10 TIPS FOR PLASTIC INJECTION MOLDING

PART DESIGN

Designing a plastic part or component to be produced by injection molding involves an assortment of considerations. While it is a complex undertaking that requires greater instruction than can be provided here, what follows are some simple tips that should help.

1 MANUFACTURABILITY
Optimize part design for ease, speed and effectiveness of manufacturing processes, including tooling, molding and finishing

2 MOLDABILITY
Consider how well a mold will work being injected with molten plastic, and whether the part can be ejected effectively

3 WALL THICKNESS
Less than 0.25” preferably; core thicker walls; maintain uniform thickness; smooth transitions for necessary variations

4 DRAFT ANGLE
Include at least a minimum draft angle (½˚) for easier part ejection; some textured surfaces require greater draft angles

5 AVOID SHARP EDGES & CORNERS
Use fillets with inside radii of at least 50% of nominal wall thickness; outside radii should be 150% of nominal wall thickness

6 RIBS, BOSSES & GUSSETS
Max thickness should be 50% to 60% of the nominal wall; use ribs, bosses and gussets for wall and part stability

7 THREADS
Add radius to crests and roots; avoid thin edges at ends; 32 threads per inch max; consider part ejection for internal threads

8 MATERIAL
Only some plastics will meet certain requirements; consider the resin shrinkage; additives can be used, if necessary

9 AVOID UNDERCUTS
Protrusions or indentations that impede ejection will require mechanical side actions in the mold, if possible

10 MISCELLANEOUS
Surface finish or texture; gate location; tolerances and critical dimensions; secondary operations and assembly

QUESTIONS?
Contact us to speak with a consultant → 909-981-9662

PRECISION MOLDED PLASTICS

PrecisionMoldedPlastics.com