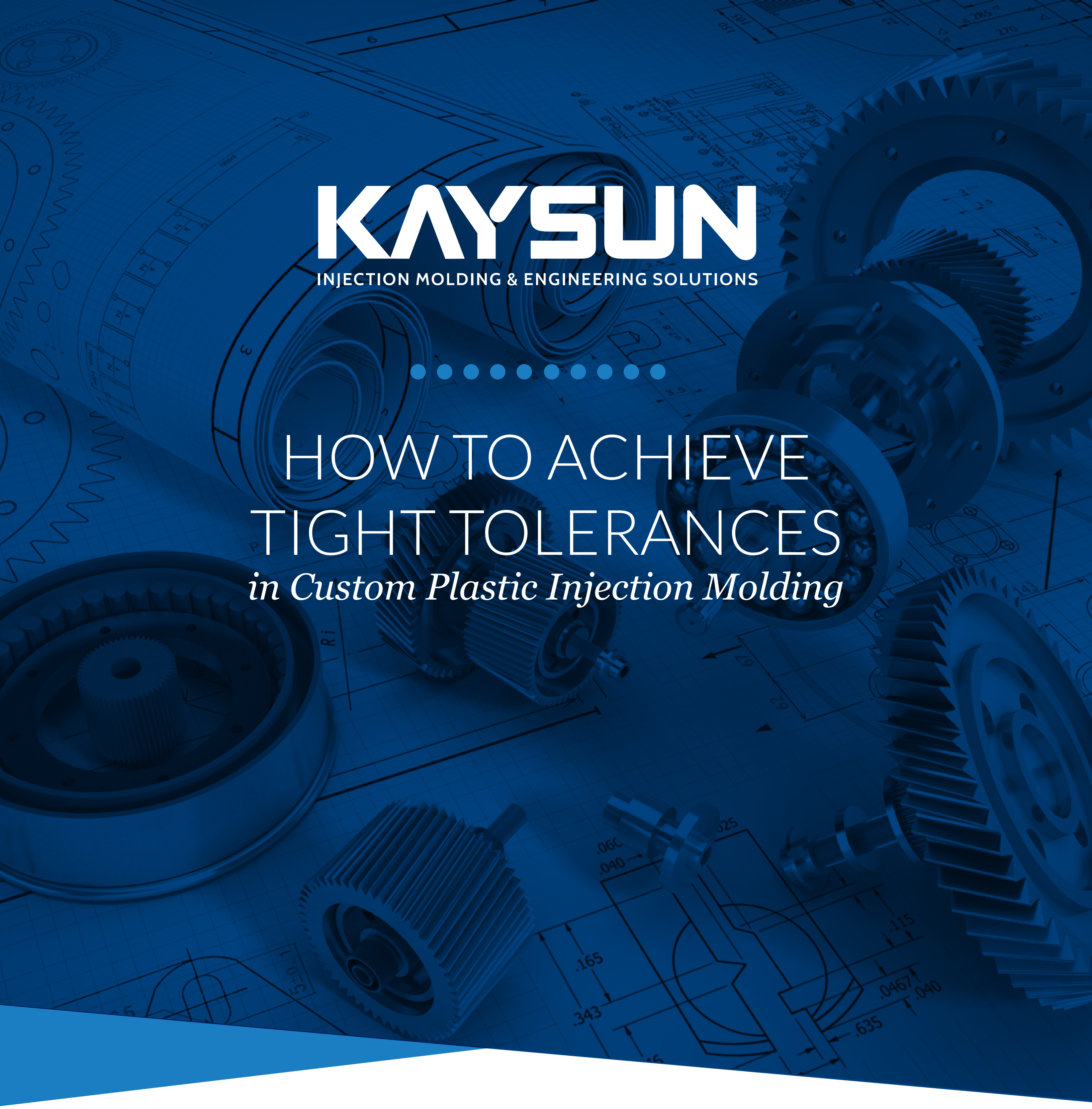


KAYSUN

INJECTION MOLDING & ENGINEERING SOLUTIONS

HOW TO ACHIEVE TIGHT TOLERANCES *in Custom Plastic Injection Molding*

Tight tolerance is a term often tossed around loosely by both injection molders and manufacturers. However, if tight tolerances are not adhered to correctly within a design, parts and components could underperform or possibly fail, resulting in a tooling and/or process overhaul. Tight tolerance is serious business, especially for plastic injection-molded complex parts.

In general, a typical tight tolerance for injection molding is +/-.002 inches (.050mm), while a very tight tolerance is +/-.001 inches (.025mm). There are many factors that impact injection molding tolerances — **part design and complexity; material selection; tooling; process design and control** — all of which require precision to meet exacting part specifications.

Part Design and Complexity

PART DESIGN IS THE SINGLE BIGGEST FACTOR IN CONTROLLING TIGHT TOLERANCES.

Making improvements during the design phase of a project not only helps ensure that repeatable tight tolerances can be achieved during production, but also helps improve manufacturability, quality, and end-user satisfaction, all while reducing costs.

Part geometry, overall size, and wall thickness requirements can all have an influence on tolerance control. Thick walls may have differential shrink rates within the thick sections, making it difficult to hold tight tolerances since the variable shrink can "move" within the section.

Part size has an impact if the dimension with the tight tolerance is large (it's easier to hold tight tolerances in smaller areas). The larger the dimensions, the larger the overall shrinkage, which makes it more challenging to control and maintain tight tolerance.

PART COMPLEXITY CAN IMPACT TOLERANCE IF SHRINK AND WARP ARE NOT REPEATABLE.

That's why upfront discussions between the manufacturing and design teams are absolutely essential for working out the best possible part design to minimize shrink and warp. It's important to note that:

- **Part complexity also impacts tooling design and material flow**, because filling the parts quickly, maintaining proper tooling temperature, and managing the cooling process are important for tolerance control.
- **Moldfill analysis** is needed for accurate predictions regarding mold heating and cooling, shrinkage, and warpage — all of which affect tolerances.
- **Temperature** is another key consideration during design. In fact, if the tight-tolerance part will be exposed to high/low temperature extremes in normal operation, where it will expand and contract, achieving tight tolerance may not even possible.

Design for Manufacturability (DfM) provides optimal control

DfM is the process of proactively designing products to optimize all facets of manufacturing. Tolerances, rules, and best practices are defined by guidelines. This methodology aligns engineering and production in the initial design phase so any problems can be addressed early in the product development process. Potentially costly issues that could impact manufacturability — tooling changes, resin selection, tolerances, secondary processing — are identified and addressed.

Successfully executing DfM for complex injection-molded plastic applications requires considerable technical expertise, which underscores the importance of partnering with an experienced molder.

THE VALUE OF CONDUCTING MOLDFILL ANALYSIS

Done during the design phase, a moldfill simulation helps ensure high-quality parts and optimizes cycle times and tooling trials. A computer program predicts the flow of plastic during all phases of the injection molding process, including flow, pack, and cooling. Other parameters that can be evaluated with moldfill analysis include:

- Type of gates
- Gate locations
- Vent locations
- Runner systems
- Tool temperatures
- Processing conditions
- Flow-induced shear stress
- Weld line location
- Air trap prediction
- Sequential valve gating
- Cavity/part temperature differential
- Coolant temperature
- Coolant flow rate
- Shrinkage
- Warpage

Material Selection

Material selection decisions must be made early in the design process. Different resins can affect the production of different tolerance sizes for the same part, so a tradeoff must be made between tolerance expectations and the physical properties of the resin.

SHRINKAGE HAS A BIG IMPACT ON TIGHT TOLERANCE.

Different materials have different shrink rates — the higher the shrink rate, the less repeatable a tolerance becomes. Amorphous materials generally hold tighter tolerances than crystalline materials. Why? Because crystalline materials go through a phase change from a crystalline solid (densely packed structure) to an amorphous melted fluid (less dense structure), resulting in a volume change. Amorphous materials, on the other hand, stay amorphous when melted and don't experience a drastic volume change, so they shrink less.

The challenge of crystalline materials
The degree of crystallinity can be made to vary within crystalline materials, so holding size is difficult. Parts molded in "cold" tools don't have the environment to develop their full crystalline potential. Parts with thick walls provide a well-insulated center area for crystal structure to develop, whereas crystal growth is quenched in the "skin" of the part in contact with the tool wall. Materials such as acetal and polyethylene can even continue to grow crystals below room temperatures.

So, to create crystalline parts with tight-tolerance dimensions, the injection molding environment must be designed to ensure that the highest state of crystallinity is achieved quickly. In general, high tool temperatures provide the best environment for crystal growth. Crystal growth can also be maximized by adding a nucleating agent to the melt, which provides sites throughout the melt that stimulate crystal growth.

THERMAL EXPANSION'S EFFECT

Holding tight tolerances can be a challenge with most resins because of their high thermal expansion rates (fillers help reduce this). Even though plastic parts can be held to tight tolerances in a climate-controlled environment, this doesn't mean they'll maintain these dimensions as the temperature changes. This must be considered when plastic parts are combined with other material types such as metals, or when the end use occurs in an environment with extreme temperature swings.

Tooling

Tool design, material, and cavitation all impact the ability to achieve a required tolerance. The need to heat and cool tools, and the number of cavities in the tool, can make holding tight tolerances more challenging. If tooling is not designed to provide consistent, repeatable cooling, shrink rates increase and tight tolerances are harder to achieve.

Custom injection-molded parts often require complex tools. These handle features like undercuts or threads, which typically require more tool components. There are other components that can be added to a tool to form complex geometry: rotating devices (using mechanical racks and gears), rotational hydraulic motors, hydraulic cylinders, floating plates, and multi-form slides. These components must quickly transfer heat away from the melted plastic. Special cooling line placements and component materials can do this.

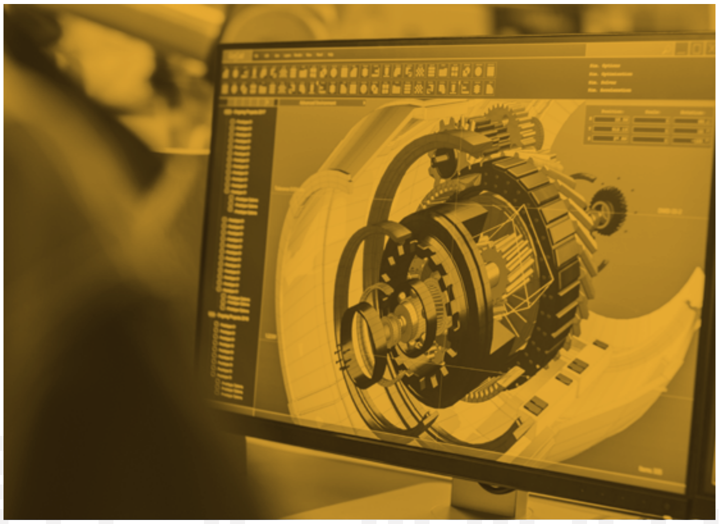
Also consider:

- **Grade of steel**
- **Location of waterlines** (to maximize cooling and minimize warping)
- **Gate types and locations** (to achieve optimum flow, fill pressure, cooling time; gate locations shouldn't impact part performance or appearance: flow marks, shrinkage, warping)
- **Draft, surface finish, and ejection design**, which can affect dimensional stability

Process Design and Control

Almost any reasonable design looks good on paper (or even as a prototype) — but that doesn't mean making it is a sure thing. Setting up the ideal process for the part, and being able to repeat it, is the key to molding tight tolerance parts.

Proper process control and process development ensure the part does not experience unnecessary pressure or stress during the molding process. Matching pressure curves helps eliminate the lot-to-lot variation that's common in the industry. Conducting injection molding operations in a climate-controlled facility also reduces process variation.



Controlling the process in real time
For complex plastic parts or components, temperature transducers can be strategically placed in tools (and hot manifold tools) to monitor and control the process in real time. Sensors can also be placed on the surface of the tool as a backup, in case the cooling lines or unit fail. Upper and lower limits are set on these sensors to monitor the cooling rate.

OTHER TOOLING DESIGN CONSIDERATIONS THAT IMPACT TOLERANCE

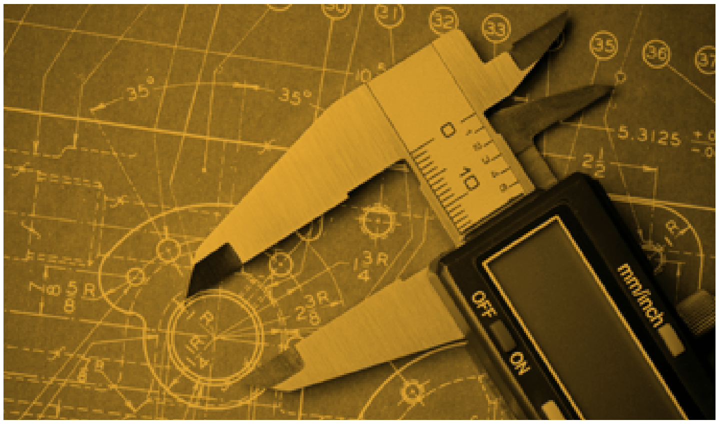
Establishing a production-capable process with in-tool pressure sensors, key components of scientific molding, is important for establishing a benchmark to refer back to when making changes (tooling material, process, or molding machine changes). This way, the process can be monitored and documented so that it can be set up and repeated accurately in the future.

Because no two identical machines ever run the same, creating a pressure curve is a must if a tool needs to be run in a different machine. Matching cavity pressure curves assures that the variation in processing conditions on the alternate machine still produces the same part.

"LOOSE" MAY WORK, TOO

Tight tolerances may not be required in your part design, depending on the intended end use. Many designers automatically set a tolerance in the CAD drafting software, and all dimensions are toleranced to that number. In reality, the part may not need tight tolerances. Designers need to understand that tighter tolerances may equate to increased production costs and development costs.

Tolerances should be as generous as possible in the design phase, unless they're required for proper fit or function of the part/assembly. Achieving tight tolerances may be a more expensive process, so if it's not required, loosen up the tolerance a bit to reduce production costs.



Contact Kaysun to learn how their experienced engineering and design teams can help you meet your tight tolerance goals and improve the profitability of your next injection molding project.

KAYSUN
INJECTION MOLDING & ENGINEERING SOLUTIONS
www.kaysun.com
920-686-5800

Follow us on: