WHITEPAPER

THE ROLE OF GEOMETRIC DIMENSIONING & TOLERANCING (GD&T) AND HOW TO CAPITALISE ON ITS VALUE PROPOSITION
Contents

Executive Summary .................................................................................................................................................. 3

Section 1: Benefits of GD&T .............................................................................................................................. 3

Section 2: Differences between ASME and ISO .................................................................................................... 3

Section 3: GD&T Standards Supported by PC-DMIS .......................................................................................... 3

Section 1: Benefits of GD&T ................................................................................................................................ 4

Section 2: Differences between ASME and ISO (terms and definitions) ................................................................ 5

Size tolerances: ASME Y14.5 compared to ISO 14405-1 and ISO 5459 ................................................................. 5

ASME Y14.5 – 2009 .............................................................................................................................................. 5

ISO 14405-1: 2010 with envelope requirement specified  ....................................................................................... 6

Profile tolerances: ASME Y14.5 compared to ISO 1101 ....................................................................................... 6


ISO 1101: 2004 ..................................................................................................................................................... 7

ISO 1101: 2012 (added support for unequally disposed profile) ............................................................................ 7

Section 3: GD&T Standards Supported by PC-DMIS and examples for features of size, material condition
modifiers, simultaneous requirements and Profile tolerances .............................................................................. 8

PC-DMIS support for features of size ..................................................................................................................... 8

Geometric tolerance with modifier ...................................................................................................................... 9

Datum feature with modifier .................................................................................................................................... 9

Calculating Maximum Material Boundary (MMB) ................................................................................................. 9

Example of a Position tolerance with a modifier on secondary datum ................................................................. 11

Simultaneous requirement: Position and Profile ................................................................................................... 12

Simultaneous requirement (multiple FCFs referencing the same DRF) ............................................................... 13

Simultaneous datum shift .................................................................................................................................. 13

PC-DMIS Simultaneous Evaluation ..................................................................................................................... 14

Simultaneous Report ......................................................................................................................................... 15

Datum Shift applied simultaneously to Position and Profile FCFs .................................................................... 16

Example of PC-DMIS report for Profile per ASME and ISO ............................................................................ 17

ASME Y14.5 - Two actual values (MAX and MIN) ............................................................................................ 17

ISO 1101 - Single actual value MEAS = largest MAX or MIN x 2 .................................................................... 17
Executive Summary

This whitepaper describes the engineering standards and benefits of Geometric Dimensioning and Tolerancing (GD&T), along with the supported attributes in the PC-DMIS measurement software.

Section 1: Benefits of GD&T

‘Benefits of GD&T’ discusses how manufacturing processes benefit from GD&T:

- Geometric tolerances vs. plus and minus tolerances
- Datum features with modifiers providing additional tolerance
- GD&T as an international language that provides concise specifications, with one interpretation that ensures correlation with suppliers globally and parts that meet functional requirements

Section 2: Differences between ASME and ISO

‘Differences between ASME and ISO’ explains the different GD&T standards and shows examples of where terms are different but have the same definition (features of size) and examples of the same geometric tolerance but different definitions (profile):

- Features of size – ASME Y14.5 compared to ISO 14405-1: 2010
- Profile tolerance – ASME Y14.5 compared to ISO 1101 (2004 and 2012)

Section 3: GD&T Standards Supported by PC-DMIS

‘GD&T Standards Supported by PC-DMIS’ shows examples of GD&T attributes for:

- Features of Size
- Material Condition Modifiers
- Calculating and applying Datum Shift
- Simultaneous requirements for Position and Profile tolerances
- Calculating and report actual values for Profile
Section 1: Benefits of GD&T

Major industries have become more global, resulting in the decentralisation of design, manufacturing and inspection, demanding the dimension and tolerance requirements to more precisely state the functional requirements of the produced parts. To accomplish this, geometric tolerances need to replace the traditional plus and minus tolerancing practices used to control form, orientation, location and profile of part features. The use of plus and minus tolerancing is reserved for size dimensions for features of size. Section 2 provides the common definitions for specific size characteristics controlled with plus and minus tolerances per ASME and ISO and identifies the differences. Section 2 also provides definitions for Profile tolerances per ASME and ISO and identifies the differences.

Geometric tolerances controlling orientation, location and profile require reference to a Datum Reference Frame (DRF) which represents the function of the part in the assembly with its mating parts. Datum features of size specified with material condition modifiers for maximum material boundary (MMB) or least material boundary (LMB) allow additional tolerance for the considered features when the datum feature departs from its maximum material condition (MMC) or least material condition (LMC) state. The additional tolerance from the datum feature's departure from its MMC or LMC size is not added to the tolerance zone(s) for the feature(s) referencing the DRF, which is a common misunderstanding in industry. This additional tolerance is applied directly to the datum feature and allows the degrees of freedom constrained by the datum to be released and translated and/or rotated within the MMB or LMB boundary. Section 3 provides an example of calculating an MMB and shifting the datum feature within this boundary, until the deviation of the considered features are minimised. Section 3 also provides definitions and an example of the simultaneous requirement for position and profile tolerances.

GD&T is defined in two standards, ASME Y14.5 and ISO 1101. Drawings produced according to these standards yield precise definitions for each of the 14 geometric tolerance types used to control form, size, orientation and location, resulting in a single, precise interpretation of the part specifications. This ensures parts produced globally and in conformance to the drawing specifications meet the functional requirements of the final product.

In summary, GD&T provides the tools for designers to specify the maximum tolerances required for the function of the final product in its assembled state. Geometric tolerances referencing DRFs represent how parts fit with the mating parts in the assembly, which ensures that when parts conform to their geometric tolerances they will function as designed. Drawings produced in accordance to GD&T standards (ASME and ISO) will have one precise interpretation regardless of where the parts are produced globally.
Section 2: Differences between ASME and ISO (terms and definitions)

Size tolerances: ASME Y14.5 compared to ISO 14405-1 and ISO 5459

Features of Size (internal and external): A single cylinder, sphere, circle and two opposed parallel planes.

The default definitions for features of size per ASME are the same in ISO, but only when the envelope requirement is specified (Size ISO 14405). Note: The terms for some of the size characteristics per ASME and ISO are different but their definitions are the same.

Figure 1 identifies each of the specific size characteristics as defined in ASME Y14.5 – 2009.

Figure 1

ASME Y14.5 – 2009
- Local size: Any individual distance at any cross section of a feature of size.
- Unrelated actual mating envelope: A similar perfect feature counterpart expanded within an internal feature or contracted about an external feature, and not constrained to any datum(s).
- Related actual mating envelope: A similar perfect feature counterpart expanded within an internal feature or contracted about an external feature while constrained either in orientation or location or both to the applicable datum(s).

Figure 2 identifies each of the specific size characteristics with the corresponding ISO terms as defined in ISO 14405-1: 2010. Note: ISO 14405-1 does not cover datum features of size; this is defined in ISO 5459: 2011.
ISO 14405-1: 2010 with envelope requirement specified

- Local size (two-point): The distance between two opposite points taken on the feature of size.
- Global size: The size of the associated feature established from the extracted feature(s) with maximum inscribed criterion (internal feature) or minimum circumscribed criterion (external feature).
- There is no ISO term for this however ISO 5459: 2011 provides a graphical example (same as ASME) and a brief definition: Second associated feature with orientation constraint from the first associated feature.

The following size specifications are not supported by ASME and are only supported by ISO. Refer to ISO 14405-1: 2010 for definitions.

- Calculated size
  - Circumference, area and volume diameter
- Rank-order size
  - Maximum, minimum, average and median size
- There are 14 additional specification modifiers that provide specific size types not covered by the envelope requirement modifier
  - LP, LS, GG, GX, GN, CC, CA, CV, SX, SN, SA, SM, SD and SR

Profile tolerances: ASME Y14.5 compared to ISO 1101

ASME Y14.5 – 2009 and Y14.5.1M – 1994

- Bi-lateral and unequally disposed profile tolerances have two actual values, one for the largest deviation in the plus material direction and one for the largest deviation in the minus material direction. No single value can be compared to the tolerance in the feature control frame (FCF) to determine conformance.
ISO 1101: 2004
- Bi-lateral profile tolerance has a single actual value that can be compared to the tolerance in the feature control frame to determine conformance. Single actual value = largest deviation (amongst plus and minus material directions) x two.

ISO 1101: 2012 (added support for unequally disposed profile)
- Bi-lateral and unequally disposed profile tolerances have a single actual value that can be compared to the tolerance in the feature control frame to make a conformance decision. Single actual value = largest deviation (amongst plus and minus material directions) x two.
- Refer to ISO 1101: 2012 for definition of unequally disposed profile.
Section 3: GD&T Standards Supported by PC-DMIS and examples for features of size, material condition modifiers, simultaneous requirements and Profile tolerances

PC-DMIS supports the following national and international standards:

- ASME Y14.5 – 2009 Dimensioning and Tolerancing
- ASME Y14.5.1M – 1994 Mathematical Definition of Dimensioning and Tolerancing Principles
- ISO 1101 – 2004 Geometrical Tolerancing
- ISO 14405-1: 2010 Linear Sizes – Envelope requirement

Currently adding PC-DMIS support for revisions to the following standards:

- ISO 1101 - 2012 (revisions will be supported in PC-DMIS 2016 – 2017)
- ISO 14405 - 1 - 2010 (revision will be supported in PC-DMIS 2016 - 2017)

PC-DMIS support for features of size

Cylinder, sphere, circle and width features (internal and external) support the following math type: Least squares, maximum inscribed and minimum circumscribed.

Examples of math type for circles:

Least Squares  Minimum Circumscribed  Maximum Inscribed

Figure 5
Geometric tolerance with modifier

![Image of geometric tolerance with modifier](image)

Figure 6

0.376 – URAME (actual hole diameter)  0.370 – MMC diameter  0.006 – Bonus Tolerance

Datum feature with modifier

![Image of datum feature with modifier](image)

Figure 7

**Calculating Maximum Material Boundary (MMB)**

Maximum material boundary (MMB) = MMC size plus the applicable geometric tolerance. Primary datum MMB is just the MMC size and does not include the geometric tolerance. The secondary datum MMB is the MMC size plus the orientation tolerance and the tertiary MMB is the MMC size plus the position tolerance.
Datum shift is the amount the datum can translate and/or rotate within the MMB.

PC-DMIS calculates a datum bonus value which represents the MMB for the axis of the related actual mating envelope of the datum feature. The datum feature axis is allowed to shift within this boundary while simultaneously shifting the geometric tolerance zones, minimising the deviations of the considered features.
Example of a Position tolerance with a modifier on secondary datum

Figure 10

Means this:

Figure 11

The axis of datum D and the four .010 diameter position tolerance zones can translate simultaneously within the maximum material boundary of datum D axis.
Simultaneous requirement: Position and Profile
When multiple Position or Profile FCFs reference the same DRF and less than 6 DOF are constrained, they must be evaluated simultaneously.

![Figure 12](image1)

When multiple Position or Profile FCFs reference the same DRF and MMB is specified on 1 or more datum features, they must be evaluated simultaneously.

![Figure 13](image2)

When an FCF has the notation SEP REQT beneath it, simultaneous evaluation is not required for that FCF.
Simultaneous requirement (multiple FCFs referencing the same DRF)

The axis of datum E and the tolerance zones of the three FCFs must be translated simultaneously within the maximum material boundary of Datum E. The tolerance requirements must be meet simultaneously with a single common datum shift.
PC-DMIS Simultaneous Evaluation

After creating all the feature control frames for position and profile, the user is required to open the Simultaneous Evaluation dialogue window and select the FCFs that require simultaneous evaluation.

Each of the selected FCFs in the measurement routine will be unmarked and the Simultaneous Evaluation command will group the FCFs in the report and apply the datum shift simultaneously to all the selected FCF commands.

PC-DMIS Simultaneous Evaluation supports multiple position or profile FCFs or combining multiple position and profile FCFs.
Simultaneous Report

<table>
<thead>
<tr>
<th>Feature</th>
<th>MEAS</th>
<th>NOMINAL</th>
<th>+TOL</th>
<th>-TOL</th>
<th>TRAY</th>
<th>BONUS</th>
<th>OUTTOL</th>
</tr>
</thead>
<tbody>
<tr>
<td>CYL 1</td>
<td>1.0055</td>
<td>1.0000</td>
<td>0.0010</td>
<td>0.0035</td>
<td>0.0155</td>
<td>0.0000</td>
<td></td>
</tr>
<tr>
<td>CYL 2</td>
<td>1.0052</td>
<td>1.0000</td>
<td>0.0010</td>
<td>0.0052</td>
<td>0.0152</td>
<td>0.0000</td>
<td></td>
</tr>
<tr>
<td>CYL 3</td>
<td>1.0054</td>
<td>1.0000</td>
<td>0.0010</td>
<td>0.0054</td>
<td>0.0154</td>
<td>0.0000</td>
<td></td>
</tr>
<tr>
<td>CYL 4</td>
<td>1.0053</td>
<td>1.0000</td>
<td>0.0010</td>
<td>0.0053</td>
<td>0.0153</td>
<td>0.0000</td>
<td></td>
</tr>
<tr>
<td>CYL 5</td>
<td>0.3750</td>
<td>0.3750</td>
<td>0.0050</td>
<td>0.0030</td>
<td>0.0059</td>
<td>0.0000</td>
<td></td>
</tr>
<tr>
<td>CYL 6</td>
<td>0.3750</td>
<td>0.3750</td>
<td>0.0050</td>
<td>0.0050</td>
<td>0.0050</td>
<td>0.0000</td>
<td></td>
</tr>
<tr>
<td>CYL 7</td>
<td>0.3750</td>
<td>0.3750</td>
<td>0.0050</td>
<td>0.0050</td>
<td>0.0050</td>
<td>0.0000</td>
<td></td>
</tr>
<tr>
<td>CYL 8</td>
<td>0.3756</td>
<td>0.3750</td>
<td>0.0050</td>
<td>0.0056</td>
<td>0.0056</td>
<td>0.0000</td>
<td></td>
</tr>
<tr>
<td>CYL 9</td>
<td>0.3757</td>
<td>0.3750</td>
<td>0.0050</td>
<td>0.0057</td>
<td>0.0057</td>
<td>0.0000</td>
<td></td>
</tr>
<tr>
<td>E1CYL 12</td>
<td>2.5000</td>
<td>2.5000</td>
<td>0.0200</td>
<td>0.0200</td>
<td>0.0150</td>
<td>0.0000</td>
<td></td>
</tr>
</tbody>
</table>

**Figure 17**
Datum Shift applied simultaneously to Position and Profile FCFs

Figure 18
Section 3: GD&T Standards Supported by PC-DMIS and examples for features of size, material condition modifiers, simultaneous requirements and Profile tolerances

Example of PC-DMIS report for Profile per ASME and ISO

**ASME Y14.5** - Two actual values (MAX and MIN)

**ISO 1101** - Single actual value MEAS = largest MAX or MIN x 2

Figure 19

Figure 20