

# Design Best Practices for Flawless Injection Molded Components

Injection molding is one of the most cost-effective manufacturing methods, but only if your design is optimized for the process.

Delivering Quality,  
Innovation, and  
Reliability for Your  
Manufacturing Needs

✍ No MOQs

📧 Quote within 48 hours

~ Vertically Integrated

## 1 Choose the Right Material from the Start

- Polypropylene → lightweight, chemical-resistant, cost-effective
- ABS → rigid, strong, easy to process
- Nylon → impact-resistant, excellent for wear-and-tear applications

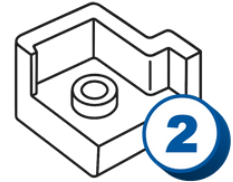
👉 **Pro Tip:** Each resin shrinks differently during cooling.



## 2 Maintain Uniform Wall Thickness

Consistent wall thickness = consistent quality.

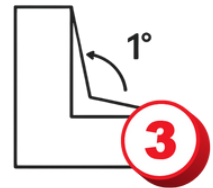
- Target 2–4 mm for most materials.
- Use ribs, gussets, or bosses to add strength without bulky walls.



## 3 Add the Right Draft Angles

Draft angles are essential for smooth part ejection from the mold.

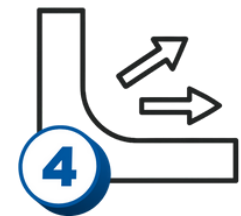
- Smooth surfaces: at least 1° per side
- Textured surfaces: 3° or more



## 4 Eliminate Stress Concentrations

Sharp corners and abrupt changes in geometry can create weak points. Instead, use fillets and radii to spread stress evenly.

- Add fillets (0.5–1 mm radius minimum) at inside corners.
- Use ribs to strengthen flat areas.
- Support bosses properly to prevent cracking.



## 5 Plan Ahead for Gates & Parting Lines

Gate location affects how resin flows into the mold, which impacts both performance and appearance.

- Place gates in non-visible areas.
- Collaborate early with your mold maker to reduce weld lines.
- Consider parting line location to minimize flash and cosmetic concerns.



## 6 Validate with Simulation & Prototyping

Don't wait until production to discover flaws. Testing early saves time and money.

- Moldflow analysis → simulates resin flow to predict warping, short shots, or weld lines.
- 3D printed prototypes → validate geometry and fit before investing in tooling.

