General Tolerance Standards in Manufacturing: ISO 2768, ISO 286, and GD&T

By GEMS-MFG Team

I. Overview of Tolerance Standards

Tolerance standards provide guidelines on the acceptable amount of dimensional variation in parts, covering aspects such as form, fit, and function. Understanding how to apply tolerances ensures that parts are produced to the desired quality and can be assembled seamlessly in mass production. In manufacturing, tolerances are necessary because no process is entirely free from variation. A tolerance defines the acceptable range within which a dimension may vary without negatively affecting part function or performance. Establishing a tolerance allows engineers to manage the balance between part quality, manufacturing capability, and cost.

Tolerance management for precision manufacturing is vital to ensure proper fit between parts and function correctly, especially when components are produced in different locations or by different suppliers. Tolerance standards allow manufacturers to control the allowable variation in part dimensions, ensuring reliable fits, proper assembly, and optimal performance across industries such as automotive, aerospace, electronics, and medical devices. This guide will explore the technical aspects of ISO 2768, ISO 286, and Geometric Dimensioning and Tolerancing (GD&T), offering insights into their use, application, and real-world examples.

Key Tolerance Systems:

1. **ISO 2768**: Governs general tolerances for dimensions and geometry when no specific tolerances are noted.
2. **ISO 286**: Establishes tolerance classes and fit systems, especially for mating components like shafts and holes.
3. **GD&T**: Provides a more sophisticated system for controlling part features beyond linear dimensions, focusing on the geometry and orientation of part surfaces.

Key Considerations for Applying Tolerances:

1. **Material properties**: Different materials expand, contract, or deform differently. Proper tolerance design accounts for these variations.

2. **Manufacturing process**: Certain processes, such as CNC machining or injection molding, have inherent limitations in precision that influence tolerance decisions.

3. **Cost-effectiveness**: Tighter tolerances often require advanced tooling or machining processes, significantly raising production costs.

II. ISO 2768: General Tolerances for Linear, Angular, and Geometric Dimensions

The process of vacuum casting is relatively straightforward, yet it involves precise execution to ensure the highest part quality. Here's a breakdown of the core steps involved:

ISO 2768 is a widely used standard for general tolerances, particularly in situations where no specific tolerances are provided on engineering drawings. It simplifies the design and production process by providing default tolerances for basic linear, angular, and geometric dimensions. ISO 2768 is divided into two parts:

- **ISO 2768-1**: Tolerances for linear and angular dimensions.
- **ISO 2768-2**: Geometrical tolerances for form and position.

Key Aspects of ISO 2768:

1. **Linear Dimensions**: Defines tolerances for features like length, width, height, and diameter based on specific dimension ranges. These are grouped into four tolerance grades—**fine (f)**, **medium (m)**, **coarse (c)**, and **very coarse (v)**. The grades provide flexibility depending on the required precision.
2. **Angular Dimensions**: Tolerances for angles control the variation in the orientation of part features like chamfers, bevels, and tapers.
3. **Geometrical Features**: Covers form tolerances such as straightness, flatness, and cylindricity, ensuring that parts maintain the correct shape within the allowable limits.

Expanded Table 1: Linear Dimensions Tolerance (ISO 2768-1)

Dimension Range (mm)	Fine (f)	Medium (m)	Coarse (c)	Very Coarse (v)
0.5 to 3	±0.05	±0.10	±0.20	±0.50
3 to 6	±0.05	±0.12	±0.25	±0.60
6 to 30	±0.10	±0.20	±0.50	±1.00
30 to 120	±0.15	±0.30	±0.80	±1.50

Application in Design:

ISO 2768 is ideal for less critical parts that do not require highly specific tolerances. For example, when designing non-interfacing components, it can reduce the time spent calculating individual tolerances and improve manufacturing efficiency.

Example:

A sheet metal part designed with a **50 mm width** under the **medium tolerance class (m)** will have an allowable deviation of ±0.3 mm, meaning the actual width can range from **49.7 mm to 50.3 mm**.

III. ISO 286: Limits and Fits for Holes and Shafts

ISO 286 is the international standard for defining the system of fits and tolerances between mating parts such as shafts and holes. It establishes standardized **tolerance grades** and **fit classes**, ensuring that components fit together appropriately depending on the application requirements (e.g., free movement, tight assembly, or interference).

Fit Systems:

- **Clearance Fit**: Ensures that the shaft is always smaller than the hole, allowing free movement.
- **Interference Fit**: The shaft is larger than the hole, creating a tight, forceful fit that may require press-fitting tools.
- **Transition Fit**: A balance between clearance and interference, allowing a tight fit with the potential for slight movement.

International Tolerance (IT) Grades:

The **IT grades** define the amount of allowable deviation in part size. They range from **IT01** (highest precision) to **IT18** (lowest precision). The lower the grade number, the tighter the tolerance.

Expanded Table 2: IT Grades (ISO 286)

Nominal Size (mm)	IT6	IT7	IT8	IT9	IT10
18 - 30	10 µm	16 µm	25 µm	40 µm	64 µm
30 - 50	12 µm	20 µm	30 µm	48 µm	75 µm
50 - 80	15 µm	24 µm	37 µm	60 µm	97 µm

Common Fits (ISO 286):

- **H7/h6**: A common clearance fit used in mechanical assemblies where parts need to move freely, such as gears and bearings.
- **H7/p6**: A typical interference fit for components that must be locked together, like shafts pressed into housings.

Expanded Table 3: Common Fits (ISO 286)

Fit Type	Hole Tolerance	Shaft Tolerance	Application
Clearance Fit	H7	h6	Bearings, rotating shafts
Interference Fit	H7	p6	Press-fit gears, pulleys
Transition Fit	H7	k6	Aligning shafts and couplings

IV. GD&T (Geometric Dimensioning and Tolerancing)

GD&T provides a detailed system for specifying tolerances on both size and geometry of parts. It is a more advanced and comprehensive approach compared to ISO 2768 and ISO 286. The purpose of GD&T is to control not just the size of parts, but their overall form, orientation, and location, ensuring that parts function together under real-world conditions.

Key Concepts in GD&T:

1. **Datums**: The reference points or planes from which measurements are made.

2. **Tolerance Zones**: GD&T defines the permissible limits of variation for part features in three-dimensional space.
3. **Symbol-Based Language**: Uses a set of symbols to communicate tolerances in an efficient, unambiguous manner.

Geometric Control Categories:

1. **Form Tolerances**: Control the shape of features (e.g., flatness, straightness).
2. **Orientation Tolerances**: Control the orientation of features relative to datums (e.g., perpendicularity, parallelism).
3. **Location Tolerances**: Control the location of features relative to datums (e.g., position, concentricity).
4. **Runout Tolerances**: Control the variation in surface rotation (e.g., circular runout, total runout).

Expanded Table 4: Common GD&T Symbols

Symbol	Control Type	Description
–	**Flatness**	Controls how flat a surface must be
⊥	**Perpendicularity**	Ensures features are perpendicular to a datum plane
⊕	**Position**	Controls the location of a feature in relation to a datum
↔	**Parallelism**	Ensures two features remain parallel to each other

Application in Design:

GD&T is particularly useful in applications requiring high precision or complex geometries. It allows engineers to specify how much deviation is permissible while ensuring functionality.

Example:

In a complex assembly, a bolt hole's **position tolerance** might be defined as **⊕ 0.1 mm**, ensuring that the hole's center can deviate by up to 0.1 mm from the true position. This ensures proper assembly with other components even if some slight deviation occurs.

V. How to Choose the Right Tolerance

Choosing the right tolerance involves balancing the part's functionality, manufacturing capability, and cost. Overly tight tolerances can lead to increased production costs, while overly loose tolerances can compromise part performance and fit. The key is selecting the most suitable tolerance grade based on the specific application, production process, and functional requirements of the part.

Factors to Consider When Choosing Tolerances:

1. **Functionality**:
    - **Critical Dimensions**: Identify dimensions that directly affect part performance, such as bearing fits, gear spacing, or moving parts. These typically require **tight tolerances** to ensure precision and smooth operation.
    - **Non-Critical Dimensions**: Features like exterior finishes or cosmetic dimensions may tolerate larger deviations, reducing manufacturing difficulty and cost.
2. **Material Selection**:
    - Different materials react differently to machining or forming processes. For instance, **plastics** and **elastomers** often exhibit more variability due to material flexibility or shrinkage, whereas **metals** can hold tighter tolerances.
    - Example: An injection-molded **plastic** part may have looser tolerances (e.g., ISO 2768-m), while a **machined aluminum** part might demand tighter control (e.g., ISO 2768-f).
3. **Manufacturing Process Capability**:
    - Tolerances should reflect the inherent capabilities of the manufacturing process. Processes like **precision CNC machining** can achieve tighter tolerances (micron-level), while processes like **casting** or **injection molding** might only support medium or loose tolerances due to material shrinkage or tool wear.
    - Example: In **mold making**, precision molds require tight tolerances to ensure the accuracy of the molded part, but the molded part itself might not need the same level of tightness.
4. **Assembly Fit**:
    - If a part interfaces with other components, it's important to define tolerances that ensure proper assembly. Standards like **ISO 286** provide fit classifications, from **clearance fits** (looser tolerance) to **interference fits** (tighter tolerance), to guide tolerance choices for mating parts.

- Example: A **H7/h6** fit is commonly used for shafts and holes to achieve a smooth sliding fit without excessive play.

5. **Cost Implications**:

   - Tighter tolerances typically require more precise machinery, more rigorous quality control, and possibly additional finishing operations, all of which can raise production costs. By selecting the **loosest tolerances** that still meet functional requirements, manufacturers can often reduce costs.

   - Example: For a part that doesn't require high precision, specifying ISO 2768-m (medium tolerance) instead of ISO 2768-f (fine tolerance) could cut production costs while maintaining functionality.

Tolerance Grades and Their Applications:

Different standards offer tolerance grades that manufacturers can select based on the precision required.

1. **ISO 2768 Grades**:

   - **f (fine)**: Suitable for parts requiring high accuracy, typically used in **machined components**.

   - **m (medium)**: The most commonly used grade, balancing performance and cost for **general-purpose parts**.

   - **c (coarse)**: Applied to parts with less critical dimensions or where functionality allows more deviation, often for **castings** or **welded parts**.

   - **v (very coarse)**: Used in less precise manufacturing processes like **fabrication** or **structural parts**.

2. **ISO 286 Grades**:

   - **IT5-IT7**: Used for **high-precision fits**, like in bearings or high-tolerance assemblies.

   - **IT8-IT11**: Common for **general machine parts**, providing a balance between precision and manufacturability.

   - **IT12-IT16**: Used in **low-precision parts** where large variations in dimensions are acceptable, such as **structural or bulk parts**.

3. **GD&T Tolerances**:

- **Feature Control Frames**: Define the allowable deviation for features such as **flatness**, **perpendicularity**, **cylindricity**, or **true position**. GD&T allows for precision control in complex parts or assemblies, where both form and orientation tolerances are crucial.
- Example: A **true position tolerance** of ⊕0.05 mm for a hole ensures that it is precisely located within an assembly.

Step-by-Step Guide for Selecting Tolerances:

1. **Analyze Design Requirements**: Review the part design and identify critical features that need tighter control (e.g., interfaces, moving parts) versus non-critical features (e.g., cosmetic surfaces, mounting holes).
2. **Consult Standards**: Refer to relevant tolerance standards (ISO 2768, ISO 286, GD&T) based on the type of part and the fit required. Use general tolerances for simpler parts, fits from ISO 286 for shaft-hole systems, and GD&T for more complex geometric controls.
3. **Evaluate Manufacturing Methods**: Match the tolerances with the capabilities of your production process. For instance, tighter tolerances are achievable in CNC machining but may require secondary operations like honing or grinding in casting.
4. **Consider Assembly**: Choose tolerances that ensure proper mating between parts. For example, parts that slide together should have a **clearance fit**, while parts that must not move relative to one another may need an **interference fit**.
5. **Assess Cost-Performance Trade-Off**: If the part's function allows, choose looser tolerances to reduce production costs. If precision is crucial, such as in aerospace or medical devices, opt for tighter tolerances and invest in high-precision manufacturing processes.

Example:

In the design of a **precision medical device**, engineers might specify **GD&T true position tolerance** for critical hole alignments (⊕0.01 mm), but use ISO 2768-m tolerances for less critical features like the device's external housing. This ensures both precise functionality and cost-effectiveness in production.

VI. Understanding Tolerance Stack-Up

Tolerance stack-up occurs when multiple tolerances accumulate across several components or features in an assembly. If not properly controlled, this accumulation can result in misalignment, poor fit, or functional failure of the product. Below are the **Usual Methods to Control Tolerance Stack-Up:**

1. **Worst-Case Method**:

   Assumes all tolerances are at their maximum or minimum limits. This is a conservative approach and often results in tighter tolerances, leading to increased manufacturing costs.

2. **Statistical Tolerance Analysis**:

   Recognizes that not all tolerances will simultaneously be at their extremes. This method applies statistical tools like root sum squares (RSS) to balance between tighter tolerances and manufacturing feasibility.

3. **GD&T Positional Tolerances**:

   GD&T provides an effective way to control tolerance stack-up by defining a “zone” for features like holes or pins. This allows slight deviations while ensuring assembly functionality.

Example:

In an automotive engine assembly, several bolts must align with their respective holes. A worst-case tolerance stack-up might predict significant alignment issues, requiring tighter tolerances across the entire design. By using GD&T with positional tolerances (e.g., ⊕ 0.2 mm), engineers can allow more deviation for each hole while ensuring overall assembly success.

VII. Inspection and Quality Control

Proper tolerance control is only as effective as the inspection methods used to verify compliance with design specifications. Inspection methods should be chosen based on the type of tolerance applied and the precision required.

Inspection Techniques:

1. **Coordinate Measuring Machine (CMM)**:

   Measures parts in three dimensions with high accuracy, ideal for verifying GD&T features such as form, orientation, and location tolerances.

2. **Go/No-Go Gauges**:

   Simple gauges for quickly checking if a dimension is within tolerance (common for hole/shaft fits from ISO 286).

3. **Optical Measuring Systems**:

   Non-contact measurement tools that can be used for inspecting complex geometries, surface profiles, or tight tolerances where contact measurement might damage the part.

Quality Control in Manufacturing:

To ensure that tolerances are maintained throughout production, manufacturers implement quality control measures, including:

1. **In-process inspection**:

   Regular checks during manufacturing to catch deviations early.

2. **Final inspection**:

   Comprehensive measurement of finished parts to ensure they meet all tolerance requirements before assembly or shipment.

3. **Statistical Process Control (SPC)**:

   Using statistical data from manufacturing processes to monitor and control tolerance deviations, reducing the occurrence of defects.

Example:

For a CNC-machined automotive part requiring a positional tolerance of ⊕ 0.1 mm, a CMM would be used to measure the deviation of critical features relative to their true position. This ensures precise assembly with mating components like bearings or shafts.

VIII. Conclusion

In practice, engineers often combine ISO 2768, ISO 286, and GD&T to fully define the tolerances on a part. ISO 2768 can be applied for general dimensions, ISO 286 for mating features, and GD&T for critical geometric control. The key is selecting the right standard for the right feature, ensuring the part meets performance requirements without unnecessary manufacturing cost.

The use of **ISO 2768**, **ISO 286**, and **GD&T** in manufacturing plays a vital role in controlling part dimensions, ensuring compatibility, and maintaining high-quality production standards. Each standard offers unique approaches to managing tolerances, allowing manufacturers to choose the best system based on part functionality, manufacturing capabilities, and cost considerations. By applying these standards effectively and combining them with robust inspection processes, manufacturers can reduce errors, minimize rework, and optimize production efficiency.

Are you looking for a reliable supplier who has good demand of above tolerance standards that result in high quality assurance for your projects? **GEMS-MFG** is the comprehensive solution provider here for you. As a one-stop custom manufacturer, we provide a wide range of services, including rapid prototyping, mold making, injection molding, CNC machining, die casting, and more. Whether your requirements involve intricate prototypes or precision parts, GEMS-MFG is committed to delivering an efficient and cost-effective solution tailored to your needs. Contact us today [**INFO@GEMS-MFG**] to explore our offerings and receive an instant quote. Your manufacturing goals are our priority.