Core & Cavity Design Guide

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

Based on article:

https://nylonplastic.com/core-cavity-injection-molding-design/

Quick Overview

In injection molding, the core and cavity represent the two primary halves that form every plastic part you've ever held. The cavity forms the external surface of your part — the side customers see and touch — while the core creates internal features like ribs, bosses, and hollow sections. Think of the cavity as the mold's "negative space" that defines your part's outer geometry, and the core as the "positive shape" that fills what would otherwise be solid plastic. This partnership between core and cavity isn't just academic terminology; it's the fundamental design constraint that determines draft angles, cooling efficiency, ejection strategy, and ultimately your per-part cost.

Getting core and cavity design right at the DFM (Design for Manufacturability) stage separates projects that run smoothly from those plagued by flash, warpage, and stuck parts. The parting line — where core and cavity meet — is where most molding defects originate. A well-engineered core-cavity relationship accounts for material shrinkage (ranging from 0.2% for unfilled ABS to 2.5% for polyethylene), incorporates adequate draft angles that vary by material type and texture depth, and positions cooling channels where they'll actually remove heat from thick sections rather than just the mold steel. This guide distills decades of tooling experience into actionable design principles you can apply before sending your CAD file for tooling quotes.

The parting line is where your core and cavity halves separate, and its placement is the single most consequential decision in mold design. A flat, planar parting line along a single plane is the simplest and least expensive option — but real-world parts rarely offer that luxury. Stepped parting lines follow part geometry when features prevent a flat split, adding roughly 10-15% to tooling cost while resolving undercut situations without side actions. When parts have holes or features perpendicular to the mold opening direction, side actions (also called slides or lifters) extend the parting line into three dimensions, increasing both tooling complexity by 20-40% and cycle time by 2-5 seconds per side action.

Engineering Notes

Understanding Core and Cavity Fundamentals

In injection molding, the core and cavity represent the two primary halves that form every plastic part you've ever held. The cavity forms the external surface of your part — the side customers see and touch — while the core creates internal features like ribs, bosses, and hollow sections. Think of the cavity as the mold's "negative space" that defines your part's outer geometry, and the core as the "positive shape" that fills what would otherwise be solid plastic. This partnership between core and cavity isn't just academic terminology; it's the fundamental design constraint that determines draft angles, cooling efficiency, ejection strategy, and ultimately your per-part cost. Getting core and cavity design right at the DFM (Design for Manufacturability) stage separates projects that run smoothly from those plagued by flash, warpage, and stuck parts. The parting line — where core and cavity meet — is where most molding defects originate. A well-engineered core-cavity relationship accounts for material shrinkage (ranging from 0.2% for unfilled ABS to 2.5% for polyethylene), incorporates adequate draft angles that vary by material ty

Parting Line Strategy: Where Form Meets Function

The parting line is where your core and cavity halves separate, and its placement is the single most consequential decision in mold design. A flat, planar parting line along a single plane is the simplest and least expensive option — but real-world parts rarely offer that luxury. Stepped parting lines follow part geometry when features prevent a flat split, adding roughly 10-15% to tooling cost while resolving undercut situations without side actions. When parts have holes or features perpendicular to the mold opening direction, side actions (also called slides or lifters) extend the parting line into three dimensions, increasing both tooling complexity by 20-40% and cycle time by 2-5 seconds per side action. Strategic parting line placement minimizes visible witness lines on cosmetic surfaces. For consumer-facing products, position the parting line along natural edges or transitions rather than across large flat faces. When working with glass-filled materials, keep the parting line at least 0.5mm away from sharp corners to prevent chipping during ejection. For multi-cavity tools, ensure the parting line plane is consistent across a

Draft Angles by Material: The Numbers That Matter

Draft angle requirements vary dramatically by material type, and the "standard 1°" rule of thumb leads to heartbreak with certain resins. Polypropylene and polyethylene, with their high shrinkage rates (1.5-2.5%), can often release with as little as 0.5° draft on the core side — but only if surface finish is polished to at least SPI B-2. ABS and polycarbonate demand 1° minimum on the cavity and 1.5° on the core, while glass-filled nylon requires 1.5-2° minimum on all surfaces because its abrasive filler content creates higher friction during ejection. POM (acetal) is particularly unforgiving: its high crystallinity and low coefficient of thermal expansion mean it grips cores tenaciously, requiring 2-3° draft on deep core features. Texture depth changes everything about draft requirements. Every 0.025mm (0.001 inches) of texture depth requires an additional 1° of draft per side — so a heavy leather grain texture at 0.1mm depth needs 4° of draft beyond the base material requirement. This is why textured consumer products often have noticeably tapered sidewalls when you look closely. For deep drawn cores exceeding a 3:1 length-to-diame

Cooling Channel Design: The Cycle Time Multiplier

Cooling accounts for 60-80% of injection molding cycle time, making cooling channel design the highest-leverage knob for reducing per-part cost. Conformal cooling — where channels follow the part contour rather than running in straight drilled lines — can reduce cooling time by 30-50% on complex geometries, though it adds 15-25% to tooling cost due to the need for 3D-printed or diffusion-bonded inserts. For conventional drilled channels, the rule of thumb is channel diameter equal to 1.5-2 times the wall thickness of the surrounding steel, with center-to-center spacing of 3-5 channel diameters. Channels closer than 3 diameters create diminishing returns while channels farther than 5 diameters leave hot spots that extend cycle time. Core cooling presents the greater challenge because cores have less thermal mass and are surrounded by hot plastic on more sides than the cavity. Baffles and bubblers solve this by directing coolant into the core interior. A bubbler — essentially a tube within a drilled hole that brings coolant to the core tip — works well for cores up to 40mm diameter, maintaining a temperature differential of less than

Ejector Placement: Freeing the Part Without Damage

Ejector pin placement determines whether your part pops out cleanly or emerges with stress marks, deformation, and the telltale circular witness lines of poorly planned ejection. The fundamental rule: eject where the part is strongest, not where it's most convenient for the mold base layout. Eject against ribs and bosses from behind, never against thin unsupported walls that will deflect. Ejector pins should contact the part through the B-side (core side), and the total ejector area must be sufficient to overcome the part's adhesion to the core — which for deep-draw parts with minimal draft can exceed 50 MPa of holding force. For parts with fine surface finish requirements, blade ejectors or stripper plates eliminate pin marks entirely on cosmetic surfaces. A stripper plate adds roughly $2,000-4,000 to tooling cost but can be the difference between acceptable and premium surface quality on clear polycarbonate optics or high-gloss automotive trim. When pins are unavoidable, place them on non-cosmetic surfaces, ribs, or areas that will be post-processed. The minimum pin diameter should be 4 times the part's wall thickness at the eject

Industry Application Matrix

Class A surface + tight assembly tolerances PC, PEEK, PP (medical grade) Cleanroom molding, zero flash tolerance Thin walls (0.8-1.2mm) + cosmetic finish High-temperature molding, thick sections Cost Decision Framework: Core-Cavity Design Economics Tooling cost for a single-cavity mold with a planar parting line starts at $3,000-8,000 for simple geometries in P20 steel. Each side action adds $1,500-3,500. A stepped parting line adds 10-15% to the base tooling cost. For volume analysis, the break-even between a simple 2-plate mold and a complex multi-slide mold depends on annual volume: <5,000 parts/year: Simple tooling with manual deflashing. Tooling ROI at 3,000 parts. 5,000-50,000 parts/year: Mid-range tooling (P20/H13 steel) with automated ejection. Optimal tool life 250K-500K shots before major refurbishment at 40-60% of original cost. 50,000-500,000 parts/year: Premium tooling (H13/S136 hardened to 48-52 HRC) with conformal cooling. Tool life exceeds 1 million shots. Per-part amortization drops below $0.10 at 300K+ units.

Common Core-Cavity Defects and Solutions

Insufficient clamp force, worn parting line surfaces, or material viscosity too low at injection pressure Verify clamp force "e 3-5 tons per square inch of projected area. Restore parting line by spot-facing or re-grinding mating surfaces flat within 0.01mm. Reduce melt temperature 10-15°C if material is over-shearing. Insufficient draft, vacuum lock on deep cores, or excessive shrinkage gripping Increase core draft by 0.5-1.0°. Add air poppet valves ($150-300 each) to break vacuum on deep draws. Apply TiN or DLC coating (0.002-0.005mm thick) to reduce friction coefficient below 0.1. Consider core cooling delay of 2-4 seconds. Rib thickness exceeds 60% of nominal wall at attachment point Reduce rib root thickness to 50-60% of wall thickness. Add 0.5-1.0mm radius at rib-wall junction. Increase packing pressure by 15-25% and extend packing time by 2-3 seconds. If sink persists, add a subtle texture to the opposite surface. Differential cooling between core and cavity, uneven shrinkage from wall thickness variation

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/