Engineering Tolerance Guide

Dedicated engineering reference guide for buyers, engineers, and sourcing teams

Based on article:

https://nylonplastic.com/engineering-tolerance-guide-plastic-parts/

Quick Overview

Plastic parts do not hold tolerances like metal parts - and expecting them to is the single most common cause of production delays, cost overruns, and supplier disputes in injection molding and CNC machining. A turned aluminum part can reliably hold plus or minus 0.025 mm; an injection molded PA66 part in the same geometry will struggle to hold plus or minus 0.15 mm once moisture conditioning, mold wear, and process variation are factored in.

This guide translates ISO 2768, DIN 16901, and decades of production data into practical tolerance tables for each manufacturing process and material. Use these numbers at the design stage to avoid the expensive discovery that your plus or minus 0.05 mm drawing note was never achievable in the first place.

Not all plastics are created equal when it comes to dimensional stability. The key differentiators are: shrinkage (higher = wider tolerance band), moisture absorption (nylon swells, PP does not), and coefficient of thermal expansion (CTE) (determines how much a part changes size between molding and room temperature). The table below shows realistic tolerance expectations for a 100 mm long feature in a well-designed production mold.

Engineering Notes

Tolerance Capability by Manufacturing Process

plus or minus 0.05-0.10 mm Thermal expansion during cutting plus or minus 0.025-0.05 mm Injection Molding (unfilled) plus or minus 0.10-0.30 mm Shrinkage variation 0.1-0.3% plus or minus 0.08-0.20 mm Anisotropic shrinkage; mold wear plus or minus 0.15-0.30 mm Layer resolution; powder bed shrinkage plus or minus 0.10-0.20 mm Post-cure shrinkage; support marks

Material-Specific Tolerance Guidance

Not all plastics are created equal when it comes to dimensional stability. The key differentiators are: shrinkage (higher = wider tolerance band), moisture absorption (nylon swells, PP does not), and coefficient of thermal expansion (CTE) (determines how much a part changes size between molding and room temperature). The table below shows realistic tolerance expectations for a 100 mm long feature in a well-designed production mold. plus or minus 0.08-0.15 mm plus or minus 0.08-0.15 mm plus or minus 0.12-0.25 mm +0.5-1.5% dimension at 50% RH plus or minus 0.08-0.18 mm +0.3-0.8% (reduced by glass content) plus or minus 0.15-0.35 mm plus or minus 0.08-0.20 mm plus or minus 0.10-0.20 mm

DIN 16901: The Plastic-Specific Standard

DIN 16901 defines tolerance grades specifically for plastic molded parts, recognizing that plastics have larger and more variable shrinkage than metals. It uses a series of tolerance groups based on nominal dimension range. For a 100 mm feature, DIN 16901 fine tolerance corresponds to approximately plus or minus 0.18 mm for unfilled semi-crystalline materials like PA66 - roughly 6x the tolerance that ISO 2768-m (medium) specifies for machined metal of the same size. This standard, not ISO 2768, should be referenced on plastic part drawings to establish realistic expectations with mold makers and molding suppliers.

Design Rules for Plastic Tolerances

Specify tolerances only where needed: Do not apply blanket tolerances to entire parts. Every tolerance on a drawing costs money - the mold maker must hold it, the molder must verify it, and both will charge for it. Use general tolerances for non-functional surfaces (ISO 2768 or DIN 16901 reference) and specific tolerances only for bearing fits, seal surfaces, and assembly interfaces. Add 0.05 mm per 100 mm for moisture-sensitive materials: Nylon (PA6/PA66) parts change dimension by 0.5-1.5% between dry-as-molded and 50% RH equilibrium. A 100 mm PA66 feature that measures 100.00 mm fresh from the mold will measure 100.50-101.50 mm after conditioning. Either specify the measurement condition (dry or conditioned) or widen the tolerance to absorb the moisture effect. Mold tolerance is not part tolerance: A mold cavity machined to plus or minus 0.01 mm will not produce parts at plus or minus 0.01 mm. The molding process adds variation from: shrinkage (1-2% of dimension), mold temperature fluctuations (plus or minus 3 deg C = plus or minus 0.03 mm on 100 mm), and packing pressure variations. Budget 3-5x the mold tolerance for the final pa

Industry Application Matrix

Syringe plungers, luer fittings, inhaler bodies plus or minus 0.05-0.10 mm on critical seals ISO 13485; functional rather than dimensional validation Connector housings, sensor brackets, fluid fittings plus or minus 0.10-0.20 mm Temperature range -40 to +120 deg C; must fit after thermal cycling Phone cases, laptop housings, wearable bands plus or minus 0.08-0.15 mm on cosmetic gaps Gap-and-step visible quality metric; 0.1 mm gap visible to user Gear housings, bearing seats, pump bodies plus or minus 0.10-0.25 mm Must maintain fit after oil/chemical exposure and temperature cycling

Cost Decision Framework

Tolerances drive mold cost non-linearly: A mold designed for plus or minus 0.20 mm might cost $12,000. The same part geometry tightened to plus or minus 0.10 mm adds $5,000-8,000 for higher-precision machining, hardened steel, and conformal cooling. Tightening further to plus or minus 0.05 mm adds another $8,000-15,000 - bringing the total to 2-3x the cost for a 4x tighter spec. The process trade-off: If the part truly needs plus or minus 0.05 mm or better, injection molding may be the wrong process. CNC machining from plastic stock achieves plus or minus 0.05 mm at lower tooling cost ($0 mold, $15-50/part machining) for volumes under 500. Above 5,000 pcs, the per-part machining cost usually exceeds the amortized mold cost. Decision rule: Design parts at plus or minus 0.15 mm for injection molding as the baseline. Tighten only the features that absolutely require it - bearing seats, seal grooves, snap-fit engagement surfaces. Each tightened tolerance adds cost; each unnecessary tolerance guarantees disputes.

Common Defects and Solutions

Out-of-tolerance after conditioning Part measures in-spec dry but out-of-spec after moisture Nylon absorbed 1.5-2.5% moisture, swelling 0.5-1.5% Specify conditioning before measurement; widen tolerance or use GF grade Cavity-to-cavity or shot-to-shot variation over 0.1 mm Process instability: melt temp plus or minus 5 deg C, hold pressure drift Stabilize process within plus or minus 3 deg C and plus or minus 50 PSI; add SPC on critical dimensions Warpage causing out-of-spec Part twists after ejection, dimensions shift Differential cooling; anisotropic GF orientation Use mold flow analysis; balance cooling; reposition gates for symmetric fill Tool wear exceeding tolerance Cavity dimensions growing over production run GF abrasion on soft steel; high injection velocity at gate Upgrade to H13/D2; hard chrome wear surfaces; monitor every 25K shots

RFQ Checklist

- Application environment: temperature, moisture, UV, chemicals, sterilization, or outdoor exposure.
- Mechanical requirements: load, stiffness, impact, wear, friction, creep, and fatigue life.
- Drawing requirements: tolerance class, critical dimensions, surface finish, threads, inserts, and inspection method.
- Production needs: prototype or production quantity, expected annual volume, target unit cost, and lead-time window.
- Material notes: preferred grade, color, reinforcement, flame rating, certification, and substitute-material flexibility.

Need manufacturing support?

Share your drawing, target material, working environment, tolerance requirements, and quantity. Nylon Plastic can help evaluate manufacturability, material alternatives, and production quotation details.

Contact: nova@nylonplastic.com | https://nylonplastic.com/contact/