Injection molding DFM checklist

Nine things worth checking on a plastic part before the STEP file leaves your computer, with the number each one is measured against. These are the thresholds the Fabdose analysis engine loads out of its own rule files.

SIX A GEOMETRY CHECK CAN SETTLE decided from the shape alone, face by face

1. **Draft angle, every face.** 2° smooth, 3° textured. Parting surfaces and undercut faces are exempt, they were never meant to carry draft.
2. **Undercuts, by severity.** Under 5° usually ignorable. 5 to 15° lifter. 15 to 30° slide core. Past 30° redesign, or a mold technique that shows on the quote.
3. **Wall thickness vs. the resin.** ABS bottoms out at 1.14mm, PP at 0.635mm, PEEK at 0.508mm. A 1.0mm wall is fine in PP and too thin in ABS. Table below.
4. **Wall thickness uniformity.** Within 30% of the part's *dominant* thickness, not the average. Ribs built at 40 to 60% of nominal are excluded, not flagged.
5. **Flow length ratio.** Length over wall thickness. Above 75 is thin-wall molding: hard to fill, short shots appear.
6. **Shrinkage.** Above 1.5% typical gets a dimensional compensation note. POM at 1.8% and PP at up to 2.0% are the two that trip it.

THREE IT CANNOT, SO CHECK THEM YOURSELF on an injection molded part Fabdose does not flag these from the geometry today

7. **Ribs.** 40 to 60% of nominal wall, spaced 3x wall or more, no taller than 3x wall. Past 60% you are designing a sink mark into the face on the other side.
8. **Bosses.** Wall at 60% of nominal or under. OD 2.0 to 2.5x the ID. Same sink mark logic, except a boss usually sits where you can see it.
9. **Holes and internal corners.** Depth under 3x diameter, min diameter 0.5mm. Internal R at least 0.5mm, ideally half the wall. External at least 0.3mm.

WALL THICKNESS AND SHRINKAGE, BY RESIN wall figures are what your part is compared against; shrinkage is a note, not a limit

Resin	Min wall	Max wall	Shrinkage	Resin	Min wall	Max wall	Shrinkage
ABS	1.14 mm	3.56 mm	0.7 to 1.6 %	**PA**	0.76 mm	2.92 mm	0.3 to 1.2 %
PC	1.016 mm	3.81 mm	0.5 to 0.7 %	**POM**	0.76 mm	3.05 mm	0.4 to 2.0 %
PC/ABS	1.14 mm	3.56 mm	0.2 to 0.5 %	**PMMA**	0.64 mm	3.0 mm	0.2 to 0.5 %
PP	0.635 mm	3.81 mm	1.0 to 2.0 %	**PEEK**	0.508 mm	3.0 mm	1.0 to 1.7 %
TPU	0.64 mm	3.0 mm	1.0 %	**TPE**	0.64 mm	3.0 mm	Not available

Where these numbers come from. They are read straight out of the rule files that ship inside the Fabdose desktop app: the injection molding rule set for the process limits, and the materials file for the per-resin figures, which were compiled from resin supplier datasheets. None of it is an average of what other sites publish. Millimetre values are unrounded because several were converted from imperial datasheet figures. PC shrinkage is for unfilled grades, glass filled runs lower. TPE varies too much by type for one number to mean anything. If a figure here disagrees with your molder, your molder is talking about your tool, your gate and your resin lot, and they win.

Run the first six on your own part. Fabdose reads a STEP file on your computer and names the faces that fail, by face number and by location in the model. First five analyses free, no card. fabdose.app/download How the engine is tested: fabdose.app/verification