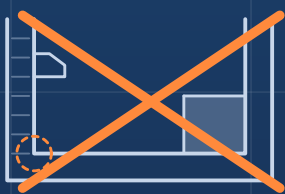


9 RULES BEFORE YOU PAY FOR A MOLD

ENCLOSURE CHECKLIST

Nine reasons the enclosure you 3D printed cannot be mass produced, and how to fix every one of them before you commit thousands of dollars to an injection mold.



PRINTS FINE
CAN'T BE MOLDED



MOLD READY
SHIPS IN VOLUME

PREPARED BY
JOHN TEEL

DOC NO.
ENC-09

REV
A

SOURCE
[PREDICTABLEDESIGNS.COM](https://predictabledesigns.com)

Enclosure Design Manufacturability Checklist

Nine reasons a 3D printed enclosure cannot be mass produced, and the fix for each

The enclosure you 3D printed is likely impossible to mass produce. Your printer will happily build shapes all day long that no factory could ever produce in volume, and these problems usually cannot be patched at the end. I have seen plenty of designs that had to be completely redesigned before they could be manufactured, which means you need to know these rules while you are still designing your part.

This checklist covers nine reasons a 3D printed enclosure cannot be mass produced, counting down to the mistake that multiplies your entire manufacturing bill. For every reason you will find what goes wrong in the mold and exactly what to do instead, so run through the whole list before you commit thousands of dollars to tooling.

THE NINE REASONS AT A GLANCE

#	WHAT YOUR PRINTER LET YOU DO	WHAT IT COSTS YOU IN THE MOLD
#9	No draft angle on the walls	Scratched parts that refuse to eject
#8	Non-uniform wall thickness	Warping, sink marks, and hidden voids
#7	No support ribs	A thin shell that flexes like a cheap toy
#6	Sharp internal corners	Cracks in the part and a mold that wears out
#5	Snap fits tuned on your printer	Snap fits that will not close in production plastic
#4	Nowhere for the mold to leave its marks	A parting line across the product's face
#3	Ignoring how the plastic flows	Incomplete parts and weak, visible weld lines
#2	Undercuts	Side actions that add cost and wear out
#1	Too many separate plastic pieces	A separate mold, and a separate bill, for each

HOW TO USE THIS CHECKLIST

The reasons run in countdown order from #9 to #1, the same order as the video. For every one you will find what goes wrong in the mold, the exact fix, and how to apply it while you are still designing your part.

#9

Never Leave the Walls Without a Draft Angle

PART EJECTION

WHAT GOES WRONG IN THE MOLD

- Mass producing a plastic enclosure means injection molding, where melted plastic gets injected into a steel mold, and that one mold can produce hundreds of thousands of parts.
- Your 3D printed enclosure almost certainly has perfectly vertical walls, which was no problem for your printer.
- In a mold, every part has to slide out, and a vertical wall drags against the steel the whole way out, scratching the surface and sometimes refusing to eject at all.

DO THIS INSTEAD

THE FIX	HOW TO APPLY IT
Add a degree or two of draft	Draft is just a slight taper on every wall that runs in the direction the mold opens. A degree or two is the usual starting point.
Add more draft on textured surfaces	If the surface is textured instead of smooth and polished, you will need a little more draft for the part to release cleanly.

ONE MOLD, HUNDREDS OF THOUSANDS OF PARTS

Every rule in this checklist exists because a mold has to release the part cleanly every single cycle, and a problem like this usually cannot be patched at the end. It has to be designed out before the steel is cut.

#8

Never Let the Wall Thickness Vary

WALL DESIGN

WHAT GOES WRONG IN THE MOLD

- Injection molded plastic shrinks as it cools, and thick sections cool a lot slower than thin ones.
- That uneven cooling is what makes parts warp, and it creates sink marks, which are the small dents that show up on the surface behind every thick feature.
- Really thick sections can even form voids, which are hollow pockets inside the plastic where the material pulled apart as it shrank.
- Your printer never showed you any of this, because a printed part never goes through that shrinking. A chunky 3D printed design comes out perfect, and the exact same design comes out of a mold warped and dented.

DO THIS INSTEAD

THE FIX	HOW TO APPLY IT
Keep the main wall thickness uniform	Keep your main wall thickness as uniform as possible, usually in the 2 to 3 mm range for most plastics.
Core out thick areas from the back	If you need a thick area for strength, do not make it one solid mass. Core it out from the back so the walls stay consistent, then get the strength back with ribs.

#7

Never Thin the Walls Without Adding Ribs

STIFFNESS

WHAT GOES WRONG IN THE MOLD

- Your 3D printed prototype was probably built with thick walls and solid sections, because thick sections cost you nothing but a little extra filament.
- Then you thin everything down to uniform 2 to 3 mm walls for molding, and the whole part turns into a thin hollow shell.
- On top of that, molded ABS is noticeably less stiff than the PLA most people print with, so a design that feels rock solid coming off your printer can flex like a cheap toy coming out of a mold.

DO THIS INSTEAD

THE FIX	HOW TO APPLY IT
Add ribs inside the part	Ribs are thin walls of plastic added to the inside of the part that act like structural beams. They give you the stiffness back without thickening the walls.
Use gussets on tall features and corners	Gussets, which are small triangular braces, do the same job for tall features and corners.
Keep each rib to half the wall thickness	Keep each rib to about half the thickness of the wall it attaches to. Make a rib too thick and it creates the exact sink mark problem from the last section, right on your cosmetic surface.

THE ONE PLACE YOU WANT THINNER PLASTIC

A thin rib might sound like it breaks the uniform wall rule, but ribs are the one place you want thinner plastic on purpose. The spot where a rib meets the wall is thicker than either one by itself, so a thin rib keeps that spot from turning into a thick section.

#6

Never Leave a Sharp Internal Corner

STRESS AND TOOLING

WHAT GOES WRONG IN THE MOLD

- Sharp corners are a double mistake, one in the plastic part and one in the steel mold.
- In the plastic part, a sharp inside corner concentrates stress, so when the part gets dropped or flexed, that corner is where the crack starts.
- In the steel mold, a sharp corner on your part means a sharp corner cut into the steel, which costs more to machine and wears out faster than a rounded one.
- Your printer renders corners fine either way, so nobody ever thinks about them.

DO THIS INSTEAD

THE FIX	HOW TO APPLY IT
Round every corner, inside and out	A good starting rule is an inside radius of at least half your wall thickness.
Match the outside radius to the wall	Make the outside radius equal to the inside radius plus the wall thickness. That keeps the wall thickness constant as it wraps around the corner, so the corner itself does not become the thick section you just worked to eliminate.

#5

Never Trust a Snap Fit You Tuned on Your Printer

ASSEMBLY

WHAT GOES WRONG IN THE MOLD

- Everyone has done this. You print a part, test the snap fit, tweak the model by a fraction of a mm, and print it again until it closes with a perfect click.
- All of that tuning was specifically for your printer and the filament or resin you printed with. Production molded ABS or polycarbonate has different stiffness, different friction, and different shrinkage, so the snap that works perfectly on your printer might not close at all in the production plastic.
- Printed snaps tend to break along the layer lines, so people beef them up to survive on the printer, which leaves them too stiff to flex properly once they are molded in tougher plastic.

DO THIS INSTEAD

THE FIX	HOW TO APPLY IT
Use screws unless you truly need snaps	Snap fits can take a lot of iterations to get right even at the mold level, so if you do not absolutely have to have them, screws tend to be simpler.
Test critical snaps in the real plastic	CNC machining is a good way to do that, because you can have a prototype machined from real ABS or polycarbonate and get a much better feel for how the production snap will behave.
Leave room to adjust the fit	Leave yourself some room to adjust the fit after the first molded samples come back.

A HARDWARE ACADEMY MEMBER'S CALL

A member of my Hardware Academy ran into exactly this issue with snap fits, and after our review he decided to go with three plain screw holes instead, to reduce the amount of snap fit optimization needed.

#4

Never Ignore Where the Mold Leaves Its Marks

COSMETICS

WHAT GOES WRONG IN THE MOLD

- Every injection molded part comes out of the mold with marks on it. There is a parting line where the two mold halves meet, small circles where the ejector pins pushed the part out, and a scar where the plastic was injected.
- Those marks are not a defect. They are just part of the injection molding process, and they have to land somewhere.
- Your printer leaves none of these, so people design as if they do not exist. Then the first molded samples show up with a parting line running straight across the product's face and ejector circles right where the customer's thumb rests.

DO THIS INSTEAD

THE FIX	HOW TO APPLY IT
Decide where the marks go yourself	Decide where these marks go instead of letting the molder decide for you.
Run the parting line along a natural edge	Run the parting line along a natural edge of the design, where it disappears into the shape instead of cutting across a face.
Hide the ejector pins and the gate	Keep the ejector pins and the injection point on surfaces the customer never sees, and settle all of it with your molder before the design is final.

#3

Never Design Without Thinking About Plastic Flow

MOLD FILLING

WHAT GOES WRONG IN THE MOLD

- Molten plastic enters the mold through a small opening called the gate, and from that point it has to race outward and fill every corner of the cavity before it freezes.
- Put the gate far away from a thin section, and the plastic solidifies before it gets there, leaving you with what molders call a short shot, which is a part that is literally incomplete.
- When the flow splits around a hole and meets up again on the other side, it forms a weld line, which is a visible seam that is also the weakest spot in the part.

DO THIS INSTEAD

THE FIX	HOW TO APPLY IT
Plan the gate while you design	Think about where the gate will go while you are still designing, and keep the path from the gate to the farthest thin feature as short as you can.
Talk to your molder before the design is final	Moving the gate also moves where the weld lines land, so a quick conversation with your molder before the design is final can keep them off the surfaces your customer sees.

WHY THE GATE IS A DESIGN DECISION

The gate location decides both whether the part fills completely and where its weakest seam ends up. That is why it belongs in your design process, and not just in the molder's setup after the design is final.

#2

Never Accept an Undercut Without Trying to Remove It

TOOLING

WHAT GOES WRONG IN THE MOLD

- An undercut is any feature that traps the part in the mold, like a snap fit, a side hole, or a groove around the outside of the part.
- Your printer builds these without any problems, because 3D printing does not require the part to be removed from a mold. A basic mold only opens in one direction, and an undercut physically locks the part inside.
- Molders can solve this with side actions, which are extra pieces of the mold that slide in from the side and pull out of the way before the part ejects. They work, but every side action adds cost to the mold, adds a mechanism that wears out, and adds one more thing that can go wrong on every single cycle.

DO THIS INSTEAD

THE FIX	HOW TO APPLY IT
Redesign the feature so it is not an undercut	This is possible more often than people think, so try it before you accept a side action.
Put a hole in the wall below the undercut	This one is less obvious. Put a hole in the wall directly below the undercut, and the mold can form the feature through that hole with no side action needed.

THE TRICK THAT HAS SAVED THOUSANDS IN TOOLING

That one hole has saved people thousands of dollars in tooling, and once you know about it, you will start spotting those holes under the snap fits in commercial products everywhere.

#1

Never Design In More Plastic Pieces Than You Need

TOOLING COST

WHAT GOES WRONG IN THE MOLD

- Every other mistake on this list relates to a single part. This one multiplies across your whole product, and it does the most financial damage.
- Your printer makes a five-piece enclosure feel free, because printing five parts costs about the same as printing one.
- Every separate plastic piece in your design needs its own mold, and every mold costs thousands of dollars, often tens of thousands.

DO THIS INSTEAD

THE FIX	HOW TO APPLY IT
Design for the fewest separate pieces	Design your product to require the fewest separate plastic pieces possible, and combine parts wherever you can.
Use a stock part wherever one will do	Use a stock part wherever one will do the job, so you are not paying for a mold you do not need.
Ask your molder about a family mold	If you are stuck with a few small pieces, ask about a family mold, which forms several different parts in one mold as long as they are all about the same size.

WHAT THE MOLDS FOR MY OWN PRODUCT COST

The molds for my own product cost over \$100,000. My manufacturer believed in the product enough to finance that cost themselves and amortize it by charging me an extra dollar per unit for the first 100,000 units. That deal saved my launch, but it also taught me exactly what every extra plastic piece costs.

Run This Checklist Before You Pay for a Mold

Nine boxes to tick before your enclosure design goes to tooling

- ☐ **Draft:** Every wall that runs in the direction the mold opens has a degree or two of draft, with a little more on any textured surface.
- ☐ **Wall thickness:** The main walls are uniform, usually 2 to 3 mm, and every thick area is cored out from the back instead of left as a solid mass.
- ☐ **Ribs:** Stiffness comes from ribs and gussets on the inside of the part, with each rib at about half the thickness of the wall it attaches to.
- ☐ **Corners:** Every corner is rounded inside and out, with an inside radius of at least half the wall thickness and an outside radius equal to the inside radius plus the wall.
- ☐ **Snap fits:** Any snap fit you cannot replace with screws has been tested in the actual production plastic, with room left to adjust after the first molded samples.
- ☐ **Mold marks:** The parting line runs along a natural edge, and the ejector pins and the injection point sit on surfaces the customer never sees, all settled with your molder.
- ☐ **Plastic flow:** The gate location is planned, the path from the gate to the farthest thin feature is as short as possible, and the weld lines land off the visible surfaces.
- ☐ **Undercuts:** Every undercut has been redesigned away or formed through a hole in the wall below it, and any side action that remains is a deliberate choice.
- ☐ **Piece count:** The product uses the fewest separate plastic pieces possible, with stock parts wherever they will do the job and a family mold considered for the small pieces.

Get Experienced Engineers On Your Enclosure Design

If you want experienced engineers reviewing your enclosure design before you commit thousands of dollars to a mold, you can get that kind of review from me and other experts inside my Hardware Academy.

[JOIN AT PREDICTABLEDESIGNS.COM/ACADEMY](https://predictabledesigns.com/academy)