KAYSUN

INJECTION MOLDING & ENGINEERING SOLUTIONS

TOOLING DESIGN
for Custom Injection Molding

Custom injection molding is a high-precision manufacturing process that injects molten plastic into carefully designed tooling, where the plastic cools and hardens into the specified part. The part is then ejected from the tool and sent on for secondary processes, if necessary.

The tool consists of two halves: the movable or B side, and the stationary or A side. The void created within a closed tool (part geometry or wall section) receives the molten plastic, and is referred to as the part cavity. Depending on production needs, "multi-cavity" tools can be designed to create multiple cavities, or more identical parts, during the same cycle.

Designing the tool and its various components is a highly technical, often complex process that requires precise scientific know-how to produce top-quality parts that maintain tight tolerances. Strict attention must be paid to:

- Selecting the proper grade of tool steels so components that come in contact with each other do not wear out prematurely
- Determining steel hardness to maintain the proper balance between wear and toughness
- Placing water lines to maximize cooling and minimize warping
- Calculating gate/runner sizing specifications for proper filling and minimal cycle times
- Identifying the best shut-off methods for tooling durability over the life of the program

During the injection molding process, molten plastic flows into the tool cavity through channels called runners. The direction of flow into the part cavity is controlled by gates at the end of each channel. The system of runners and gates must be precisely designed to ensure even distribution of molten plastic. Proper placement of cooling channels in tool walls for water or hot oil circulation is required for achieving final parts with homogeneous physical properties and repeatable dimensions.

In general, custom injection-molded parts include features like undercuts, threads, and tight tolerance dimensions. Sophisticated tooling components such as slides, lifters, rotating or unthreading cores (using racks and gears), rotational hydraulic motors, hydraulic cylinders, servo drivers, floating plates, and multi-form collapsible cores are required.

MAIN STAGES OF TOOLING

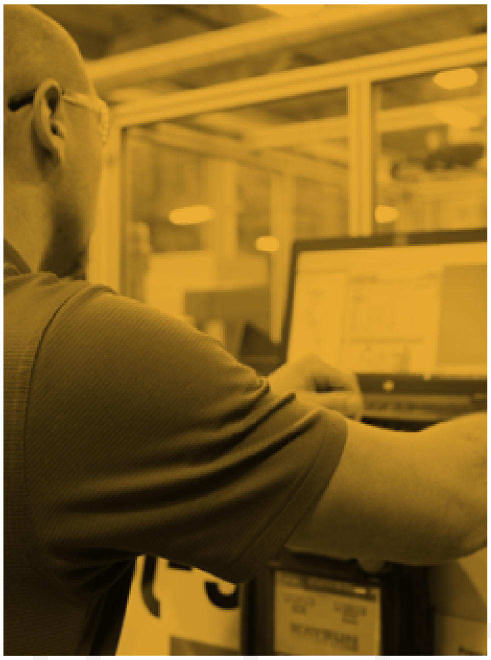
STAGE 1: Manufacturability and Feasibility

In this initial stage, engineers encompassing the disciplines of design, tooling, materials, manufacturing, and quality work together to determine product specifications, tooling component functionality and materials, operational constraints, and any needed product enhancements and improvements. The team especially focuses on:

- The physical and chemical properties of the selected plastic in order to choose the proper tool steel and gate type for the application
- Mold fill analysis with various iterations for gate locations to optimize fill and process parameters, determine the best location for RJG pressure transducers, and identify proper vent locations
- 3D tool design concepts are generated and reviewed internally, then presented to the customer for final approval related to gate marks, ejector pin marks, and split lines that will appear on the plastic part
- Manufacturability review confirming standard plastic injection design practices and tooling details to achieve the most robust design possible
- Tooling specifications and tooling sources finalized
- Completing a comprehensive process failure mode effects analysis (PFMEA)

STAGE 2: Design

- The tool builder is given the design specifications and concepts needed to generate preliminary 2D and 3D designs
- The preliminary design models are used to determine adequate cooling, sufficient ejection, refine the mechanical details for functionality, and many other details defined in our rigorous design review process
- Upon review and approval of the models, the detailed design is finalized



STAGE 3: Construction

- Materials and components are ordered
- The tool builder’s progress is closely monitored and on-site meetings are held
- The completed tooling is inspected against a comprehensive checklist

STAGE 4: Bring the Tool In-House for the Initial Sample

- Initial sampling using scientific molding practices is carried out; RJG cavity pressure transducers in the tool accurately document the filling profile over time
- Processing parameters are recorded
- Sample parts are measured, tested, and evaluated against specifications

STAGE 5: Qualify Tool for Mass Production

- Any needed dimensional adjustments are made
- Tool construction is verified and the process is detailed and documented for maximum repeatability with minimal setup time
- Optimized parts are resampled and submitted to the customer for final approval
- Upon final approval from the customer, the production process is launched

STEEL VS. ALUMINUM

Most tools are made from hardened or pre-hardened steel. Hardened steel, which is heat treated after machining, has superior wear resistance compared to pre-hardened steel and lasts longer. Steel tools are more expensive than tooling made from other materials such as aluminum. However, they are more durable and support higher production levels before needing to be replaced.

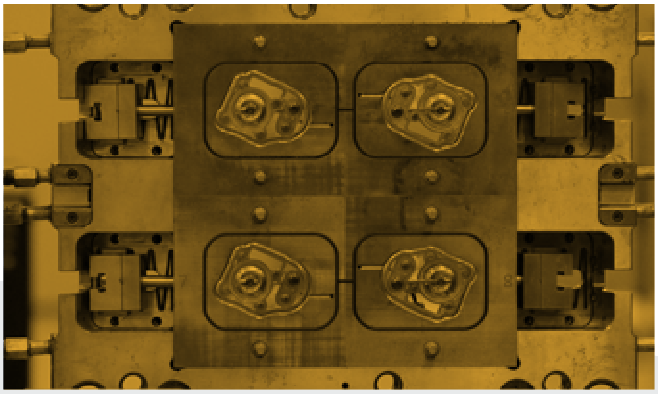
Tooling engineers must take steel hardness into account. Harder steel is more brittle and will crack upon flexing. Therefore, it is not a good choice for tool components that are subjected to side loading or impact.

Because of its rapid cooling characteristics, aluminum is sometimes used for tooling. Since it is easier to machine than steel, aluminum can also reduce the time required for building the tool, which translates to faster turnaround and production cycles.

However, even hardened aluminum is softer than steel, making it difficult to weld, maintain, and keep from wearing out rapidly. Aluminum is best suited for prototypes and short runs.

Depending on the product and tooling design, hybrid tools can sometimes be built that are mostly steel but use aluminum in low-wear areas to transfer heat. However, aluminum is not a good choice for complex parts or harder, glass-filled plastics because of premature wear on tooling, runner systems, and gates. Copper alloys are sometimes used as an aluminum replacement when rapid heat dissipation is required.

Both steel and aluminum tools can be coated with special materials such as nickel-boron and nickel-teflon (0.0002 to 0.0004 inches in thickness). Coatings improve wear resistance, reduce friction, and extend tooling life, especially when molding glass fiber reinforced plastics.



PROTOTYPING FOR INJECTION MOLDING PRODUCTION

Prototyping for injection molding production hinges on the development of a single-cavity pre-production tool. When tested, this low-production tool provides tangible insights.

Part design is either proven out or needed improvements revealed in areas that typically affect the final shape and function of a complex injection-molded part, including:

- Plastic materials testing and selection
- Part geometry feasibility and optimization
- Gate location
- Production readiness and timeline reduction

KEY COMPONENTS OF TOOLING DESIGN

Gates

Gates are openings at the end of runners that direct the flow of molten plastic into the tool cavity. Gates vary in size and shape depending on the part design and plastic type. Design engineers must consider a number of factors to determine gate types and locations to achieve optimum flow, fill pressure, cooling time, and dimensions/tolerance. It is imperative to locate gates where they won't impact part performance or appearance (vestige, flow marks, shrinkage, warping).

Draft

One essential aspect of tooling design is the ease with which final parts are removed, thereby preventing surface damage. This is accomplished by applying a draft angle, or taper, to the tool walls. The required degree of draft angle depends on several factors, including design of the part, material, tool cavity depth, surface finish, and texture. Typically when part sidewalls are angled only a few degrees, it creates enough space for easy part removal when the tool is opened. Generally, the deeper the cavity, the more required draft. Draft angles typically vary from about 1 to 5 degrees.

Finish and Texture

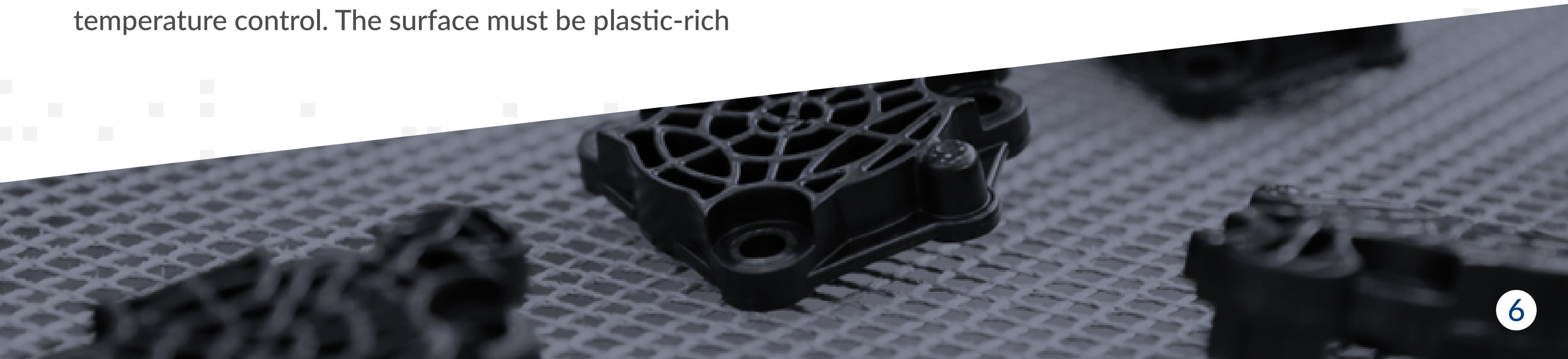
Molding surface cooling is critical for determining part surface finish. For example, a smooth surface finish on a 50-percent glass-filled plastic depends on proper temperature control. The surface must be plastic-rich with the glass fiber slightly deeper in the part, requiring a hotter tool and about a 10% longer cooling time.

Tooling can also be modified to apply a texture or pattern to the injection-molded part surface, which eliminates finishing steps. Texture can also provide better part function, such as enhanced grip or reduced wear from friction. Depending on the type, depth, and location of texture, draft may need to be adjusted to facilitate part ejection.

Manufacturability

The main goal of tooling design is to create complex injection-molded parts with high manufacturability. In other words, refining a high-quality process that is simple and efficient, long-lasting, easy to operate and maintain, and that meets all customer specifications at the lowest possible cost. Fulfilling these expectations depends on designing the best tooling option for each customer's needs.

To accomplish this, tooling decisions must be made in the earliest design stages. The toolmaker must be involved as early as possible to provide a realistic machining perspective on part design, tolerances, tooling design, selected materials, and associated costs. Taking this step up front is the best way to eliminate wasted effort and rework that adds significant cost to the tooling budget. Part design and tooling design are dependent on each other and should be done concurrently whenever possible.



Lifecycle Costs

Toolmaking is one of the greatest expenses in the production process and, understandably, one of the greatest customer concerns. Properly designing, building, and maintaining tooling for each part requires a highly skilled team of engineers and technicians using the latest in sophisticated design and manufacturing technologies. Labor cost can be optimized, however, by working closely with an experienced, efficient tooling team that makes informed decisions on material selection and design trade-offs early in the design process.

In an effort to save costs up front, some companies source tooling according to price, looking only for the lowest bid. Buyer beware. There is usually a reason behind lowball machining/tooling bids, including:

- Poor quality
- Poor repeatability
- Inferior tooling
- Improper materials
- Skilled labor gaps
- Waste/rework

4 WAYS AN EXPERIENCED CUSTOM INJECTION MOLDER ENHANCES TOOLING

When you commit to working alongside an experienced custom injection molder to source, design, and use tooling for your complex applications, you proactively protect your brand and your bottom line.

Kaysun has the deep engineering and tooling design experience necessary to navigate the many project variables that ensure the best quality, consistency, and price by:

1. **Offering design for manufacturability (DfM) studies, part design review, and suggested modifications** that eliminate or greatly reduce tooling challenges early in the design phase, thereby lowering costs while enhancing quality, reliability, and longevity
2. **Leveraging scientific molding expertise** to engineer a production process with top-quality functionality that aligns with the tooling design
3. **Synchronizing project timelines with your team's expectations** by ensuring the tool is operating at peak performance to meet market launch needs and maximize profitability
4. **Proactively maintaining the tool** so the original part design isn't compromised over the course of many production runs

Some companies trying to beat a deadline may select a tool vendor quickly, hoping “things turn out right.” However, a lack of due diligence can lead to oversights that take much longer to straighten out. **Although rushing tooling might get the first shots completed quickly, chances are final part production will fall behind schedule.**

The best way to get maximum value for your tooling budget is to consider lifecycle costs, not up-front costs. **Quality and repeatability is achieved by working with an experienced custom injection molder that, first, takes the time to completely understand your needs, goals, and production expectations — and then designs the best possible tooling to meet those needs.**

Up-front costs for quality tooling may be higher compared to cheaper vendors or offshore suppliers. However, the return on investment comes quickly in the form of higher part quality, fewer defects, greater throughput, longer-lasting equipment, and greater customer satisfaction and loyalty.

Contact Us today to request a quote for your next complex injection molding project.

KAYSUN
INJECTION MOLDING & ENGINEERING SOLUTIONS

www.kaysun.com
920-686-5800

Follow us on: