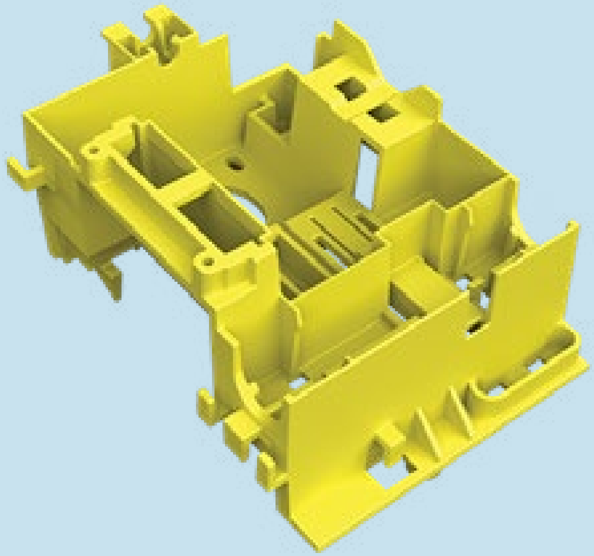
DESIGN

11 Design Tips to Reduce Manufacturing Costs

PROTOLABS

Injection Moulding

Everyone wants to save money on manufactured parts, and depending on what process you're designing for and where you are in your product life cycle, cost can be removed through well-designed parts. Here are a handful of helpful tips to help you to design cost-efficient parts for any manufacturing process at Protolabs.



1 Keep an Eye on Undercuts

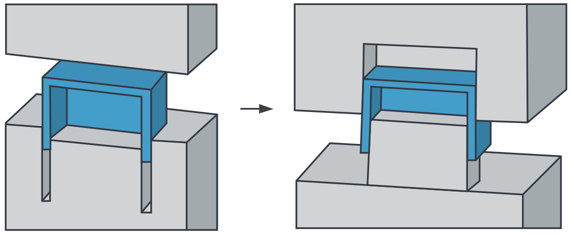
Undercut features complicate and, in some cases, prevent part ejection. Get rid of them if you can, but maybe that's not possible, if, for example, you need a side action, sliding shutoff or pick out.
One alternative may be using sliding shutoffs and pass-through cores, or by changing the parting line and draft angles to provide an easier mould build. These reduce tooling costs as you avoid additional pieces to the mould that add to manufacturing costs. In addition to the rise in manufacturing costs of using hand-loaded inserts, this also may have an impact on your piece part price because of longer cycle times and manual mold operation.

2 Avoid Unnecessary Features

Textured surfaces, moulded part numbers, and company logos look cool, but be prepared to pay a bit extra for these and other non-mission critical features. That said, permanent part numbers are a requirement for many aerospace and military applications. Use a mill-friendly font such as Century Gothic Bold, Arial, or Verdana (san-serif fonts), keep it above 20 pt., and don't go much deeper than 0.25mm - 0.38mm. Also, be prepared to increase draft if part ejection is a concern.

3 Use Core-Cavity Method

If you need an electronics housing or similar box-shaped part, you can either sink the wall cavities deep into the mould base, requiring long thin tools to machine ribs into the mould, or machine the aluminium material down around the core and mould the part around it. The latter approach is known as a core cavity, and is a far more cost-effective method of moulding tall walls and ribbed surfaces. Better yet, this makes it easier to provide smooth surface finishes, adequate venting, improved ejection, and can eliminate the need for super-steep draft angles.

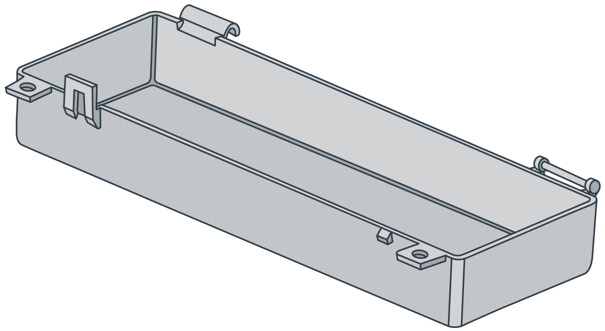


▶ **Using a core cavity, as shown, can be a cost-effective method of moulding tall walls and ribbed surfaces.**

4 Try Self-Mating Parts

Maybe you're designing a snap-together case for some medical components, or two interlocking halves of a portable radio. Why build two mating parts when you can make one? Redesign the snaps so that the halves can be fit together from either direction, thus building a so-called "universal" part. Only one mould is needed, saving production expenses up front. And you can now mould twice as many of one part, instead of half the quantities of two.

Maybe you are after a higher volume of parts? You can still achieve high volumes using aluminium tooling with two-, four-, or eight-cavity moulds depending on size and part geometry. That can reduce your piece part price, although this would impact your tooling costs.



▶ **This is an example of one half of a self-mating part, which fits together in either direction with its other half, building a "universal" part.**

5 Leverage Multi-Cavity and Family Moulds

Got a family of parts that all fit together? How about multiple moulding projects at one time? There's no reason to build a mould for each individual part, provided A) everything is made of the same plastic, B) each part is roughly the same size (e.g., have similar processing times), and C) all elements be squeezed into the same cavity, while still allowing for proper mould functioning.

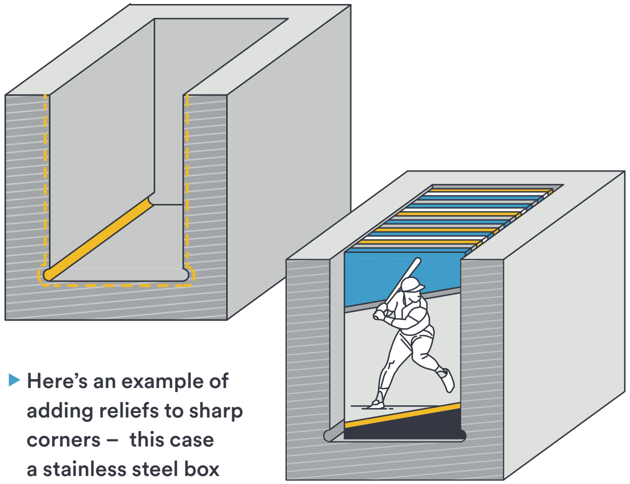
In addition, maybe you can join some of those parts with a living hinge? This method is a great way, for example, to mould two halves of a clamshell-style container. These parts would otherwise need a pin-type assembly to open and close. The only caveat here is that a flexible and tough material must be used, such as polypropylene (PP).

CNC Machining

1 Provide Relief to Corner Pockets

Consider the corners of a machined pocket – perhaps the inside of an electronics housing, or a bracket used to capture the body of a rectangular component. One common design oversight is leaving the intersection of the vertical walls on those part features perfectly sharp. To illustrate, think about machining a stainless steel box. The only way to get the perfectly square vertical corners needed is with electrical discharge machining (EDM), or, multiple flat plates bolted together. Both can be slow and expensive processes.

Instead, we'll equip one of our machining centers with the smallest end mill available to clean out the corners. In 304 stainless steel, that means a 0.8mm end mill, which leaves a corner radius of 0.4mm. That's pretty sharp, but the depth is limited. The length of most steel-cutting end mills in this size range maxes out at five times the cutter. Machining with small end mills such as this is also slow and delicate work, driving up the cost of your project because of added milling time.



▸ Here's an example of adding reliefs to sharp corners – this case a stainless steel box that holds a collection of baseball cards – which helps achieve the perfectly square corners needed to fit the cards.

A more budget-friendly approach is machining a relief in each corner of the pocket. This removes that pesky radius, leaving a U- or C-shaped clearance instead. It also allows for far deeper pockets – by cutting a 6.35mm wide relief in each corner, functionally sharp corners to around 32mm in depth are possible. And by switching to aluminium or even plastic, pocket depths twice that of steel are possible. Best of all, designing pockets in this manner reduces part cost, because larger end mills can be used and material removal rates increased accordingly.

2 Avoid Text Until Moulding

Similarly, text engraving is an aesthetically pleasing but time-intensive operation, one that might be best to avoid if possible. Here again, a ball end mill is used to trace whatever letters, numbers, and symbols are called for on the CAD model. It looks cool, and might be a valid requirement on your machined part, but it's probably more appropriate on injection-moulded parts, where additional machining time is amortized over higher part volumes. Because of our toolsets for metals vs. plastics, we have a minimum feature size of 0.90mm in metals and 0.51mm in plastic.

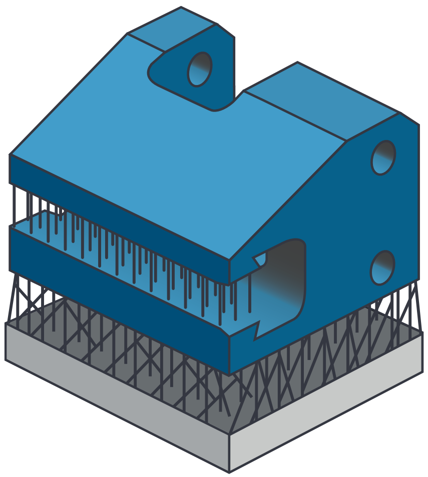
3 Be Cautious of Thin Walls and Features

Our standard part tolerance is +/- 0.13mm. If you have a feature that is 0.51mm or smaller, our automated quoting system will highlight it as a thin-wall geometry, but keep in mind that we'll still allow it to be machined – so the machined part may differ slightly from your original design. Any thin walls that are 0.5mm. or less are not only subject to breakage during the machining operation, but may flex or warp afterwards. Beef them up as much as your part design allows.

3D Printing

1 Optimise the Design

Well-designed 3D-printed parts follow many of the same rules as those made with injection moulding. Use gradual transitions between adjoining surfaces. Eliminate large differences in cross-section and part volume. Avoid sharp corners that often create residual stress in the finished workpiece. Watch that thin unsupported walls don't grow too tall, or else buckling or warping may occur. Also, surfaces with shallow angles tend to leave ugly "stair-stepping" that makes them unsuitable for cosmetic features – flatten them out where possible.



Support structures on DMLS parts help prevent curl that can occur during the sintering process.

2 Throw Out Tradition

The most dramatic 3D printed part designs leverage 3D's ability to create "organic" shapes, such as honeycombs and complex matrices. Don't be afraid to use these shapes, provided doing so creates a lighter, stronger part. Nor should you fear placing holes – lots of them – into your part design. With traditional manufacturing, drilling holes in a solid block of material increases part cost and waste. Not so in the additive world, where more holes mean less powder and less processing time. Just remember, 3D printed holes don't need to be round. Quite often, an elliptical, hexagonal, or free-form hole shape would better suit the part design and be easier to print.

3 Consider Next Steps in the Design Cycle

Just because you can print parts with lots of holes, however, doesn't mean you should, especially if the plan is to make lots of such parts later on. Because 3D printing offers tremendous design flexibility, it's easy to paint yourself into a corner by not considering how parts will be manufactured post-prototyping. Based on our examples at the start of this design tip, an increasing number of companies are finding 3D printing suitable for end-use parts, but many parts will transition from printing to machining, moulding, or casting as production volumes grow. That's why it's important to perform a design for manufacturability analysis early on in the design cycle, assuring cost-effective production throughout the part's life cycle.

Read Full Cost-Saving Design Tips Online By Service

- Injection Moulding
- CNC Machining
- 3D Printing

PROTOLABS

Contact Us

Proto Labs, Ltd.
Halesfield 8,
Telford,
Shropshire,
TF7 4QN

P +44 (0) 1952 683047
E customerservice@protolabs.co.uk