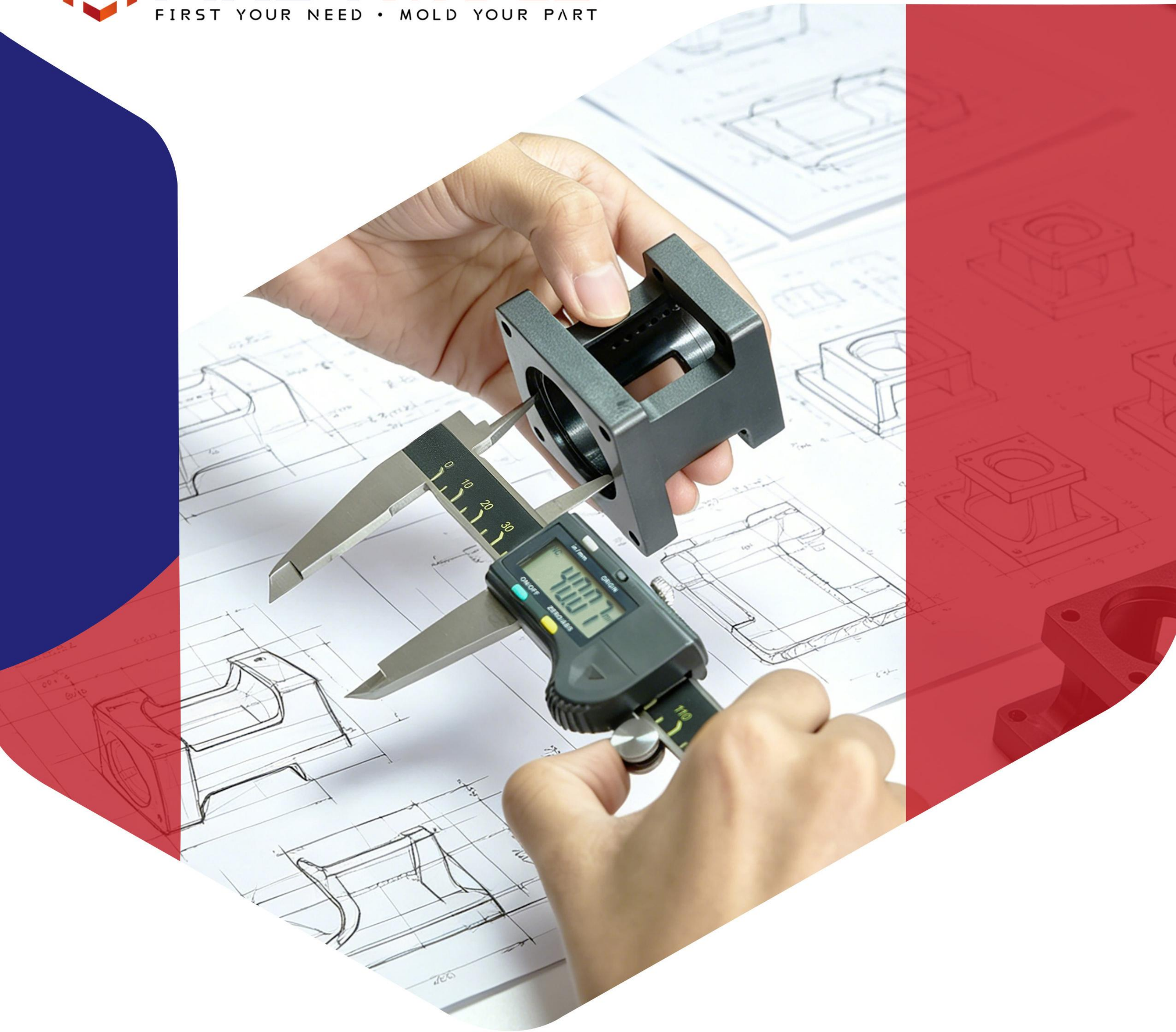

TOLERANCES

INJECTION MOLDING

Contents

1. Introduction

In plastic product design and manufacturing, tolerance is not only a dimensional range on a drawing. It is also a balance point between product function, manufacturing capability, and production cost. Its purpose is to define the allowable variation within which a part can still meet assembly and functional requirements despite normal manufacturing variation.

Injection molded parts are affected by material shrinkage, part design, mold strategy, molding process, and measurement method. Therefore, the accuracy logic used for metal machining should not be directly applied to plastic molded parts. The same “±0.10 mm” requirement may carry very different manufacturing difficulty and risk when applied to a small hole, snap-fit, sealing surface, or long housing dimension.

Through this guide, First Mold aims to help design engineers, purchasing teams, and project teams build a clearer tolerance judgment framework, improve DFM communication, reduce mold modification risks, and make more reliable manufacturing decisions.

Professional and standardized injection molding workshop

2. Basic Concepts

2.1 Basic Definition of Tolerance

Injection molding tolerance refers to the permissible deviation between the actual dimension of a plastic part and the nominal dimension specified on the drawing. Because plastic parts are affected by material shrinkage, tooling design, and molding process conditions, final dimensions naturally vary from design values. Tolerances define the acceptable dimensional range for production and inspection.

Example: 100.00 ± 0.10 mm

A dimension marked as 100.00 ± 0.10 mm means that the nominal dimension is 100.00 mm. The acceptable upper limit is 100.10 mm, and the acceptable lower limit is 99.90 mm. Therefore, the tolerance zone extends from 99.90 mm to 100.10 mm, with a total tolerance of 0.20 mm.Parts measured within this range meet the specified dimensional tolerance.

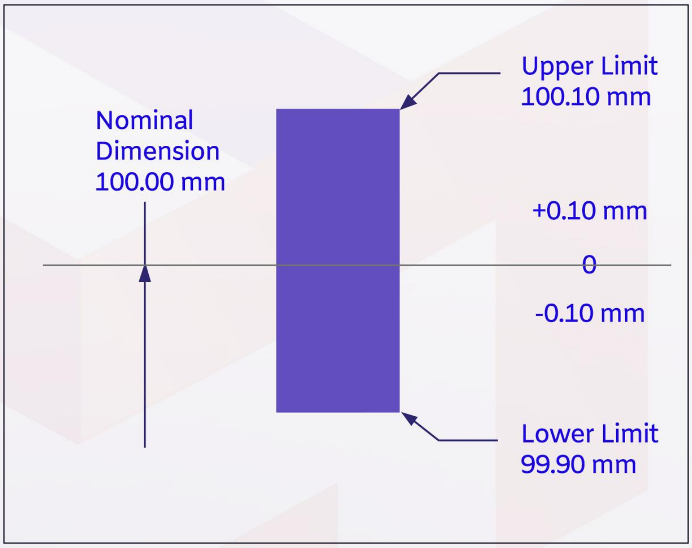

Figure 1. Tolerance Example: 100.00 ± 0.10 mm

Note: 10 ±0.10 mm is one of the most common tolerance expressions. In some functional areas, such as hole-and-shaft fits or assembly clearance, engineers may use asymmetric tolerances, such as 10 +0.10 / -0.00 mm. Symmetrical tolerances should be used whenever possible during the initial design phase.

2.2 Common Types

Depending on the function, assembly method, and appearance requirements of the part, tolerances are commonly divided into dimensional tolerance, geometric tolerance, assembly tolerance, and cosmetic tolerance. Different tolerance types control different risks and should not be replaced by one uniform “±0.10 mm” requirement.

2.2.1 Dimensional Tolerance

Dimensional tolerance controls the acceptable variation of basic sizes, such as length, width, height, wall thickness, hole diameter, and spacing. In injection molding, it is affected by material shrinkage, mold compensation, gate location, packing, and cooling behavior. Its purpose is to ensure that the part can still fit and function within the allowed dimensional range.

2.2.2 Geometric Tolerance

Geometric tolerance controls the shape, orientation, and positional relationship of features, not just their size. GD&T, or Geometric Dimensioning and Tolerancing, is a standardized drawing language used to communicate these geometric relationships.
For example, the overall length of a housing may be within tolerance, but if a large surface warps, the part may still show uneven gaps or poor assembly contact.

2.2.3 Assembly Tolerance

Assembly tolerance focuses on whether multiple parts can fit together correctly. It does not only evaluate whether one dimension is acceptable, but also whether multiple dimensional variations still allow the final assembly to function as intended.
Even if each individual dimension is within tolerance, tolerance stack-up may still cause interference, looseness, misalignment, or uneven gaps.

2.2.4 Cosmetic Tolerance

Cosmetic tolerance controls appearance quality that can be seen by the eye or felt by touch. It is not always the same as dimensional tolerance and usually needs to be evaluated together with cosmetic standards, limit samples, texture requirements, and acceptance criteria.

For parts with high cosmetic requirements, even if the dimensions meet the drawing requirements, defects such as flash, sink marks, mismatch, parting line offset, or warpage may still cause the part to be rejected.

Table 1. Simple Comparison of Common Tolerance Types

Tolerance Type	What It Mainly Controls	Typical Risks
Dimensional Tolerance	Basic part sizes such as length, thickness, hole diameter, and spacing	Poor fit or functional failure
Geometric Tolerance	Shape, orientation, and position of features	Warpage, misalignment, or poor contact
Assembly Tolerance	Fit and function between multiple parts	Interference, looseness, uneven gaps, or assembly failure
Cosmetic Tolerance	Visible or touchable appearance quality	Appearance rejection despite acceptable dimensions

2.3 Factors Influencing the Tolerances

2.3.1 Part Design

Part design is the first factor affecting injection molding tolerance. Uneven wall thickness, local thick sections, deep ribs, bosses, long structures, large flat surfaces, and complex snap-fits all change melt flow, cooling behavior, and shrinkage distribution.

Wall thickness consistency is especially important for dimensional stability. Wall thickness variation causes different cooling rates in different areas, which can lead to uneven shrinkage, warpage, sink marks, or assembly deviation. When thick sections are unavoidable, coring, ribs, or structural optimization should be considered to reduce local shrinkage risk. Features that affect screw fastening, snap-fit retention, housing alignment, or sealing should be identified as critical dimensions during DFM instead of being treated as ordinary dimensions.

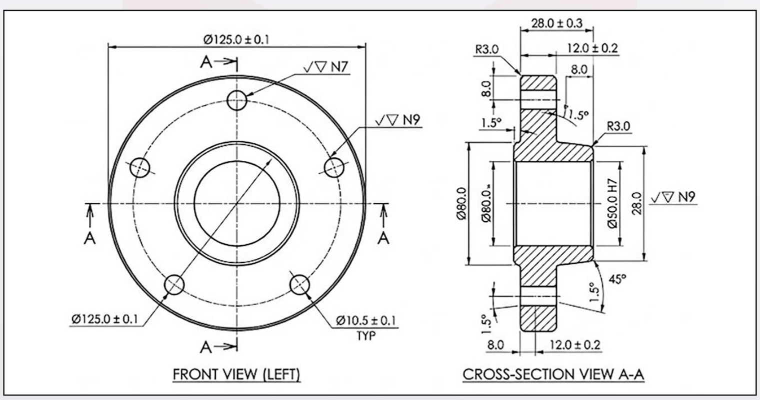

Figure 2. Injection-Molded Flange Drawing

2.3.2 Material Selection

Material selection is the foundational factor influencing the shrinkage and dimensional stability of an injection-molded part. Different resins exhibit vastly different shrinkage behaviors; even within the same material family, specific resin grades, filler types, and glass fiber (GF) content can lead to significant dimensional variances. Understanding the inherent stability of polymers is essential for setting realistic tolerance expectations.

[Download: First Mold Comprehensive Resin Shrinkage Reference Table]

Material grade should be confirmed before mold design begins because mold dimensions are calculated based on shrinkage compensation. If the material changes after tooling starts, shrinkage compensation, gate strategy, and cooling design may need to be reviewed again, which may lead to delays or mold modifications.

Shrinkage is commonly calculated as:

Shrinkage = (Original size – Cooled size) / Original size × 100%

Note: While this formula is fundamental for mold compensation, it should be used as a guideline for design rather than the sole determinant for final production tolerances.

2.3.3 Tool Design

Tool design is the key step in translating drawing dimensions into actual molded part dimensions. A mold must consider product geometry, material shrinkage compensation, gate location, cooling system, parting line, ejection method, venting, and adjustment space for critical dimensions.

Cooling has a major effect on tolerance stability. Uneven cooling can cause inconsistent shrinkage across different areas, leading to warpage, sink, or dimensional drift. Gate location is also important because it determines the melt entry direction and pressure path, affecting filling balance, weld lines, fiber orientation, local shrinkage, and cosmetic quality.

Critical dimensions may require a steel-safe strategy, meaning the mold retains material for controlled adjustment after trial molding. If T1 samples show dimensional deviation, a steel-safe strategy allows gradual correction through targeted mold modification rather than irreversible early machining.

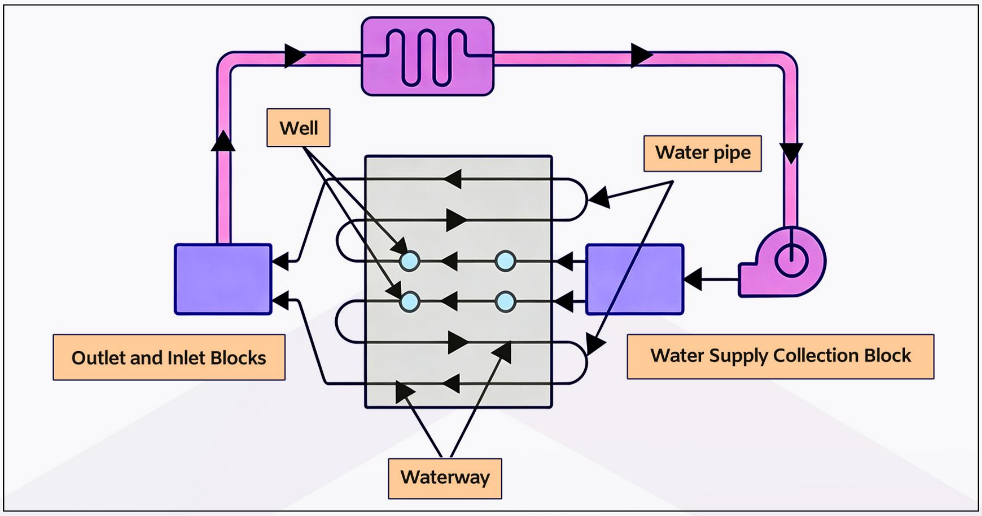

Figure 3. Mold Cooling Design

2.3.4 Process Control

Even when part design, material selection, and tool design have been confirmed, injection molded parts may still show dimensional variation during trial molding or production. Injection speed, packing pressure, packing time, mold temperature, melt temperature, and cooling time all affect filling, compensation, cooling, and post-ejection dimensional stability.

For example, insufficient packing may cause sink and local undersize dimensions; insufficient cooling may cause post-ejection deformation; unstable mold temperature may create batch-to-batch dimensional differences.

For critical dimensions, process parameters should not only target successful molding, but also establish a stable process window verified by trial data and dimensional inspection.

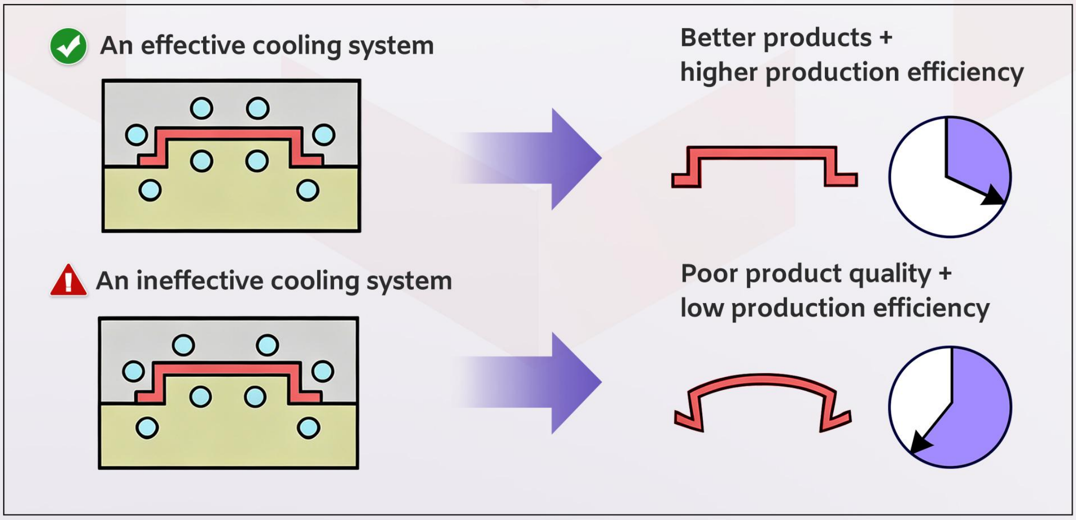

Figure 4. Effective vs. Ineffective Cooling

2.4 Injection Molding Tolerances vs. Machining Tolerances

Injection molded parts and machined metal parts both require tolerances, but their tolerance logic is different. Machined dimensions are mainly controlled by machine accuracy, tools, toolpaths, fixturing, and inspection methods. Injection molded parts are also affected by mold precision, but material shrinkage, cooling, ejection, post-shrinkage, moisture absorption, and environmental conditions play a much larger role.

Table 2: Comparison of Tolerance Logic: Injection Molding vs. Metal Machining

Item	Injection Molding	Metal Machining
Main Logic	Control molded part variation after shrinkage and cooling	Control material removal accuracy
Main Variables	Resin, mold, gate, cooling, process	Machine, cutter, fixture, toolpath
Dimensional Risk	Shrinkage, warpage, post-shrinkage, deformation	Cutting error, setup error, thermal deformation
Adjustment Method	Process tuning or mold modification	Parameter adjustment or re-machining
Standard Risk	Do not directly apply machining tolerance logic	General machining tolerances may apply

3. Tolerance Standards and Reference Systems

Tolerance standards for injection molded parts act as a common language for engineering communication across regions and organizations. Selecting a suitable standard is not only an administrative choice, but an engineering decision that balances material shrinkage, part structure, manufacturing capability, and cost.

3.1 ISO 20457

ISO 20457 is an international reference standard for tolerances and acceptance conditions of plastic molded parts. It is useful for general molded components such as plastic housings, brackets, covers, and connectors.

For ordinary dimensions, ISO 20457 can provide a practical tolerance reference. For functional areas, such as snap-fits, sealing surfaces, hole patterns, cosmetic gaps, and assembly interfaces, tolerances should still be individually specified and validated.

Table 3. ISO 20457 Tolerance Series Guide

Tolerance Series	Typical Use
Series 1	General non-critical dimensions
Series 2	Dimensions with assembly or stability requirements
Series 3	Critical functional dimensions requiring stronger control
Series 4	Special high-accuracy dimensions; not recommended as default

3.2 DIN 16742

DIN 16742 is commonly used for plastic molded part tolerances in German and European engineering projects. One important value of DIN 16742 is that it does not treat all dimensions the same. It distinguishes between W, or tool-specific dimensions, and NW, or non-tool-specific dimensions. W dimensions are mainly formed by mold geometry and are usually easier to control. NW dimensions are more affected by shrinkage, warpage, cooling, ejection, and process variation.

3.3 GB/T 14486—2008

GB/T 14486—2008 is a Chinese reference standard for dimensional tolerances of plastic molded parts. This standard can support general dimensional tolerance discussion for molded plastic parts, especially for ordinary linear dimensions. Its positioning is closer to ISO 20457 and DIN 16742 than to general machining tolerance standards.

Table 4. Simplified Reference of MT Tolerance Grades

Material	High-Precision Dimensions	General Dimensions	Unmarked Dimensions
ABS	MT2	MT3	MT5
PC	MT2	MT3	MT5
PMMA	MT2	MT3	MT5
PA(unfilled)	MT3	MT4	MT6
PA(glass-fiber reinforced)	MT2	MT3	MT5
POM	MT3–MT4	MT4–MT5	MT6–MT7
PP	MT2–MT3	MT3–MT4	MT5–MT6
PE	MT5	MT6	MT7

3.4 ISO 2768

ISO 2768 is a common general tolerance standard for engineering drawings. It is mainly used for linear and angular dimensions without individual tolerance indications.
However, ISO 2768 is not a plastic molding-specific tolerance standard. If it is directly applied to high-shrinkage materials, thin-wall parts, or large molded plastic components, it may create unrealistic expectations for manufacturability, cost, and inspection.

Table 5. ISO 2768 Tolerance Classes

Tolerance Class	Typical Use
f	Higher accuracy general mechanical dimensions
m	Common default level for ordinary mechanical drawings
c	Dimensions with lower accuracy requirements
v	Non-critical and low-accuracy dimensions

3.5 Measurement Methods and Inspection Considerations

Measuring molded parts is essential for confirming quality and compliance with design requirements. Common measurement methods include coordinate measuring machines (CMMs), optical measurement systems, micrometers, calipers, laser scanners, and profile projectors. The method should be selected according to the tolerance requirement, feature type, part rigidity, and inspection purpose.

For injection molded parts, measurement conditions are as important as measurement tools. Measurement time, environmental temperature, reference datum, fixturing method, and whether the part is measured in a free state or constrained state can all affect the result. For tight tolerances or critical dimensions, the measurement method should be confirmed before tooling and documented in the inspection plan.

Figure 5. CMM Inspection

Figure 6. Dimensional Check

4. Tolerance Design Optimization

Tolerance optimization for injection molded parts is not about tightening every dimension. It is about matching tolerance requirements with function, material, structure, tooling, and production capability.

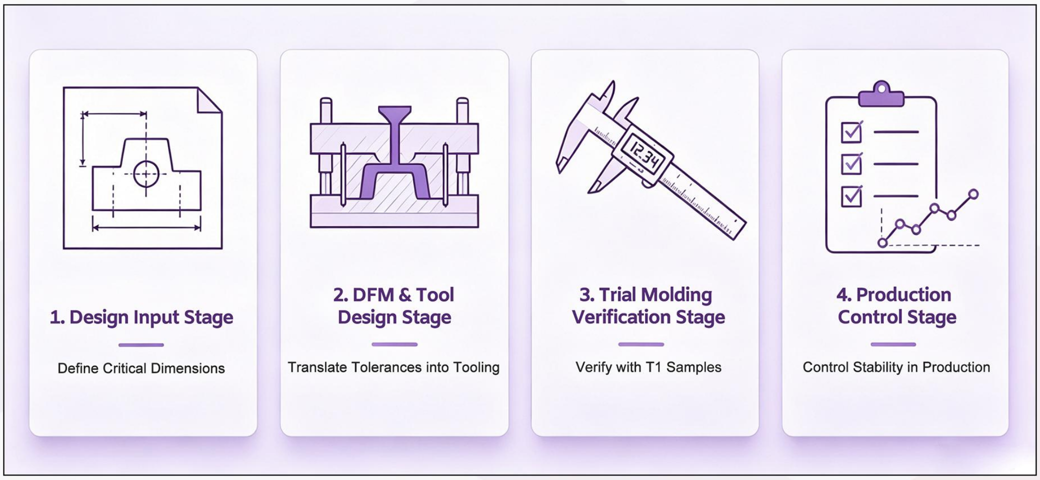

Figure 7. Tolerance Design Optimization

4.1 Design Start Stage

First, separate ordinary dimensions from critical dimensions. Ordinary outside dimensions may follow standard tolerances. Dimensions that affect assembly, sealing, positioning, appearance, or movement should have dedicated tolerance requirements, measurement datums, and acceptance methods. Tight tolerances should not be committed before the material is confirmed.

4.2 DFM and Tool Design Stage

During DFM, confirm whether critical dimensions are manufacturable. Key factors include material shrinkage, wall thickness variation, gate location, cooling system, parting line, ejection method, and multi-cavity consistency. Critical areas such as holes, snap-fits, sealing surfaces, and inserts should retain reasonable adjustment space in the mold.

4.3 Trial Molding Verification Stage

After T1 samples, critical dimensions should be measured, and dimensional deviation should be traced to material shrinkage, mold dimensions, process parameters, uneven cooling, or measurement method. Assembly parts should also be tested in real assembly conditions to confirm gaps, snap force, sealing performance, and movement feel.

4.4 Production Control Stage

In production, critical dimensions should be managed through an inspection plan, including first article inspection, patrol inspection frequency, measurement equipment, and acceptance criteria. For multi-cavity molds, tight tolerances, or high-shrinkage materials, cavity-to-cavity variation, material batches, mold temperature fluctuation, and process stability should be monitored continuously.

5. Injection Molding Tolerance Checklist

Table 6. Pre-Tooling Tolerance Review Framework

Review Dimension	Key Verification Points	Engineering Objective
Drawing Definition	General tolerance standard, critical dimensions, tolerance tightness, measurement datum, GD&T requirements, and consistency between 3D model and 2D drawing	Ensure that tolerance requirements are clearly defined, measurable, and aligned with function, assembly, and cosmetic requirements
Material Behavior	Exact resin grade, shrinkage range, high-shrinkage or hygroscopic behavior, post-shrinkage risk, glass fiber or filler content, and cosmetic performance requirements	Evaluate whether the selected material can support the required dimensional stability and whether shrinkage compensation assumptions are reliable
Part Structure	Wall thickness variation, local thick sections, rib design, boss design, snap-fit design, long spans, large flat surfaces, sealing areas, and insert features	Identify structural features that may create uneven shrinkage, warpage, sink marks, deformation, or tolerance stack-up risk
Tooling Strategy	Gate location, cooling layout, parting line, ejection method, venting design, steel-safe allowance, and multi-cavity control	Verify whether the mold design can translate drawing requirements into stable molded dimensions while retaining adjustment capability for critical features
Inspection Method	Measurement timing, measurement temperature, equipment selection, datum consistency, fixturing method, FAI requirement, Cp/Cpk or trend data, and cosmetic acceptance criteria	Ensure that dimensional and cosmetic acceptance are evaluated under consistent, agreed, and repeatable inspection conditions

6. Conclusion

Injection molding tolerances are not solely determined by mold precision. They result from the combined effects of part design, material behavior, mold design, molding processes, measurement methods, and mass production control. Every tolerance requirement specified on the drawing must ultimately be realized through the mold, the manufacturing process, and the inspection system. Proper tolerance design does not mean specifying tighter tolerances for all dimensions, but rather identifying the key dimensions that truly affect functionality, assembly, appearance, and stability.

First Mold has over a decade of manufacturing experience in mold making and injection molding, supporting customers from DFM review and tooling to trial molding and production. Our engineering and molding teams provide practical manufacturability feedback before tooling begins, helping customers reduce mold modification risks and improve production stability through clearer tolerance planning.

First Your Need
Mold Your Part

Contact Details

https://firstmold.com
sales@firstmold.com
+86 13925326660


Mold Making


Injection Molding


CNC Machining


3D Printing


Die Casting


Sheet Metal