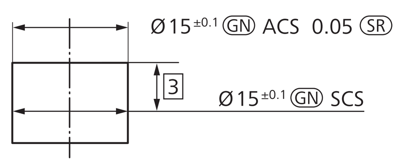
GPS – Geometric Product Specifications

ZEISS ACADEMY
METROLOGY

Sizes and modifiers

Sizes are dimensions of enclosed geometric features, e.g. diameter of cylinders and circles or distances of parallel surfaces. The type of metrology evaluation can be stipulated here by indicating specification modifiers.



LP	Local Point: analysis of the measurement points as a two-point dimension (based on the midpoint of a Gaussian circle)
GX	Global Maximum: evaluation of the measurement points as an inscribed circle / inscribed cylinder (MICI / MICY).
GN	Global Minimum: evaluation of the measurement points as a circumscribed circle / circumscribed cylinder (MCCI / MCCY).
GC	Global Chebyshev: evaluation of the measurement points as per the Chebyshev minimum method (MZCI / MZCY).
GG	Global Gauss: evaluation of the measurement points as a Gauss best fit (LPCI / LPCY).

So-called rank-order sizes can be indicated as a supplement to the modifiers:

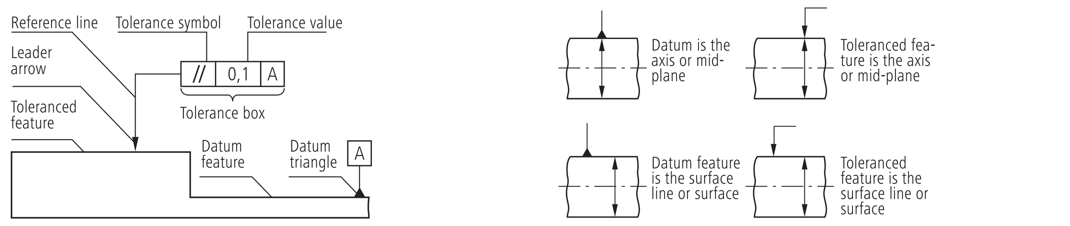
SX	Statistical Maximum: largest value of the measured values
SN	Statistical Minimum: smallest value of the measured values
SA	Statistical Average: mean value of the measured values
SM	Statistical Median: median of the measured values
SR	Statistical Range: range of the measured values
SD	Statistical Mid-Range: mean value of SX and SN

Additional information stipulates requirements for the measurement and evaluation:

ACS	Any Cross Section: inspection in any (possible) circular section.
SCS	Specific Cross Section: inspection only in the circular section (usually indicated by a theoretical dimension).
ALS	Any Longitudinal Section: inspection in each (possible) longitudinal section.
CZ	Common Zone: joint inspection of the characteristics in a common tolerance zone.
SZ	Seperate Zone: inspection of the characteristics independently of each other (in separate tolerance zones).
TED	Theoretically Exact Dimension: theoretical dimension without tolerance to indicate the ideal location, e.g. 25 or 60°.

Form and location tolerances

Form tolerances limit the deviations of an individual feature from its geometrically ideal form. Orientation, location and runout tolerances limit the errors of the mutual location of two or more features. One or more features can be specified as datum features.



○	Roundness (form tolerance): the tolerance zone is limited in the measuring plane perpendicular to the measuring axis by two concentric circles a distance t apart.
—	Straightness (form tolerance): the tolerance zone is limited by - a cylinder of diameter t (⌸ of an axis or a derived median line) - two parallel planes a distance t apart (⌸ of an edge) - two parallel lines a distance t apart (⌸ of a surface line).
⌭	Cylindricity (form tolerance): the tolerance zone is limited by two coaxial cylinders a distance t apart. ⌭ limits implicitly: ○ and ⌸ axis and ⌸ surface and ⏥ surface
⏥	Flatness (form tolerance): the tolerance zone is limited by two parallel planes a distance t apart. ⏥ limits implicitly: ⌸ surface or ⌸ derived median line

Unlike form tolerances, location tolerances almost always require one or multiple datums:

//	Parallelism (orientation tolerance): the tolerance zone is limited by two parallel lines or planes a distance t apart, which are parallel to the datum. // limits implicitly: ⏥ ⌸ surface or ⌸ derived median line/axis
⊥	Perpendicularity (orientation tolerance): the tolerance zone is limited by two parallel planes a distance t apart (or a cylinder with Ø t) which are perpendicular to the datum. ⊥ limits implicitly: ⏥ ⌸ surface or ⌸ derived median line/axis
∠	Angularity (orientation tolerance): the tolerance zone is limited by two parallel planes a distance t apart (or a cylinder with Ø t) which are at the defined angle to the primary datum. ∠ limits implicitly: ⏥ ⌸ surface or ⌸ derived median line/axis
⌖	Position (location tolerance): the tolerance zone is limited by a cylinder of diameter t coaxial to the datum (position with Ø sign). For additional tolerancing, see ISO 5458. ⌖ limits implicitly: ⌸ ⏥, if applicable also ⌭ ⊥ and ⏥ plane
⌯	Symmetry (location tolerance): the tolerance zone is limited by two planes symmetrical to the datum axis or datum plane at a distance t apart. ⌯ limits implicitly: ⌸ center plane ⏥ center plane if applicable ⏥ center plane
◎	Coaxiality or concentricity (location tolerances): the tolerance zone is limited by a cylinder or circle of diameter t which is coaxial or concentric to the datum. ◎ Coaxiality limits implicitly: ⌸ axis ⏥ axis ○ concentricity
⌒	Line profile (profile tolerance with and without datum): the tolerance zone is limited by two curves which envelop circles of diameter t whose mid-points are on a curve of geometrically ideal form. If applicable: a datum limits the position
⌓	Profile of a surface (profile tolerance with and without datum): the tolerance zone is limited by two surfaces which envelop spheres of diameter t whose mid-points are on a curve of geometrically ideal form. If applicable: a datum limits the location.
↗	Circular radial or axial run-out (run-out tolerances): the tolerance zone is limited by two concentric circles or parallel planes a distance t apart, which are located by datum A-B. ↗ Circular radial run-out limits: ○ ◎ ↗ Circular axial run-out limits: ⌸ measuring line
⌰	Total radial or axial run-out (run-out tolerances): the tolerance zone is limited by two concentric cylinders or parallel planes a distance t apart, which are located by datum A-B. ⌰ Total radial runout limits: ○ ⌭ ◎ ⌸ ⏥. ⌰ Total axial run-out limits: ⌸ ⏥

Additional drawing entries

In many cases additional, usually circled characters can be entered in the feature control frame next to the tolerance (and/or sometimes also next to a datum).

Ⓜ	Maximum material requirement: Ⓜ permits the addition of unused dimension tolerance portions to the toleranced form or location error. Example (simple case): cylinder diameter 6 mm and the axis straightness tolerance t = 0.5 mm (see image to the right). If the actual diameter is 5.0 mm, the straightness of the axis can deviate up to 1.5 mm.
Ⓛ	Minimum material requirement: Ⓛ enables the addition of unused dimension tolerance portions (away from the material side) to the toleranced form or location error. Example (simple case): cylinder diameter 6 mm and axis straightness tolerance t = 0.5 mm (see image to the right). If the actual diameter is 6.0 mm, the straightness of the axis can deviate up to 1.5 mm.
Ⓡ	Reciprocity requirement: the R-requirement enables the "reversal" of Ⓜ or Ⓛ, i.e. the addition of unused form and location tolerances to the dimension tolerance.
Ⓔ	Envelope requirement: as per ISO 8015, dimension tolerances and form and location tolerances must always be viewed independently of each other. By inputing Ⓔ on the dimension tolerance, the entire tolerance width, including form deviations, are limited to the dimensional tolerance. Thus, in the example to the right, the external envelope (dimension+form) may not exceed the diameter of 6.0 mm. If this is already utilized, e.g. by the dimensional tolerance, no more form deviations may occur.
Ⓐ	Axis as a toleranced feature: to illustrate that not the surface but rather the axis or center plane (center line) should be toleranced, a Ⓐ can be input in the drawing (in 3D drawings necessary).
Ⓕ	Free state: the inspection of the (elastic or plastic, non-rigid) workpiece must be performed in the unfixtured state (only formed by gravity) (as per ISO 10579).
UZ	Asymmetric tolerance zone (with profile form tolerances): the tolerance zone is moved by the value t from the material center outwards (in the example to the right, the zone is entirely outside of the material).
Ⓟ	Projected tolerance zone: the tolerance zone has been moved by t entirely outside of the workpiece in order e.g. inspect relevant locations for subsequent assembly later.

Datum and tolerance direction limitations

Datum features can also be limited in their configuration and direction of action:

><	Datum feature only acts as an orientation feature. The position coordinates are not considered.	⌖ Ø 0.2 A ><
[PL]	Plane: datum feature only functions as a plane. Other parameters of the datum feature (e.g. origin coordinates) are not considered.	⌖ 0.6 A[SL] ...
[SL]	Straight Line: datum feature functions only as a straight line. Other parameters of the datum feature (e.g. origin coordinates) are not considered.	A, 12°
[PT]	Point: datum feature functions only as a line. Other parameters of the datum feature (e.g. orientation information) are not considered.	
⟨//B⟩	Direction indicator: the tolerance should only be inspected in direction B. In the example to the right, the parallelism must only be inspected parallel to the datum B.	// 0.03 A ⟨//B⟩
⟨//B	Intersection plane indicator: the tolerance should only be inspected in the intersection plane B. In the example, straightness must be inspected perpendicular to B.	— 0.03 A ⟨⊥B
A↔B	Tolerance zone limitation: the tolerance must be inspected only in the area between A and B.	— 0.5 A↔B
t/...	Tolerance zone limitation: the tolerance must contain the value 0.5 in every section of the length 100 (in the example to the right).	— 0.5 / 100
A→B	Variable tolerance zone: the tolerance width changes from 0.3 mm (with A) linear up to 0.5 mm (with B).	— 0.3 - 0.5 A→B
⌀	Circumferential zone: the tolerance applies to all line and surface features surrounding the entire workpiece in the viewing plane.	⌓ 0.03 B

Tolerance indications for associations and filters

In addition, indications can be made in the tolerance box for the association (calculated best fit) of the features and for filtering, e.g. // 0.1Ⓧ G50 A or ○ 0.1 Ⓝ S50-150 or ○ 0.1 F3. The following applies:

Ⓧ	Inscribed feature: the toleranced and measured feature must be evaluated as an inscribed circle / inscribed cylinder (MICI / MICY).
Ⓝ	Circumscribed feature: the toleranced and measured feature must be evaluated as a circumscribed circle / circumscribed cylinder (MCCI / MCCY).
Ⓖ	Gaussian feature: the toleranced and measured feature must be evaluated as a Gaussian best fit (LPCI/LPCY).
Ⓒ	Minimum feature: the toleranced and measured feature must be evaluated in accordance with the Chebyshev minimum method (MZCI/MZCY).
Ⓣ	Tangential feature: the toleranced and measured feature must be evaluated as an external tangential feature (as per the Chebyshev minimum method) (OTPL).
G	Gaussian filtering: the standard Gaussian filter must be used as a digital filter. The indication "G50" means a low-pass filter with 50 waves per revolution. "G50-150" would be a band-pass filter.
S	Spline filtering: the spline filter must be used as a digital filter. The indication "S50" means a low-pass filter, "S50-150" means a band-pass filter with 50-150 waves per revolution.
F	Fourier analysis: the evaluation is performed using Fourier analysis. Here "F3" limits the analysis to the third harmonic vibration (orbiform curve form).

Important ISO standards for the GPS

ISO 1101	GPS – tolerances of form, orientation, location and run-out
ISO 5458	GPS – Position tolerancing
ISO 5459	GPS – Datum and datum systems
ISO 8015	GPS – Geometric tolerancing – Fundamentals – Concepts, principles, rules
ISO 2692	Form and position tolerancing, maximum material requirement
ISO 10579	GPS – Dimensioning and tolerancing – non-rigid parts
ISO 12180	GPS – Cylindricity
ISO 12181	GPS – Roundness
ISO 12780	GPS – Straightness
ISO 12781	GPS – Flatness
ISO 14405-1	GPS – Dimensional tolerancing – part 1: linear dimensions
ISO 14405-2	GPS – Dimensional tolerancing – part 2: dimensions other than linear sizes

AD	If a different standard or work standard becomes applicable for a technical drawing in addition to the GPS standards (or if these are replaced), this can be performed in the tolerance box by adding the "AD Name of the particular standard."

Carl Zeiss Industrielle Messtechnik GmbH
73446 Oberkochen/Germany

Phone: +49 (0) 7364/ 20-6336
Fax: +49 (0) 7364/ 20-3870
Email: info.metrology.de@zeiss.com
Internet: www.zeiss.com/imt

There is additional information on geometric product specifications (and the differences to the ASME standardization) in the seminars and books of the ZEISS Metrology Academy.
Books can be ordered here: probes.zeiss.com