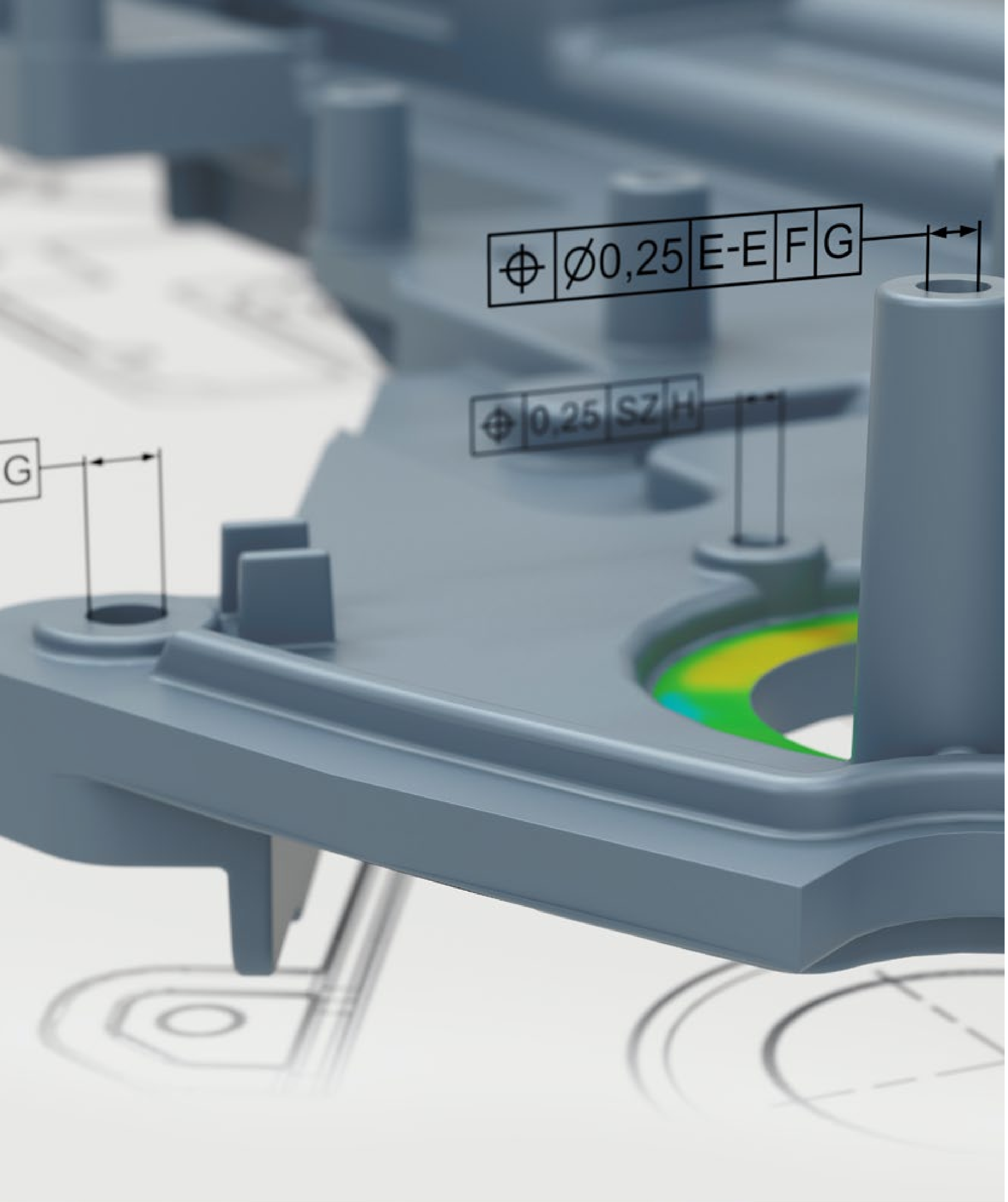
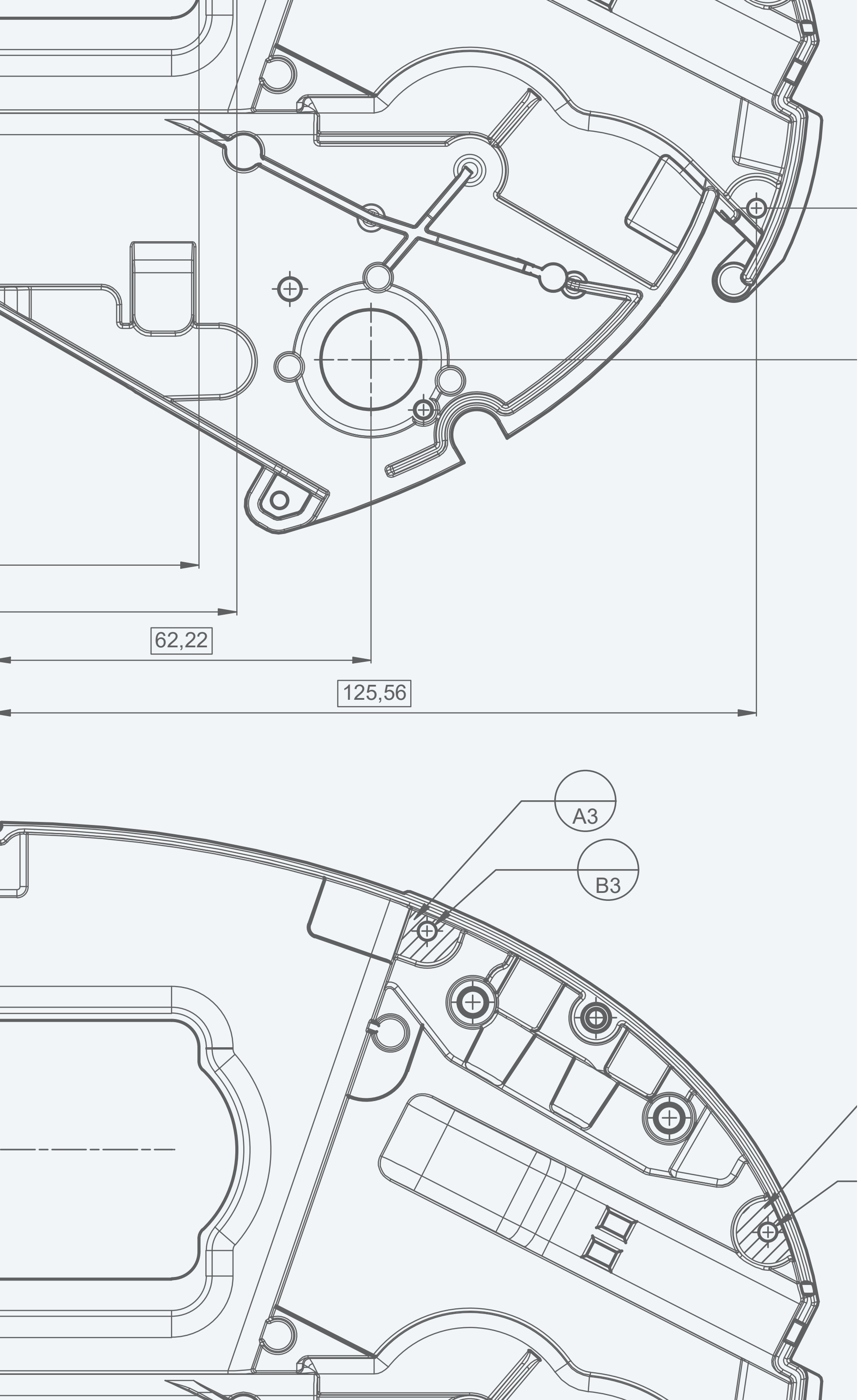


WHITE PAPER

Geometric Dimensioning and Tolerancing (GD&T)

Increase quality
and reduce costs



Content

Summary

Tolerances ensure that parts perform their intended function and are interchangeable. When dimensions and geometries fall short of or exceed the specified tolerance, this can result in problems during assembly, or even in malfunction parts.

When tolerancing parts, Geometric Dimensioning and Tolerancing (GD&T) plays an important role. GD&T describes the type and form of permitted geometric deviations in the part with extreme accuracy.

Compared to conventional tolerancing, a product specification that uses GD&T has several advantages:

- Simplified manufacturing and component testing
- Optimal comparability
- Option to combine tolerances
- More usable tolerance in round and cylindrical elements
- No tolerance aggregation through chain dimensioning
- Suitable for complex, freeform parts
- Component testing with product and manufacturing information (PMI / FTA) support

Both GD&T and the measuring results of the tolerance checks are expressed as pure numerical values. However, interpreting these numbers and translating them into specific corrective actions is a challenge. For this reason, graphical representation methods in the form of deviation flags have been established to visually identify the type and form of deviations.

GD&T is most easily inspected using software that color-codes nominal and actual values and uses vectors to represent these deviations.

The rules, symbols, definitions, requirements, default values and recommended methods to determine and interpret GD&T are established in systems of standards, although only two see global acceptance and use:

- ISO GPS (geometric product specification), established by the International Organization for Standardization
- ASME Y14.5, established by the American Society of Mechanical Engineers (ASME)

The two systems differ primarily in their basic principles of tolerancing, their mode of representation, their methods of computation and their types of tolerances. Before implementing GD&T, it must be decided which system should be applied.

Tolerancing in Manufacturing

Today's modern manufacturing machinery can achieve very high degrees of accuracy. Nevertheless, workpieces continue to suffer from undesired deviations in their nominal dimensions and geometries. To limit these deviations (and their resulting effects), manufacturers and metrologists rely on tolerances. Tolerances define what deviations from the nominal value are acceptable – and when a part is considered "not OK" (NOK) and thus unusable without improvement.

Tolerances ensure that

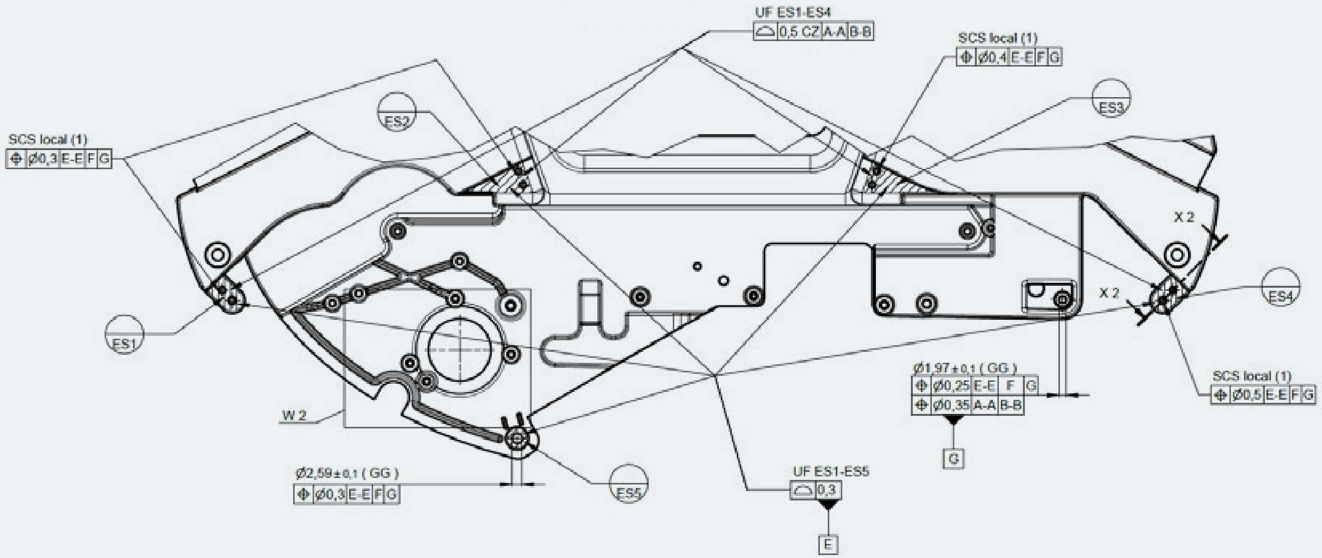
parts perform their intended function and are interchangeable. When dimensions and geometries fall short of or exceed the specified tolerance, it can result in assembly difficulties or even malfunctioning parts.

Defining tolerances requires subtlety

Tolerances that are too generous necessarily result in manufacturing variability. Workpieces that deviate from one another require additional labor during replacement or assembly. In a worst-case scenario, workers are forced to hunt for pairs of parts that can still be put together. This requires a lot of time and thus leads to cost increases.

In turn, tolerances that are too tight require manufacturing machinery with extreme positioning accuracy and repeatability; wear caused by age must be necessarily avoided or offset. Furthermore, component testing becomes very time-consuming. Manufacturing and assembly costs, along with quality assurance costs, skyrocket.

The rule of thumb, therefore, is as follows: Tolerances should always be as loose as possible but as tight as necessary.



Advantages

of GD&T

When tolerancing parts, Geometric Dimensioning and Tolerancing (GD&T) plays an important role, since it describes the type and form of permitted geometric deviations in the part with extreme accuracy.

GD&T is designed to ensure that parts have the desired shape, fitting accuracy and function. This makes GD&T an indispensable tool for producing high-quality parts.

GD&T is applied in every step of the process chain, such as design, production and quality assurance. While it is used for specification purposes during part design, it gives the metrologist exact specifications for verifying the part.

Compared to conventional tolerancing, a product specification that uses GD&T offers several advantages:

Simplified manufacturing and component testing

Tolerances are defined in a set alignment with datum systems that reflect the assembly situation. These are geometrically ideal, virtual fixtures against which the part to be inspected is aligned. This means that the part is not looked at by itself but rather according to its subsequent functionality and assembly capability. This simplifies both manufacturing and subsequent metrological quality assurance.

Optimal comparability

The datum system also ensures that measuring results can be easily compared, e.g., between individual manufacturing locations of a company or between suppliers and customers.

Option to combine tolerances

For counterparts, GD&T can be considered in relation to the dimensional tolerance, which allows for more latitude in manufacturing and quality assurance, and thus reduces the number of NOK parts. For both tolerances, a grand total is defined as a virtual gauge that cannot be exceeded. This way, if the dimensional tolerance deviation is very small, the geometric dimensioning deviation can be greater accordingly. The technical term for this principle is maximum material requirement.

More usable tolerance in round and cylindrical elements

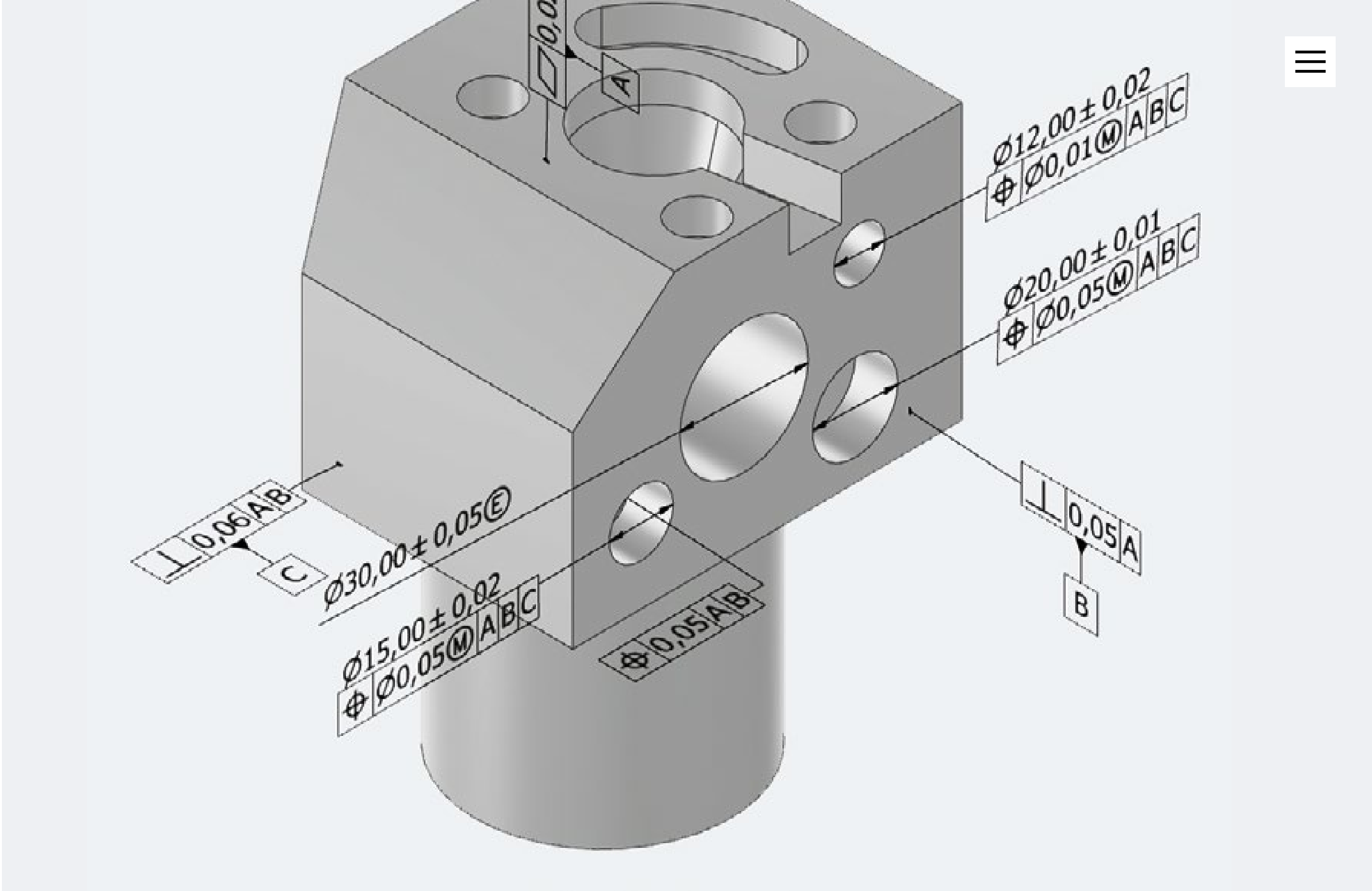
Compared to conventional tolerancing, GD&T also supports circular or spherical tolerance zones. This increases the usable tolerance for round and cylindrical elements during manufacturing by 57 % without impacting the function of the part. At the same time, the circular or spherical tolerance zone reflects the characteristic of round and cylindrical elements better than the classic +/- tolerance, which only works with flat tolerance zones.

No tolerance aggregation through chain dimensioning

In design, GD&T makes it easier to calculate tolerances, since negative effects caused by chain dimensioning – and with it undesirable tolerance aggregation – are avoided entirely.

Suitable for complex, freeform parts

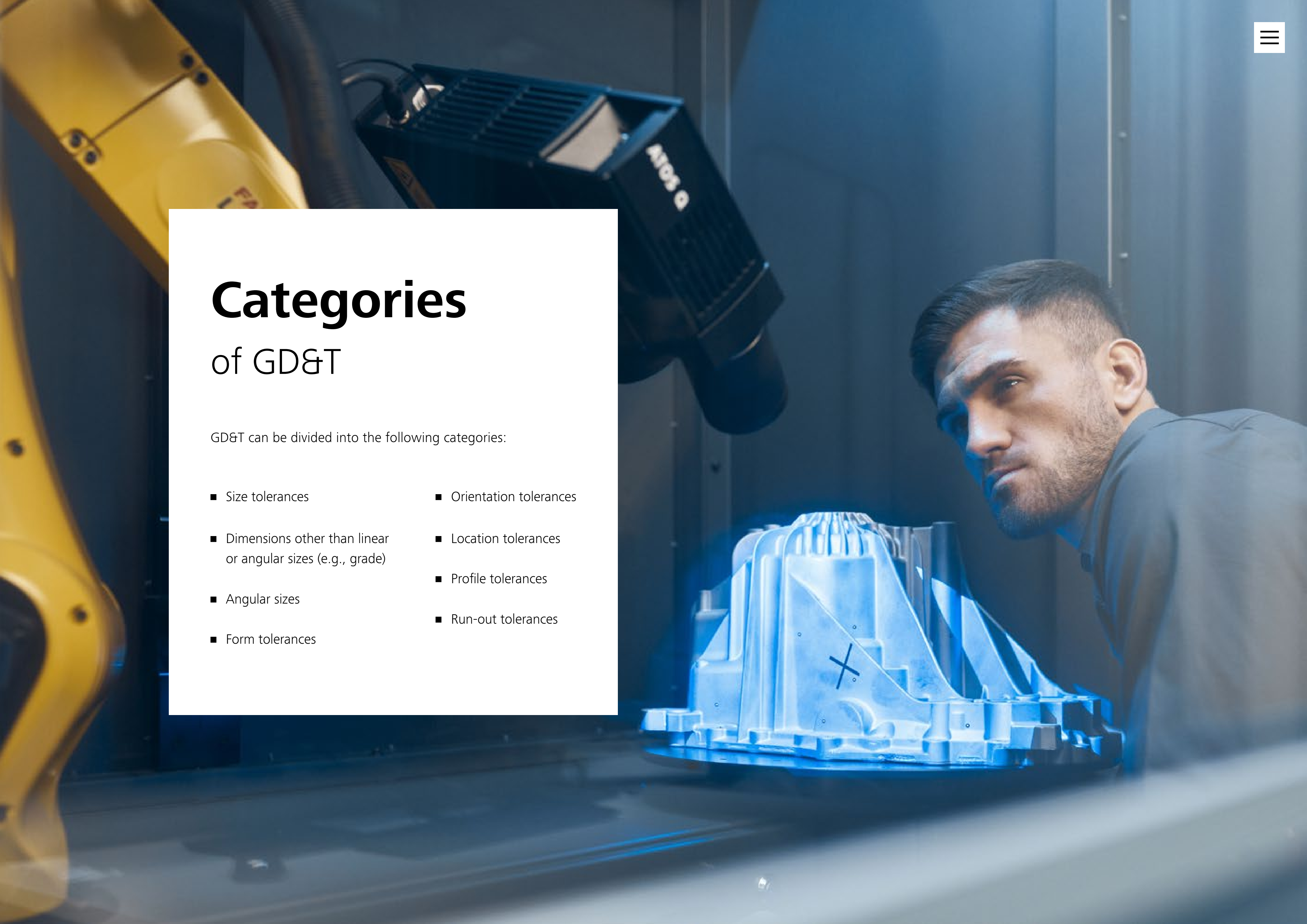
Compared to conventional tolerancing, GD&T can also be used to describe freeform tolerance zones and restricts even the most complex freeform surfaces. If a datum system is not used, only the toleranced form deviation is defined. If a datum system is used, the maximum position and orientation deviations are restricted.



Component testing with product and manufacturing information (PMI/MBD) support

The flaw with conventional tolerancing is that it is ambiguous. GD&T, on the other hand, can help achieve 100 % certainty in OK/NOK decisions, making it suitable for automation and digitalization applications, such as product and manufacturing information (PMI). PMI data contains information relevant to manufacturing. All drawing elements, such as dimensions and their relationships with geometric elements, are saved directly to the CAD data in machine-readable format. Modern metrology software can read this information and automatically add the corresponding inspections and traverse paths in order to, e.g., facilitate automated decisions regarding dimensional part quality. This, in turn, leads to a reduction of potential error sources.

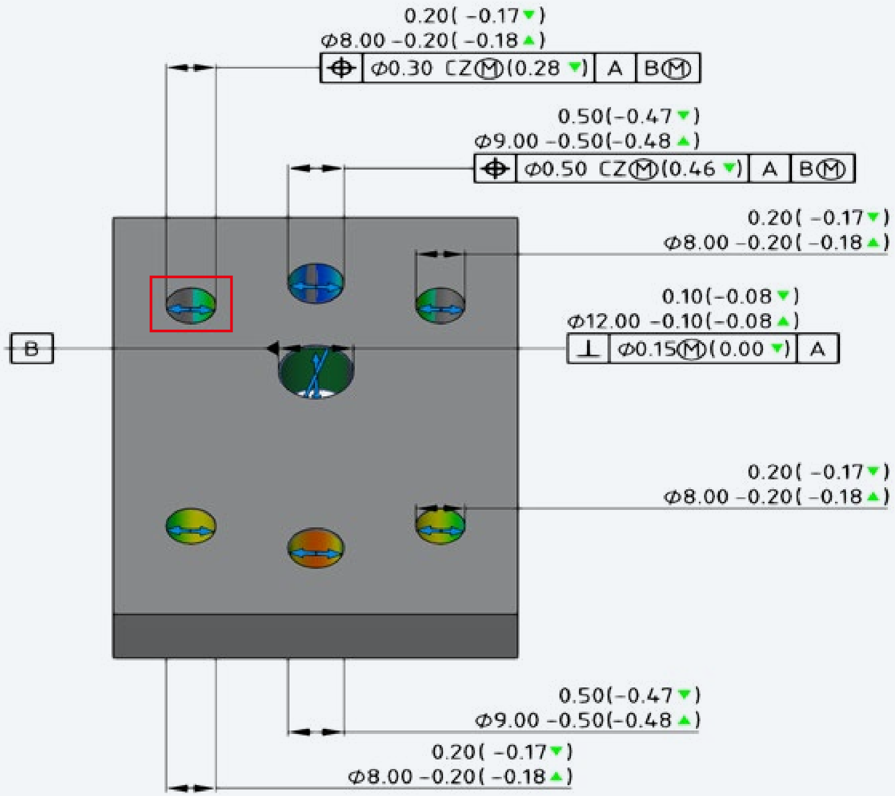
.

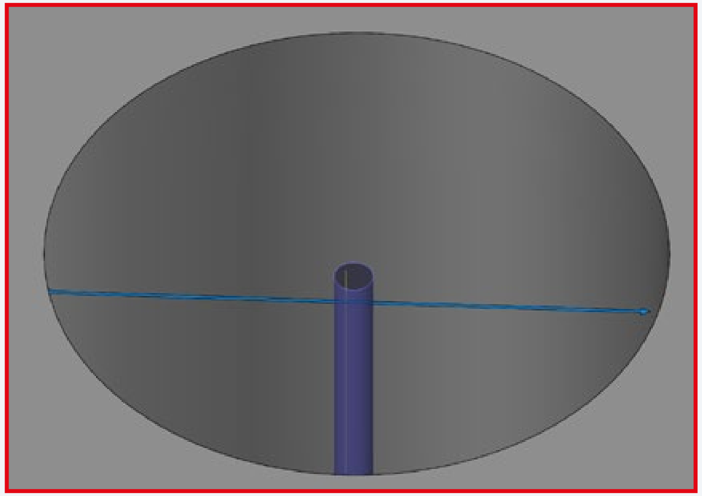


Categories
of GD&T

GD&T can be divided into the following categories:

- Size tolerances
- Dimensions other than linear or angular sizes (e.g., grade)
- Angular sizes
- Form tolerances
- Orientation tolerances
- Location tolerances
- Profile tolerances
- Run-out tolerances





Tolerance Zones

GD&T always makes use of tolerance zones. A tolerance zone describes the acceptable area in which a given deviation (dimension, form, position, orientation) may occur in geometric elements.

As long as the deviations remain within the tolerance zone, it is ensured that, e.g., the function of one or more geometric elements, or even of the entire part (depending on the chosen tolerances) is fulfilled. The type of deviation is of no consequence.

This requires carefully chosen tolerances that unambiguously and unmistakably describe the function of the part.

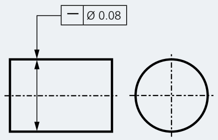
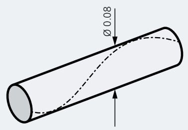
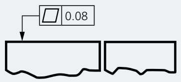
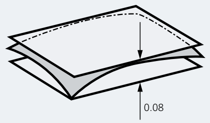
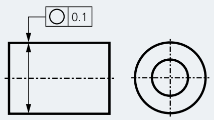
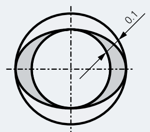
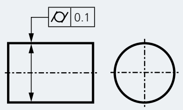
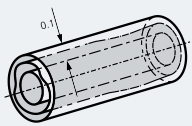
The basis for the desired nominal value as well as the alignment of the tolerance zone is formed by datums (e.g., a single geometric element, such as a cylinder or plane) or datum systems (a combination of various geometric elements arranged in a hierarchical structure).

Representation

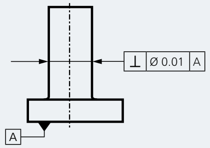
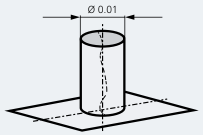
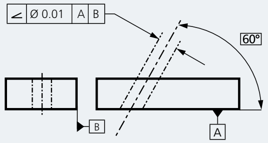
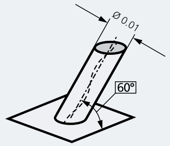
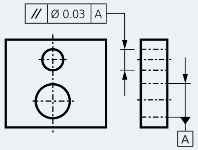
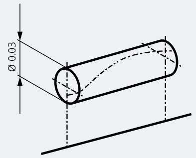
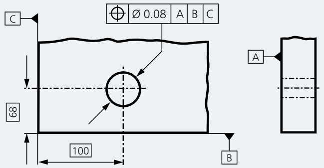
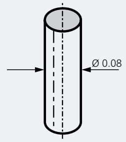
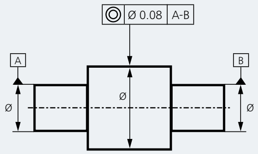
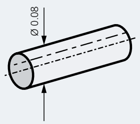
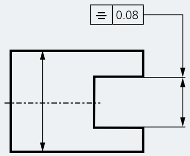
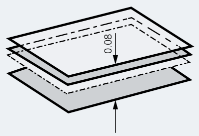
of GD&T

An international language of symbols has been developed to represent GD&T in drawings.

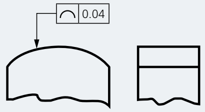
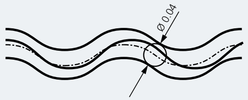
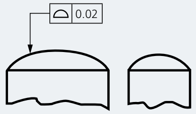
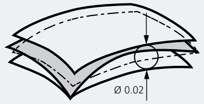
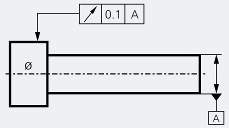
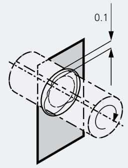
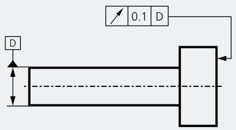
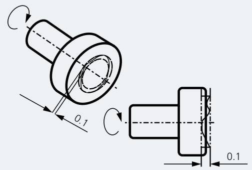
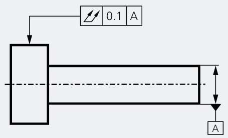
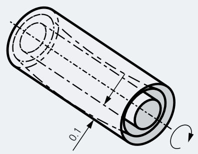
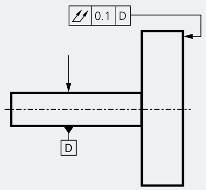
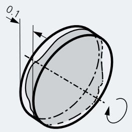
Form Tolerances

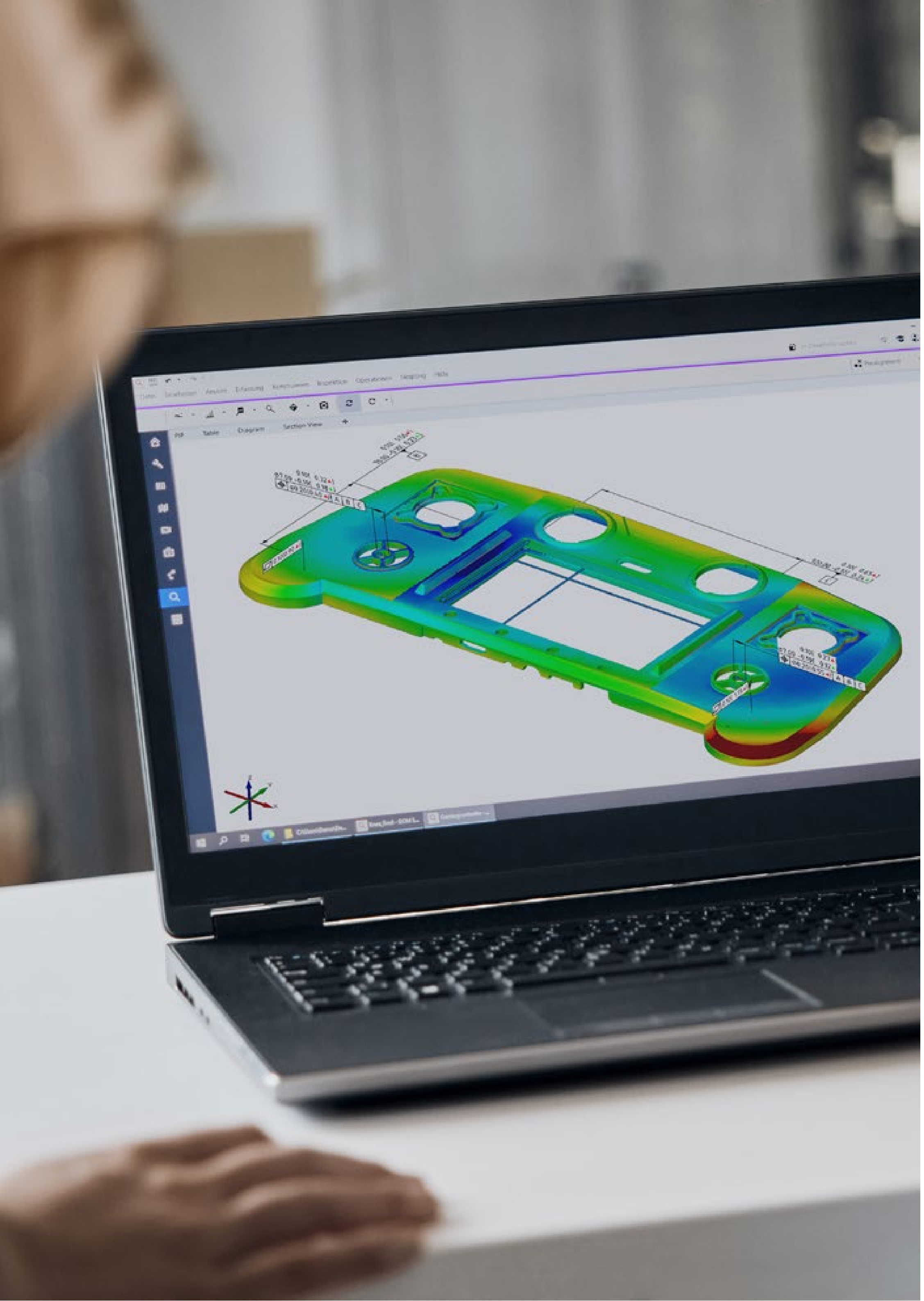
Symbol and Tolerated Properties		Drawing Specification and Explanation		
		Drawing Specification	Tolerance Zone	Explanation
—	Straightness			The extracted median line (extracted axis) of the cylinder connected to the feature control frame shall be contained within a cylindrical tolerance zone with a diameter of 0.08.
⏥	Flatness			The extracted surface shall be contained between two parallel planes 0.08 apart.
○	Roundness			The extracted circumferential line of each cross-section of the cylinder lateral surface shall be contained between two concentric circles in the same plane at a distance of 0.1.
⌭	Cylindricity			The extracted cylinder lateral surface shall be contained between two coaxial cylinders with a radial distance of 0.1.

Position and Orientation Tolerances

<table>
<tr><th colspan="2" rowspan="2">Symbol and Tolerated Properties</th><th colspan="3">Drawing Specification and Explanation</th></tr>
<tr><th>Drawing Specification</th><th>Tolerance Zone</th><th>Explanation</th></tr>
<tr><td colspan="5">Orientation Tolerances</td></tr>
<tr><td>⊥</td><td>Perpendicularity of a line (axis) to a surface</td><td>
</td><td>
</td><td>The extracted median line (extracted axis) shall be contained within a cylinder of diameter 0.01, perpendicular to the datum plane A.</td></tr>
<tr><td>∠</td><td>Angularity of a line (axis) to a datum surface</td><td>
</td><td>
</td><td>The extracted median line (extracted axis) shall be contained within a cylinder of diameter 0.01, which is inclined at a theoretically exact angle of 60° to the datum plane A.</td></tr>
<tr><td>//</td><td>Parallelism of a line (axis) to a datum line</td><td>
</td><td>
</td><td>The extracted median line (extracted axis) shall be contained within a cylinder of diameter 0.03, which is parallel to the datum straight line A.</td></tr>
<tr><td colspan="5">Location Tolerances</td></tr>
<tr><td>⌖</td><td>Position of a line</td><td>
</td><td>
</td><td>The extracted median line (extracted axis) shall be contained within a cylindrical tolerance zone with a diameter of 0.08, whose axis is perpendicular to datum plane A, at a distance of 68 to datum plane B and at a distance of 100 to datum plane C (theoretically exact location of the bore axis).</td></tr>
<tr><td>◎</td><td>Coaxiality of an axis</td><td>
</td><td>
</td><td>The extracted median line (extracted axis) of the cylinder connected to the feature control frame shall be contained within a cylinder of diameter 0.08 that is coaxial to the common datum axis A-B.</td></tr>
<tr><td>⌯</td><td>Symmetry of a median plane</td><td>
</td><td>
</td><td>The extracted median surface shall be contained between two parallel planes 0.08 apart, which are symmetrical to the datum median plane.</td></tr>
</table>

Position and Orientation Tolerances

Symbol and Tolerated Properties		Drawing Specification and Explanation		
		Drawing Specification	Tolerance Zone	Explanation
Profile Tolerances				
	Profile form of any line			In each section, parallel to the projection plane, the extracted profile line shall be contained between two equidistant lines enveloping circles with a diameter of 0.04, the centers of which are situated on a line having the ideal geometrical form.
	Profile form of any surface			The extracted surface shall be contained between two equidistant surfaces enveloping spheres with a diameter of 0.02, the centers of which are situated on a surface having the ideal geometrical form.
Run-Out Tolerances				
	Circular run-out (radial)			The extracted circumferential line, which is perpendicular to the datum straight line A in each cross-section, shall be contained between two concentric circles with a radial distance of 0.1 in the same plane.
	Circular run-out (axial)			In each cylindrical section, whose axis coincides with the datum axis D, the extracted line shall be contained between two circles of distance 0.1 arranged perpendicular to datum axis D.
	Total run-out (radial)			The extracted surface shall be contained between two coaxial cylinders with a radial distance of 0.1, whose axes coincide with the datum straight line A.
	Total run-out (axial)			The extracted surface shall be contained between two parallel planes 0.1 apart, perpendicular to the datum straight line D.

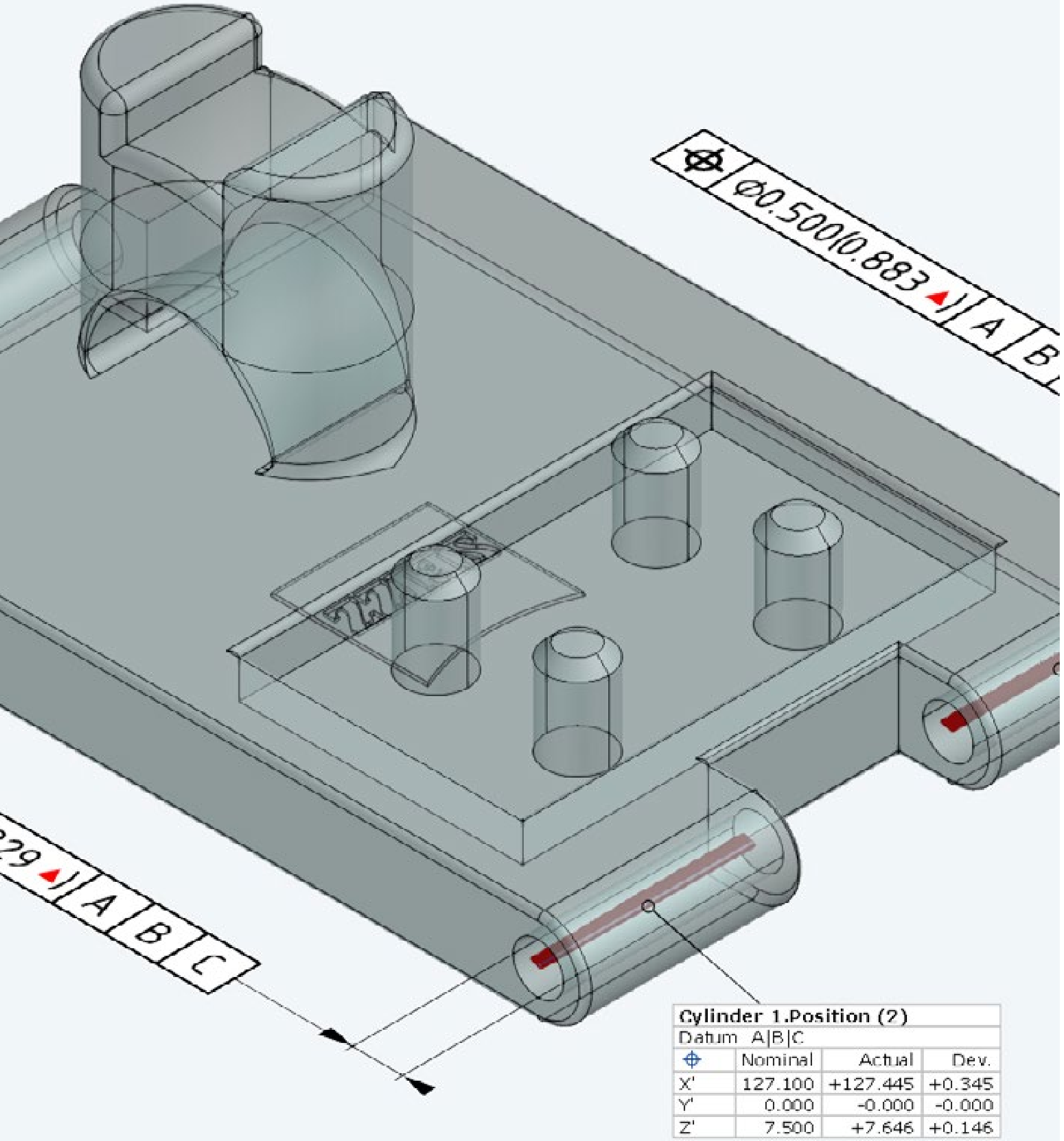


Interpretation
of GD&T

Both GD&T and the measuring results of the tolerance check are expressed as pure numerical values. However, interpreting these numbers and translating them into specific corrective actions is a challenge: The numbers contain neither information on the orientation in which the inspected part element may need to be corrected nor what quantity of material must be removed or added at which location. They merely reflect the width or diameter of the computed tolerance zone.

For this reason, graphical representation methods, e. g., in the form of deviation flags, have been established to visually identify the type and form of deviations. These can also be supported by indicating deviations in individual axes.

GD&T is most easily inspected using software that color-codes nominal and actual values and uses vectors to represent deviations.



Practical Example

Software-assisted interpretation of position deviations

In this practical example, a part of an additively manufactured plastic cube is measured by a 3D measuring system and then ZEISS INSPECT runs a nominal–actual comparison of the GD&T. The user receives specific information for each inspected geometric element from the software.

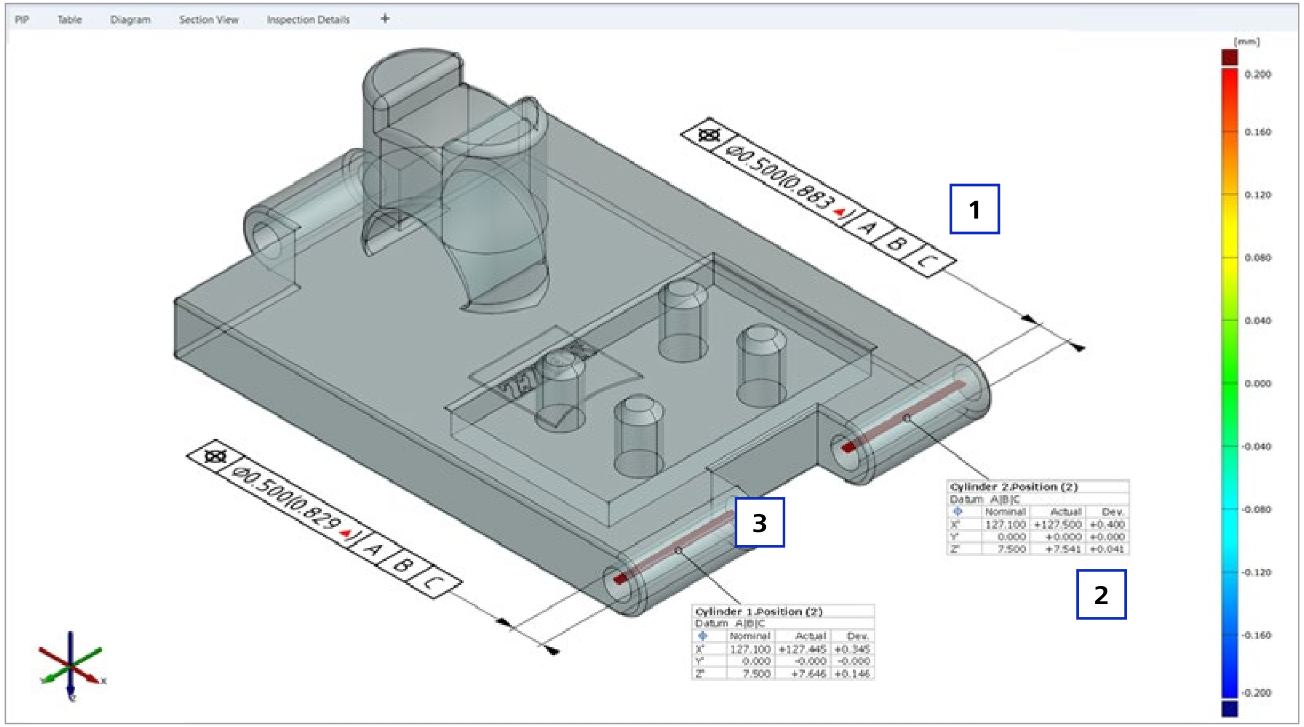
Step 1

User inspects the positions
of Cylinders 1 and 2

1 The drawing specification states that the extracted median line (extracted axis) must lie within a cylindrical tolerance zone with a diameter of 0.50 mm. The display shows that both inspected cylinders do not meet this specification (red arrow).

2 The user sees an overview of the actual and nominal values for the extracted cylinder axis as well as the deviation between actual and nominal (Nominal, Actual, Deviation) in the X, Y and Z directions. This makes it clear in which orientation the part must be corrected in the manufacturing process.

3 To further simplify the interpretation of the inspection results, the software also displays the deviations in different colors.



Results display in ZEISS INSPECT: position inspection with orientation-based correction information

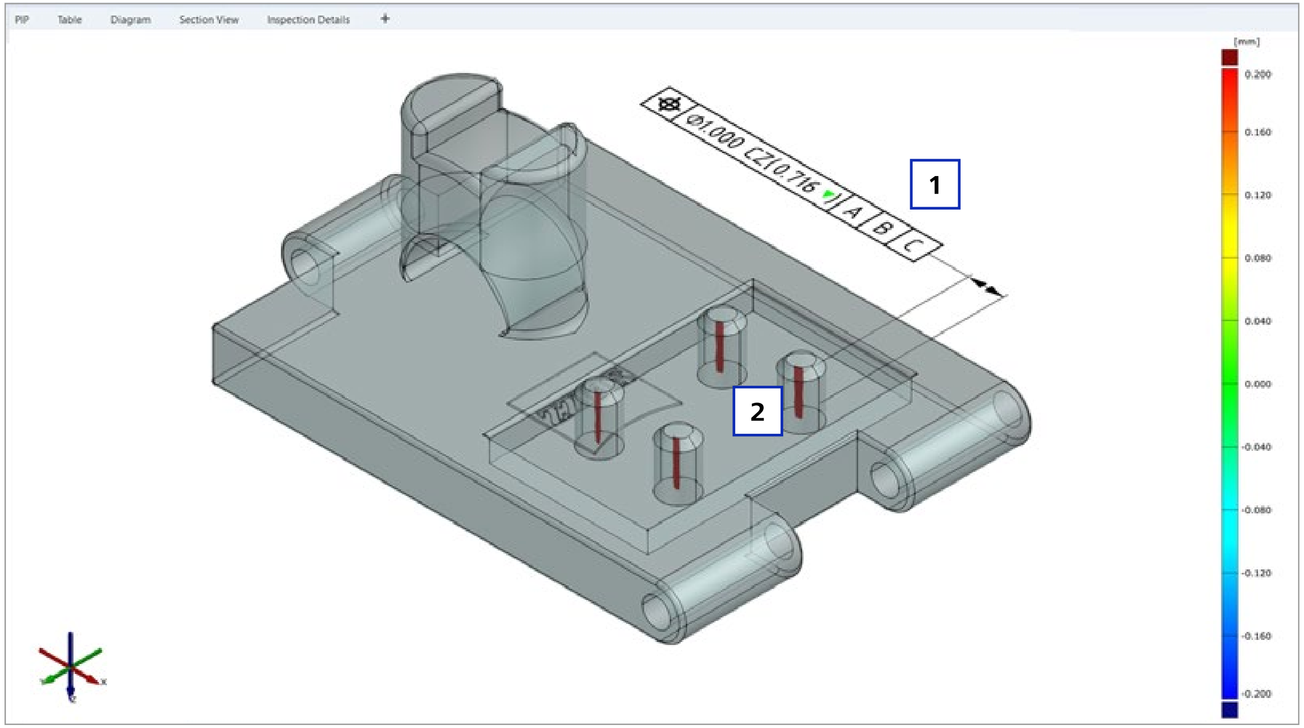
Step 2

User looks at the positions of another group of cylinders

Check for location relative to datum system and distances from one another (simultaneously)

1 The drawing specification states that the extracted median line (extracted axis) for all four cylinders must lie within a cylindrical tolerance zone with a diameter of 1.00 mm. The green arrow shows the user that this is the case here. The largest deviation detected is 0.72 mm. The initials “CZ” (combined zone) taken from the design drawing indicate that the tolerance zone applies simultaneously to multiple geometric features.

2 The software displays the tolerance zones in blue. The display also includes color-coded vectors (yellow/orange) to indicate the deviation from the middle of the tolerance zone. The user sees the deviation vectors not in their original size, rather upscaled (e.g., 10x magnification). This simplifies interpretation: Both the orientation and the extent of the deviation become clearly visible and, based on this, specific corrective actions can be derived if needed.



Results display in ZEISS INSPECT: position check with tolerance zone visualization and exaggerated scaling of deviation vectors

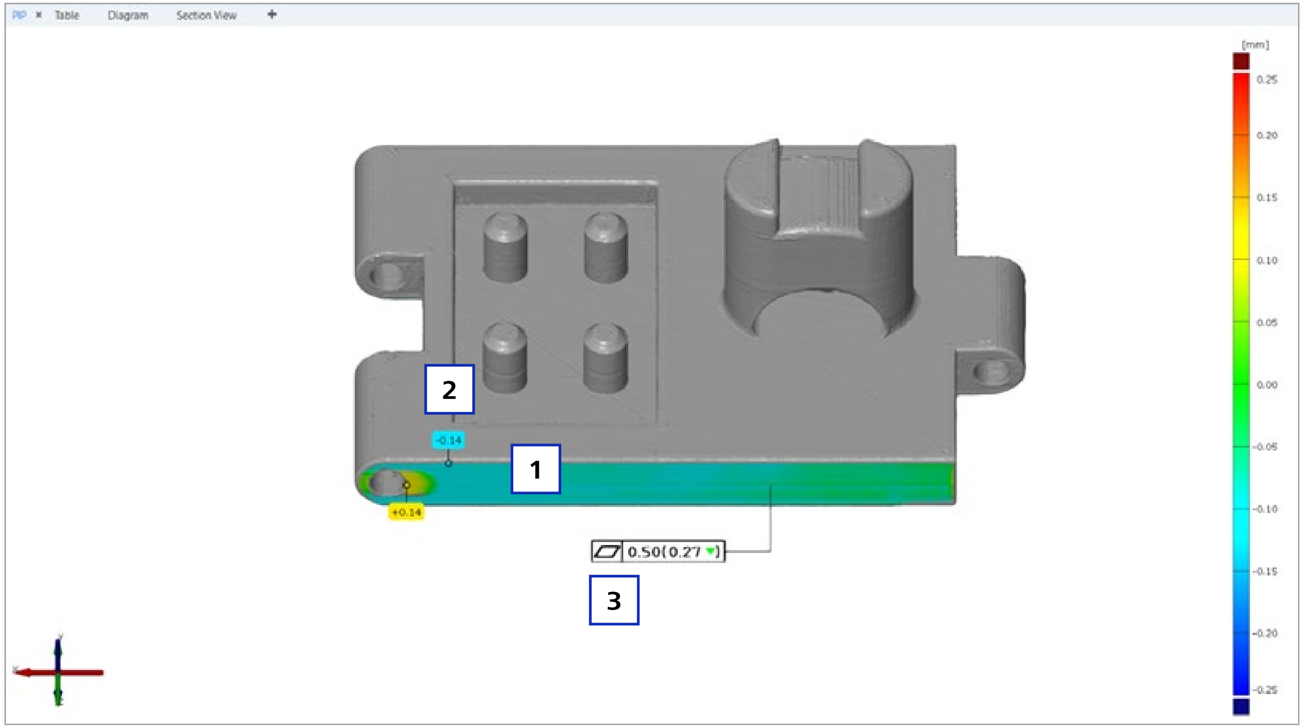
Step 3

User checks flatness

1 The software generates a color deviation representation of the examined surface.

2 The software also shows minimum/maximum deviation labels. These represent the deviation values of the points with the farthest and closest distance to the middle of the tolerance zone.

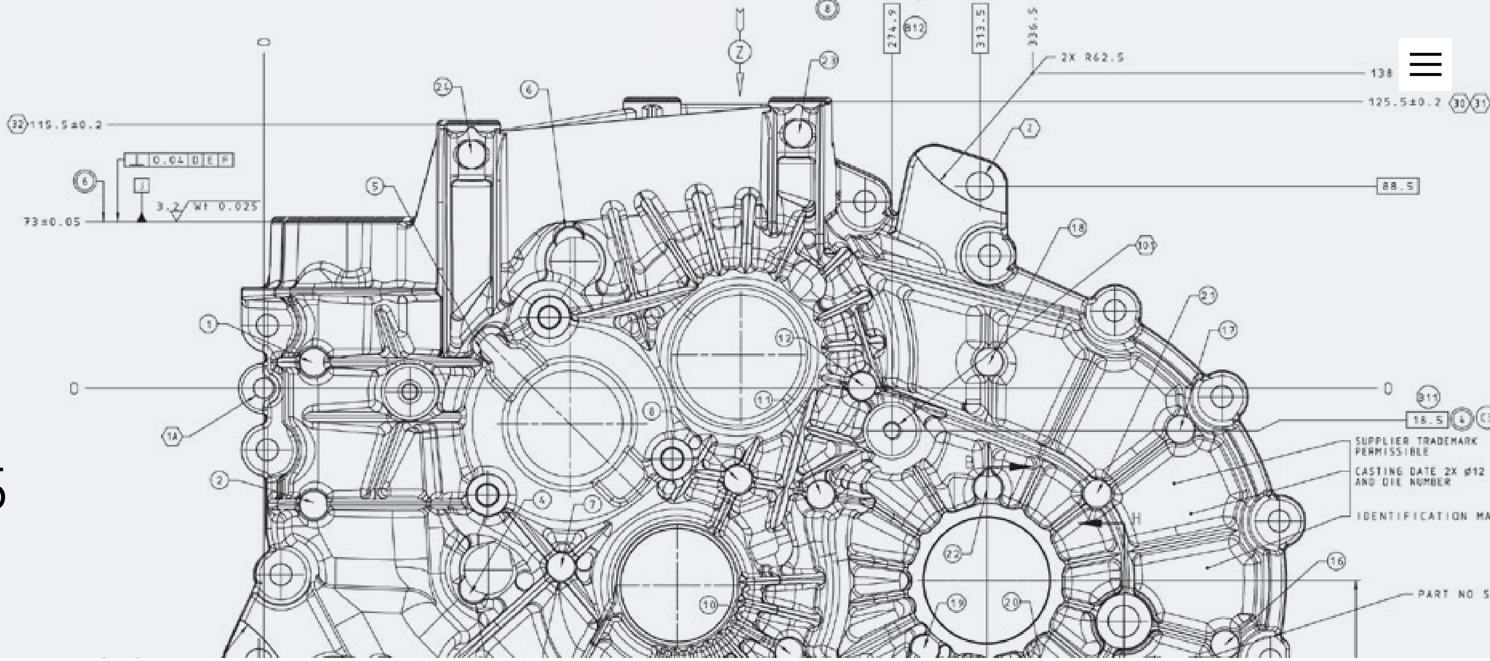
3 According to the design specification, the measured surface between two parallel planes must be 0.50 mm apart. The green arrow shows that the examined surface meets the tolerance specifications. Although deviations were detected, they are still within the tolerance zone.



Results display for flatness check in ZEISS INSPECT

Standards

ISO GPS and ASME 14.5



The rules, symbols, definitions, requirements, default values and recommended methods for determining and interpreting GD&T are established in systems of standards, although only two see global acceptance and use:

ISO GPS (geometric product specification), established by the International Organization for Standardization

ASME Y14.5, established by the American Society of Mechanical Engineers

Commonalities between systems of standards

Both systems are purely drawing standards that do not require any explicit measuring method or a specific measuring system. However, metrologists must take care to meet the requirements of the standards with consideration to measurement uncertainty. The metrologist alone is responsible for the measuring strategy.

On the other hand, responsibility for the specification to which, e.g., the drawing or a CAD data set belongs, lies solely with the design engineer, who is responsible for ensuring that the function of the part is fully described without any ambiguities. This makes GD&T the perfect tool for both standards.

Use of the systems in drawings also ensures legal certainty: A requirement (e.g., from the customer) is only valid if the specification is also listed on the drawing or other documents. This means other, unlisted specifications cannot be asserted.

Differences between systems of standards

The two systems differ from one another primarily in their basic principles of tolerancing, their mode of representation, their methods of computation and their types of tolerances. The main differences are briefly explained below.

Basic tolerancing principle
Under the ISO system, tolerancing follows the independency principle: Every tolerance recorded on the drawing (dimension, form, location) is considered individually and must be evaluated independently of the other tolerances. For sizes, this means that the dimensional tolerances and geometric tolerances are defined independently of one another. Relationships between the tolerances of individual geometric elements are only toleranced if necessary for the functionality of the part.

Under the ASME system, tolerancing of a part focuses on assembly capability and follows the envelope principle: When it comes to sizes, dimension, form and location are not considered disconnected from one another. Instead, the toleranced dimension also determines the geometric characteristics of the part. Dimensional and form deviations of surfaces must lie within the dimensional tolerance. A separate, independent tolerancing of characteristics of individual geometric elements occurs only if required for the functionality of the part.

Representation type
The systems differ in terms of their drawing data. To be able to read drawings and, therefore, tolerance specifications correctly, the user must be highly familiar with the type of representation used by each system.

Computation methods
Both systems also differ in the use of datums and datum systems, the measuring of the toleranced elements, and the computation of deviations.

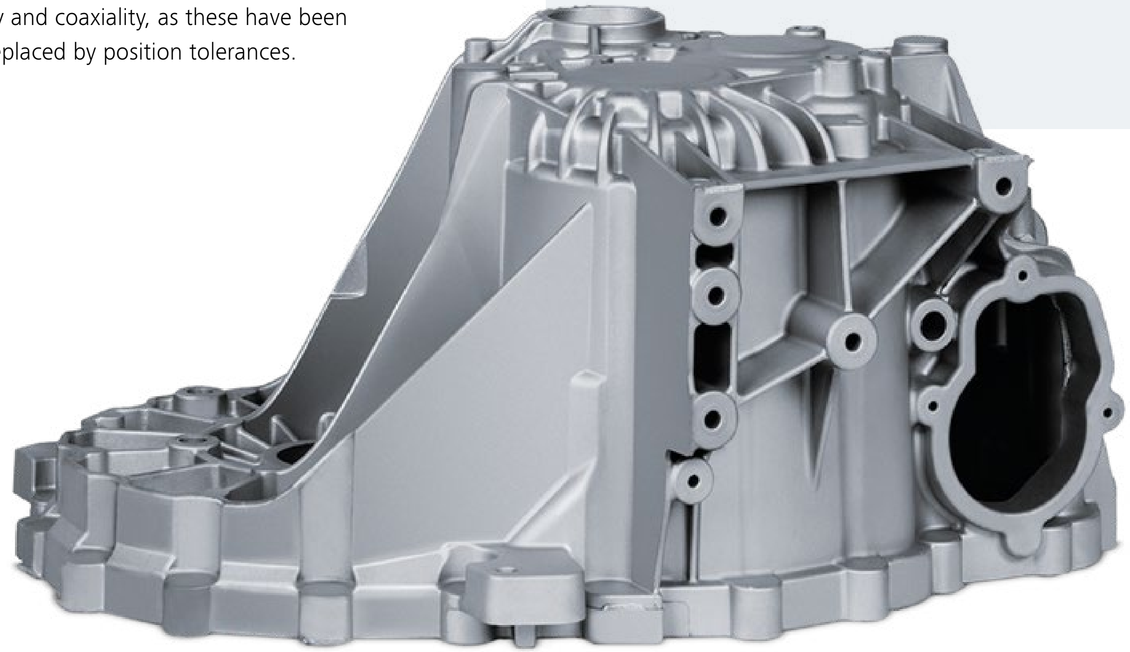
Tolerance types
Since 2018, the two systems have differed in terms of the types of geometric tolerances used. ASME Y14.5 does not use symmetry and coaxiality, as these have been completely replaced by position tolerances.

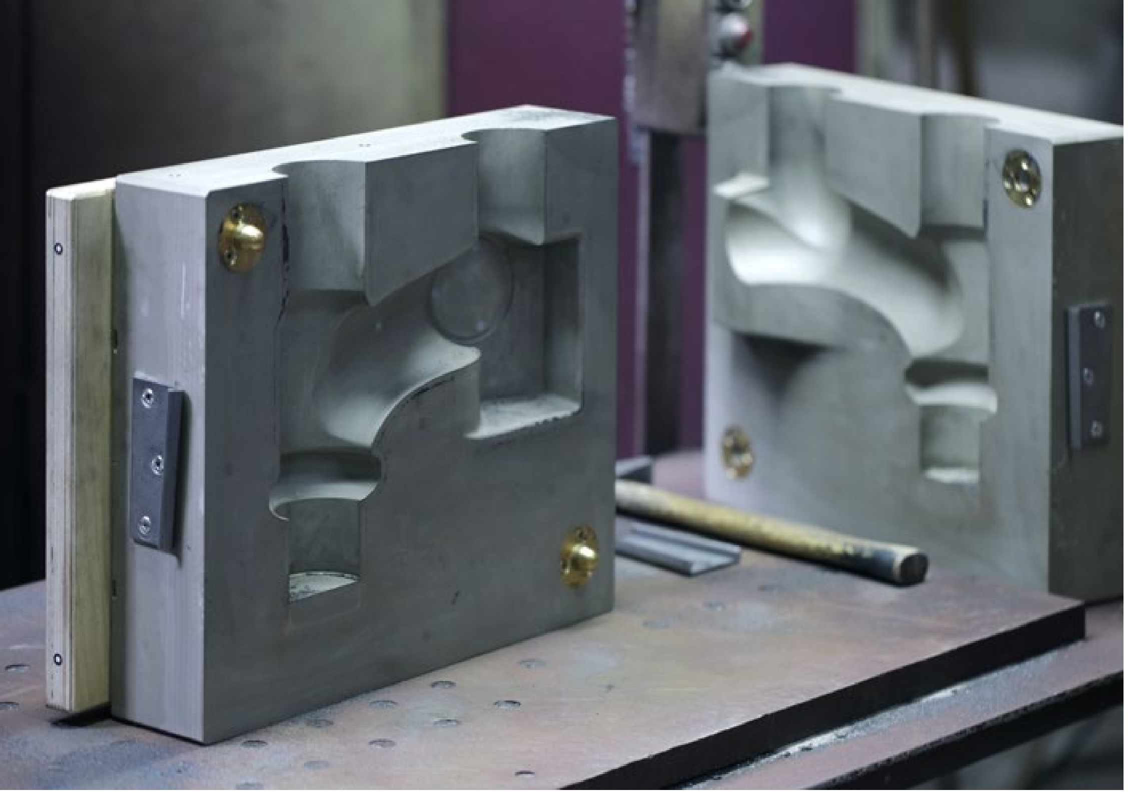
Before implementing GD&T, it must be decided which system should be applied.

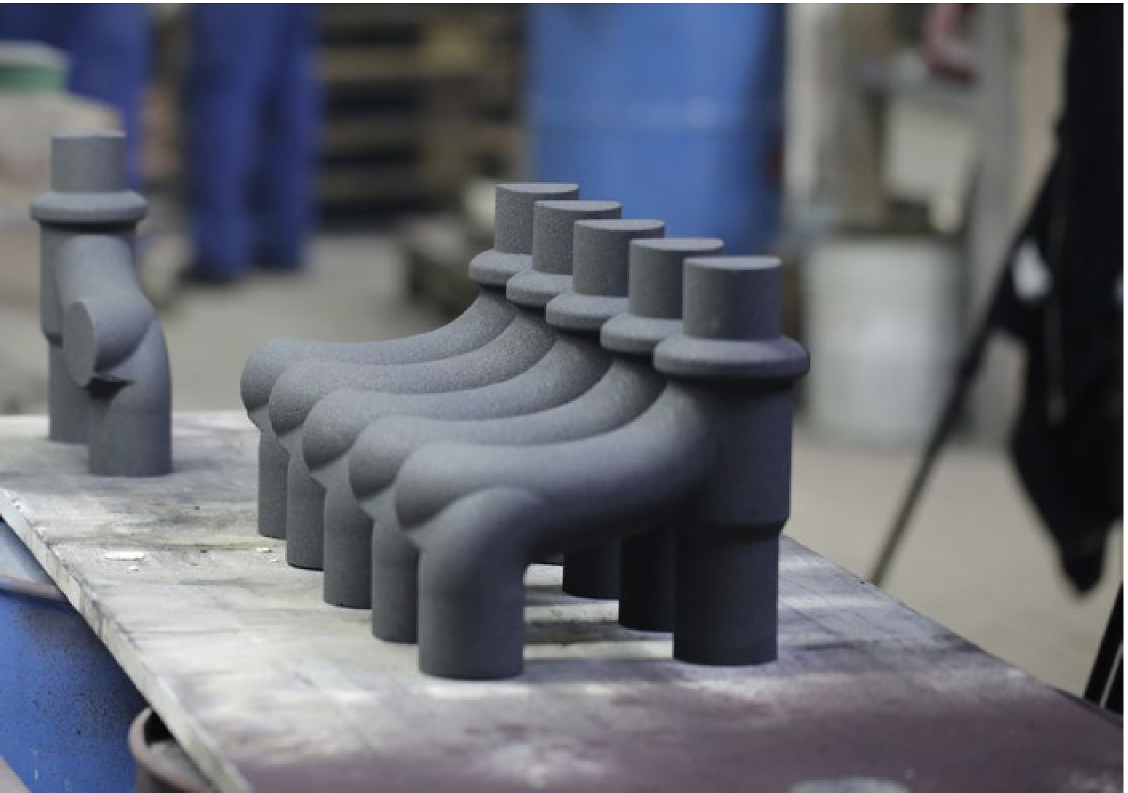
Possible decision-making criteria are:

- Which system has already been established in the company, e.g., in other departments or at other locations?
- Which system do customers and suppliers use?

A list of the main ISO and ASME standards for GD&T specification and verification can be found in the appendix to this document.







Application Note

GD&T analysis of a core box

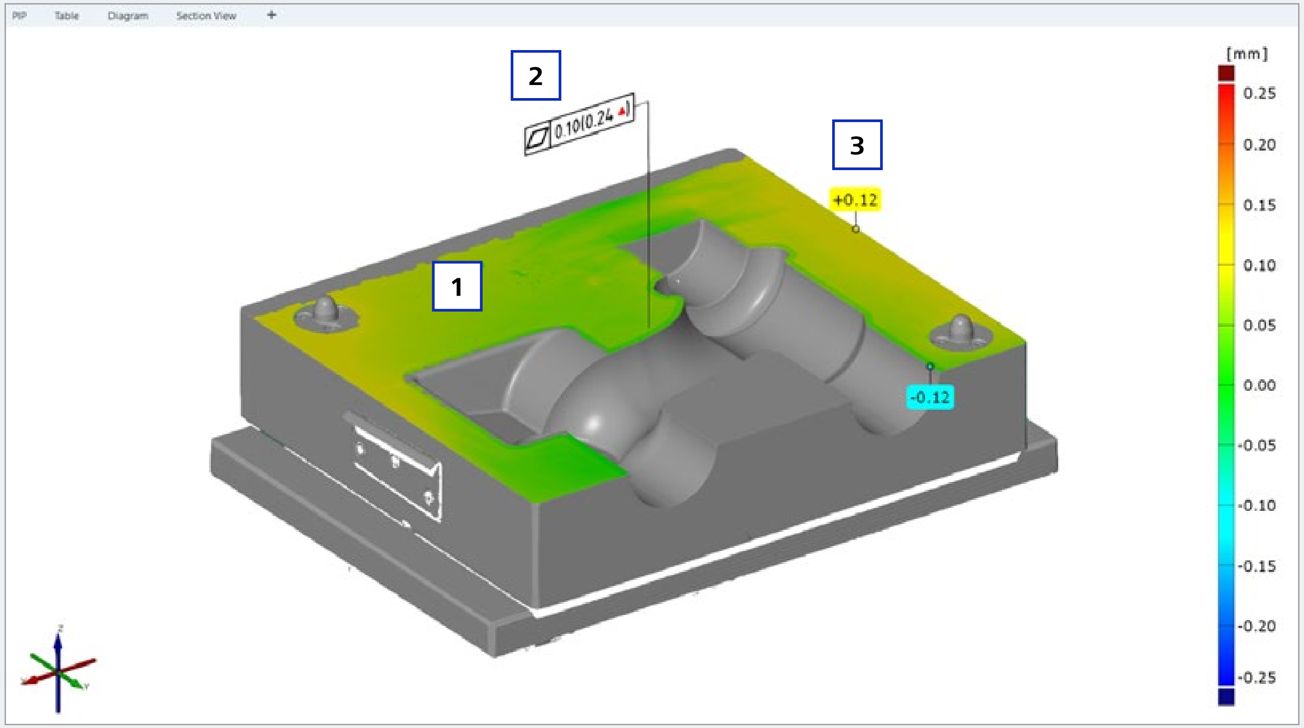
In foundries, work is done using what are known as “cores”. These are displacers that are put inside the casting mold. Cores help create hollow spaces in cast parts as well as complex object geometries. Cores are manufactured using core boxes, which are filled with a molding material (e.g., furan resin sand).

To guarantee the cores are manufactured properly, the shapes of the functional alignment elements in the core box must meet the design specifications. The alignment elements also must be arranged together properly, otherwise the two halves of the core box cannot be joined cleanly. If the two halves do not seal correctly, the sand used as the molding material will seep out of the core box during injection. Even the gas pumped into the core box to harden the sand will leak out.

The design engineer saves the prescribed GD&T in the drawing. The metrologist then measures the core boxes with a 3D measuring system. A software-based GD&T inspection is then conducted to verify compliance with the prescribed GD&T and, with it, the quality of the core box. ZEISS INSPECT is used, in which analysis is possible according to both ISO standards and ASME Y14.5.

Application note

Results in ZEISS INSPECT – flatness



GD&T inspection on a core box: flatness check – results display in ZEISS INSPECT

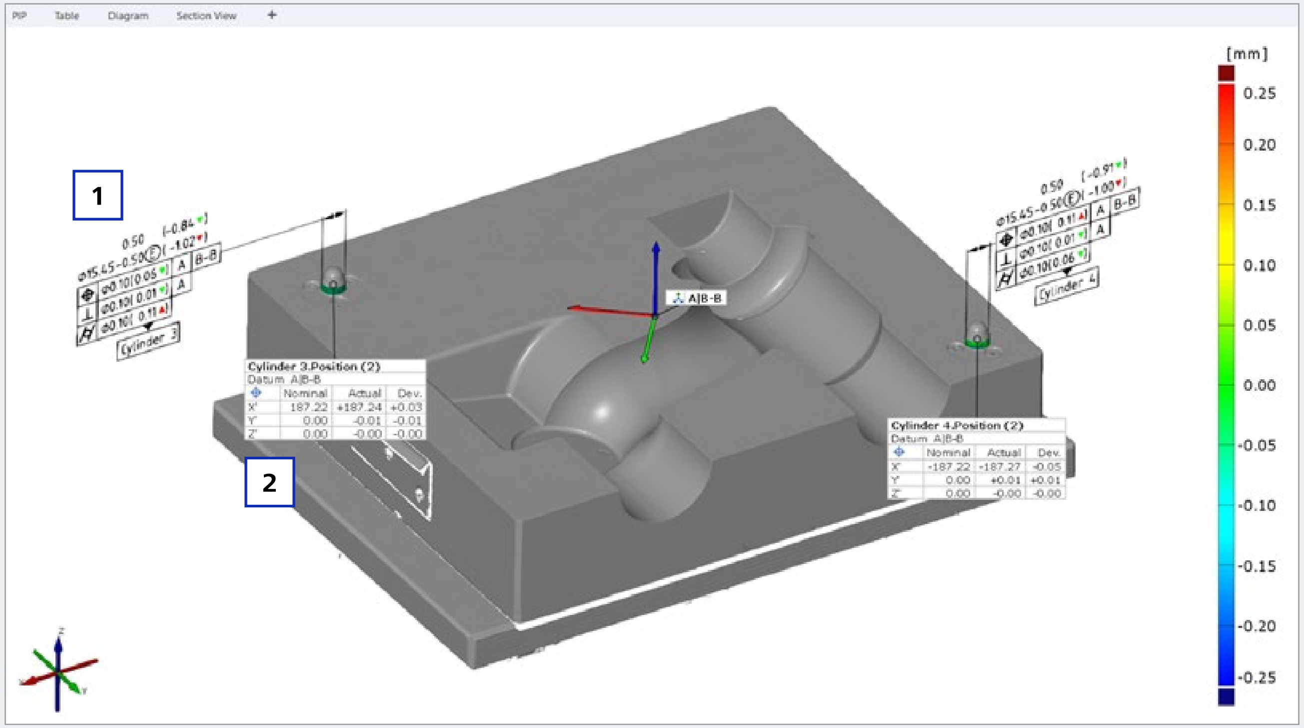
1 The user checks the plane used to align the halves of the core box and then decides accordingly whether they can be assembled seamlessly. The inspection result and any deviations are displayed in color.

2 According to the drawing specification, the measured surface between two parallel planes must be 0.10 mm apart. The check shows that this is not the case (red arrow): The permitted tolerance is exceeded. A distance of 0.10 mm would have been accepted, however, the distance here is 0.24 mm.

3 The software also shows minimum/maximum deviation labels. These represent the deviation values of the points with the farthest and closest distance to the middle of the tolerance zone.

Conclusion: The manufacturing supervisors recognize that the core box must be reworked, especially the areas highlighted with the minimum/maximum deviation labels. Otherwise the two core box halves will not seal cleanly and the core shooting will not proceed as desired.

Results in ZEISS INSPECT – position check

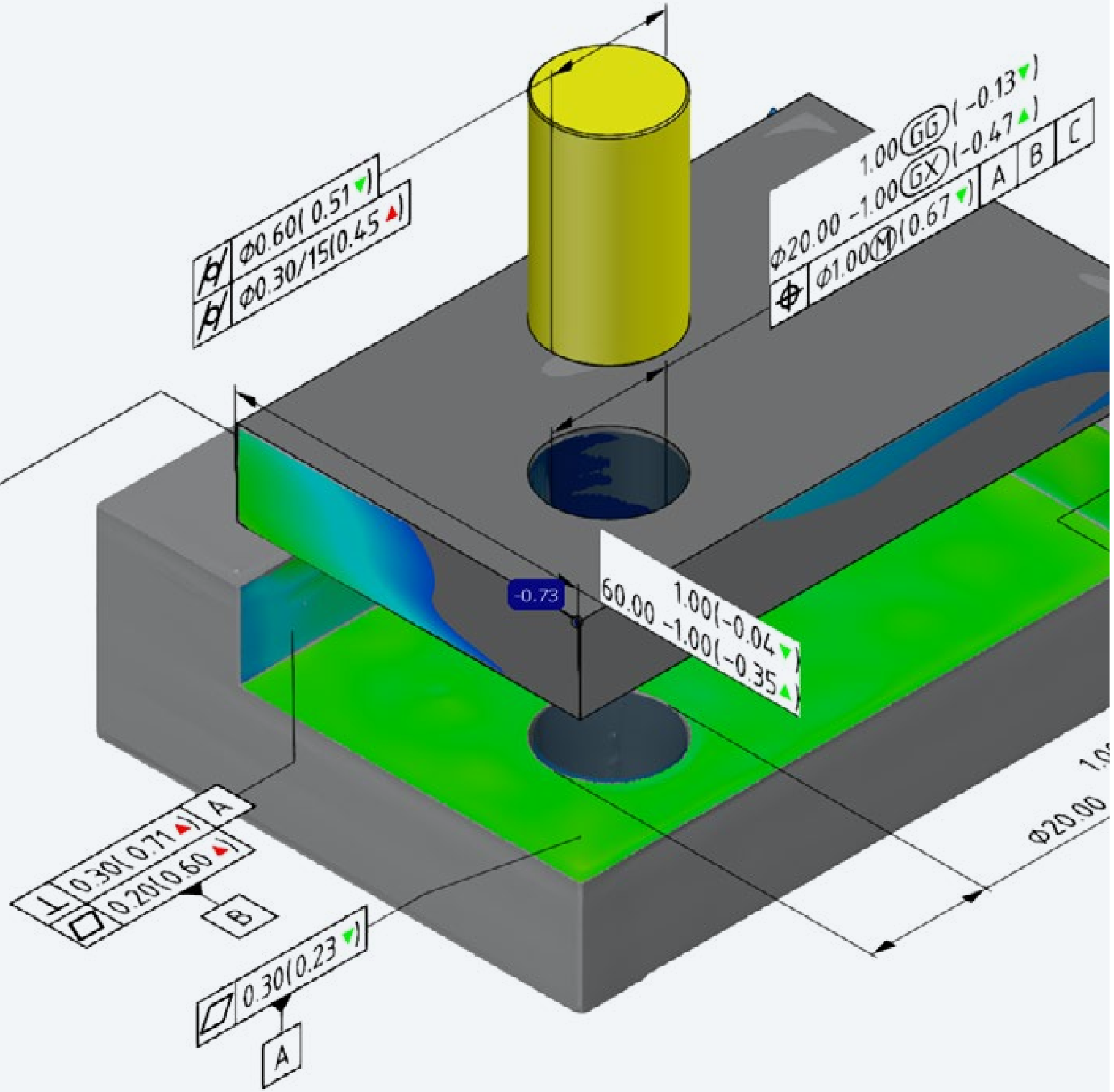


GD&T inspection on a core box: cylinder position check – results display in ZEISS INSPECT

1 Now the user checks whether the pins used to seal the core box halves meet the nominal specifications. The software displays the design specifications along with each inspection result:

- By entering the encircled “E” at the dimensional tolerance for Cylinder 3, the design engineer indicates that the geometric element must be checked with a minimum circumscribed element and a two-point size. This means the separate GD&T calculations for the pin cannot be considered independently of each other but must be set in relation to one another. The overall check shows that the tolerance zone is exceeded (red arrow).
- Checking the position of a line: The extracted median line (extracted axis) must lie within a cylindrical tolerance zone with a diameter of 0.10 mm. The specification is met (actual = 0.05 mm).
- Checking the perpendicularity of a line (axis) to a surface: The extracted median line (extracted axis) must lie within a cylinder with a diameter of 0.10 mm that is perpendicular to Datum Plane A. The specification is met (actual = 0.01 mm).
- Checking cylinder shape: The extracted cylinder lateral surface must lie between two coaxial cylinders with a radial distance of 0.10 mm. The specification is not met (actual = 0.11 mm).

2 The user sees an overview of the actual and nominal values for the extracted cylinder axis as well as the deviation between actual and nominal (Nominal, Actual, Deviation) in the X, Y and Z directions. This makes it clear in which orientation the position of the pin must be corrected in the manufacturing process.



eLearning

A beginner's guide to GD&T and further GD&T inspections

Those who would like to learn more about geometric dimensioning inspections can visit the ZEISS Quality Training Center and take the free eLearning module: ZEISS INSPECT (GD&T Analysis) Starter eLearning.

This module discusses the basic GD&T workflows in easy-to-understand concepts. You will not only gain insight into the theoretical principles but also become familiar with practical analytical options in ZEISS INSPECT.

Topics

- Use GD&T Quick Creation
- Visualize problematic areas with deviations
- Create and use datum systems for GD&T checks
- Create individual and combined GD&T checks
- Work with material requirements

To ZEISS Quality Training Center

Appendix

ISO and ASME Standards for GD&T Specification and Verification

The main ISO standards

ISO 1, Geometrical product specifications (GPS) – Standard reference temperature for geometrical product specification and verificationISO 286-1, Geometrical product specifications (GPS) – ISO code system for tolerances on linear sizes – Part 1: Basis of tolerances, deviations and fits

ISO 1101, Geometrical product specifications (GPS) – Geometrical tolerancing – Tolerances of form, orientation, location and run-out

ISO 1660, Geometrical product specifications (GPS) – Geometrical tolerancing – Profile tolerancing

ISO 2692, Geometrical product specifications (GPS) – Geometrical tolerancing – Maximum material requirement (MMR), least material requirement (LMR) and reciprocity requirement (RPR)

ISO 5458, Geometrical product specifications (GPS) – Geometrical tolerancing – Pattern and combined geometrical specification

ISO 5459, Geometrical product specifications (GPS) – Geometrical tolerancing – Datums and datum systems

ISO 14253-1, Geometrical product specifications (GPS) – Inspection by measurement of workpieces and measuring equipment – Part 1: Decision rules for verifying conformity or nonconformity with specifications

ISO 14253-2, Geometrical product specifications (GPS) – Inspection by measurement of workpieces and measuring equipment – Part 2: Guidance for the estimation of uncertainty in GPS measurement, in calibration of measuring equipment and in product verification

ISO 14405-1, Geometrical product specifications (GPS) – Dimensional tolerancing – Part 1: Linear sizes

ISO 14405-2, Geometrical product specifications (GPS) – Dimensional tolerancing – Part 2: Dimensions other than linear or angular sizes

ISO 14405-3, Geometrical product specifications (GPS) – Dimensional tolerancing – Part 3: Angular sizes

ISO 17450-1, Geometrical product specifications (GPS) – General concepts – Part 1: Model for geometrical specification and verification

ISO 17450-2, Geometrical product specifications (GPS) – General concepts – Part 2: Basic tenets, specifications, operators, uncertainties and ambiguities

The ISO GPS system contains over 100 standards in all.

ASME Y14.5

This system is not divided into individual standards. All concepts, principles and rules can be found in the current version, **ASME Y14.5:2018**.

ASME Y14.5.1 contains the mathematical definitions of the GD&T principles.

Carl Zeiss GOM Metrology GmbH
Schmitzstraße 2
38122 Braunschweig
Germany

Phone: +49 531 39029-0
info.optical.metrology@zeiss.com
www.zeiss.com/optical-metrology

EN_White paper GD&T Digital Version CZ-03/2022