Injection molding design guidelines

stratasys
DIRECT MANUFACTURING

Contents

Injection molding design guidelines

When your product is ready for production, your biggest concern is most likely lead time. That's understandable – it can make a sizeable impact on meeting your launch deadlines and budget. Designing for manufacturability right from the start will heavily influence how fast your product goes to market. The design not only needs to look and function as it should but also be able to stand up to the stresses of manufacturing. If it can't, expect expensive and time-consuming reworks that can spike the budget and delay production.

For injection molding specifically, unsuccessful design for manufacturability can result in parts shrinking, warping, cracking or even breaking – setbacks you can't afford to take the time to remedy. To avoid these and execute a successful design transfer, there are certain considerations to be aware of. For instance, if you're working with a contract manufacturer, engage them early on in development. They'll help guide you and ensure your part will be produced as intended. Keep highly detailed diagrams and notes about your part and share them regularly. Opening lines of communication upfront will help save time down the road.

Beyond this, it comes down to properly designing the part. To get you started, the following are the top design considerations for injection molding with Stratasys Direct Manufacturing.

Wall section **considerations**

Wall thickness

On average, the wall thickness of an injection molded part ranges from 2 mm to 4 mm (0.080 inch to 0.160 inch), but can be as thin as 0.05 mm (0.020 inch) when produced with thin wall injection molding.

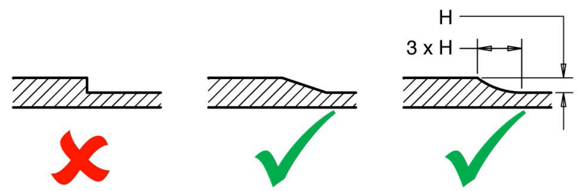
To achieve higher cost savings, consider a thinner wall thickness, as long as that thickness is consistent with the part's function and meets all mold filling considerations. As expected, parts cool faster with thin wall thicknesses, making for shorter cycle times and more parts per hour. Thinner parts also weigh less, requiring less plastic per part.

Uniform walls

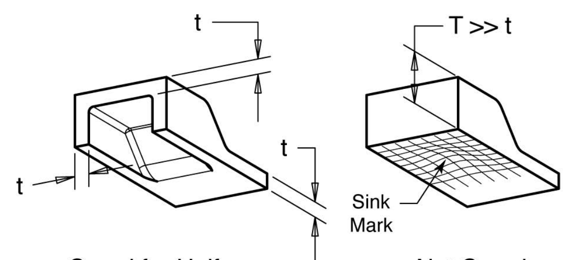
Uniform wall thickness makes the injection molding process more efficient because the molten plastic doesn't have to be forced through varying restrictions and can also help prevent issues later on as the part cools. When walls aren't uniform, the thinner sections will cool first followed by the thicker sections. As the thicker sections cool, they'll shrink and start to build stress near the boundary area between the two thicknesses. Because the thin sections harden first, they won't eject from the mold, but the thick section will, leading to warping, or twisting, of the part. If wall thickness variability is severe, the part can crack during removal.

When your design doesn't allow for uniform walls

If your design doesn't allow for uniform walls, make the change in thickness as gradual as possible. Coring and adding gussets are two options.



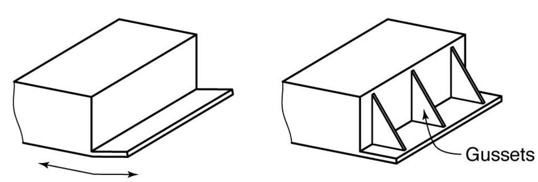
Coring: A method in which plastic is removed from the thick area to help keep wall sections uniform. It essentially eliminates the issue altogether.



Cored for Uniform Wall Thickess

Not Cored

Add Gussets: Gussets are support structures that're designed into the part to reinforce areas, such as walls or bosses, to the floor. Keeping them no more than 60 percent of the nominal wall's thickness helps reduce the possibility of warping.

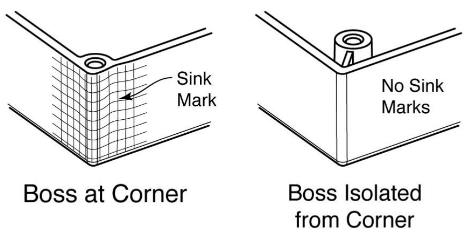


Warping

Gussets Minimize Warping

Voids and shrinkage

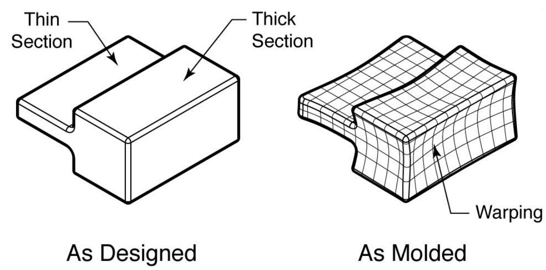
Wall thickness comes into play with voids and part shrinkage as well. The intersection between different thicknesses can cause troublesome shrinkage problems, particularly where ribs, bosses and other projections are attached. If the wall that a projection is attached to is thicker than other walls – taking a longer time to solidify – the nominal wall will shrink as the projection shrinks, resulting in a sunken area.



Minimize shrinkage by maintaining rib thickness between 50 and 60 percent of the attached wall's thickness. When it comes to bosses, isolate them in a corner, rather than incorporating them directly into the corner, to avoid producing thick walls that may cause sink.

Warpage

The dynamic between thin and thick wall sections, and their respective cooling times, can contribute to warping, as well. As the thicker sections cool, they shrink, taking some of the material from the unsolidified parts with them.



Other factors that impact the risk of warping include:

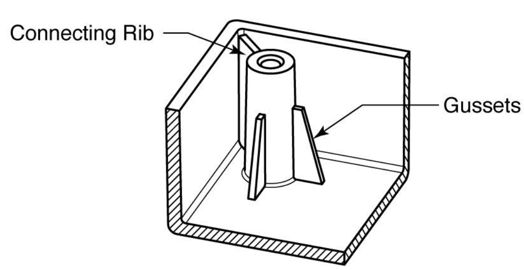
- Molding process conditions
- Injection Pressures
- Cooling rates
- Packing problems
- Mold temperatures

Follow resin manufacturers' process guidelines for the best results.

Bosses

To minimize sinking, boss wall thickness should be less than 60 percent of the nominal wall. If the boss isn't in a visible area, however, the wall thickness can be increased to allow for additional stresses imposed by self-tapping screws.

The base radius should be a minimum of 0.25 X thickness . Strengthen bosses by incorporating gussets at the base or by using connecting ribs attached to nearby walls.



Ribs

Ribs increase a part's bending stiffness, by increasing the moment of inertia, without adding thickness, as demonstrated in the below equation.

Bending stiffness = E (young's Modulus) x I (Moment of Inertia)

Take the following into consideration when working with ribs:

Rib thickness

Rib thickness should be less than wall thickness to minimize sinking. The recommended thickness shouldn't exceed 60 percent of the nominal thickness. Attach ribs to the corner radii as much as possible.

Rib intersections

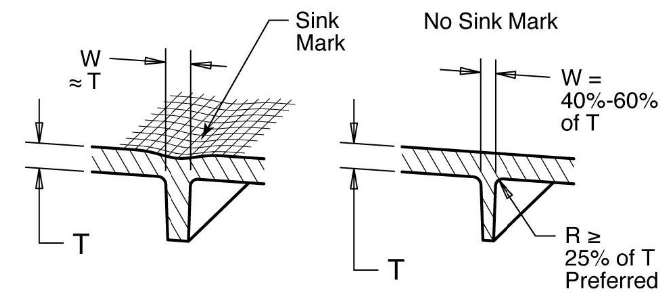
Material thickness will be greater at rib intersections. To avoid excessive sinking from occurring on the opposite side, employ coring, or another means of material removal.

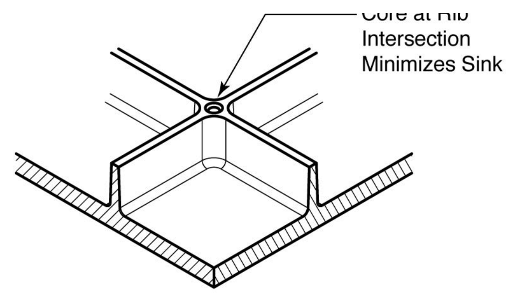
Rib guidelines

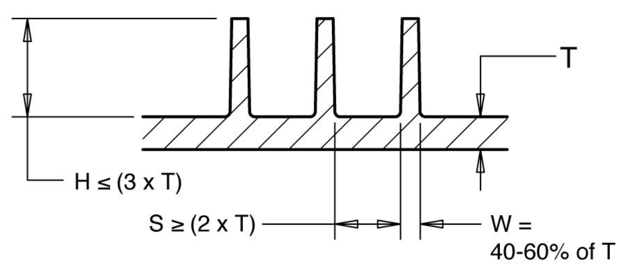
A rib's height should be limited to less than three times its thickness. Use multiple ribs to increase bending stiffness rather than one very tall one.

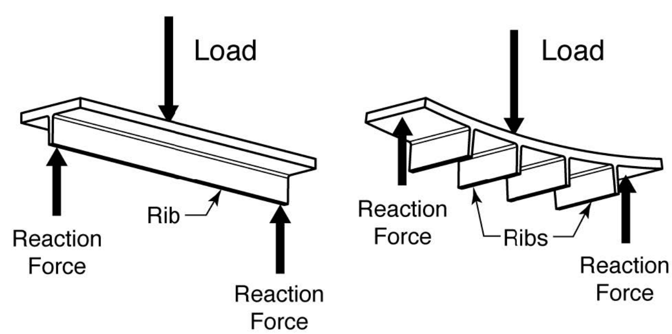
Rib/load effect on stiffness

Orientate ribs to provide maximum bending stiffness to a part. Pay attention to a rib's orientation to the bending load to increase stiffness.









Draft and texture

Applying draft, or a taper, to a part greatly improves its ability to withstand molding stresses during the cooling process and to successfully eject from the mold once solidified. Designing in as much draft as possible will help ease part ejection.

The ideal draft angle depends on the part's depth and required end-use function. Typically, one or two degree depth of draft with an additional 1.5 degrees per 0.25 mm depth of texture is enough to do the trick. If draft isn't acceptable due to design considerations, a side action mold may be required.

Textures, lettering and draft

Textures and lettering can be included on mold surfaces for end user or factory purposes. Texture or letter depth is somewhat limited, and extra draft needs to be added to the design to allow for part removal and to minimize the risk of dragging or marring the part.

Draft for texturing is somewhat dependent on the part design and specific desired texture. However, in general, 1.5 degrees minimum per 0.025 mm (0.001 inch) depth of texture, in addition to the normal draft, is sufficient. For office equipment, such as laptops, we recommend a texture depth of 0.025 mm (0.001 inch) and minimum draft of 1.5 degrees. More may be needed for heavier textured surfaces, such as leather, in which case a depth of 0.125 mm/0.005 inch and a minimum draft of 7.5 degrees is required.

Sharp corners

Sharp corners greatly increase stress concentration, which, when high enough, can lead to part failure. They often appear in non-obvious places, such as a boss attached to a surface or a strengthening rib.

Watch the radius of sharp corners closely because the stress concentration factor varies with radius for every given thickness. Stress concentration is high for R/T values less than 0.5. For R/T values over 0.5, the concentration decreases. The inside radius should be a minimum of one times the thickness. At corners, however, the suggested inside radius is 0.5 times the material thickness and the outside radius is 1.5 times the material thickness. If your part design allows, use a larger radius.

In addition to reducing stresses, the fillet radius provides a streamlined flow path for the molten plastic, resulting in an easier to fill mold.

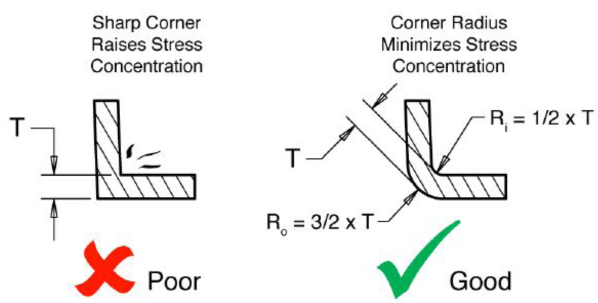


Figure 14: Radius Recommendation

Inserts

Inserts for machine screws or other fasteners, are often made of brass and enable many assembly and disassembly cycles. Inserts in injection molded parts are installed using one of the following methods:

Ultrasonic insertion

Ultrasonic insertion is when an insert is vibrated into place by using an ultrasonic transducer, called the horn. Ultrasonic energy is converted into thermal energy through the vibration, melting the plastic for the insert. It's done rapidly with short cycle times and low residual stresses.

For optimum performance, design a custom horn for each application and ensure good melt flow characteristics for a successful insertion process.

Thermal insertion

Using a heated tool, like a soldering iron, the insert is heated until it melts the plastic and then is pressed into place. As the plastic cools, it shrinks around the insert, securing it in place.

The advantage to this method is the special tooling that is inexpensive and simple to use. However, it's possible to overheat the insert or the plastic, resulting in a non-secure fit and degradation of the plastic.

Molded-in

Core pins can be used to hold inserts in place while the injected plastic completely encases the insert during the mold cycle. Molding in inserts may slow the cycle because inserts have to be hand loaded, but the process provides excellent retention and eliminates secondary operations, such as ultrasonic and thermal insertion methods. For high volume production runs, an automatic tool can load the inserts, but keep in mind it could increase the complexity and cost of the mold.

Living hinges

Living hinges are thin sections of plastic that connect two segments of a part together and allow it to hinge open and closed. Materials used in molding living hinges must be flexible, such as polypropylene or polyethylene.

Typically, these hinges are incorporated into containers used in high volume applications, such as toolboxes and CD cases. Well-design hinges should be able to flex more than a million cycles without failure.

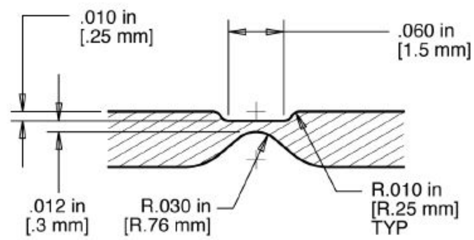


Figure 17:
Living Hinge Design for Polypropylene and Polyethylene

Over-molding

The overmolding process is when a flexible material, called the overmold, is molded into a more rigid material called the substrate. If properly selected, the overmold will form a strong bond with the substrate.

Bonding agents are no longer required to achieve optimum bond between substrate and overmold.

Insert molding

For insert molding, a pre-molded substrate is placed into a mold and the flexible material is shot directly over it. The advantage of this is conventional, single-shot injection molding machines can be used. It's also the most widely used overmolding process.

Final words

Once you've taken these guidelines into account and have a design ready for production, we will work with you to make any additional tweaks to further ensure it's ready to be produced en masse. While only one step in the product development lifecycle, these design considerations can help save you time and money to get your product to market faster. Contact Stratasys Direct Manufacturing to work with experts on ensuring your design is ready for manufacturing.