

Six Early DFM Decisions That Define Injection Molding Success



Explore six key early decisions in DFM for injection molding to avoid common pitfalls, ensure production success, reduce costs and shorten timelines.

Before design for manufacturing (DFM) became widely adopted, the development path for injection molded parts often followed a rigid and inefficient journey. A 3D model and drawing arrived with a purchase order, the lead-time clock started and mold build advanced with the expectation that parts would be made exactly as specified. Changes were requested but repeatedly denied for a host of reasons ranging from not enough time to make changes or simply “I want what I want.” Inevitably, manufacturability problems surfaced after sampling, when designs were already difficult to change and teams were seeking to minimize lead time.

The results were predictable: parts stuck due to insufficient draft, thick walls caused warpage, gate changes were suggested and declined, and processing adjustments were the last line of defense to compensate for

Six decisions determine whether programs launch cleanly, validate quickly and stay in spec as volumes increase.

design limitations. Tolerances were relaxed just enough to pass inspection, while long-term production stability remained uncertain. This back-and-forth became accepted as normal until advances in tooling, materials and process control made it clear these pitfalls were avoidable. That realization shifted the industry from reactive troubleshooting to proactive DFM, a process that encouraged early decisions and collaboration and determined whether a part succeeds in production or struggles to get there.

As part of a successful DFM process, six early decisions are the keys to determining whether programs launch cleanly, validate quickly and stay in spec as volumes increase. They're purposefully interdependent because optimizing one decision in isola-

Successful molded part development programs apply DFM principles in the earliest stages of design. Source: Velocity

tion without considering how it impacts the others is not likely to lead to positive outcomes. Success within these areas will likely bring success through shortened timelines, reduced cost and increased odds that what works from the start will get you through to the finish.

1. PARTING LINE: SET THE STAGE FOR EVERYTHING ELSE

Every molded part needs a defined path out of the tool. Establishing a parting line early sets orientation, identifies where the draft begins, pinpoints impacts to functional and cosmetic surfaces and clarifies which features require movement and frames gating options.

Considerations

- Pull details into the line of draw whenever possible. This improves tolerance adherence and simplifies steel conditions.
- Model the parting line from the outset, especially on complex surfaces, to convey design intent and avoid late discoveries.
- Expect a witness at every steel intersection. Avoid placing tight tolerances directly across the split line because the witness can consume most of the allowed tolerance.
- Avoid splitting radii on the parting line when you can because it complicates the steel and increases flash risk.
- Use draft checks early and often, confirm parting line locations and highlight modeling problems before architecture is locked.

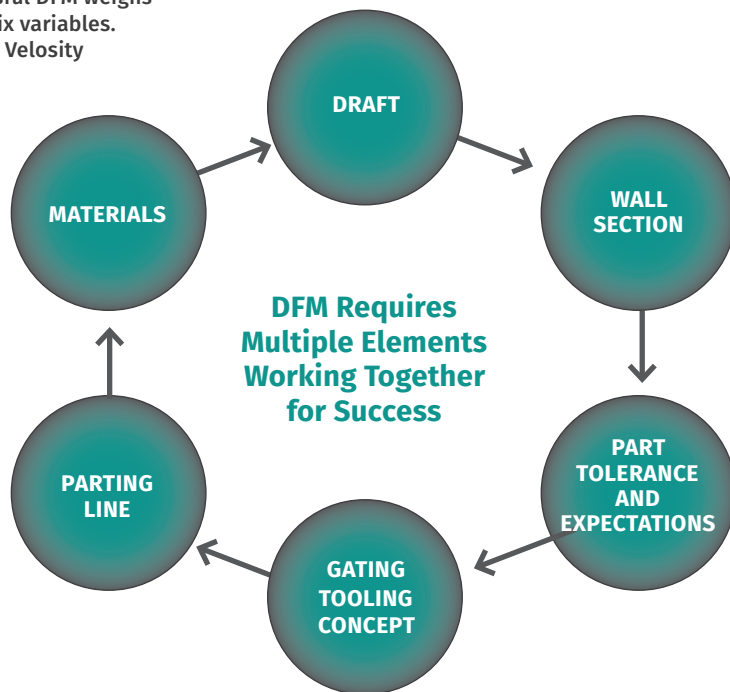
2. CHOOSING MOLD SIDES: DECIDE WHO HOLDS THE PART AND WHY

With the split defined, determine the orientation in the mold and which side retains the part on opening. This decision balances cosmetics, detail distribution, ejection feasibility and gating.

Considerations

- The A-side of the mold is usually the cosmetic or visual side of the part. The B-side is typically the ejector side and often contains the features that cause the part to stay in the mold when it opens.
- If the side that naturally holds the part cannot tolerate ejector marks or if the part is symmetrical and it's unclear which side will retain it, the design must intentionally guide the part to the correct side using draft, feature placement, additional ejection or surface finish.

Successful DFM weighs these six variables.
Source: Velocity



- For deep parts, draft alone may not be enough to release the part cleanly. Consider ejection methods early to prevent dragging, scuffing or part damage.
- On large parts or parts with long flow lengths, the chosen orientation should also support effective gate locations and material flow.

3. MATERIALS: SHRINK, FLOW AND CAPABILITY BEGIN HERE

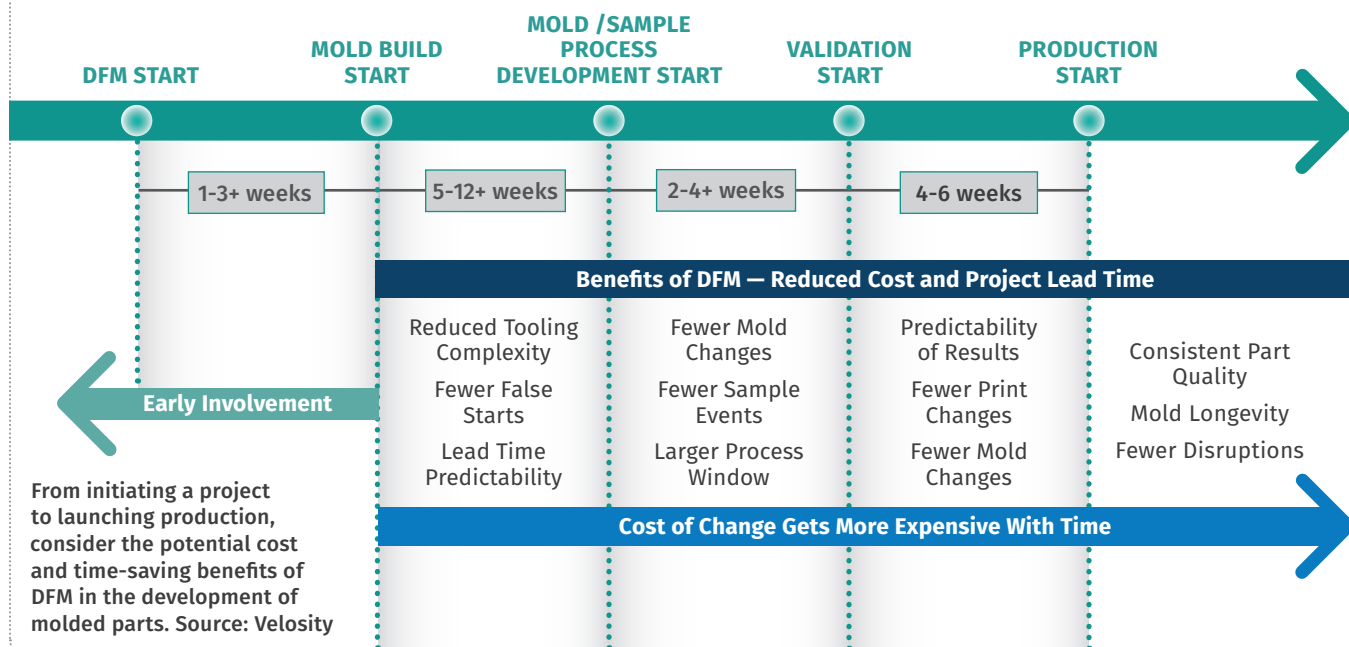
Plastic materials form the backbone of DFM. Resin selection influences many key molding factors, such as part tolerances, draft sensitivity and the ability to fill thin walls. For example, two similar designs can behave very differently in amorphous vs. (semi) crystalline resins. Balancing molding requirements with part function and commercial needs can be a challenge since no single material is suitable for every application.

Factors influencing material selection include mechanical properties, appearance, shrinkage, tolerance capability, viscosity and regulatory or agency ratings. All plastics shrink as they cool, and the uniformity of this shrinkage typically governs many aspects of part quality.

Considerations

- Choose materials early enough to guide wall thickness targets, gate type and placement, and suitability for cosmetic and tolerance requirements.
- Expect wall-dependent shrinkage: Thicker sections shrink more while thinner sections shrink less. The larger the part, the bigger the impact of small shrink differences. ►

Molded Part Development Timeline



- Shrink is influenced by material composition, part wall thickness, pressures needed to fill, gate size and mold/melt temperature, which are all variables that aren't fully known at concept, so designing with flexibility in mind is critical.
- For fiber-filled/crystalline resins, understanding gate placement and impact to flow direction and critical features will help manage expectations impacted by differential shrink and warp.

4. GATING AND MATERIAL FLOW:

DON'T TRADE FLOW FOR CONVENIENCE

Part orientation in the mold helps determine gating options, while material viscosity dictates how far the resin can flow through the part's wall thicknesses. Locating gates for convenience instead of flow can create compromises that are difficult to mitigate later.

Cosmetic and functional requirements must be balanced with material flow. Problems arise when flow takes a back seat to part requirements, or worse, when gating is not considered at all. Many part designers view gating as a minor annoyance with little impact on function. Gates can create higher-stress areas, cause blush that affects

Material	Shrink, in/in
ABS	0.004-0.006
Acetal	0.018-0.025
LCP	0.001-0.003
Polycarbonate	0.005-0.007
Polypropylene	0.010-0.025
TPE/TPU	0.015-0.026

Shrinkage rates for various resins. Source: Velocity

appearance, produce knit lines around cores that compromise both appearance and strength, and leave gate vestiges that interfere with fit or impede part function.

Considerations

- Flow from thick to thin. Avoid long, obstructed paths that force high pressure and create pack inefficiencies.
- Plan where knit lines land, keeping them off highstress or high-cosmetic regions when possible.
- Vestige is inevitable; choose gate type (e.g., valve, hot tip, edge, subgate) with removal method, automation potential and cycle time in mind.
- If the "ideal" location conflicts with aesthetics or assembly, identify the tradeoff early and consider minor geometry tweaks, texture strategies or finishing steps intentionally, not after T1.

5. DRAFT: EJECTION AND SHUTOFFS BOTH MATTER

Draft is not optional for molded parts, but the where and how much are context dependent. There are two distinct draft conversations: draft to release the part (ejection) and draft at the shutoffs where A- and B-side steels meet.

Considerations (ejection)

- Model draft into the CAD from the start; don't leave it as a print note. This conveys and preserves design intent and avoids miscommunication that can have real cost and lead-time implications or last minute thickness swings.
- As a baseline, a target of ~1 degree on polished molding surfaces should be the goal. For textured surfaces, a common rule of

thumb is ~1.5 degrees per 0.001 in. texture depth or a bit more on visual faces.

- Minimal draft can be tolerated more easily on the ejector side, but expect increased risk of scuffing, sticking and a narrower process window.

Considerations (shutoffs)

- Treat shutoffs as precision components: angle, mold design integrity and steel robustness govern long-term flash and maintenance.
- Butt/flat shutoffs run parallel to the parting line; slide-by/wrap-around shutoffs require an angle of ~5 degrees preferred; ~3 degrees may be acceptable in some cases. Anything less than that is prone to damage and long-term maintenance issues.
- Model the shutoff geometry and transitions so steps in the part are intentional and functionally/cosmetically acceptable.

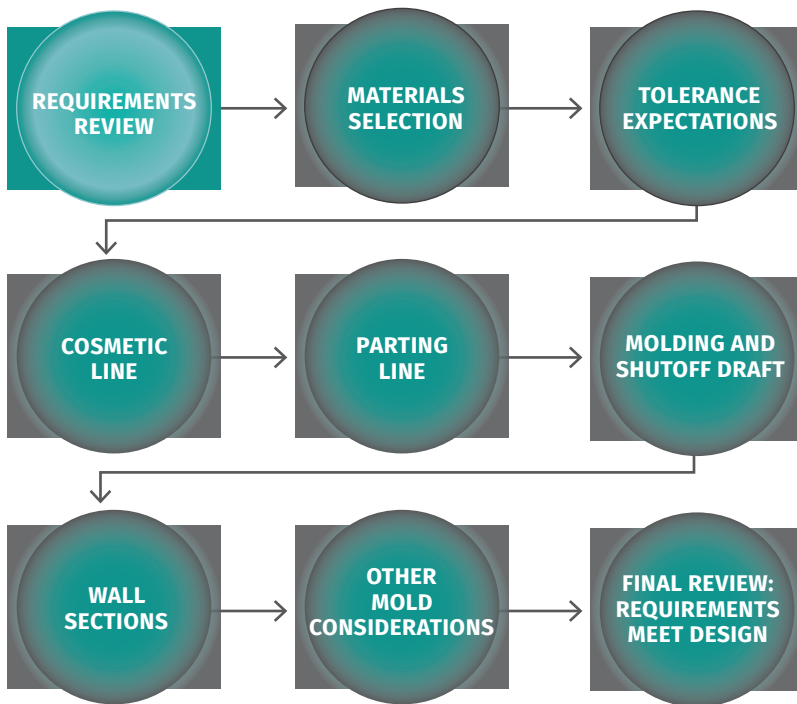
times, while walls that are too thin increase the risk of short shots, high injection pressure, flash and unstable processing.

Once nominal wall thickness is established, maintaining uniformity becomes the priority. Nonuniform walls produce nonuniform shrinkage, which can cause sink, warp, bubbles, voids, stress and dimensional variation. While perfect uniformity is rarely achievable, gradual transitions should be favored over abrupt changes.

Considerations

- Extreme thickness invites quality issues. Overly thick walls increase cooling time (part cost) and risk sinks/voids, while overly thin walls force high pressure and raise short shot/flash risk.
- Size ribs and bosses to ~66% of the main wall to which they're attached. For critical cosmetic faces, aim closer to ~50% while ensuring fill reliability.

- Watch bosses attached to curved surfaces. Walls can balloon in thickness, leaving few coring options and high sink potential.
- Core thick regions where feasible but avoid thin steel that creates hot spots or is damage prone.
- Balance wall thickness with draft so the resulting wall sections remain within targets after modeling.



Follow these steps to apply DFM to your next molded part development project. Source: Velocity

DFM RESOURCES AND STRUCTURED COLLABORATION

As product design and development teams navigate tighter schedules and increasingly complex geometries, aligning designs with injection molding requirements early in development has never been more important. A strong DFM approach ensures part intent matches molding realities, reducing risk, minimizing late-stage surprises and improving overall product performance. By reviewing materials selection, parting line strategy, molding and shutoff draft, and wall-thickness consistency early on, teams can resolve manufacturability concerns while changes are still fast and cost effective. [PT](#)

6. WALL SECTION: BUILDING ON THE BACKBONE OF THE PART

Wall thickness directly affects part strength, weight, cycle time and overall moldability. The nominal wall serves as the backbone of the part and defines the primary flow path for the material. As space constraints tighten and assemblies are pushed to be smaller and lighter, wall thickness limits are increasingly challenged, making balanced DFM decisions more critical. There is no single ideal wall thickness, but material viscosity, part size, flow length and performance requirements all influence where to start. Walls that are too thick invite sink marks, voids and longer cooling

ABOUT THE AUTHOR: Mike Fuller is the director of technical solutions at Velocity with more than 30 years of experience in engineering injection molded parts, spanning prototype development to high-volume manufacturing across medical, defense and consumer product sectors. He currently applies his expertise to advise on plastic part design for molding. Specializing in evaluating part designs for molding, he plays a pivotal role in ensuring the success of new product introductions. Velocity's Design for Manufacturing (DFM) Knowledge Center showcases the importance of designing early to deliver better device solutions. The hub features technical guidance, best practices and training materials that help teams ask the right questions at the right time. Learn more at velocity.com/resources/dfm-resource-center. Contact: (763) 657-4341; contactus@velocity.com