Verification of components by 3D-CAD data

Language code to ISO 639-1: en

ICS 01.100.01, 35.240.10

Descriptors : CAD; casting; measurement technique; three-dimensional

Contents

Earlier editions:

	(Name)	Datum	Signature
Created by	Meier-Burkamp - VTA - arq	2015-01-20	sgd. (Meier)
Checked by	Kusnik - VTA - abpe	2015-01-27	sgd. (Kusnik)
Approved by	Knödler - VPH – p1qg	2015-02-03	sgd. (Knödler)

1 Area of Application

This VN applies, in its entirety, to components used for the production of drive components for Voith Turbo GmbH & Co. KG, unless stated otherwise in drawings, standards for the individual parts or data sheets or agreed otherwise in the purchase order.
This VN is also applies to prototype parts from pre-series, near-series or series tools.

2 Scope of application

This standard applies for components which are described via a 3D-CAD solid model, and for which the production thereof is carried out:

-directly, e.g. via CNC processes from a solid
-or indirectly, e.g. by CNC-machined tool or pattern equipment

-or via RPT processes.

3 Purpose

The purpose of this standard is to simplify the dimensioning and the specification of standardized general tolerances for the surface form of components, the shape of which is described via CAD models and which is not completely dimensioned on the drawing.
This standard does not apply for raw part contours which cannot be derived from a 3D model. The tolerances apply for all areas of the part, for which tolerances cannot be specified individually.

General tolerances according to this standard apply when reference is made to such standard in drawings or associated documents (e.g. terms of delivery).

If lower tolerance values are required for functional, production or other reasons, or higher tolerance values are allowed and more economical, they must be specified individually.

4 References to standards

This standard contains specifications from other publications through dated or undated references. These references to standards are cited in the respective places in the text, and the publications are listed below. In the case of dated references, subsequent amendments or revisions of these publications are part of this standard only if they have been incorporated by virtue of such amendments or revisions. In the case of undated references, the last edition of the referenced publication applies.

DIN EN ISO 1101	Geometrical Product Specification (GPS) - Shape and position tolerancing - Tolerancing of shape, direction, location and run, General, Definitions, Symbols, Drawing entries
DIN EN ISO 5459	Geometrical Product Specifications (GPS) - Geometrical tolerancing - Datums and datum systems for geometrical tolerances
DIN EN ISO 8062-3	Geometrical Product Specification (GPS) - General dimensional and geometrical tolerances and machining allowances for castings

Unless stated otherwise, the most recent version of these standards shall apply.

5 Definitions and abbreviations

Native data:	3D-CAD Solid model, generated via a CAD system and used as a basis for deriving 2D drawings and other data records.
PMI:	"Part Manufacturing Information"
RPT:	"Rapid Prototyping"

6 Interaction between data record and drawing

A part is described in geometric terms by modeling a 3D-CAD solid model (native data).
The drawing derived from this model contains all the information required for the complete description, which is not contained in the data record.
The relationship between the 3D-CAD solid model and the drawing must be distinct and traceable.
The released drawing is a binding basis for the purchase order, production and quality assurance.
Together with the corresponding 3D-CAD solid model, the drawing represents the end product expected in this form and configuration.

Data records which are generated from the 3D-CAD solid model in other file formats must be capable of being clearly assigned to the origin data.

6.1 Data record

The 3D-CAD solid model contains as complete a description as possible of the reference geometry in the desired end state.

including:
Bevels of mold,
Chamfers,
Machining allowances,
Mold joints,
Lettering, joint identification, etc.
Casting-specific characteristics (plaster ribs, ingates, infills, testing points), insofar as they remain on the casting.

A material assignment is not absolutely essential to the verification per se, but must be carried out for other reasons.
The reference geometry is modeled to nominal dimensions.

Production-related deviations on production models (shrinkage and wear allowances) are not modeled.

The preferred data format is the PRT file or JT from Siemens NX.

6.2 Drawing

The drawing contains all information relevant to production and quality, which is not contained in the 3D-CAD model.

e.g.
Component designation
Material
Raw part weight
Identifiers (material, production date, model no.)
Material testing laboratories
Clamping and contact points
Reference to the valid data record
Datums for verification
Tolerances
Surface data
Data on residual dirt
Inspection features
References to raw part rework (repair welding)
References to leak tightness
References to casting system remnants
References to critical features

6.3 Management, release and designation of the data record

The data record which is authoritative for a particular drawing is generated by the development department concerned from the native data and saved in the material master (SAP). File format and data quality must be agreed between purchaser and supplier. The designation is as follows - "*Drawing number.file format*" (e.g. H97226449.step). The data record is released with the drawing.

6.4 Dimensioning and tolerancing

Because the part is completely described by the 3D-CAD solid model, no dimensions are required on the drawing in the first instance.
However, the accuracy the component is required to have must be clearly described on the drawing.

General tolerances are defined by indicating the permissible surface profile deviation. (See also Attachments 9.2. and 9.3.)

Surfaces and areas with limited or increased tolerances are defined by indicating tolerance zones.

Asymmetric tolerances can be indicated. (see Attachment 9.4.3.3.)

Direct dimensioning on the 2D drawing is also possible. For these dimensions, the desired tolerances must be indicated.
Dimensional tolerances entered on the drawing take precedence over the surface tolerances of the 3D-CAD solid model.

Direct dimensioning on the drawing is for only those attributes which cannot be described or which are difficult to describe via enveloping surface tolerances.
Typically, these are functional dimensions, and also shape and position tolerances, which call for a higher level of accuracy locally and/or which are supposed to bring about a selective inspection of such attributes.

6.5 Special tolerances

Casting-specific features (e.g. wall thicknesses) or customer-specific requirements are defined via special tolerances.

Special tolerances can be indicated for:
Radii
Holes
Wall and rib thicknesses
Tool offset and closing dimension deviations
Flatness
Joining and functional surfaces

6.6 Datum system

The datums required to align the component during the measurement must be shown on the drawing.
The datum planes for all 3 axis directions as defined by DIN ISO 5459 must be indicated. The datums are designated A (primary datum), B (secondary datum) and C (tertiary datum). They are indicated with the usual symbols.

Reference points (points, lines, surfaces) can also be defined if necessary. (see DIN ISO 5459).

The datums must be oriented to functional areas or clamping and locating surfaces for machining. (See also Attachment 9.1)
The "Best Fit" method is applied for the alignment only if no datums are indicated on the drawing.

7 Dimensioning of mechanical machining

Until now, machining has been dimensioned and toleranced by means of conventional methods.

Due to the breakdown into raw and finished parts, verification by 3D-CAD data is applied almost exclusively for raw parts which are not machined.

If 3D data are to be used for mechanical production, surface, form and position deviation tolerances must be defined in the data record via "PMI" (**P**art **M**anufacturing **I**nformation / **P**roductions **I**nformation **D**ata).

Voith Turbo Technical Elements (TEs) are equipped with PMIs and take precedence for creating dimensioned 3D shaped elements.

8 Measuring methods

Surfaces can be assessed by means of the following methods:

Tactile assessment
Optical assessment
Computer tomography (CT) with dimensional evaluation.

8.1 Analysis

The first step in any analysis is to align the component on the measuring unit. This requires the performance data for the datum references to be harmonized with the data record.

Next, the surface form deviation is analyzed by means of the general surface form tolerance.

Thirdly, the special tolerance zones are analyzed.

Directly indicated dimensions and tolerances must be assessed via geometric measurements and logged separately.

Inspection plans and inspection methods must be defined by the Quality Group in collaboration with Development and the supplier

8.2 Surface form tolerances

The surface form tolerance is indicated directly.

In the case of raw castings, the dimensional and/or form deviations are substantially dependent on the casting process and the casting size, characterized by the space diagonals.
Reference values for the surface form deviations achievable on castings are listed in Tables 1-5 in Chapter 9.3.

8.3 Space diagonals

The space diagonal S is determined by the largest expansion of the component. It is calculated from the nominal dimensions of the length l, the width w and the height h of the prismatic enveloping body, which delimits the user-defined component, according to the formula below:

$R=\sqrt{(l^2 + b^2 + h^2)}$

The space diagonal serves as an orientation value for selecting a degree of accuracy. (See tables in Attachment 9.3.)

9 Attachment

9.1 Datum planes and datum points

The drawings or CAD data for castings must be systematically dimensioned with datum planes and datum points, so-called reference points, such that dimension checks and subsequent machining steps correspond. These datum planes and datum points must be specified jointly by the design engineer and founder in advance. The zero position of the datum planes is precisely defined by the dimensions of the datum points.

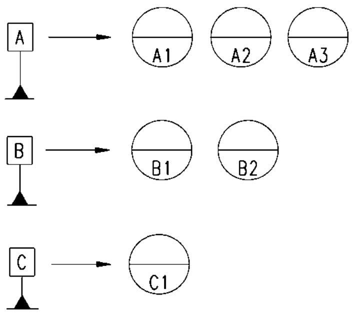


Fig 1: Datum reference – datum points

Six points (reference points) are required to limit the six degrees of freedom of a rigid body, statically determined in space. The primary datum plane "A" is fixed by three datum points A1, A2 and A3. It should correspond to the largest casting surface. The secondary datum plane "B" has the two datum points B1 and B2, which should be assigned to the longitudinal axis wherever possible. The tertiary datum plane "C" has only one datum point C1, and this should lie in the center of the casting or in the vicinity thereof. This procedure is called the **3-2-1 Method**.

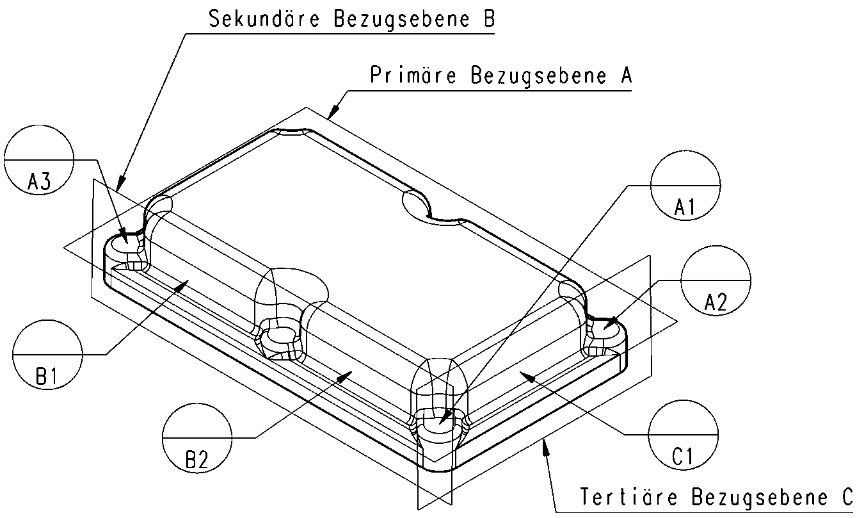


Fig 2: Datum system (schematic example)

The datum planes are positioned by the symmetrical axes of the casting. All datum points must be arranged in such a manner that they cannot be removed or altered during subsequent machining steps. Datum points should lie as far apart as possible on the outer surfaces of the casting. They can also be formed as raised or recessed surfaces. Raised datum points (reference points) are advantageous on castings with restricted shape and position tolerances. When defining the datum points, it is important to ensure that they do not fall into an area around a gate. With complex configurations, the casting can be precisely positioned by (pre-) machining reference points.

9.2 Surface form tolerances

The actual surface of the component must lie within a tolerance zone which is formed by 2 enveloping surfaces. The enveloping surfaces are at a defined distance from the theoretically exact position of the reference contour of the surface model. The enveloping surfaces are symmetrical to the reference contour. The distance of the enveloping surfaces to one another is the general tolerance.

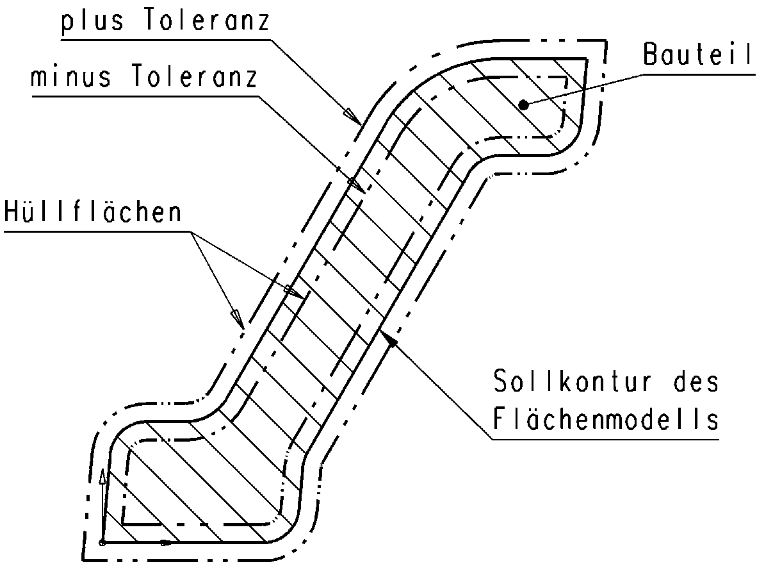


Fig 3: Example for the position of the ± tolerances on the component

The achievable tolerance values depend on the accuracy requirement, part size, manufacturing process and material. Reference values are shown in the tables below.

For wall thicknesses, an additional wall thickness tolerance must be considered if necessary.

9.3 General tolerances

The tables below contain reference values for the surface form tolerances achieved by the respective casting method. The tolerances applicable for the component concerned must be agreed between purchaser and supplier and indicated on the drawing.

Table 1

General tolerances for **sand castings from aluminum alloys** created from surface data			
Degree of accuracy	Space diagonals	General tolerance	Wall thickness tolerance
1	up to 200	2.0	±0.9
2	200 to 400	2.6	±1.0
3	400 to 700	3.2	±1.2
4	700 to 1000	4.4	±1.5
5	1000 to 1400	5.2	±1.8

Table 2

General tolerances for **chilled castings from aluminum alloys without cores** created from surface data			
Degree of accuracy	Space diagonals	General tolerance	Wall thickness tolerance
1	up to 200	1.4	+1 / -0.5
2	200 to 400	2	
3	400 to 700	2.2	
4	700 to 1000	2.6	
5	1000 to 1400	3.2	

Table 3

General tolerances for **chilled castings from aluminum alloys with cores** created from surface data			
Degree of accuracy	Space diagonals	General tolerance	Wall thickness tolerance
1	up to 200	1.8	+1 / -0.7
2	200 to 400	2.4	
3	400 to 700	2.8	
4	700 to 1000	3.2	
5	1000 to 1400	4	

Table 4

General tolerances for **die castings from aluminum alloys** created from surface data				
Degree of accuracy	Space diagonals	General tolerance	Wall thickness tolerance	Bore position tolerance
1	up to 200	1.0	+0.8 / -0.5	Ø 1.0
2	200 to 400	1.2		Ø 1.2
3	400 to 700	1.6		Ø 1.5
4	700 to 1000	2.0		Ø 1.8
5	1000 to 1400	2.4		Ø 2.0

Table 5

General tolerances for **sand castings from aluminum alloys** created from surface data			
Degree of accuracy	Space diagonals	General tolerance	Wall thickness tolerance
1	up to 200	2.5	±1.2
2	200 to 400	3.0	±1.5
3	400 to 700	3.5	±1.7
4	700 to 1000	4.0	±2.0
5	1000 to 1400	4.5	±2.2

9.4 Indication of tolerances on drawings

9.4.1 Drawing entries

The 3D-CAD data record and the drawing always belong together. If a 3D-CAD data record is passed on to the supplier, it must in principle be accompanied by the drawing (paper or electronic). This also applies for the reverse situation, i.e. if the drawing is passed on, it must be accompanied by the 3D-CAD data record. For the designation of the data record, see chapter 6.3.

Entry in drawing
The tolerance table is affixed to the drawing in accordance with VN3212.

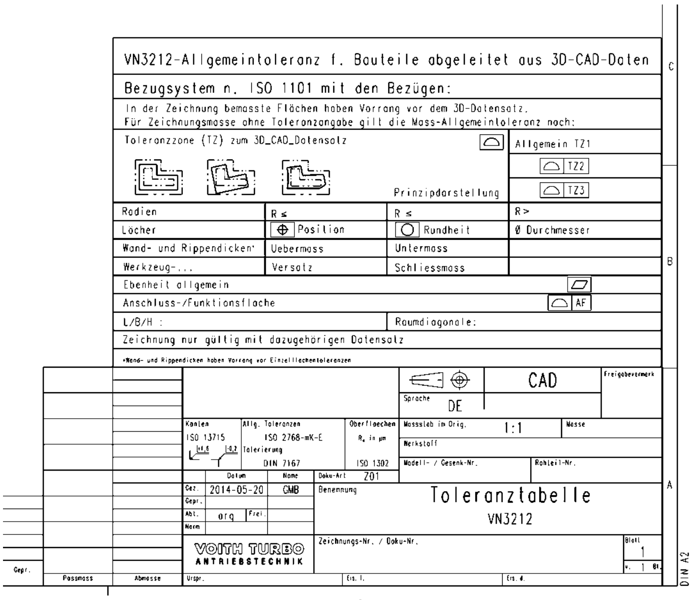


Fig. 4: Example on the position of the tolerance table on the drawing

9.4.2 Tolerance table

The individual requirements for precision of execution are defined and specified using limit values on the basis of the tolerance table below.
The entries shown in bold are component specific and made by the design engineer.
The values in the table below are merely examples. They must not be transferred indiscriminately without verification.
Explanations on the table fields are provided on the following pages.

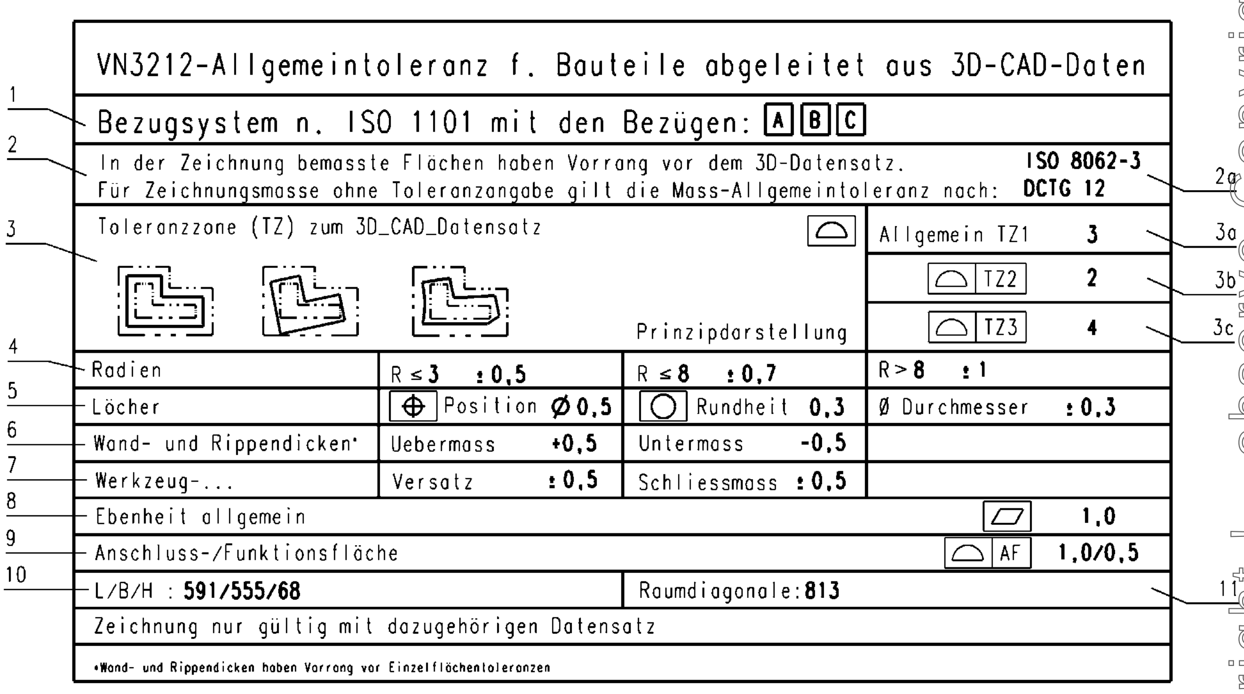


Fig. 5: Tolerance table

9.4.3 Content of the tolerance table

9.4.3.1 Datum system component (1)

A datum system to ISO 1101 must be defined for the component. The entry in field 1 denotes the datum references used.

9.4.3.2 General tolerances of elements dimensioned on the drawing (2)

If elements (surfaces, radii, distances etc.) are dimensioned directly on the drawing, the general tolerance indicated in 2a applies unless a tolerance is given at the dimension. Directly indicated tolerances take precedence over the tolerance based on 3D-CAD data.
Direct dimensioning on the drawing is to be limited to elements which cannot be described or which are difficult to describe via enveloping surface tolerances, or which are required for in-series quality monitoring.
Examples for 2a: ISO 8062-3 DCTG 12 or in individual cases ±0.5 / ±2°

9.4.3.3 General tolerance zone (3)

The fields in line 3 define the general surface tolerancing of the component
Consider 2 surfaces, equidistance to the 3D data record, which surround the component and/or which lie below the reference surface in the material. This corresponds to the surface form deviation indication according to DIN ISO 1101 with the symbol ⌓ (profile of a user-defined surface).
A tolerance zone indication of ⌓ 3, for example, denotes a ±1.5 mm tolerance relative to the reference surface. The distance from this tolerance zone to the 3D data record is specified by the design engineer.
The real component must now be completely located within this "envelope". This indication does not yet guarantee to state whether waviness, distortion, bias etc. are present within this envelope.

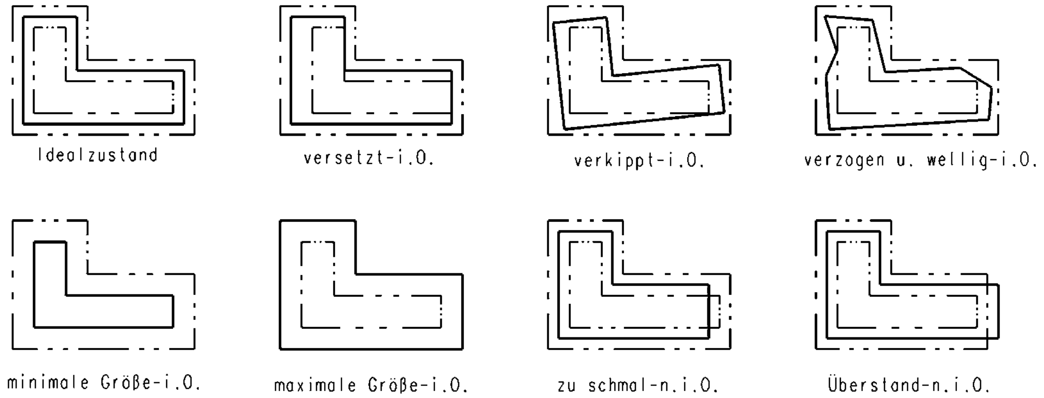
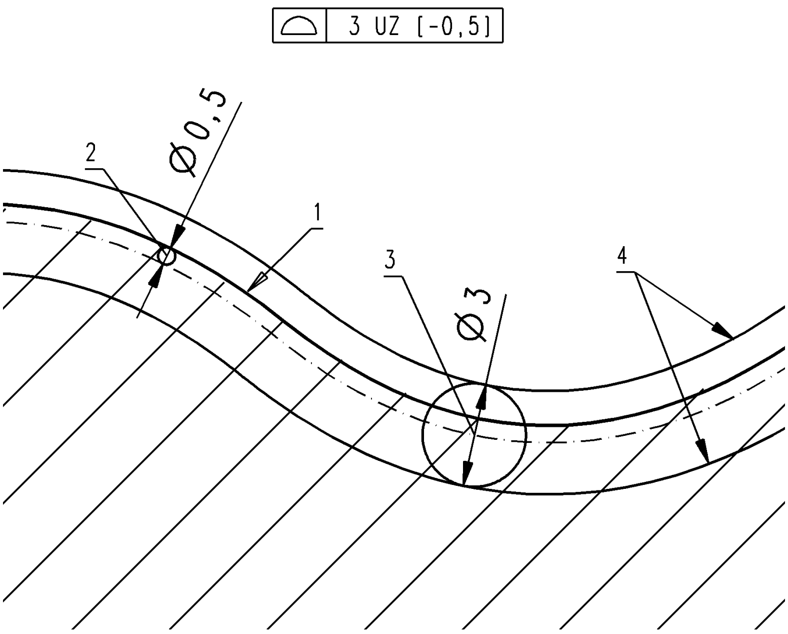


Fig. 6: Position of the real component within the tolerance zone

A different indication for plus and minus tolerance is allowed in fields 3 a-c.

If the tolerance zone is not central to the theoretically precise geometric shape, this unequally split tolerance zone must be entered using modifier UZ, as shown in Fig. 5



1 Theoretical profile in this example, the material lies beneath the profile
2 Sphere for defining the offset, theoretical profile
3 Sphere for defining the tolerance zone
4 Limits of the tolerance zone

Fig 7: Indication of an unequally split tolerance zone

The extracted (actual) surface must lie between two equidistant surfaces, which encompass spheres with a defined diameter equal to the tolerance value. The center points thereof, however, lie on a surface which corresponds to the envelope surface on a sphere, which is in contact with the theoretically precise shape, and the diameters of which are equivalent to the amount of the value which is indicated based on ZU. The direction of the offset is indicated by the sign. The "+" sign denotes "outside the material" and the "-" sign denotes "inside the material".

Additional tolerance zones (3b, 3c)
Values are entered in 3b) and 3c) only if additional tolerance zones (TZ2, TZ3) are required on the component. These can be narrower than the general tolerance zone, in order to limit important zones to a greater extent. These zones can also be larger in order to expand requirements in non-important zones. If no additional tolerance zones are used, these fields remain empty.

The position of these additional zones is indicated on the drawing via form symbol ⌓|TZ2 and a reference arrow pointing towards the surface(s), or via marking lines which define this zone. If necessary, the area can be defined by means of dimensions.

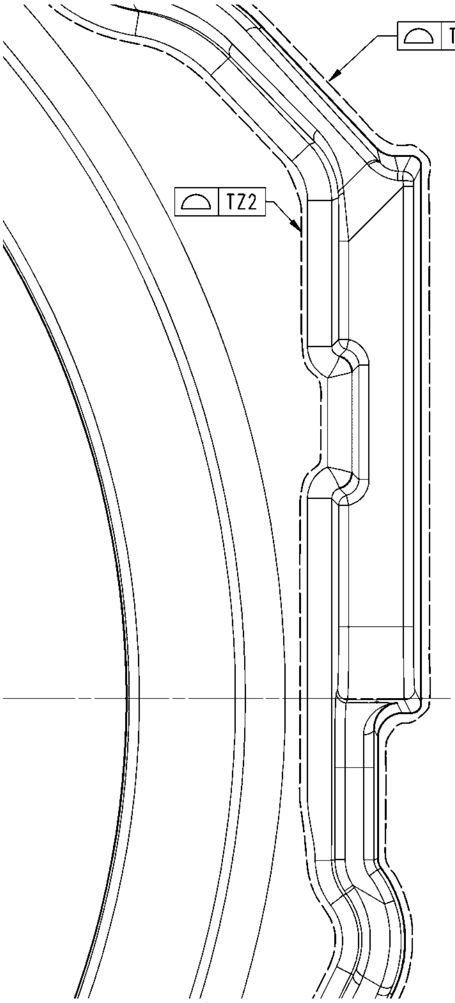


Fig. 8: Indication of additional tolerance zone

A polysurface which is complicated to describe can also be indicated in the 3D-CAD model via TEs or similar distinct methods. This requires that these identifiers are unique and cannot be lost from the data record either at Voith or at the supplier.

9.4.3.4 Radii (4)

The fields in line 4 tolerate the radii on the component that were generated by the CAD system via the radius functions thereof, i.e. not radii in the center of or within freeform surfaces.
The radii can be constant or variable over their progression.
There are three fields for "from ... to" radii for specifying tolerances for variable radii. The values are indicated as ±.

If no values are entered in the fields in line 4, the general freeform tolerance will apply

9.4.3.5 Holes (5)

The fields in line 5 tolerate round holes in the component.
The position accuracy, roundness and diameter tolerance can be specified. For position tolerance, a preceding Ø sign indicates a round tolerance field. If there is no Ø sign, the tolerance is quadratic. An intentional alignment of the quadrat must be indicated (e.g. in the comments field), otherwise the alignment will correspond to the main axes of the measuring fixture. Depending on the position and alignment of the hole, this can negatively impact the functional tolerance.

If no values are entered in the fields in line 5, the general surface form tolerance will apply.

9.4.3.6 Wall and rib thicknesses (6)

The fields in line 6 tolerate wall and rib thicknesses. Zones which have a significantly smaller third dimension depth in terms of width and height (≤ 1:4) are considered walls/ribs.

Example: A zone of 100x100 mm (no matter whether planar or curved) is still considered a wall even with a thickness of 25 mm. With a thickness of 30 mm, however, it is considered a body, which is evaluated exclusively with envelope surfaces.

If this simplistic definition does not suffice for a complex component, the walls/ribs must be indicated in detail as such on the drawing.

The tolerances must be indicated separately for a wall thickness oversize and a wall thickness undersize. An oversize can often be specified more generously, whereas an undersize quickly leads to strength and/or manufacturing problems.

Wall and rib thickness take precedence over general enveloping surfaces. Therefore, they must be inspected and analyzed separately. Where a wall has a just OK tolerance zone on the left and a just OK tolerance zone on the right, this prevents a doubling of the tolerance overall.

Example: The specified enveloping surface value is 2, i.e. ±1 mm. The wall thickness is also tolerated to ±1 mm.
Result
The wall tolerance zone inspected from the left at -1 mm → Tolerance OK.
The wall tolerance zone inspected from the right at -1 mm → Tolerance OK.
As a result, the wall is 2 mm thinner than the setpoint value, the wall thickness tolerance of ±1 mm is significantly underrun.

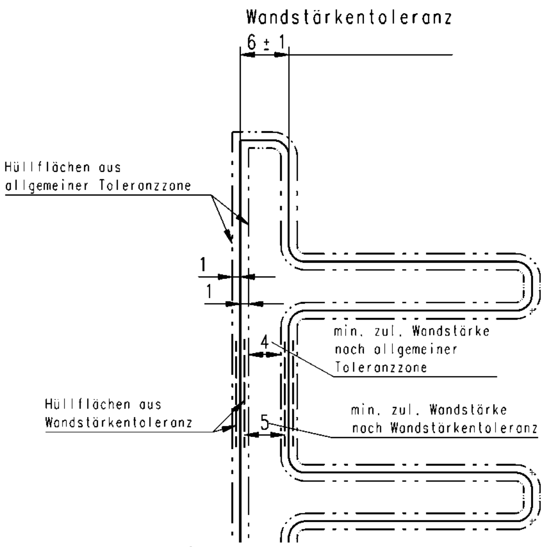


Fig. 9: Example of additional wall thickness tolerancing

If no values are entered in the fields in line 6, only the general surface form tolerance shall apply.

9.4.3.7 Tool offset and closing dimension (7)

The fields in line 7 can tolerate a possible offset or the closing dimension of the tool halves (cavities) if they are not exactly opposite one another, if slides and cores are not precisely positioned or if tools do no close in an optimum manner.

As a rule, no such additional tolerances ought to occur. Offset and closing errors must be covered within the generally applicable collective tolerances. Additional tolerances are to be entered in line 7 only if this goal cannot be achieved for technical/economic reasons.

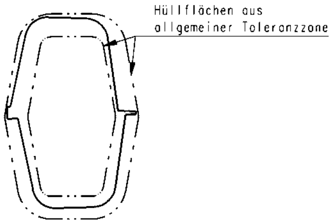
Offset tolerances describe the lateral offset of the respective tolerance enveloping surfaces per tool section in the mold parting line; the closing dimensions tolerances describe the additional deviation of the respective enveloping surfaces per tool section perpendicular to the mold parting line.

The enveloping surfaces of the top and bottom parts of the mold to be inspected can be shifted once and as a whole by the indicated value in the main partition line and/or perpendicular thereto.

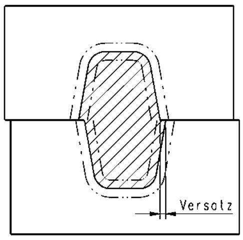
If no values are entered in the fields in line 7, only the general surface form tolerance shall apply.

Examples:

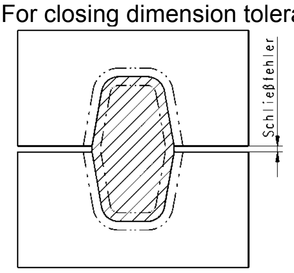
Normal case. Irrespective of offset, closing and other errors, the component must lie wholly and completely within the envelopes of the general tolerance zones.



For offset



For closing dimension tolerance



For offset and closing dimension tolerance

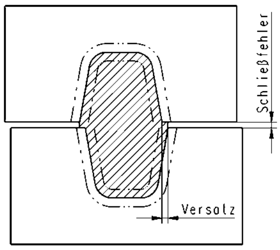


Fig 10: Examples of tool offset and closing dimension errors

9.4.3.8 Flatness (8)

Line 8 tolerates the general flatness of planar surfaces which are generated by the CAD system via its planar functions, i.e. no locally flat surfaces within a freeform surface. As a rule, only optical forecasts can and should be limited hereby.
Flatness is assessed independently of its spatial position.

If no values are entered in the field in line 8, the general surface form tolerance shall apply.

9.4.3.9 Limited flatness for joining and functional surfaces (9)

Line 9 is used if a local accuracy can no longer be guaranteed by the general surface form tolerance (TZ1, TZ2, ...). This is usually the case if there are adjacent components at these surfaces, on which other components are mounted directly or which form, in the direct vicinity of one another, a common joint, connecting surface or transition.

This tolerance indication comprises 2 tolerances. The first value defines the position of this tolerance zone in space, i.e. initially the general position on the component. The second value defines the tolerance zone of these surfaces considered for themselves, i.e. how accurate the surface as such should be. The second tolerance zone can be shifted normally (perpendicularly) within the first tolerance zone.

This tolerancing can be used for both flat and three-dimensional adjacent surfaces. There is no difference here, either in terms of method or effect.

Example:
An indication "Joint… 1.0/0.5" would mean that the marked surface must initially lie, proceeding from the overall component, within a tolerance zone of 1mm (±0.5mm). This surface must then, considered for itself, respect its own tolerance zone of 0.5mm (±0.25mm) which can be shifted normally (in parallel).

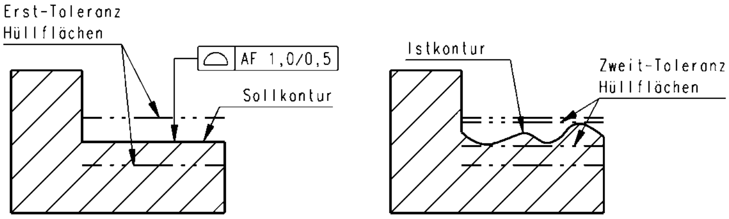


Fig. 11: Examples of terminal surface tolerancing

Terminal surfaces must be indicated on the drawing by form tolerance symbol ⌓ AF and a reference arrow pointing to the surface, or by marking lines. If necessary, the area concerned must be limited by dimensions.
If no value is entered in the field in line 9, the general surface form tolerance shall apply.

9.4.3.10 Space diagonals (10)

Length, width and height, as well as the space diagonal (see chapter 8.3) of the prismatic body which limits the component, must be entered in line 10.
These indications are for information only. They may be used to determine the required size of the measuring unit and to select the tolerance values.