APRIOS CUSTOM MFG

A PROVEN PROCESS FOR DESIGN FOR MANUFACTURING (DFM)

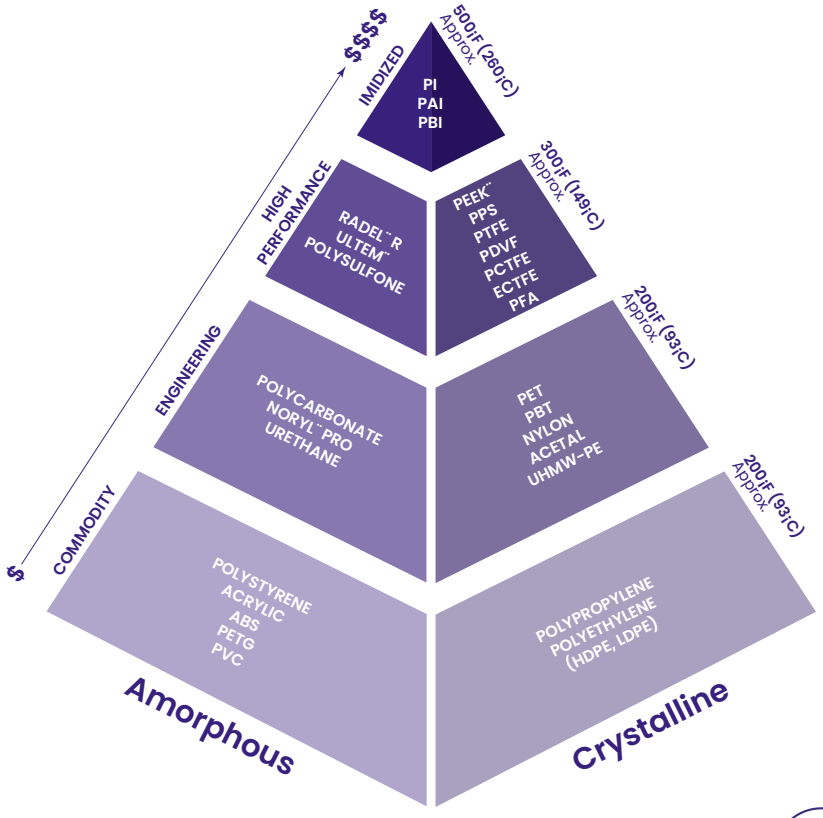
Parts designed for injection molding, whether simple or complex, can really benefit from the optimizations discovered through the design for manufacturability (DFM) process. DFM is a collaborative effort between engineering and production, ensuring that the project runs smoothly and results in high-quality parts with a great return on investment. Whether you're designing in-house or working with a design firm, bringing in an experienced injection molding and tooling production partner early can really streamline the process. DFM helps you avoid costly setbacks and delays, speeding up your time to market. While DFM for plastic injection molding covers many areas, there are a few key factors—like **material selection, wall thickness, corner radii, draft angles, and mold-flow analysis**—that can deliver immediate benefits to your project.

DFM for Injection Molding

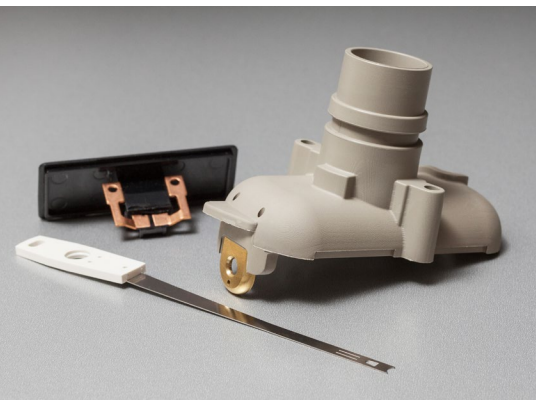
MATERIAL SELECTION

When selecting a material for a plastic injection-molded part, consideration needs to be given to the **end function and performance requirements**, such as:

- Flexibility and rigidity
- Tensile strength
- Temperature and chemical resistance
- Regulatory compliance
- Final part appearance and more



DFM for Injection Molding



Additionally, if the part requires special characteristics, your production partner will evaluate certain material characteristics such as:

- Melting point
- Cooling time
- Viscosity

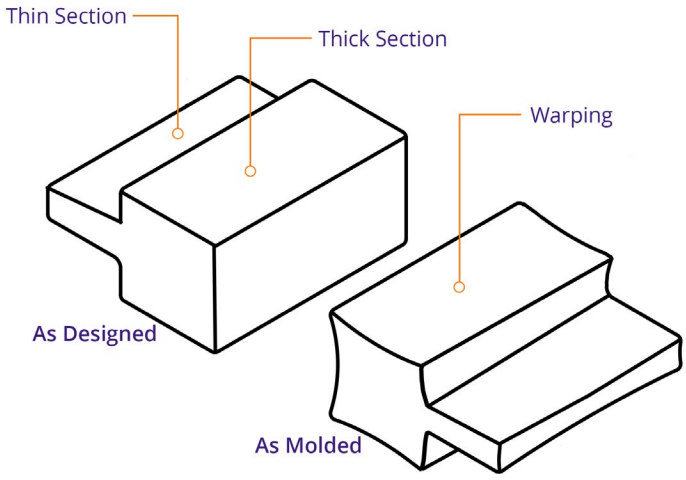
Material selection and a part's various wall thicknesses are also closely linked. For example, if a part has very thin wall sections or numerous areas that will be filled individually due to part geometry, the application may require a material with high-flow characteristics.

WALL THICKNESS

The part's geometry can influence material selection, and vice versa—material choice may affect how the part's geometry is designed.

STANDARD GUIDELINES ACROSS PLASTIC RESINS

- **Uniform Wall Thickness:** Keep primary wall thicknesses as uniform as possible to prevent issues like warping.
- **Avoid Warping:** Uniform cooling and shrinkage helps prevent twisting and curling due to non-uniform wall sections.
- **Wall Section Balance:** Avoid parts with only thick or only thin walls to prevent warping, sink, or cosmetic issues like dimples.



Warpage caused by non-uniform wall thicknesses

HANDLING THICK SECTIONS

- If necessary, core out thicker sections to maintain uniform wall thickness and add ribs for structural integrity.
- For internal features like bosses or ribs, wall thickness should be about two-thirds of the connected external wall thickness.
- Cored-out bosses help prevent issues.

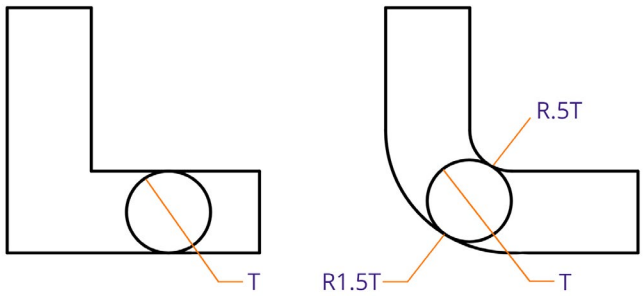
DFM for Injection Molding

CYCLE TIME CONSIDERATIONS

- Thick parts often result in longer cycle times, increasing production costs.
- Thin walls may cause difficulty filling the mold, leading to short shots and incomplete features.

CORNER RADII

As discussed, maintaining a consistent wall thickness throughout a part is critical to ensuring a high-quality product. Nowhere is this truer than around corners and wall transitions.

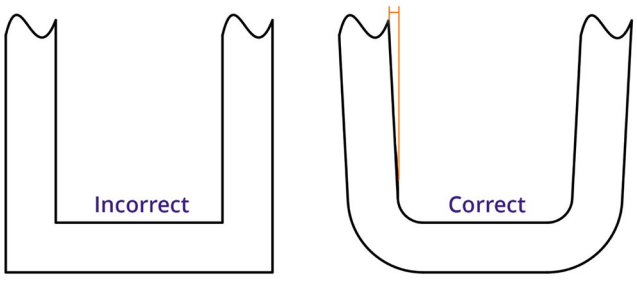


Corners should have a minimum fillet radius of 50 percent and an outer radius of 150% of the adjoining wall thickness.

- **Sharp Corners:** Sharp corners are generally impossible with most materials and geometries in injection molding, so planning for a radius is necessary.
- **External Radius:** Where possible, choose an external radius that maintains adjacent wall thicknesses to avoid issues.
- **Avoid Thick or Thin Transitions:** Transitions that are too thick or too thin can lead to warping or cosmetic problems.
- **Structural Strength:** A properly designed radius in wall transitions can provide better structural strength than transitions without or with too small a radius.
- **Ejection Benefits:** Parts with radii are easier to eject from the mold, improving the efficiency of the molding process.

DRAFT ANGLES

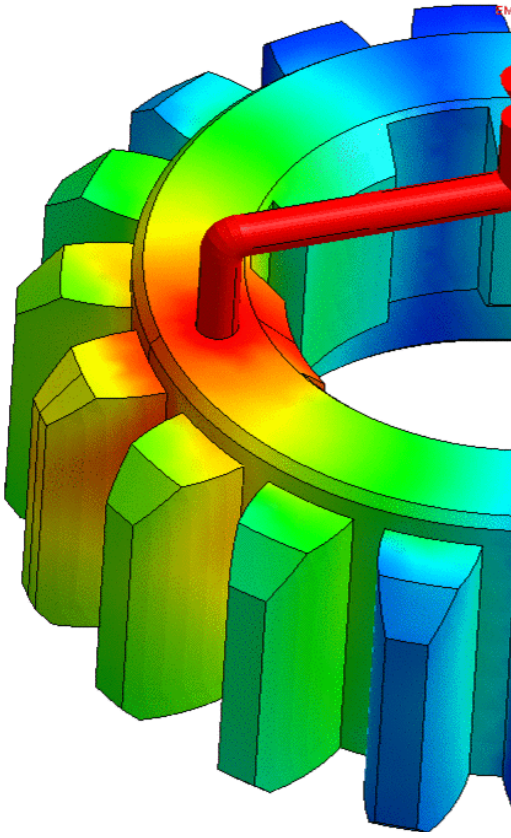
One key difference between injection molding as compared to other manufacturing methods, such as CNC machining or 3D printing, is the need to **physically eject the part** off the mold once the molding cycle has been completed. To efficiently release the part from the tool, it is imperative that it be drafted, or tapered.



One degree of draft is a good starting point

DFM for Injection Molding

When designing a part for injection molding, this should be considered throughout the process. Certain materials and geometries may require specific levels of draft. Typically, a good place to start is one degree per side. Areas where more draft may be required are textured walls or features where telescoping shutoffs are necessary. An improperly drafted feature or part can have cosmetic issues and may stick to the tool, causing the part and/or the tool itself to bend or break.



MOLD-FLOW ANALYSIS

One tool that helps to identify and mitigate common problems is a mold-flow analysis. Mold-flow analysis software simulates the production molding cycle of a part using a CAD file, an operator-chosen gate location and data of your selected material.

This simulation allows numerous parameters to be estimated and analyzed such as:

- Heating and cooling temperatures
- Part filling process
- Pressures required to fill the part

Running these simulations proactively addresses problem areas before they become an expensive and often time-consuming tooling change down the line. While these programs and simulations are immensely useful, they are technically complex and require subject-matter experts to perform. Most, if not all, tool design and injection molding firms will already have mold-flow analysis software in-house and experts on staff to analyze the results. This allows for suggestions and recommendations on how to achieve the highest-quality part.

DFM for Injection Molding

Partnering with an experienced injection molding and tooling production team early on can save you time and money, while accelerating your path to market. Aprios Custom Manufacturing offers valuable expertise for both short- and long-term success.

Visit aprios.com/dfm to see if your product qualifies for a free DFM consultation or request a quote for a DFM analysis. As an employee-owned company, Aprios specializes in Design for Manufacturing (DFM) and Design for Additive Manufacturing (DFAM) to help optimize your designs for efficient production.

We focus on high-precision, close-tolerance plastic injection molding and digital manufacturing for medical devices, aerospace, filtration, and more. With our Acceleration Station™ services, we offer the fastest route to market. Since 1977, Aprios has been providing trusted, full-service contract manufacturing solutions from design to clean room assembly.

CONTACT US

sales@aprios.com
1.763.424.2525

To schedule a call, please scan the QR code below.

https://www.aprios.com/meetings/nerickson1/dfm-consultation?uuid=3a754d1b-fd97-4ae7-8442-e8aef-558d8a4