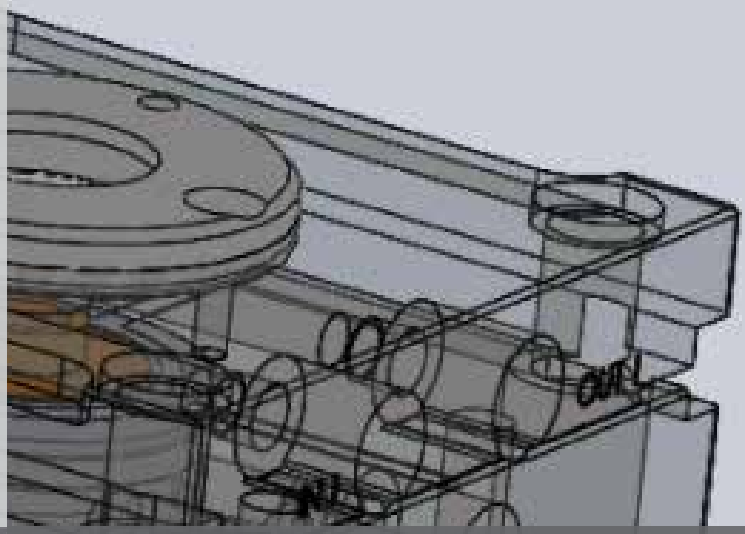
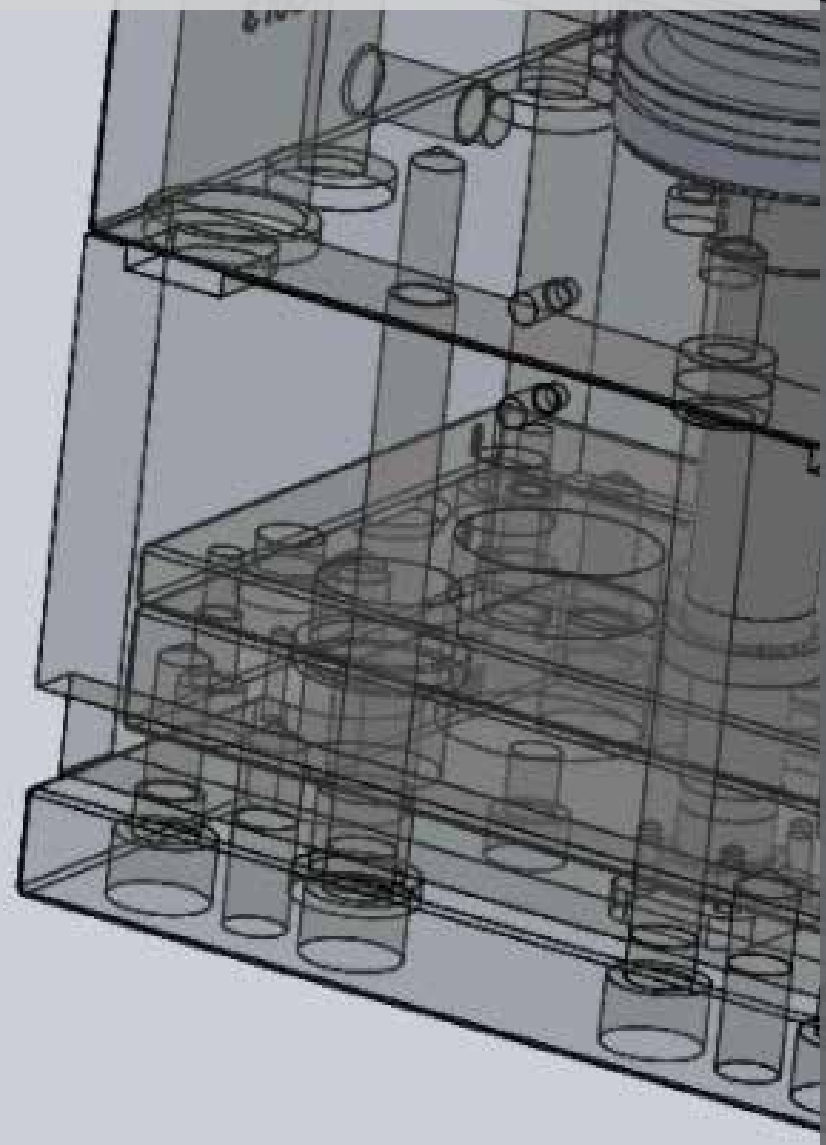
HI-TECH MOLD & TOOL, INC.

Five Design Principles for Thermoplastic Components

Success. Efficiency. Performance.

The success, efficiency and performance of your thermoplastic components begins long before you crank up production.





The decisions you make in the design phase can optimize your results, improve your products and prevent problems before they happen.

We've identified five overarching principles to consider as you design your components and plan production.

Consider these important issues and you will be on the road to more efficient, higher quality and more cost-effective production:

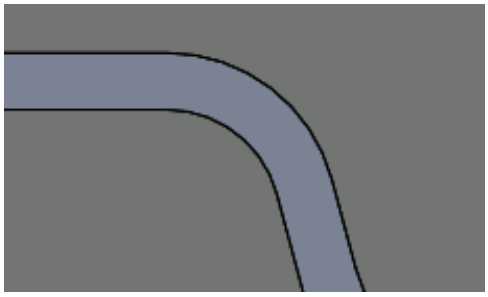
1. **Consistent or normal wall thickness: Optimal wall thickness depends on material and requirements**
2. **Injection Mold Tool Cooling: Cooling can have a significant impact on molded-in stress or reduced mechanical strength.**
3. **Geometry considerations: Ribs and rib thickness, bosses, holes, knit lines, flow lines**
4. **Material considerations: Unfilled, amorphous, semi-crystalline, filled, long fiber**
5. **Mold Considerations: Gating, ejection, mech action**

1 - Wall Thickness

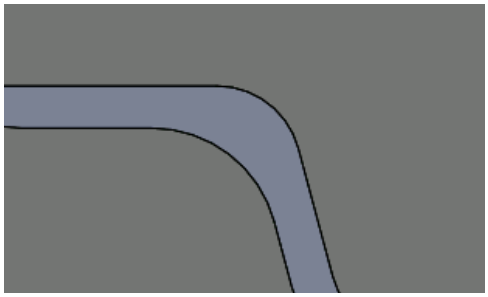
Consistent wall thickness throughout a part helps reduce molded-in stress. Thermoplastics shrink when they cool, so it's important to have various sections of the component cool down at the same rate. If the wall thickness is the same and the cooling rate is the same, the material will shrink similarly and solidify at the same time throughout.

If the component has significant wall thickness changes, the thicker cross section will still be molten and shrinking after the thinner section becomes solid. The material in the thicker wall shrinking while cooling will cause molded-in stress. If severe, molded-in stress can cause warpage or a reduction in mechanical properties.

Variations in wall thickness can also cause molding defects, such as sink marks and voids. Sink marks are often caused when a thicker adjoining wall can be seen through a wall that is similar or less in thickness. Voids can occur in thicker wall sections where material is no longer being delivered. Lastly, variations in wall thickness will also impact how the part will fill when thermoplastic is injected. Thermoplastics will always fill along the path of least resistance, so material will fill a thicker wall first.



Consistent wall thickness thru a radius.



Inconsistent wall resulting in inconsistent cooling of material.

2 - Injection Mold Tool Cooling

Cooling can have a significant impact on molded-in stress or reduced mechanical strength. If some regions of a component cool before others, the hotter regions will still be shrinking and pull on the cooled, solidified sections. This differential cooling causes molded-in stress that often results in warpage or dimensional issues.

It's important to understand and plan for mold cooling when designing a thermoplastic component. Keep in mind that mold cooling pathways are often drilled channels approximately .438 in (7/16) diameter. Other options such as bubblers, baffles, thermal pins and materials with enhanced thermal conductivity can also help with uniform cooling.

Bubblers, Baffles and thermal pins go into a sections of the tool that may need localized cooling. Using alloys with enhanced thermal conductivity for larger portions of a tool can help take more heat out of a larger area faster.

3 - Geometry Considerations

Geometry considerations are important to keep in mind when designing a thermoplastic component. The more stable a geometry is, the less likely it is that geometry will warp while cooling. For example, compare a five-sided box to an unsupported L bracket. The various sides of the box support each other, keeping them in place and preventing warping. An unsupported L bracket does not have support and will likely warp to less than 90 degrees. It's important to plan for ways to make changes if necessary. When laying out mold-cooling channels, make sure that if changes need to be made to the component geometry, water lines don't prevent these changes from being made.

Ribs and bosses need to be 70% of adjoining wall thickness to prevent sink marks. Any hole in a thermoplastic component will result in a knit line on the side of the hole opposite the gate.

Lastly, the cost of an injection molding tool will be impacted by the part geometry. Molds that can produce all geometry simply by opening and closing are going to be less expensive than those that require mechanical action to generate part geometry or are required for ejection. These mold additions for complexity will also have an impact on mold maintenance and mold longevity.

4 - Material Considerations

Choosing the right material to mold is a very important aspect of component design. There are dozens of different resins and filler systems. You should feel comfortable asking for support from component manufacturers and material suppliers – they have the expertise and experience to help you choose the best compound to produce the best component.

The design process often starts with a physical shape and then moves to material selection. While application use or function drives most of the material selection variables, shape or geometry should also play a role. Thinner walls are going to be very challenging for filled materials. Glass Fiber, Mineral and Carbon Fiber are all common fillers that reduce the potential flow length of a resin. If you're using filled materials, you may need to increase wall thickness to improve flow. There are very basic Part Advisor mold filling applications that can answer flow questions. If your material is in the database, you can get good advice from these programs.

The required aesthetics of a part often play a large role in material selection. Most thermoplastics can be precolored to any shade in the rainbow. Fillers also change the surface appearance of thermoplastics, so if your part will be seen then you may be limited based on desired aesthetics.

5 - Mold Considerations

Mold tool considerations can effect your production and final part performance. To fill the part in production, a gate or entrance into the part is required. Where you design that gate to be is important, as the gate is the location of highest molding pressure in the part. It's better to put the gate in a non-aesthetic location and a lower stress location if possible. Filled materials require larger gates and Long Fiber materials require even larger gates. Often times the gate location is dictated by thickness. Gating into the thickest section of the part will help reduce voids and porosity.

In order to produce more than one part, you need a parting line to split the mold and allow for ejection of the part. This parting line will result in a small witness line, so make sure you account for this minor aesthetic requirement. The "B" side of the mold will require ejection pins to push the part off of the mold. A component will require more ejection force if it has more geometry and draw on the "B" side.

Some geometry will require mechanical action like slides and lifters. This action enables the mold to remove steel, allowing the part to be ejected or allowing the mold to open.

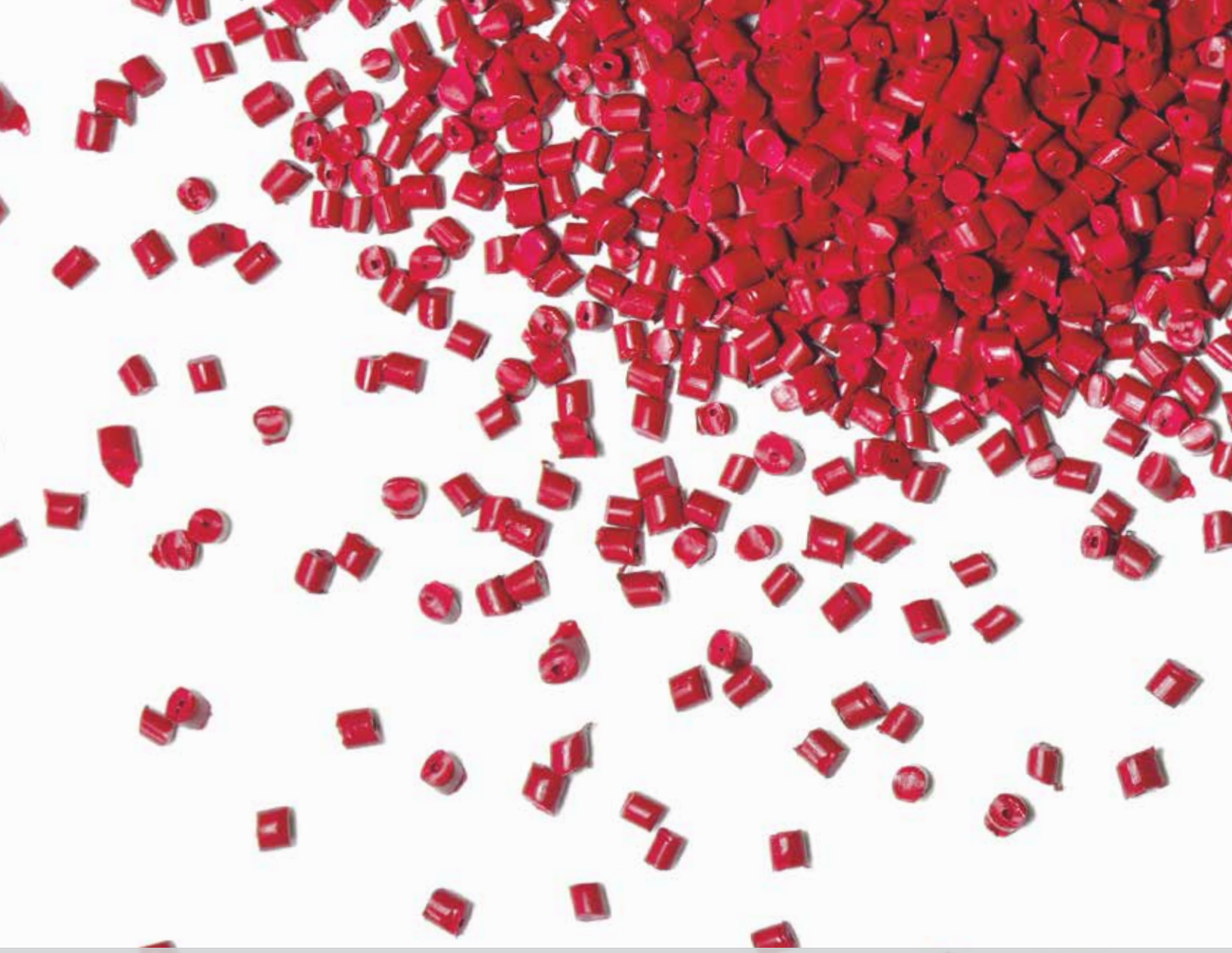
Design Efficiency, Performance and Cost-Effective Aesthetics into Your Components

There's a lot to keep in mind to successfully design thermoplastic components – it's not a one-dimensional exercise. This high-level overview is a good first step to understanding the issues and determining what you need to take into consideration from the very beginning of your design process. Hi-Tech engineers, mold makers and materials experts are available to share ideas. If you have any questions, please drop us a line at "Contact Us" and let us help. That's why we're here.

We're Here to Help

Hi-Tech is another selection resource you can tap into. Members of our team have dedicated their careers to materials research. We are happy to help you and find the right materials solution for every job. Please feel free to send us a note or question using the Contact Us form on www.hitechmoldtool.com.

We'll help you make the right selection.



~~HI-TECH MOLD & TOOL, INC.~~

One Technology Drive West | Pittsfield, MA 01201
United States