GENERAL TOLERANCES IN THE GPS CONCEPT

PREFACE
The previously published articles about geometrical product specifications explained fundamental standards of the GPS concept required to describe geometrical characteristics.

- Geometrical product specifications (GPS) – an incomplete survey
- Geometrical product specifications (GPS) – ISO 8015 basic GPS standard
- Geometrical product specifications (GPS) – ISO 14405-1, the general GPS standard for dimensional tolerancing of linear sizes
- Geometrical product specifications (GPS) – consequences on the tolerancing of features of size
- Geometrical product specifications (GPS) – application of geometrical tolerancing to step dimensions

Note: All numerical data are given in mm without indicating this unit of measurement in the following.

MEANING OF GENERAL TOLERANCES
On the one hand, the eleventh principle or functional control principle of the basic standard ISO 8015 demands

"that the specification of a workpiece is complete when all intended functions of the workpiece are described and controlled with GPS specifications."

This is the reason why any functional geometrical characteristic is defined separately by means of the symbols and tolerances the GPS system provides. On the other hand, the third principle or definitive drawing principle says that

"all specifications shall be indicated on the drawing using GPS symbology. Consequently, requirements not specified on the drawing cannot be enforced."

Hardly any drawing does not refer to the tolerance classes of the ISO 2768 standard for general tolerances. However, I have recently received many queries about how these “old” standards (ISO edition issued in 1989) agree with current general standards of the GPS system. This article answers this and many more questions relating to general tolerances.
Interpreting this principle, this means that drawings must be subject to complete dimensioning and tolerancing to communicate all specifications with supplier (manufacturer) and customer (recipient). General tolerances close the gap between concrete tolerancing of single characteristics (ensuring functional control) and the tolerancing of all characteristics (ensuring completeness in consideration of economic producibility).

Declaring tolerance classes based on general tolerance standards, you may supply any geometrical characteristic with tolerances. Since general tolerances have been developed or defined for specific production techniques, they weigh and agree on an economic production "along the way".

**HOW THESE STANDARDS ALIGN WITH THE GPS SYSTEM**

Standards on general tolerances are at the lowest level in the GPS hierarchy. They are referred to as complementary or supplementary standards. Their definitions thus supplement the information given in the basic GPS standards and all the other standards relating to geometric elements such as size, form, location, run-out, roughness and edges of bodies. The rules and principles of superior standards always apply unless a general tolerance standard suspends them.

**Example**
The sixth principle of ISO 8015 defines 20°C as a reference temperature for characteristics whose tolerance is indicated on drawings (nominal sizes, nominal conditions) and for associated measurement results (actual characteristics). DIN 16742:2013 “Plastic moulded parts – Tolerances and acceptance conditions” specifies a standard atmosphere of 23°C ± 2 K and a relative air humidity of 50% ± 10%. These specifications refer to ISO 291:2008 “Plastics - Standard atmospheres for conditioning and testing”. Both standards are actually not part of the GPS system even though the terminology and content of DIN 16742 was adapted to the GPS standards of 2013. It is thus correct to apply the "ISO 8015 DIN EN ISO 291: DIN 16742" indication on drawings of plastic parts.

Design engineers and operators therefore have to be well-acquainted with the contents of these standards (they need to know more than just the numerical value of the tolerance).

**REFERENCE TO PRODUCTION TECHNIQUES**

Standards about general tolerances focus on accuracies specific production techniques are able to achieve when ensuring the "usual shop-floor" accuracy (capability of the applied manufacturing processes and machines in consideration of the staff's skills). Referring to general tolerances, a company thus indicates to prefer a specific production technique without prescribing it (the following overview does not claim to be exhaustive).

<table>
<thead>
<tr>
<th>Process</th>
<th>Reference</th>
</tr>
</thead>
<tbody>
<tr>
<td>Casting</td>
<td>ISO 8062-3:2007 General dimensional and geometrical tolerances for moulded parts - Part 3: Tolerances and machining allowances for castings</td>
</tr>
<tr>
<td>Die forging</td>
<td>EN 10243-1:2007 Steel die forgings - Tolerances on dimensions - Part 1: Drop and vertical press forgings</td>
</tr>
<tr>
<td></td>
<td>EN 10243-2:2005 Steel die forgings - Tolerances on dimensions - Part 2: Upset forgings made on horizontal forging machines</td>
</tr>
<tr>
<td></td>
<td>EN 586-3:2002-02 Aluminium and aluminium alloys - Forgings - Part 3: Tolerances on dimensions and form</td>
</tr>
<tr>
<td>Thermal cutting</td>
<td>ISO 9013:2003 Thermal cutting - Classification of thermal cuts - Geometrical product specification and quality tolerances</td>
</tr>
<tr>
<td>Stamping</td>
<td>DIN 6930-2:2011-10 Stamped steel parts - Part 2: General tolerances</td>
</tr>
<tr>
<td>Metal cutting and forming</td>
<td>ISO 2768-1:1989 General tolerances — Part 1: Tolerances for linear and angular dimensions without individual tolerance indications</td>
</tr>
<tr>
<td>Plastic coating</td>
<td>ISO 2768-2:1989 Geometrical tolerances for features without individual tolerance indications</td>
</tr>
<tr>
<td>Rubber moulding</td>
<td>DIN 16742 Plastics moulded parts - Tolerances and acceptance conditions</td>
</tr>
</tbody>
</table>

**Table 1**
GPS - GENERAL TOLERANCES IN THE GPS CONCEPT

COMPLETENESS

As the table above shows, a reference to the general tolerance standard does not automatically guarantee complete tolerancing. Some standards only cover tolerances of size or form. In case there are no general tolerances available for direction, location or run-out, you may either refer to ISO 2768-2 or you may apply general tolerancing principles using the tolerance of profile. Switching to ISO 2768-2, you have to select a tolerance class that meets functional requirements and best suits the preferred manufacturing procedure that is neither a metal cutting or forming procedure.

CONSISTENCY

When you have to refer to more than one general tolerance standard, one and the same characteristic might have different tolerances. In these cases, the twelfth principle of ISO 8015 provides you with the following rule:

“If more than one general GPS specification is indicated in or near the title block and these specifications are contradictory, they shall be supplemented with an explanation to make it clear to which characteristics each general GPS specification applies (...). In the case of contradictory general GPS specifications (...) the most permissive one applies.”

TOLERANCING OPTIONS – EXAMPLE

BASED ON GENERAL TOLERANCES

ACCORDING TO ISO 2768

The example below (figure 3) illustrates an example without individual tolerancing. A turned part is subject to geometrical tolerancing. It is associated with tolerance class m for linear sizes and K for tolerances of form and location. The listed tolerances were taken from the indicated standards. These tolerances cover the following “regions” of the part and thus form the restricted area of the material (see figure 4).
Based on the example given in Figure 3, you apply the drawing indications as follows.

**Tolerancing ISO 8015**

The following applies to all individual geometric elements not dimensioned or tolerated directly.

- $0.2\,\text{A}\,\text{B}\,\text{C}$
  - to all real geometric elements
  - to all axes of rotationally symmetric geometric elements
  - to all medium planes of pairs of parallel opposite surfaces

referring to the datum system and the point coordinates of the current CAD data set.

**Figure 6**

By applying a tolerance of profile, you lose the classification of tolerances depending on linear dimensions. The tolerance range is symmetrical to the desired nominal form independent of what are supposed to be "dimensions". Due to a profile tolerance of 0.2 mm, all diameters have a diameter tolerance of ± 0.2 mm.

Figure 8 shows the tolerance range of the restricted area of material.

**Figure 8**

For the last 15 years, tolerances of profile have been applied to define general tolerances, especially in plastics processing and in sheet metal forming. Even the DIN 16742 standard quoted above applies the profile tolerances as general tolerances. You may indicate the tolerances on drawings as follows.

Tolerancing options – example based on a tolerance of profile according to ISO 1101

For the last 15 years, tolerances of profile have been applied to define general tolerances, especially in plastics processing and in sheet metal forming. Even the DIN 16742 standard quoted above applies the profile tolerances as general tolerances. You may indicate the tolerances on drawings as follows.
General tolerances shall simplify the design of parts based on drawing indications. Characteristics of form, location and orientation produced with the usual shop-floor accuracy are tolerated based on a tolerance class specified in the above-mentioned standards specific to the respective production technique. When you select the required tolerance class, it is important to consider the functional requirements primarily. You shall always have a look at the tolerance tables given in the standards to see the symmetrical deviations for sizes and tolerances referring to form and location. These tables help you define suitable or necessary tolerance classes. Tolerances smaller than the ones specified in the tables shall always be indicated for the respective linear size or geometric element on the drawing.

It often happens that the title block of a drawing gives a global agreement of typical tolerance classes by default. This agreement applies to an entire company or an entire group and does not relate to the specific product shown on the drawing at all. You shall thus treat this information with caution and check at least whether these global specifications are acceptable for this very product or whether comprehensive individual tolerancing is required.

It is important to check whether the relative small tolerances that do not depend on the expansion of the geometric elements observed (range of the nominal feature of size) are not too strict (see table below), especially for general tolerances of symmetry and run-out.

<table>
<thead>
<tr>
<th>Property</th>
<th>Tolerance class</th>
<th>Nominal range</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td></td>
<td>Up to 10</td>
</tr>
<tr>
<td></td>
<td></td>
<td>Over 10 to 30</td>
</tr>
<tr>
<td></td>
<td></td>
<td>Over 30 to 100</td>
</tr>
<tr>
<td></td>
<td></td>
<td>Over 100 to 1000</td>
</tr>
<tr>
<td></td>
<td></td>
<td>Over 1000 to 3000</td>
</tr>
<tr>
<td>H</td>
<td></td>
<td>0.5 (!)</td>
</tr>
<tr>
<td>H</td>
<td></td>
<td>0.1 (!)</td>
</tr>
<tr>
<td>K</td>
<td></td>
<td>0.2 (!)</td>
</tr>
<tr>
<td>L</td>
<td></td>
<td>0.5 (!)</td>
</tr>
</tbody>
</table>

Table 2
Independent of the selected tolerance class, the annex of ISO 2768-1 and of ISO 2768-2 specifies that the function of a part often allows a geometrical tolerance larger than the general tolerance values. When the general tolerance of any geometric element of the part is (occasionally) exceeded, this does not impair the function of the part.

This is, however, a rather explosive statement since it implies that generally tolerated geometrical characteristics are not relevant to the function of a part and thus do not indicate any necessity to be controlled. However, inspection planners and/or quality inspectors should not follow this statement blindly but are supposed to use their technical knowledge. At least in the product/process release phase (initial sampling) you definitely have to control generally tolerated features of size to prove that you keep the “usual shop-floor” accuracy.

As opposed to features of size, the necessity to control generally tolerated characteristics of form and location is considerably more difficult. Since you do not have any necessary symbols, such as straightness, evenness, rectangularity, symmetry and coaxiality, required for the coding of functional requirements and since technical drawings do not even indicate any clear datum, there are plenty of possible geometrical tolerances. A simple cube as shown below formally indicates the tolerances of form and location according to ISO 2768-2.

<table>
<thead>
<tr>
<th>Geometric elements</th>
<th>Geometric properties and tolerances</th>
</tr>
</thead>
<tbody>
<tr>
<td>6 faces</td>
<td>6 tolerances of evenness</td>
</tr>
<tr>
<td></td>
<td>6 tolerances of parallelism (by switching datums)</td>
</tr>
<tr>
<td></td>
<td>24 tolerances of rectangularity (by switching datums)</td>
</tr>
<tr>
<td>12 edges</td>
<td>12 tolerances of straightness</td>
</tr>
</tbody>
</table>

It is thus not possible to control these features, not even within the scope of initial sampling. What are the consequences?

• Design engineers shall indicate any tolerances of form and location relevant to the function of the part by using symbols and definitions based on ISO 1101. This leads to a necessity to control characteristics, at least within the scope of initial sampling. If these characteristics are supposed to be special characteristics, they need to become part of production control plans or test plans as characteristics that have to be controlled. Design engineers shall always cover their back and confirm the part specification by referring to the general tolerances of form and location (complete tolerancing).

• Inspection planners and inspectors are to consider constellations of geometric elements that are striking or typical in their opinion (coaxialities between inside and outside cylinders in case of cylindrical workpiece, symmetry constellations in case of prismatic workpieces). It only applies to geometric elements that have not been tolerated individually and these elements shall at least be considered in initial sampling.

**NECESSITY TO CONTROL GENERALLY TOLERANCED CHARACTERISTICS – TOLERANCING ACCORDING TO DIN 16742 FOR PLASTIC MOULDED PARTS**

The text in chapter 5.3 of this DIN standard gives a clear and definitive statement about the necessity to control.

Acceptance dimensions are all directly tolerated characteristics. All dimensions with general tolerances are not considered in the test record.

Chapter 5.6 adds:

Free form surfaces shall be specified with a profile form tolerance. The verification shall be coordinated.

**NECESSITY TO CONTROL GENERALLY TOLERANCED CHARACTERISTICS – TOLERANCING BASED ON PROFILES ACCORDING TO ISO 1101**

If profile tolerancing applies as a general tolerance for irregular surfaces, we will always assume that it is impossible to give a definitive statement about the necessity to control these characteristics. This is due to vast number of surface points tolerated like this. However, if this is your intention all the same, you will have to define the datum system with clear datums for a reproducible comparison with the nominal geometric data of the CAD model. On the other hand, you have to use selected datums of the generally tolerated surface as measuring points. If all these measuring points are within the demanded profile tolerance zone, we will assume that any other surface points will be in the specification, too. It is not unusual to define selected measuring points as special characteristics in order to agree on the necessity to control such general tolerances.
HANDLING GENERAL TOLERANCES AT THE SUPPLIER-CUSTOMER (MANUFACTURER-CONSUMER) INTERFACE

Even though design engineers are glad to use general tolerances to assist them with their work, these general tolerances, however, might turn out to be insidious in quality assurance. On the one hand, this is due to the fact that is often rather unclear whether these characteristics need to be controlled. On the other hand, the known standards on general tolerances specify that workpieces with characteristics where the general tolerances are not complied with may only be rejected (automatically) if the function is impaired. This is at least what ISO 2768 and DIN 16742 say almost verbatim and what ISO 13920 basically means. Independent of how an “automatic” rejection works in practice from a technical and organisational perspective, even a “usual” complaint about such violations will prompt the parties involved to give more or less detailed reasons for this malfunction. These parties might thus pass the complaint back and forth.

CONCLUSION

The GPS concept still supports the allocation of general tolerances to geometrical characteristics based on production techniques and thus offers the tolerancing principles of ISO 1101 for free-form surfaces. These principles provide design engineers with reasonable tools that have been tested for decades to ensure a complete tolerancing concept based on best practice. Expert groups, however, frequently demand a drawing review to check drawings for completeness, unambiguity and GPS-compliant specifications. Staff reviewing a drawing shall have in-depth knowledge of GPS standards and care for the application of the latest rules and standards.

The verification of generally tolerated characteristics is, however, rather vague since the given complementary GPS standards only consider a product that is defective since it violates general tolerances to be a defect as to quality with restrictions. The rejection of such products is not always permissible. It is thus hard to define the necessity to control these characteristics based on standards. If you want to clarify the situation nonetheless, you shall add a quality agreement to the supply agreement or indicate some further specifications on the drawing.

These considerations lead to the following conclusions.

- All characteristics controlling the function of a part have to be dimensioned and tolerated directly to communicate mandatory verification requirements.
- Design engineers shall always indicate general tolerances for the reasons discussed in this article.
- When the supplier submits a quotation, the supplier shall focus on the feasibility (producibility) of the single tolerated characteristics. The next step is to assess whether the usual shop-floor accuracy agrees with the general tolerance class applied on the drawing.
- The quality agreement between manufacturer and recipient may exclude the rejection paragraph of the respective general tolerance standard or make it more palpable.
- The agreement may even give more details about the necessity to control generally tolerated characteristics. However, we recommend you provide an indication directly on the drawing.