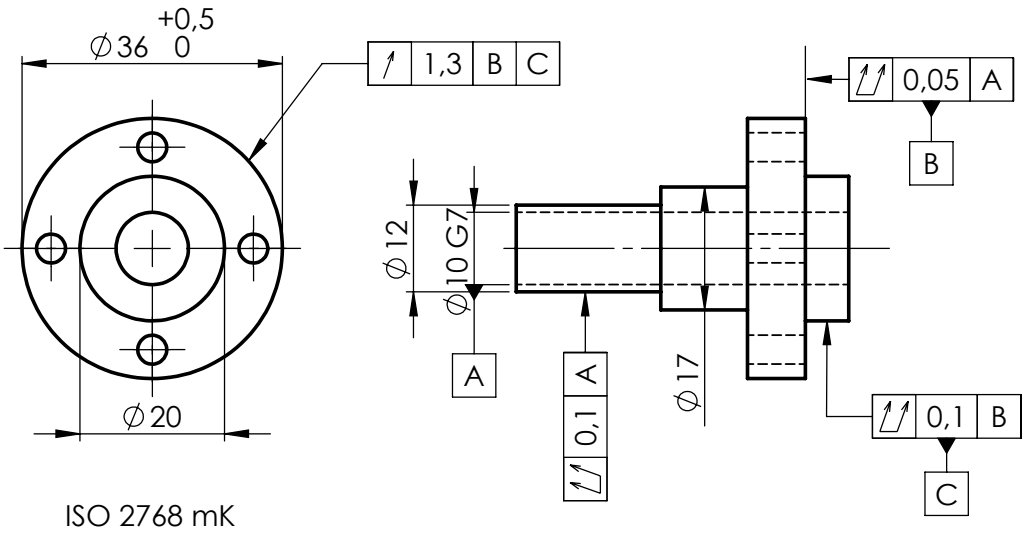


Figure 8.14 Exercise

- Describe the shapes and sizes of each of the tolerance zones?
- What is the roundness of diameter ø 20 limited to?
- What is the maximal boundary of diameter ø 36?
- What DOF's are taken by the datums B and C for the radial circular run-out tolerance on diameter ø 36?

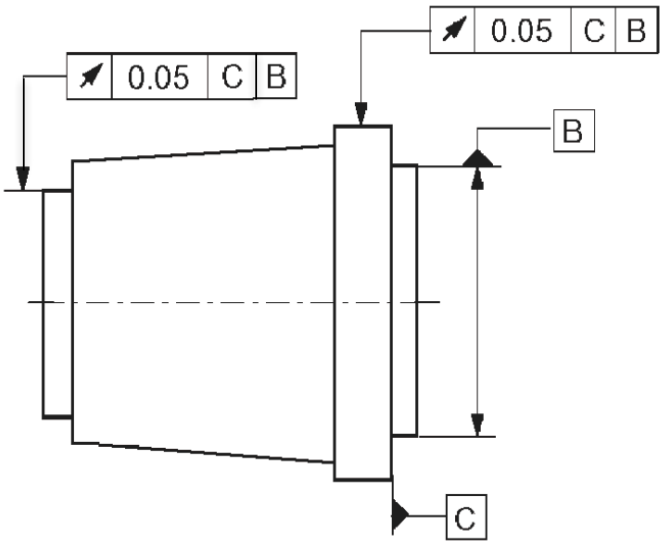


Figure 8.15 Exercise

8.3 ASSESSMENT OF RUNOUT

When verifying the radial run-out the datum axis has to be established.

For circular run-out the high (A_{max}) to low points (A_{min}) of each individual circular element need to be verified. Verifying total run-out requires the measurement of the high to low points of all surface elements.

Run-out deviation: $\delta_l = A_{max} - A_{min}$

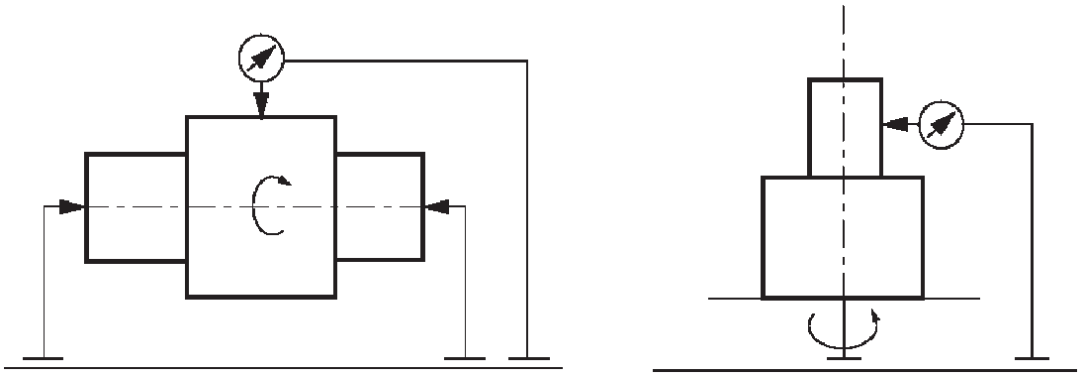


FIGURE 8.16 Assessment radial run-out with revolving workpiece

During measuring of axial run-out deviations the workpiece and indicator must be fixed in axial direction.

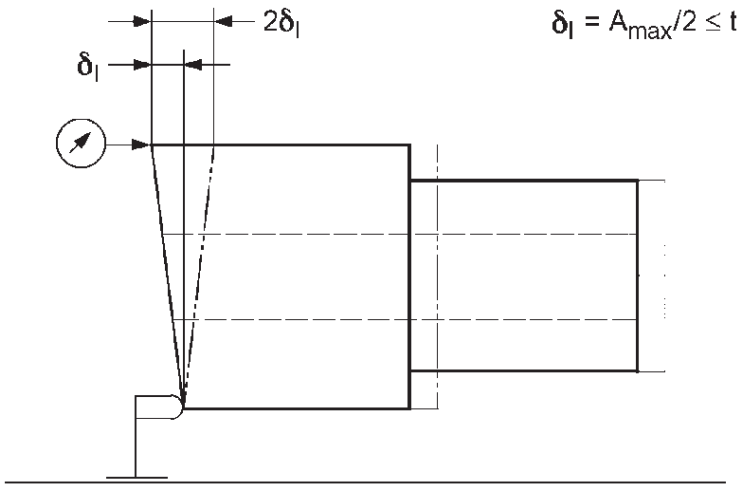


FIGURE 8.17 Assessment axial run-out

9 TOLERANCES OF PROFILE

A profile is an outline of a surface, a shape, made up of one or more features. With profile tolerancing a distinction needs to be made between tolerancing lines or surfaces. The nominal profile is to be defined by TED's or mathematical data. The tolerance zone is default equally disposed on either site of the nominal profile.

A profile tolerance can be used to limit deviations of line elements or a surface. A profile tolerance, depending on how it's applied, can affect four types of geometric characteristics: Size, form, orientation and location. When datums are used profile tolerances often control orientation and location. A profile tolerance without a datum maybe a size and/or form control.

A profile tolerance can limit:

- Form: Tolerance used without a datum
- Orientation: Datum added, indirect limitation of form
- Location: Datum and TED added, indirect limitation of orientation and form

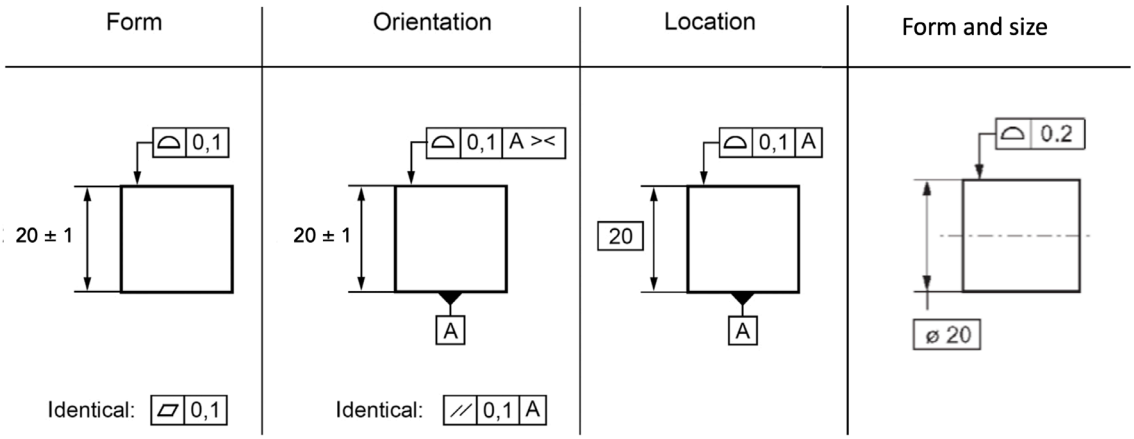


FIGURE 9.1 Use of profile requirements

9.1 LINE PROFILE

A line profile tolerance is a geometric tolerance that establishes a two dimensional tolerance zone that is normal to the nominal profile at each line element. The shape of the tolerance zone is the same as the nominal profile of the feature.

SYMBOL	TYPE OF TOLERANCE	DATUM REFERENCE	ALLOWABLE TOLERANCE MODIFIERS	ALLOWABLE DATUM MODIFIERS
⌒	Location	Required *)	Ⓕ U Z	Ⓜ Ⓛ Ⓕ
Tolerance zone line profile The profile line shall be contained between two equidistant lines enveloping circles the centers of which are situated on the nominal profile *) profile any line without a datum is a form tolerance				

TABLE 9.2 Line profile

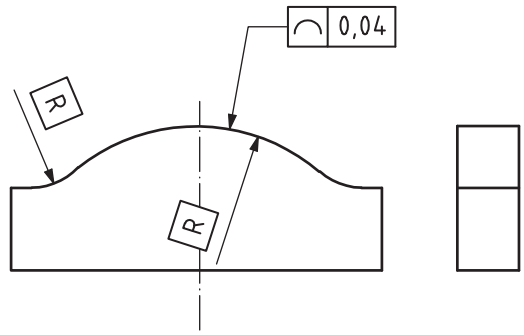


FIGURE 9.1 Line profile as form requirement ISO 1660:1987

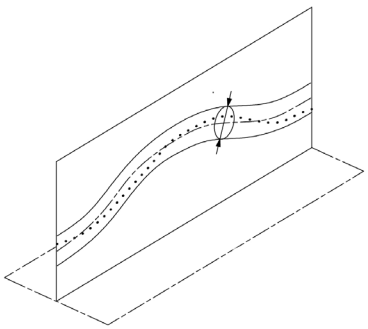


FIGURE 9.3 Interpretation

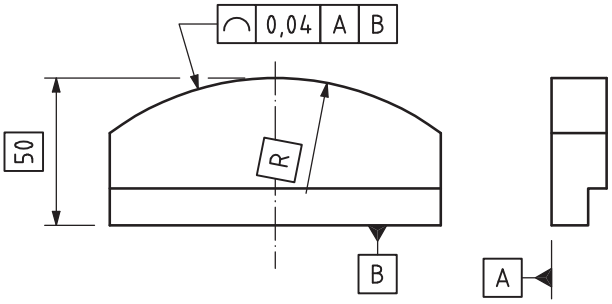


FIGURE 9.4 Line profile as location requirement ISO 1660 1987

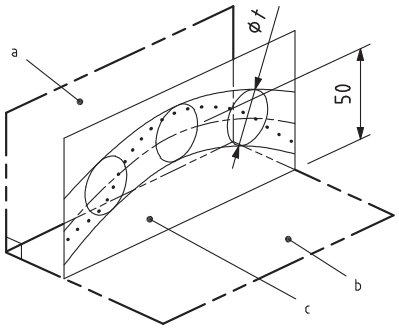


FIGURE 9.5 Interpretation

Line characteristics have an orientation so profile any line too. In older standards the orientation was given by the view holding the requirement. This has been changed with the introduction of the intersection plane.

Also was it not clear if the requirement was on one feature (ISO 8015) or on several feature and when does this combination stop. This was also not defined in previous standards.

With the introduction of ISO 1101: 2013 and ISO 1660: 2017 these ambiguous situations are solved.

United Feature (UF) is used to make several features into one feature. The between requirement defines the limits for the requirement and finally the intersection plane defines the orientation of the requirement.

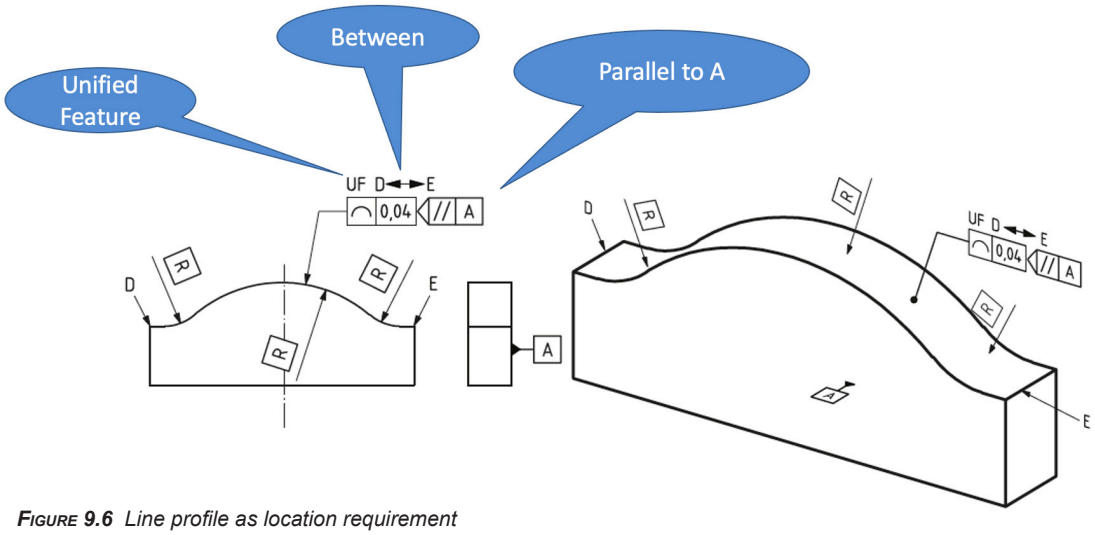


FIGURE 9.6 Line profile as location requirement ISO 1101: 2017

Profile any line is commonly used for sheet metal work.

9.2 SURFACE PROFILE

A profile of a surface tolerance is a geometric tolerance that establishes a three dimensional tolerance zone that is normal to the nominal profile at each surface element. The shape of the tolerance zone is the same as the nominal profile of the feature.

SYMBOL	TYPE OF TOLERANCE	DATUM REFERENCE	ALLOWABLE TOLERANCE MODIFIERS	ALLOWABLE DATUM MODIFIERS
⌓	Location	Required	Ⓕ U Z	Ⓕ
Tolerance zone surface profile The surface profile shall be contained between two equidistant surfaces enveloping spheres the centers of which are situated on the nominal profile				

TABEL 8.2 Surface profile

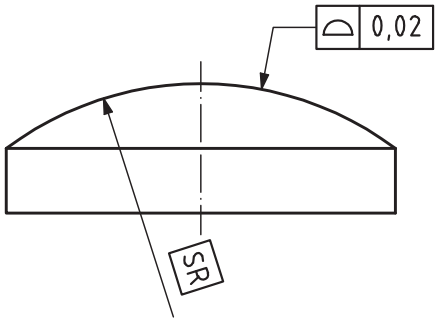


FIGURE 9.7 Surface profile as form requirement ISO 1660: 1987

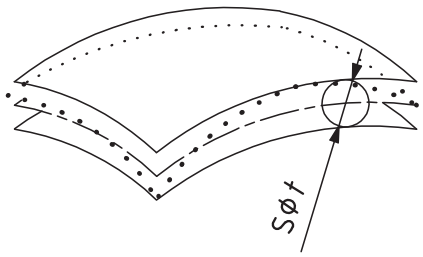


FIGURE 9.8 Interpretation

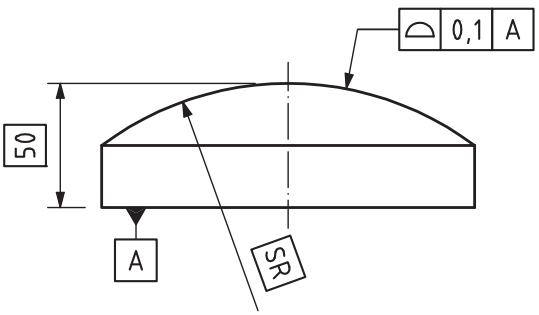


FIGURE 9.9 Surface profile, location requirement ISO 1660:1987

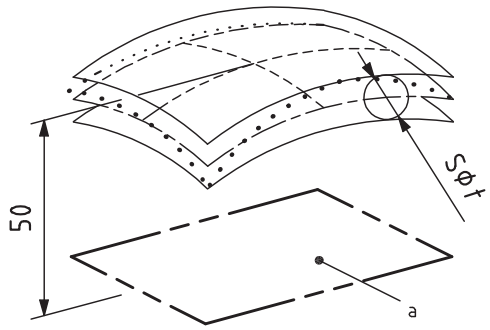


FIGURE 9.10 Interpretation

United Feature (UF) builds an one compound feature out of several single features.

Combined Zone (CZ) combines tolerance zones.

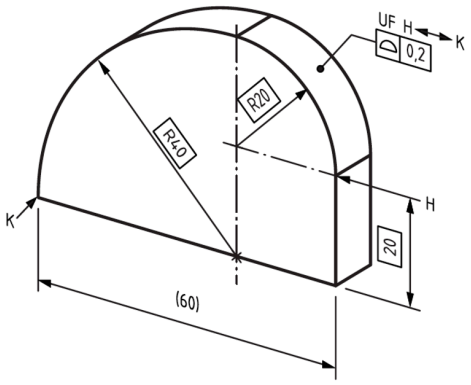


Figure 9.11 Surface profile on a 'United Feature'

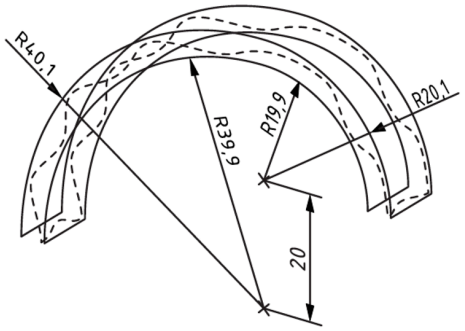


Figure 9.12 Interpretation

With ISO all requirements are independent by default and the symbol SZ (Separate Zone) is optional. Only the position characteristic needs the SZ for independency.

The all around symbol is used stating that the requirement applies to all the four features around. When the requirement is 'all around' a collection plane is used.

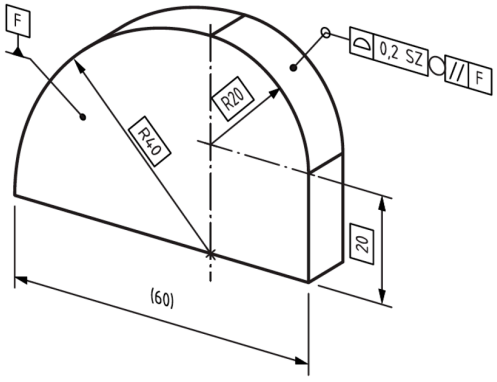


Figure 9.13 Surface profile Separate Zone

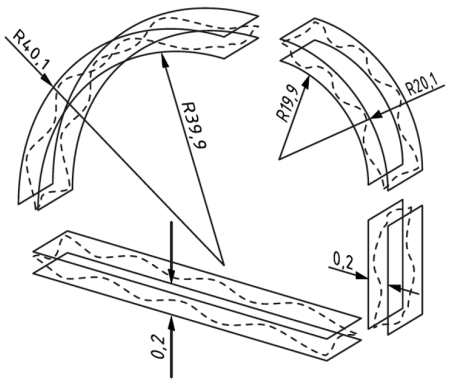


Figure 9.14 Interpretation

When using UF the other corner of the tolerance zone is a radius.
When using CZ all the edges are sharp

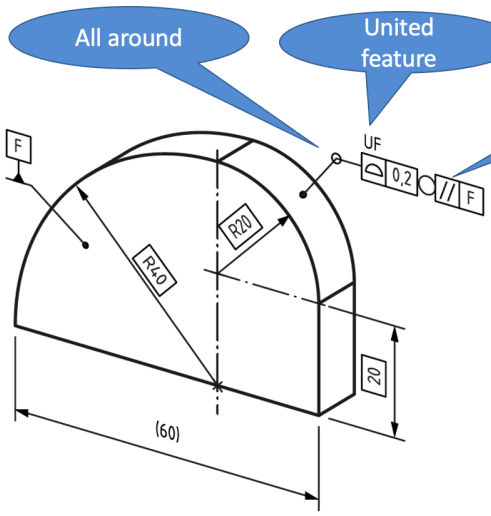


Figure 9.15 Surface profile, all around

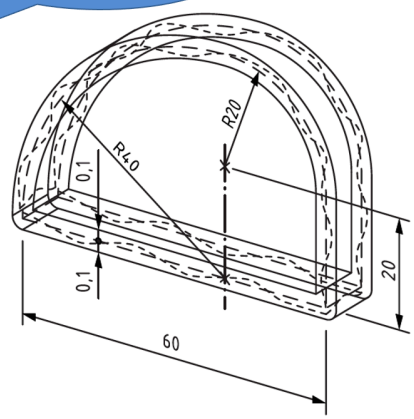


Figure 9.16 Interpretation

The lower example shows the use of the >< symbol. Datum D constrains only the rotations and not the translation.

Datums E and F are hidden features and indicated using dashed leader lines

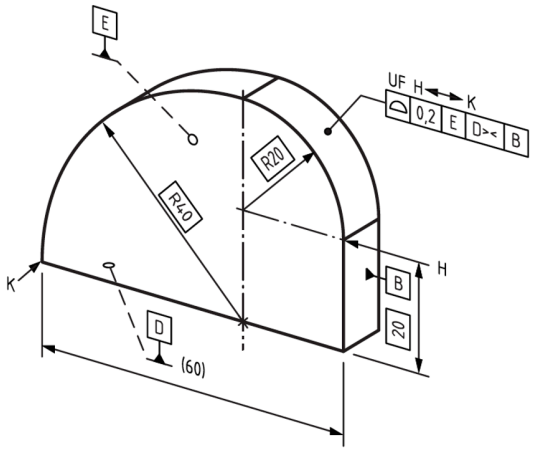


Figure 9.17 Surface profile

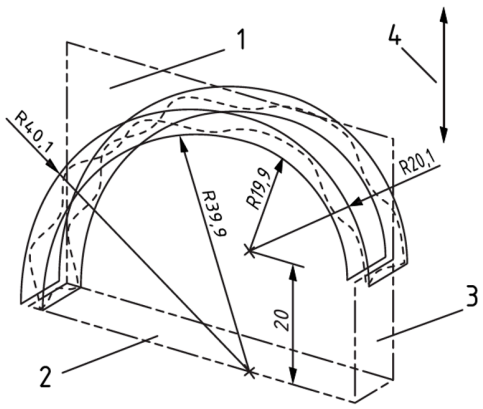


Figure 9.18 Interpretation

9.5 SURFACE PROFILE APLIED TO A CONE

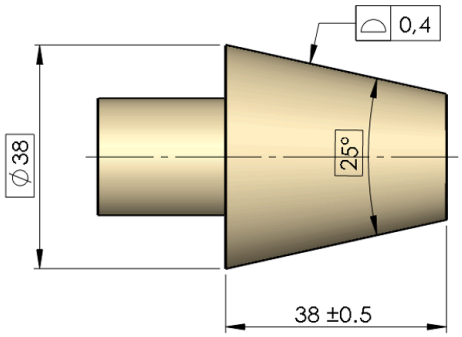


FIGURE 9.19 Surface profile as form requirement

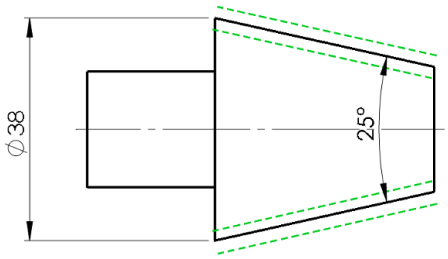


FIGURE 9.20 Interpretation

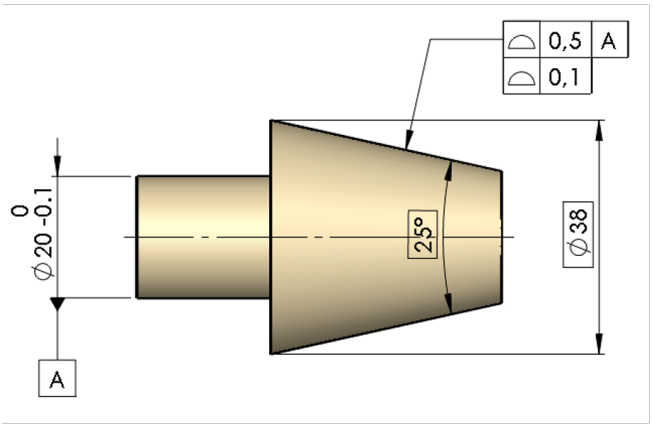


FIGURE 9.21 Radial position of the cone of 0,5mm and a form requirement of 0,1mm using surface profile

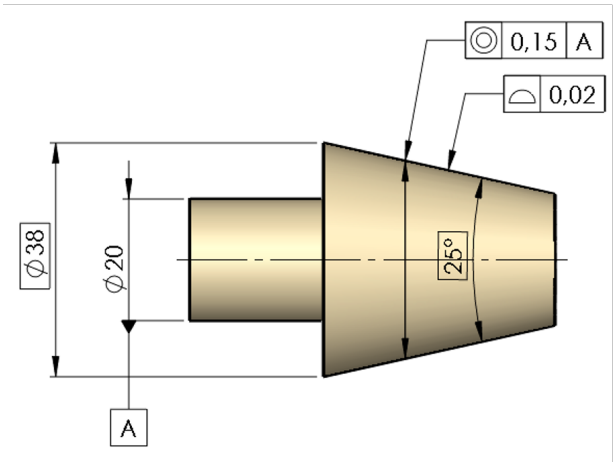


FIGURE 9.22 Radial position of the cone using coaxiality

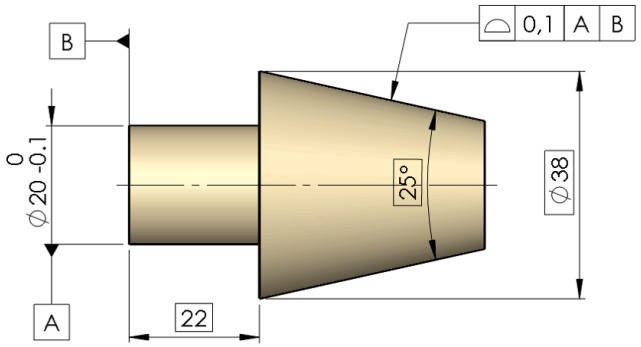


FIGURE 9.23 Radial and axial position of the cone

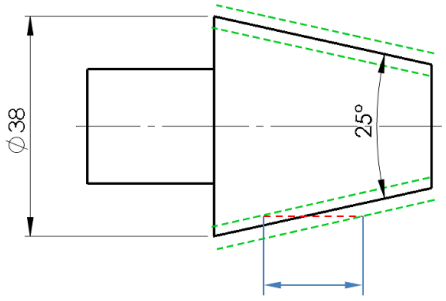


FIGURE 9.24 Interpretation

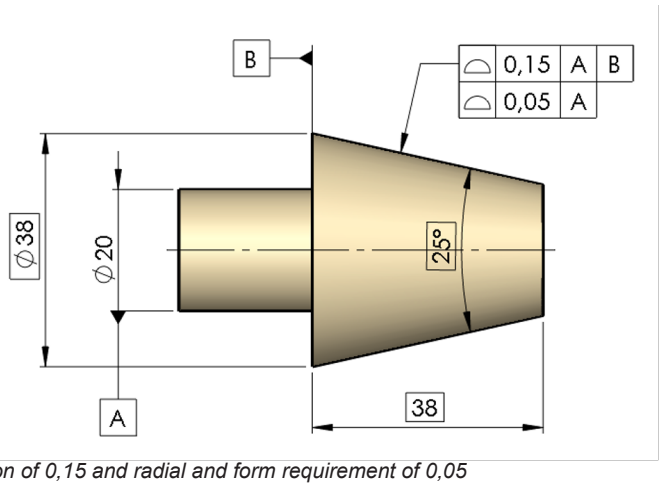


FIGURE 9.25 Axial position of 0,15 and radial and form requirement of 0,05

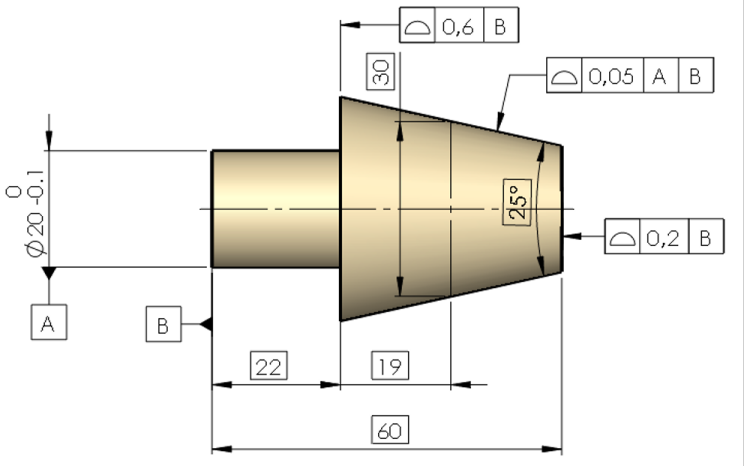


FIGURE 9.26 Using surface profile as position tolerances for the cone and the two flat surface

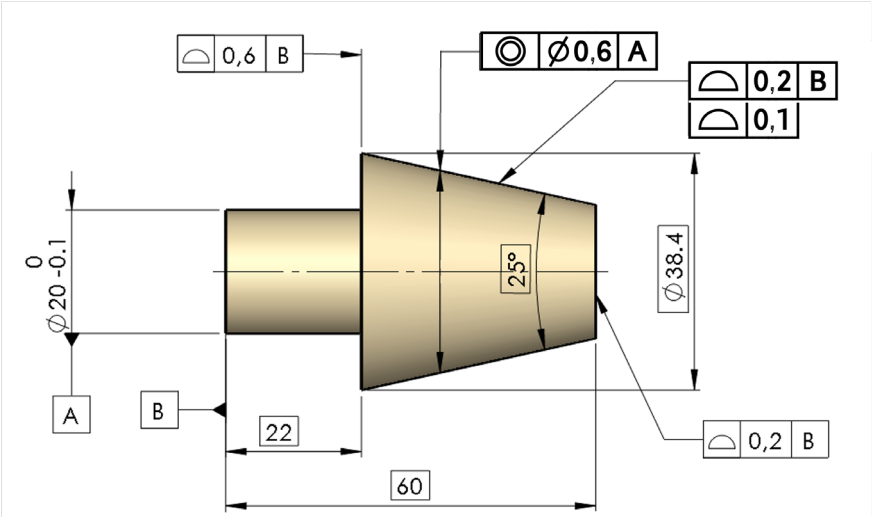


FIGURE 9.27 Radial position using coaxiality, axial requirement 0,2 and a form requirement of 0,1 Flat surfaces are positioned using surface –profile.

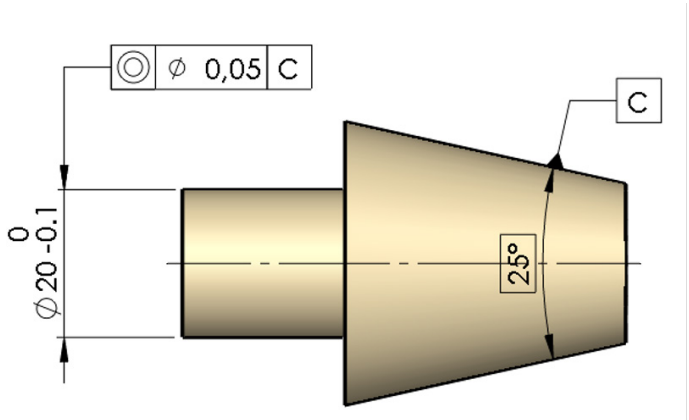


FIGURE 9.28 The axis of the cone a used as datum feature

9.6 SPECIFYING A WEDGE

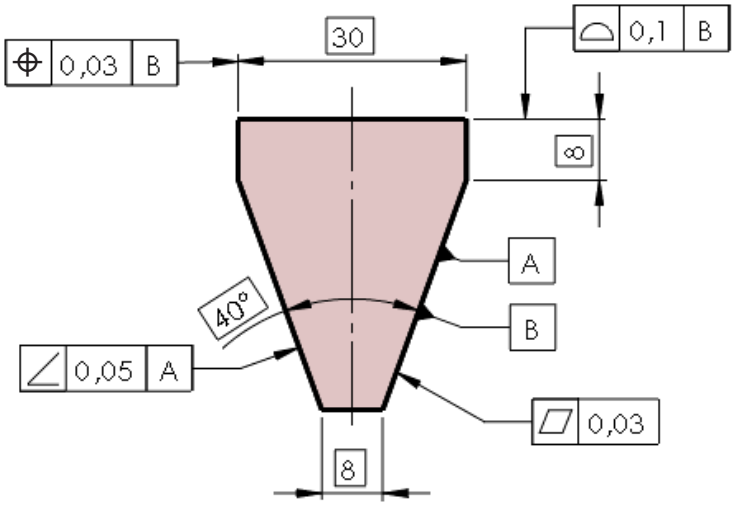


FIGURE 9.29 Specifying a wedge using angularity

10 3D PMI MBD

10.1 INTRODUCTION TO PMI

Concerning GD&T there is no difference between specifying the workpiece requirements on a 2D drawing or 3D MBD when working according the latest ISO standards as stated in tier 1.

ISO 16792:2021 is the standard on MBD. It's merely stating 'how' to put the GD&T on the model and has requirement on the CAD system. This chapter wil give a few examples.

The out of the design is merely a model with the GD&T requirements as a native file or STEP 242 file completed with several views.

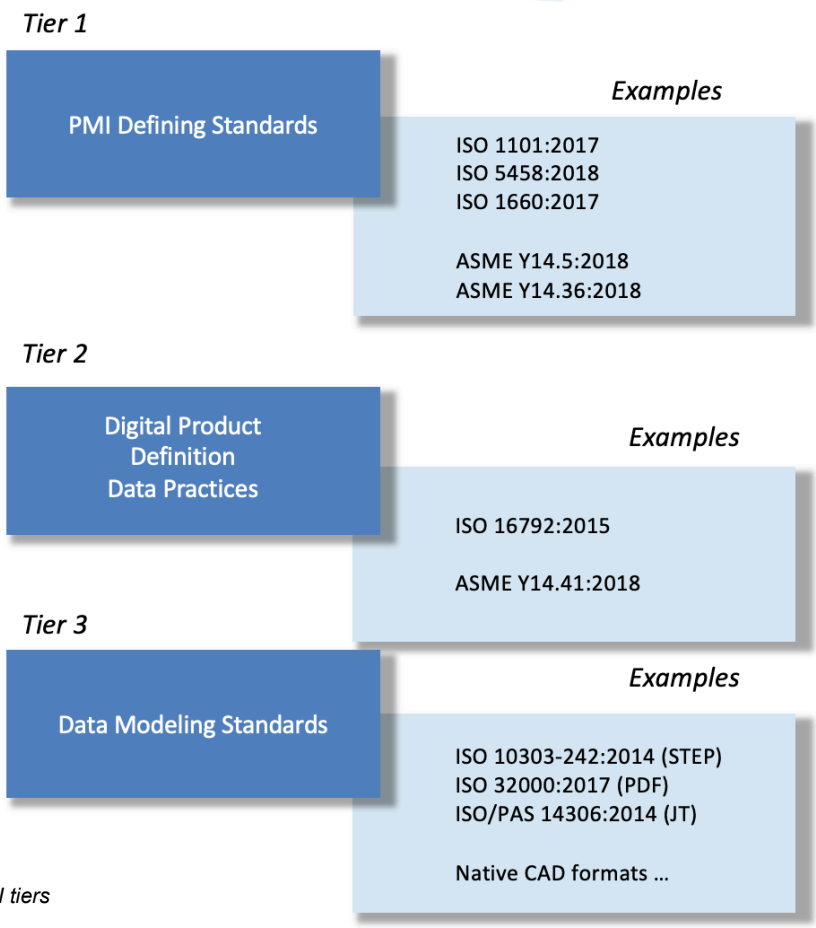


TABLE 10.1 PMI tiers

10.2 WAY OF ANNOTATING

SYMANTIC:
GD&T annotation on the model according ISO >> Software readable (CAM / CAI).

GRAPHICAL
Manual added info >> Human interface required.

Note:
All examples in this chapter are taken from ISO 16792.

10.3 MODEL COORDINATE SYSTEM

Design shall have one or more coordinate systems depicted by three mutual perpendicular line segments with the origin at the intersection.

Model coordinate system shall be right handed unless otherwise specified.

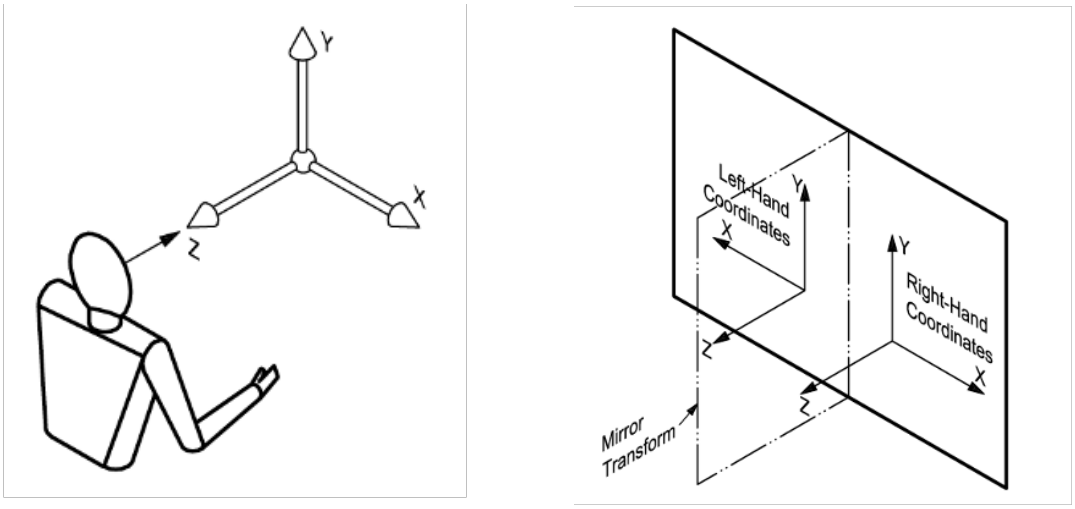


FIGURE 10.1 Model coordinate system

10.4 MODEL REQUIREMENTS

ISO 16792 states display requirements, requirements on associativity between the GD&T annotation frames and the toleranced features, placement of the GD&T annotation symbols as well as placement of welding symbols and roughness symbols.

10.4.1 DISPLAY MANAGEMENT

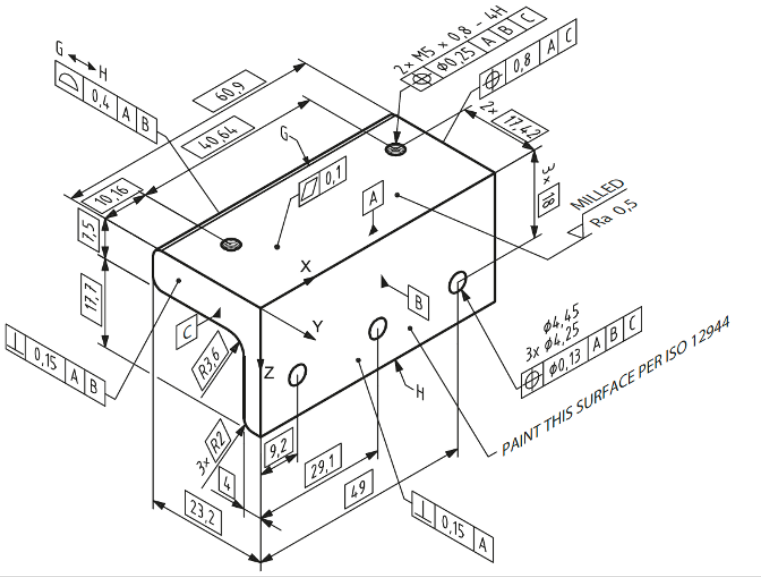


FIGURE 10.2 All annotation displayed

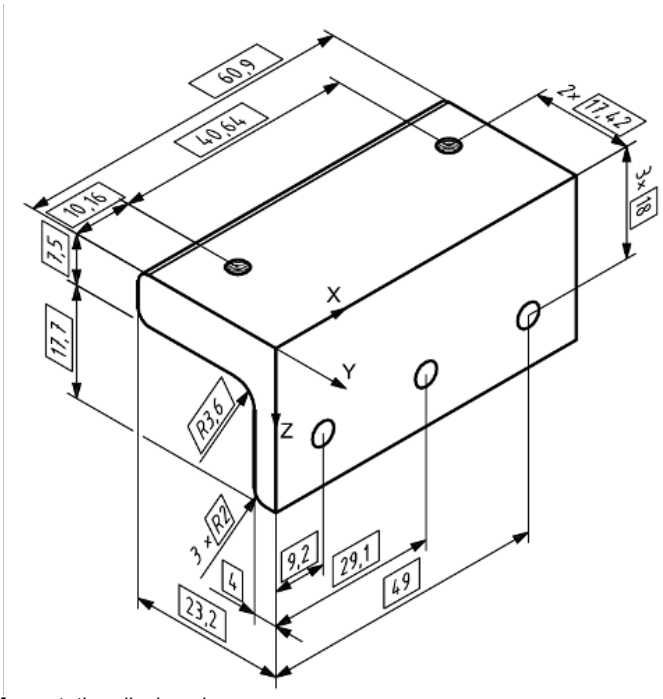


Figure 10.3 One type of annotation displayed

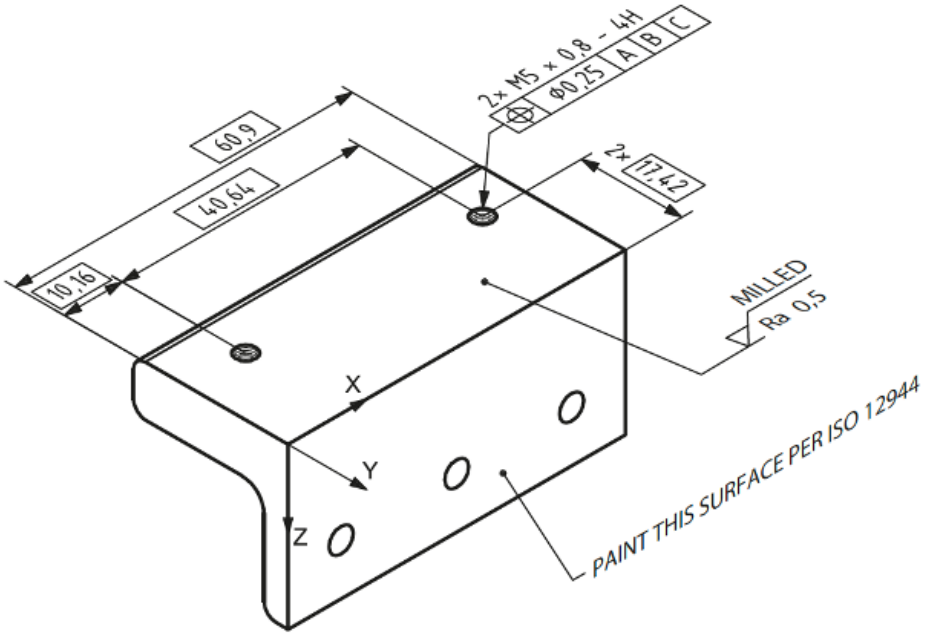


Figure 10.4 Selected annotation displayed

10.4.2 ASSOCIATIVITY QUERIES

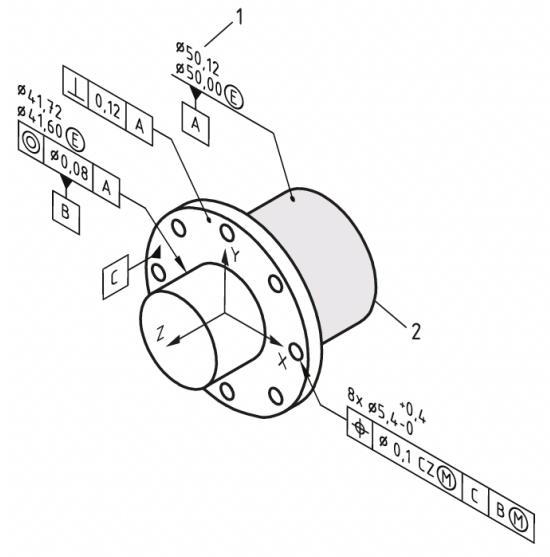


FIGURE 10.5 Size query

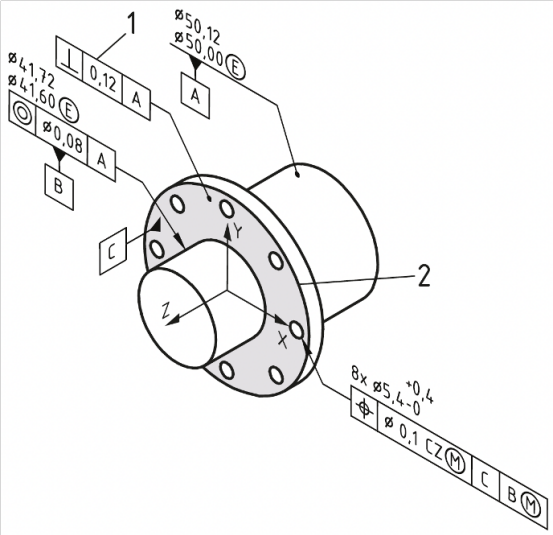


FIGURE 10.6 Geometric tolerance query

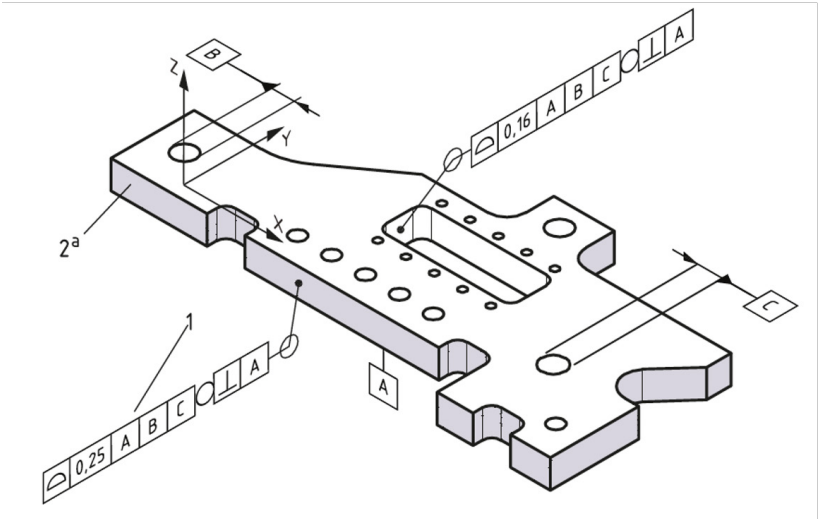


FIGURE 10.7 Surface query

10.4.3 REPRESENTING ATTRIBUTES, QUERIES

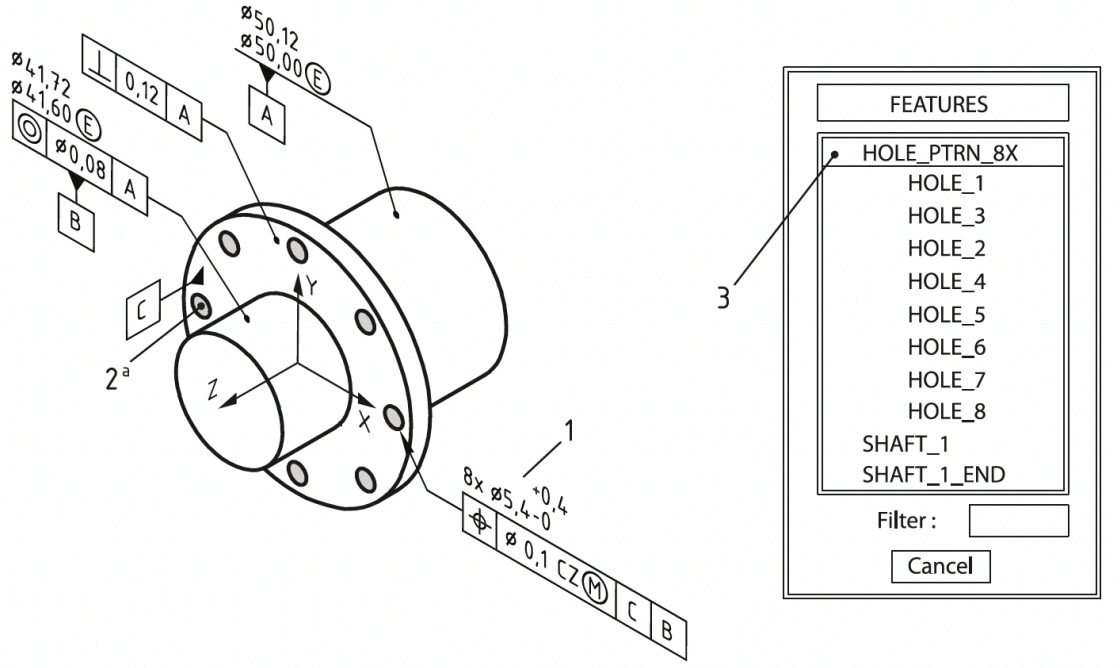


FIGURE 10.8 Representing simplified holes

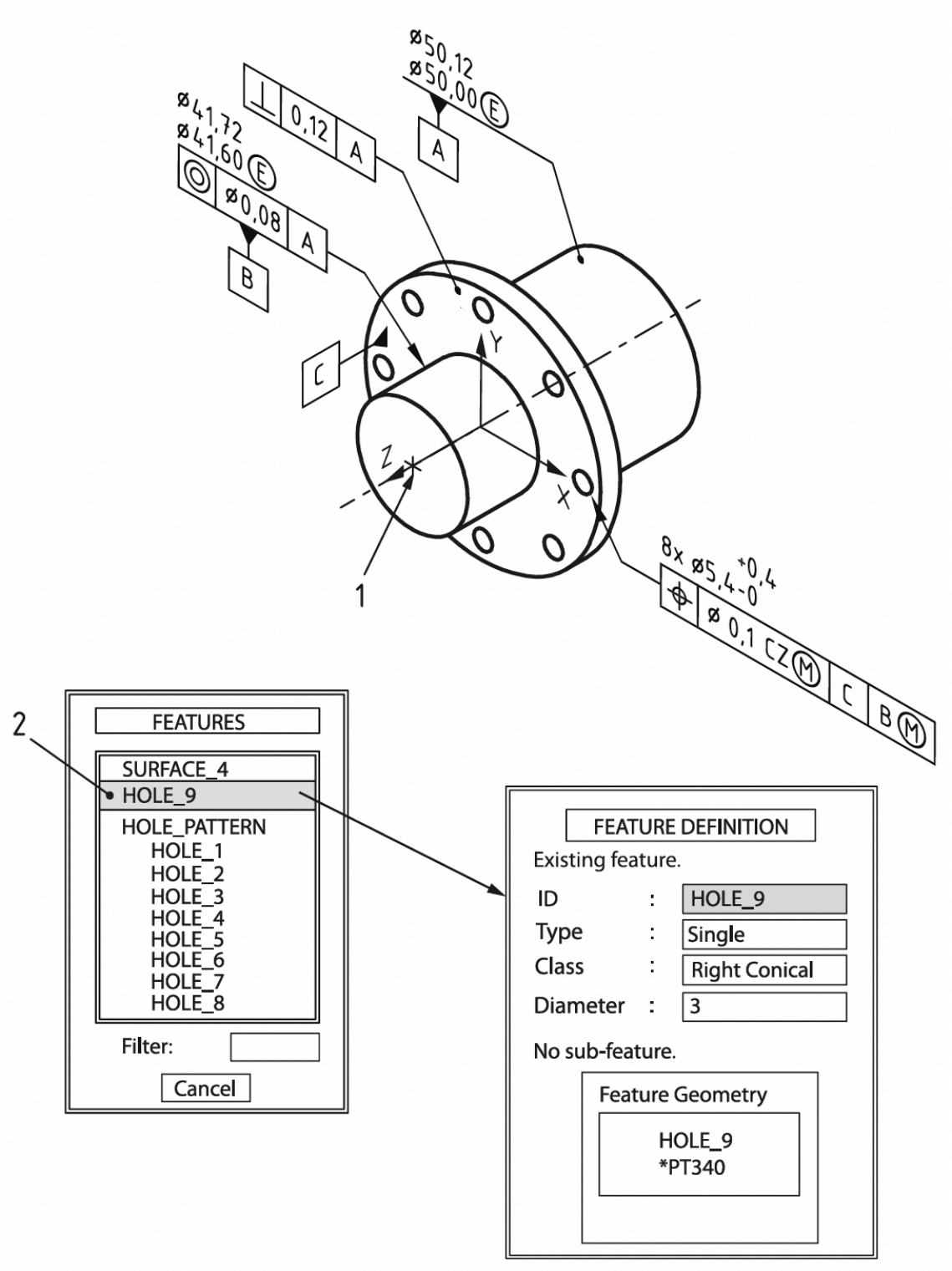


FIGURE 10.9 Pattern query

10.4.4 RESTRICTED AREA

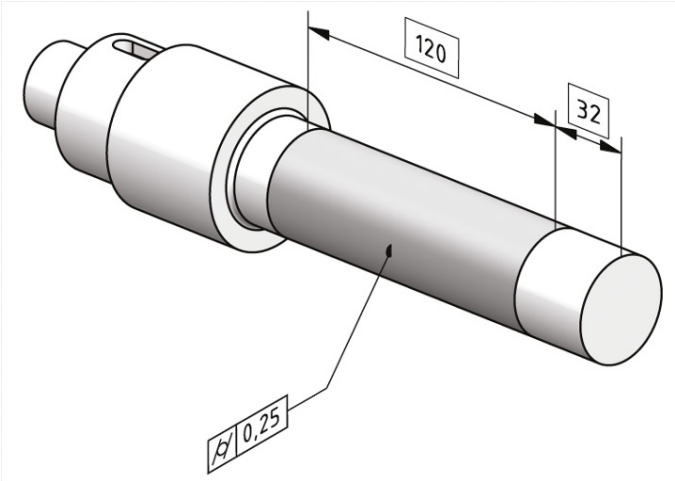


FIGURE 10.10 Restricted area using shading

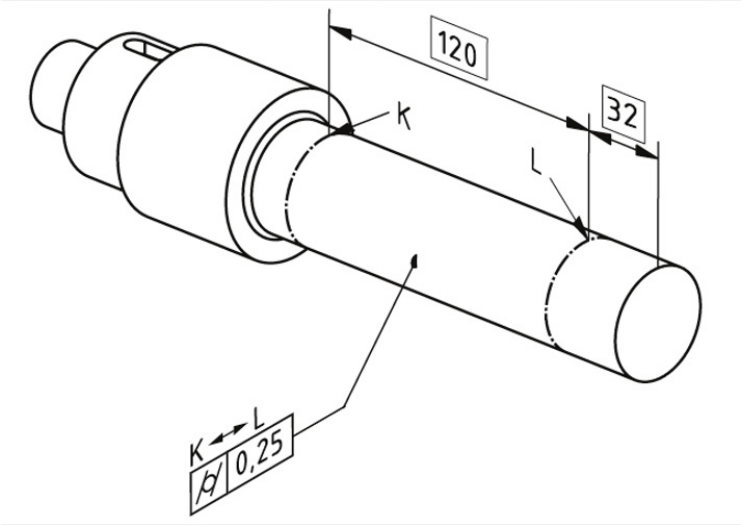


FIGURE 10.11 Restricted area using supplemental geometry

10.4.5 RESTRICTED AREA

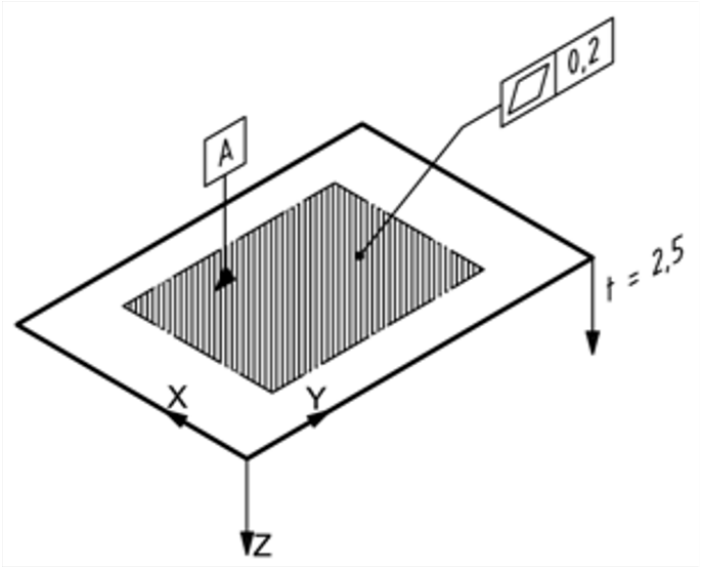


FIGURE 10.12 Partial surface as a datum

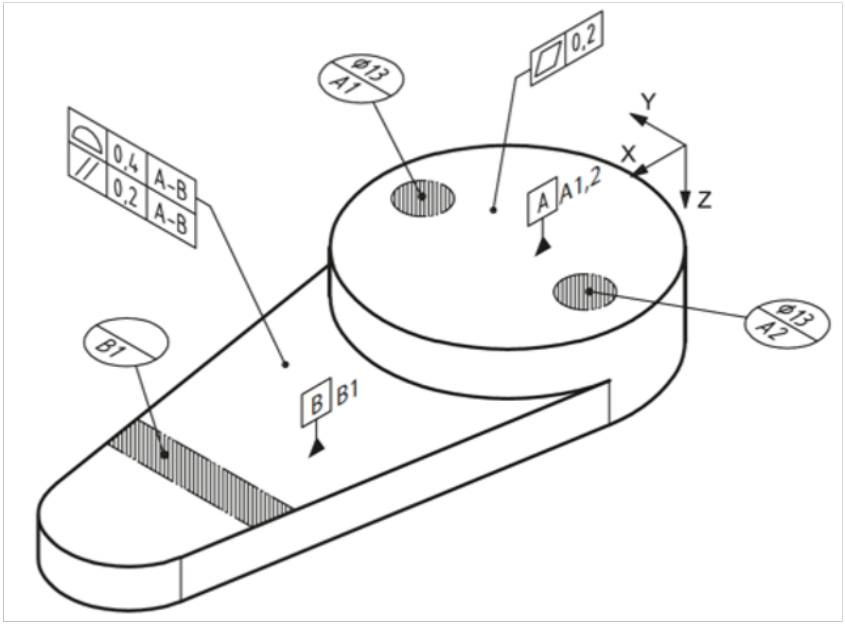


FIGURE 10.13 Datum targets

10.4.6 DIRECTION DEPENDANT TOLERANCES

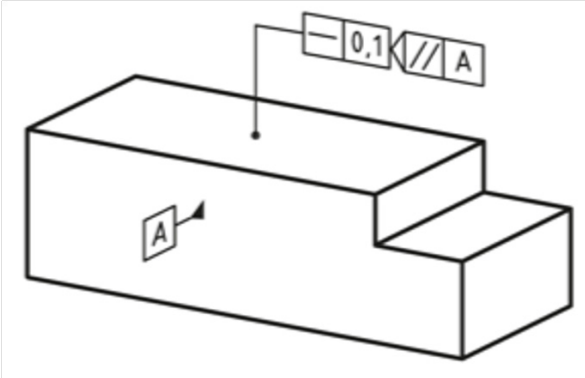


FIGURE 10.14 Straightness parallel to datum A

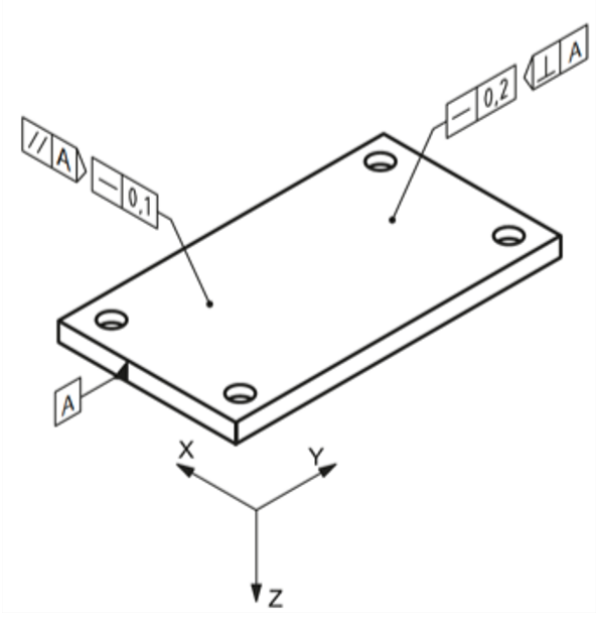


FIGURE 10.15 Direction dependent straightness

10.5 DATUMS

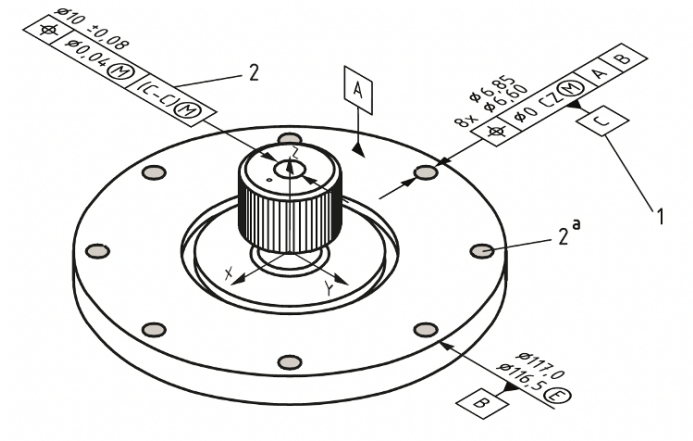


FIGURE 10.16 Indication of datum features

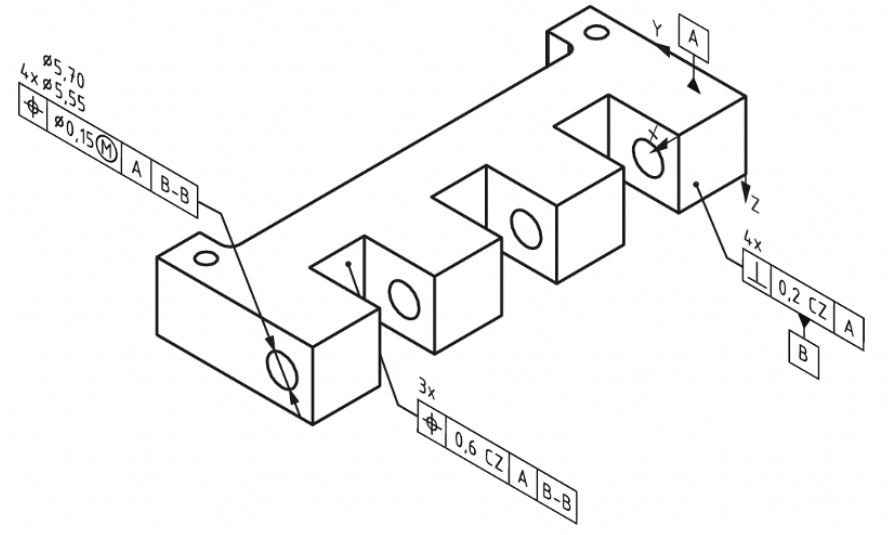


FIGURE 10.17 Indication datum CZ

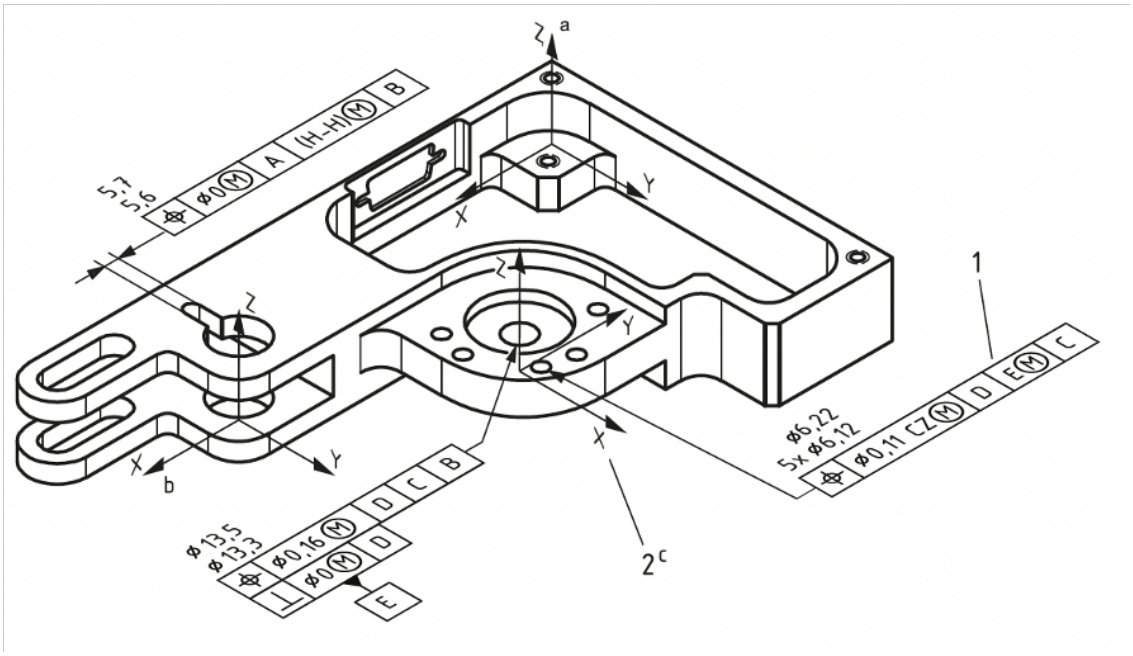
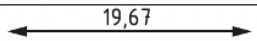
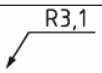
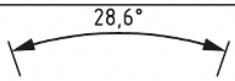
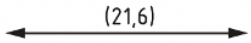


FIGURE 10.18 User defined datums

10.6 DIMENSIONS

Topic	ISO 16792		
	Model value[a]	Resolved model value[a]	Application example
Linear theoretically exact dimension (linear TED) (ISO 1101)	88,410 000 0...	88,4	
Angular theoretically exact dimension (angular TED) (ISO 1101)	28,591 827 3...	28,592	
Size (ISO 14405-1)	7,000 000 0...	7	
Linear distance	19,666 666 6...	19,67	
Radial distance (ISO 14405-2)	3,140 000 0...	3,1	
Angular dimension	28,591 827 3...	28,6	
Auxiliary dimension (ISO 129-1)	21,601 804 3...	21,6	

[a] The values shown are examples. Actual values reflect the defined precision of the model and the rounding requirements of each particular application as determined by the requirements in the data set. Refer to the decimal principle in ISO 8015.

TABLE 10.2 Resolved dimensions

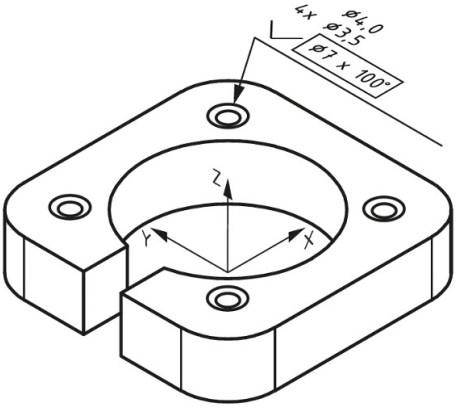


FIGURE 10.19 Coutersink holes

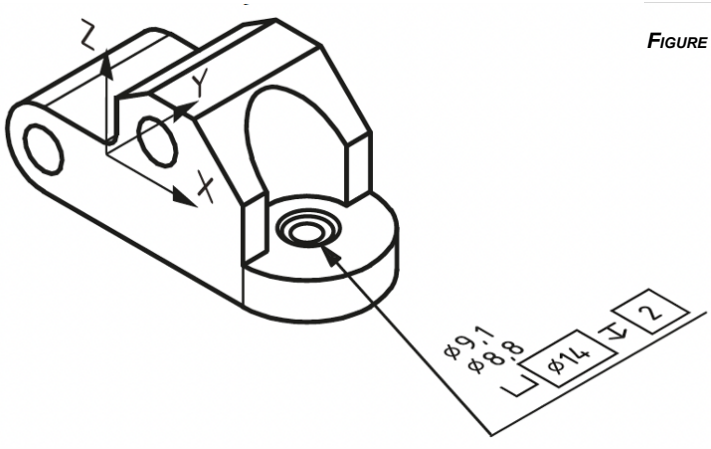


FIGURE 10.20 Counterbore hole

10.7 ROUGHNESS

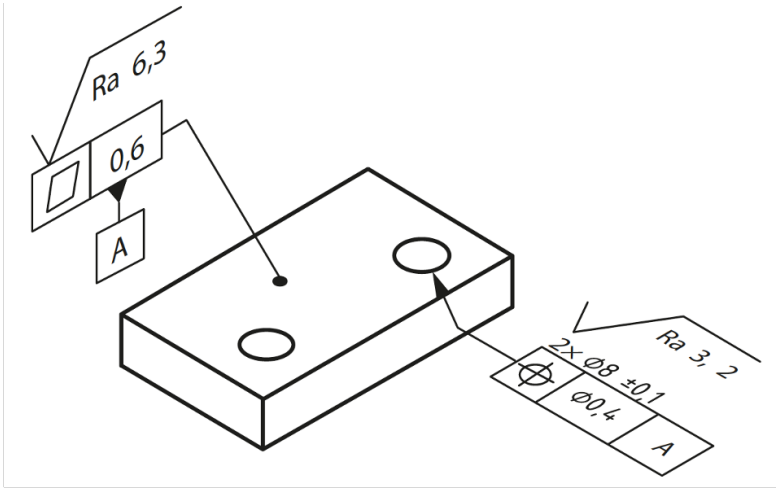


FIGURE 10.21 Surface texture

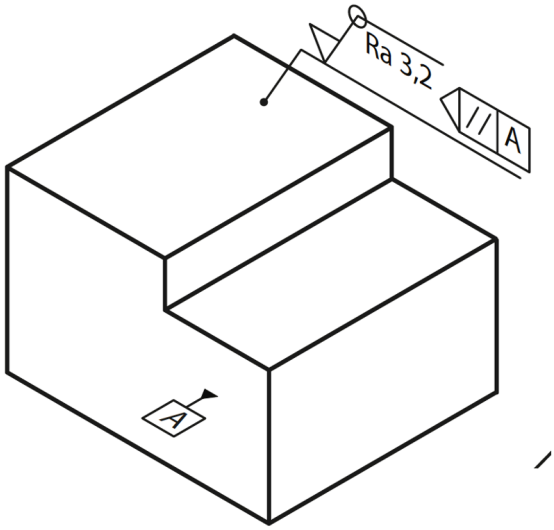


FIGURE 10.2 Orientation surface texture

11 REFERENCES

INTERNATIONAL ORGANIZATION FOR STANDARDIZATION (ISO)

Georg Henzold
Geometrical Dimensioning and Tolerancing for Design, Manufacturing and Inspection, 3nd edition, ISBN 9 780 128 240 618

Walter Jorden, Wolfgang Schütte
Form- und Lagetoleranzen: Handbuch für Studium und Praxis 10 Auflage, ISBN 9 783 446 458 475

Bernd Klein,
Toleranz Design im Maschinen- und Fahrzeugbau 4 Auflage, ISBN 9 783 110 555 103

Alex Krulikowski
ISO *Geometrical Tolerancing (ISO 2004)*

Alex Krulikowski
ISO GPS *The Ultimated Pocket Guide (ISO 2012)*

AMERICAN SOCIETY OF MECHANICAL ENGINEERS (ASME)

Don Day
The GD&T Hierarchy Pocket Guide Y14.5-2009

Paul J. Drake, Jr.
Dimensioning and Tolerancing Handbook, ISBN 9 780 070 181 311

Alex Krulikowski
Fundamentals of Geometric Dimensioning and Tolerancing 3rd Edition ISBN 9 781 111 129 828

Alex Krulikowski
The Ultimated GD&T Pocket Guide

MOST RELEVANT ISO STANDARDS

ISO 286	ISO code system for tolerances on linear sizes - Part 1: Basis of tolerances, deviations and fits
ISO 1101	Tolerances of form,orientation, location and run-out
ISO 1660	Profile tolerancing
ISO 2692	Maximum material requirement, least material requirement, reciprocity requirement
ISO 5458	Pattern and combined geometrical specification
ISO 5459	Datum and datum systems
ISO 8015	Fundamentals, Concepts principles and rules
ISO 10579	Non-rigid parts
ISO 14405-1	Dimensional tolerancing — Part 1: Linear sizes
ISO 14405-2	Dimensions other than linear or angular sizes
ISO 14405-3	Angular sizes
ISO 14638	Matrix Model
ISO 16792	Digital product definition data practices