



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

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

2

MOLDABILITY

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

7

THREADS

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

3

WALL THICKNESS

Less than 0.25" preferably; core thicker walls; maintain uniform thickness; smooth transitions for necessary variations

8

MATERIAL

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

4

DRAFT ANGLE

Include at least a minimum draft angle ($1/2^\circ$) for easier part ejection; some textured surfaces require greater draft angles

9

AVOID UNDERCUTS

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

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

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

© 2022 Precision Molded Plastics, Inc. All Rights Reserved.