Chamfer Fillet Design Guide

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

Based on article:

https://nylonplastic.com/chamfer-vs-fillet-injection-molded-parts/

Quick Overview

Chamfers and fillets are the most ubiquitous geometric features in injection-molded part design, yet they are among the least systematically specified. A chamfer—a flat, angled cut replacing a sharp 90° corner—and a fillet—a concave or convex radius transition—each serve distinct mechanical, manufacturing, and assembly functions that cannot be swapped without consequence. In a 2023 survey of mold flow analysis reports across 500+ production molds, improperly specified corner transitions ranked as the third most common root cause of part rejection, behind only sink marks and short shots. The financial impact is measurable: sharp internal corners increase mold steel stress concentrations by 3–5×, driving up tool maintenance frequency and reducing cavity life from a typical 500,000–1,000,000 shots to as few as 50,000 shots before crack initiation.

This guide provides a structured, quantitative approach to chamfer and fillet specification for injection-molded parts. We examine the stress concentration mechanics that govern fillet sizing, the mold flow and demolding forces that dictate chamfer placement, and the critical interactions with draft angles that determine whether your part releases cleanly from the tool or requires a hammer and colorful language. By the end, you will have a rules-based framework and reference tables that eliminate the guesswork from rounding corners—literally and figuratively.

The decision between chamfer and fillet hinges on the function the corner must perform. Fillets (rounded transitions) are the default choice for internal corners subjected to mechanical load, because the smooth radius distributes stress over a larger cross-sectional area, reducing the stress concentration factor (Kt) from 3.0–5.0 for a sharp corner to 1.2–1.8 for a properly sized fillet. Fillet radii should be specified as a function of the adjacent wall thickness: a minimum of 0.5× wall thickness for internal corners and 0.25× wall thickness for external corners is the industry-standard starting point. For cyclically loaded features such as snap-fit roots and living hinges, internal fillet radii of 0.6–1.0× wall thickness are recommended to suppress crack initiation at cycle counts above 10 u.

Engineering Notes

Introduction to Chamfer and Fillet Design

Chamfers and fillets are the most ubiquitous geometric features in injection-molded part design, yet they are among the least systematically specified. A chamfer—a flat, angled cut replacing a sharp 90° corner—and a fillet—a concave or convex radius transition—each serve distinct mechanical, manufacturing, and assembly functions that cannot be swapped without consequence. In a 2023 survey of mold flow analysis reports across 500+ production molds, improperly specified corner transitions ranked as the third most common root cause of part rejection, behind only sink marks and short shots. The financial impact is measurable: sharp internal corners increase mold steel stress concentrations by 3–5×, driving up tool maintenance frequency and reducing cavity life from a typical 500,000–1,000,000 shots to as few as 50,000 shots before crack initiation. This guide provides a structured, quantitative approach to chamfer and fillet specification for injection-molded parts. We examine the stress concentration mechanics that govern fillet sizing, the mold flow and demolding forces that dictate chamfer placement, and the critical interactions wit

When to Use a Chamfer vs. a Fillet

The decision between chamfer and fillet hinges on the function the corner must perform. Fillets (rounded transitions) are the default choice for internal corners subjected to mechanical load, because the smooth radius distributes stress over a larger cross-sectional area, reducing the stress concentration factor (Kt) from 3.0–5.0 for a sharp corner to 1.2–1.8 for a properly sized fillet. Fillet radii should be specified as a function of the adjacent wall thickness: a minimum of 0.5× wall thickness for internal corners and 0.25× wall thickness for external corners is the industry-standard starting point. For cyclically loaded features such as snap-fit roots and living hinges, internal fillet radii of 0.6–1.0× wall thickness are recommended to suppress crack initiation at cycle counts above 10 u. Chamfers (angled flats) serve three distinct purposes that fillets cannot replicate: (1) lead-in for assembly—a 30°–45° chamfer on a pin or boss entry face reduces the insertion force by 50–70% compared to a square edge by providing a self-centering ramp; (2) mold parting line cleanup—a 0.5–1.0 mm × 45° chamfer at the parting line masks the in

Stress Concentration Factors: The Quantitative Case for Fillets

The stress concentration factor Kt quantifies how much a geometric discontinuity amplifies the nominal stress in a loaded component. For a sharp 90° internal corner in a plastic part under tensile or bending load, Kt values range from 3.0 to 5.0 depending on the exact geometry, meaning the local stress at the corner tip is 3–5× the average stress calculated from the net cross-sectional area. For amorphous thermoplastics such as PC and PMMA, which exhibit limited plastic yielding before brittle fracture, this amplification directly translates to a proportional reduction in load-carrying capacity—a sharp-cornered PC bracket that FEA predicts will hold 200 N based on bulk material properties may actually crack at 40–67 N due to the stress concentration. Introducing a fillet radius (r) equal to the wall thickness (t) reduces Kt to approximately 1.3–1.5, recovering roughly 70% of the lost strength. The relationship follows a diminishing-return curve: increasing r/t from 0 to 0.5 drops Kt from ~3.5 to ~1.6 (a 54% reduction), while increasing r/t from 0.5 to 1.0 drops Kt from ~1.6 to ~1.3 (only a 19% further reduction). The practical desig

Mold Flow Effects: How Corners Influence Fill and Part Quality

Internal fillets and chamfers are not merely stress management tools—they fundamentally affect the polymer melt flow during injection. A sharp internal corner creates a flow stagnation zone where the melt velocity drops to near zero, causing a local reduction in packing pressure of 20–40% compared to adjacent straight-wall sections. This under-packed region is the breeding ground for sink marks (visible on the opposite surface), voids (internal porosity exceeding 1% by volume), and residual stress concentrations that manifest as warpage after ejection. Computational fluid dynamics simulations run on typical ABS and PP mold filling show that a fillet radius of 0.5 mm increases the local flow velocity at the corner by 15–25% relative to a sharp corner, sufficient to eliminate the stagnation zone for part thicknesses up to 3 mm. Chamfers interact with mold flow differently than fillets: the flat surface of a chamfer creates a more abrupt flow direction change than a radius, generating a shear rate spike of 2–4× the nominal value at the transition points where the chamfer meets the adjacent walls. For shear-sensitive materials—notably f

Draft Angle Interaction: The Hidden Dependency

Draft angles and corner features are deeply interdependent in injection mold design, yet they are often specified by different stakeholders at different stages of the design process—the product designer specifies the chamfer, and the tooling engineer specifies the draft, often without considering how the two interact. The fundamental rule: any surface that is parallel to the mold opening direction must have draft, including chamfered surfaces. A 45° chamfer on a vertical wall without draft will create an undercut that locks the part in the cavity, requiring a side action ($3,000–$8,000 per action) or, worse, manual extraction. The minimum draft on a chamfered surface is typically 0.5°–1° for the chamfer face itself, which translates to an effective draft on the boundary edges that depends on the chamfer angle. For external chamfers intended to provide assembly lead-in, the draft must be applied such that the chamfer width at the parting line remains within ±0.1 mm of the nominal dimension to ensure consistent insertion force. This often requires specifying the chamfer dimension at the parting line as the reference dimension and allo

Six Design Rules for Chamfers and Fillets

Internal Corner Fillets: r "e 0.5 × t (minimum), r "e 0.75 × t (fatigue): Never leave internal corners sharp. The cost of a fillet in mold making is negligible (a ball end mill pass adds ~30 seconds of CNC time per corner), but the cost of a cracked tool or a fractured part from a sharp corner is measured in tens of thousands of dollars in mold repair and production downtime. For parts molded in PC, PS, or acrylic—which are notch-sensitive—use r "e 0.6 × t even for non-structural features. Assembly Lead-In Chamfers: 30°–45° with 0.5–1.5 mm Face Width: For pin-in-hole assemblies (clearance fit), a 30° chamfer provides the lowest insertion force. For boss-and-screw assemblies, a 45° chamfer with 0.5–1.0 mm face width is standard. For snap-fit lead-in angles, the chamfer angle should match the snap entry angle (typically 25°–30° for cantilever snaps) to avoid a step change in insertion force. Never Mix Fillets and Chamfers on the Same Edge Cascade: A sequence of edge treatments—for example, a fillet transitioning into a chamfer and back to a fillet—creates CAD tangency discontinuities that generate visible witness lines on the molded part

Industry Application Matrix

External fillets, cosmetic chamfers r = 0.5–2.0 mm, C = 0.3–0.5 mm × 45° Uniform appearance, no witness lines, consistent surface finish Assembly chamfers, structural fillets C = 1.5–3.0 mm × 30°, r "e 0.6 × t Low insertion force, squeak-and-rattle prevention, crash safety Internal fillets, lead-in chamfers r "e 0.75 × t, C = 1.0–2.0 mm × 30° Sterilizable, no particulate traps, biocompatible finish Sealing chamfers, rib fillets C = 0.5–1.0 mm, r = 0.4–0.6 × rib thickness IP65/IP67 gasket seating, structural integrity at -20°C to +80°C

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/