KAYSUN

INJECTION MOLDING & ENGINEERING SOLUTIONS

Is Your Injection Molder Delivering DEFECT-FREE PARTS?

As an OEM, you rely on your injection molder to produce flawless Plastic components for your complex applications, on time, every time.

However, not every injection molder has the expertise and quality control measures in place to consistently hit the mark — or to earn your trust.

When it comes to achieving defect-free parts, experience makes all the difference. This eBook walks you through common molding defects, providing insights and solutions you can only get from an expert custom injection molder.

Defects Defined

There are several common injection molding defects that impact aesthetics and performance. Plastics engineering expertise is required to address:

RESIDUAL STRESS occurs as a part is cooling in the tool. Typically, design flaws or inadequate processing causes internal pressure to accumulate when the plastic must conform to the tool instead of shrinking into an organic shape. Part dimensions and all other characteristics are unaffected by residual stress, making fluctuations visually imperceptible.

The difference shows up in the field, when parts impacted by residual stress perform poorly or fail.

WARPING is an unplanned and unwanted bend, curve, or geometry in a molded part. Warping can appear either during prototyping or full-scale production, introducing potentially significant increases in project time and cost.

SINK MARKS are visual, measureable marks — usually in the form of depressions or craters — that appear on a plastic part. Beyond unappealing aesthetics, sink marks can interfere with component functionality and assembly. For example, if a plastic defibrillator housing has sink marks, the interior electronic components may be damaged by water or other contaminants.

SHORT SHOTS are voids in a molded part where plastic is physically missing. As a result, injection molded parts can be misshapen, the wrong size, or may fail to hold correct dimensions — making them unusable.

FLASH is liquid resin that escapes from a tool parting line and solidifies on the outside of a part. Flash can be extremely sharp, making general handling dangerous. It can also easily damage packaging, or cut through O-rings and seals when the part is assembled into a larger system.

Knowing about prevalent injection-molded part defects is one thing. Addressing and fixing the root causes before they occur is another — and it can only be done by an experienced custom injection molder that focuses on quality outcomes.

Other Defects to Consider

In addition to the common injection molding defects profiled, OEMs need to be aware of these potential issues:

FLOW LINES: Discolored streaks, patterns, or lines that result from varying speeds or wall thicknesses a plastic encounters as it proceeds (flows) through a tool.

VACUUM VOIDS: Surface air pockets (bubbles) caused by inconsistent solidification times between the inner sections of the part and its surface.

SURFACE DELAMINATION: Thin surface layers that can be physically peeled away from the part due to contaminated material.

WEAK WELD LINES: A line (or plane) on a part indicating insufficient bonding or solidification of plastic flowing from two different parts of the tool.

BURN MARKS: Discolorations that reveal where the plastic surface degraded due to excessive heating, rapid injection speeds, or etching from overheated trapped air.

JETTING: Wavy lines or folds on a part surface caused by the plastic's failure to stick to the tool surface due to excessive injection speeds, or the melt temperature was too low thereby increasing flow resistance. Poor gate location often contributes to jetting.

Identifying and Resolving Defects

A design for manufacturability (DfM) approach to part design and engineering that incorporates a comprehensive moldfill analysis helps experienced molders identify any potential issues.

RESIDUAL STRESS

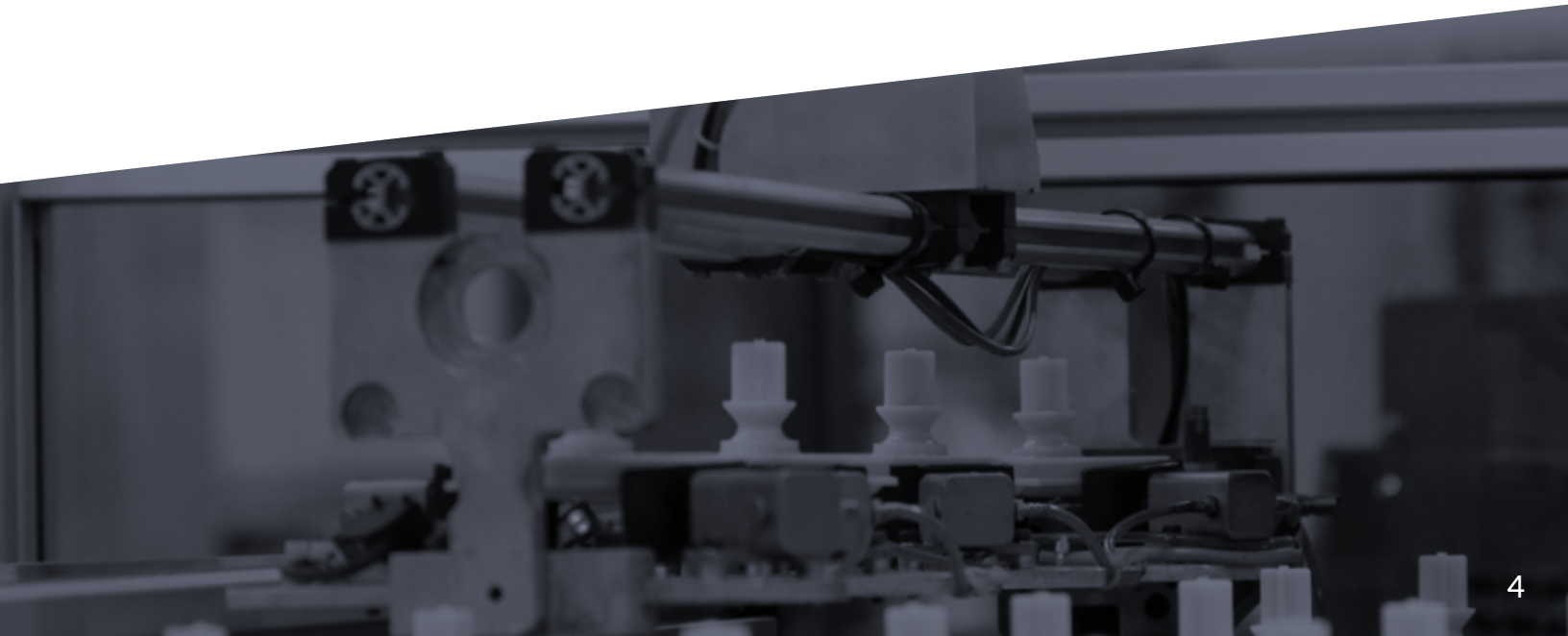
Cause: Residual stress occurs because during molding, the nature of an injected plastic and the function of the tool are at odds. The plastic wants to shrink organically and the tool restricts that shrinking.

Solution: A molder normally anticipates and compensates for residual stress by qualifying a part using the same production process and equipment time after time — which is a primary reason one injection molder cannot simply pick up where another left off in production. If a different injection molder is subsequently used, they are responsible for re-qualifying the part and ensuring consistency in their process and equipment to prevent residual stress-related defects.

WARPING

Cause: Warping is the direct result of residual stress acting on a plastic part that may have been ejected from a tool before it fully cooled and solidified. Flawed tools or plastics, production changes, or liquid plastic being too warm when injected are also potential causes. Freed from properly conforming to the tool, the plastic can act organically and warp within minutes or up to 24 hours of coming out of the tool, depending on the plastic used.

Solution: Temperature and timing are critical for prevention of warping. Careful monitoring of plastic temperature consistency prior to and during injection alleviates overheating, compromised solidification properties, and the risk of improper cooling. Likewise, the timing surrounding the release of a part from a tool must be precise. If ejected too soon, the part does not contain appropriate residual stress and is not properly solidified, so the plastic reverts to its natural properties which causes warpage.



SINK MARKS

Cause: Sink marks indicate a variety of problems, from a dirty tool to changes in plastics properties and/or equipment pressurization. They can also be caused when an adjustment is made to correct a separate molding process issue that interferes with residual stress.

Solution: Prevention of sink marks can range from minor fixes, like cleaning a dirty tool, to more complex solutions that start in the design phase — such as proper gate location, wall thickness, etc. Incorrect machine pressure could also be the culprit. Checking and calibrating machinery may resolve sink mark issues. However, calibration could also mean deviating from the original process deemed best through moldfill analysis and scientific molding measurements. Therefore, re-qualification of the part using the new process is necessary.

What is Scientific Molding?

Unlike standard molding procedures, scientific molding relies on the use of sensors and sophisticated software to monitor each phase of the injection molding process. This allows engineers real-time control and adjustment of critical variables like temperature, flow rate, fill rate, and cooling temperature to maintain precise, repeatable production of parts and products that will not fail in the field.

SHORT SHOTS

Cause: Short shots are caused by a tool cavity being improperly filled with plastic, either through not having enough material, faulty flow, or incorrect pressurization.

Solution: Short shots are prevented by maintaining the ideal pressure rate to fill out the part, as determined by applying scientific molding methodologies during the design phase. As in other aspects of production, a moldfill analysis provides feedback that engineers can use to adjust pressures and ultimately deliver fully filled, defect-free injection molded parts.

FLASH

Cause: Flash can result from general tool wear, or when an undersized press cannot produce the force required to push the two halves of the tool together during the high-pressure fill, and the tool separates at the parting line.

Solution: Tooling issues and flash go hand in hand. Therefore, expert tool design is fundamental, and largely dependent upon molder engineering experience. Tooling developed based on a solid design concept and detailed specs minimizes locking issues that generally lead to resin leaks and flash during production. In addition, it's crucial that your injection molder correctly calculates tonnage in order to use the right size injection molding machine to prevent flash.

What is Design for Manufacturability?

Design for manufacturability (DfM) aligns engineering and production during the initial design phase so any issues can be fixed early in product development. A DfM study can prevent a series of potentially costly manufacturing issues, ranging from materials compatibility to tolerances and secondary processing.

Quality-Driven Custom Injection Molding for Complex Applications

Defect-free parts are a goal, but they're never a given. The solution lies in the custom injection molding partner you choose.

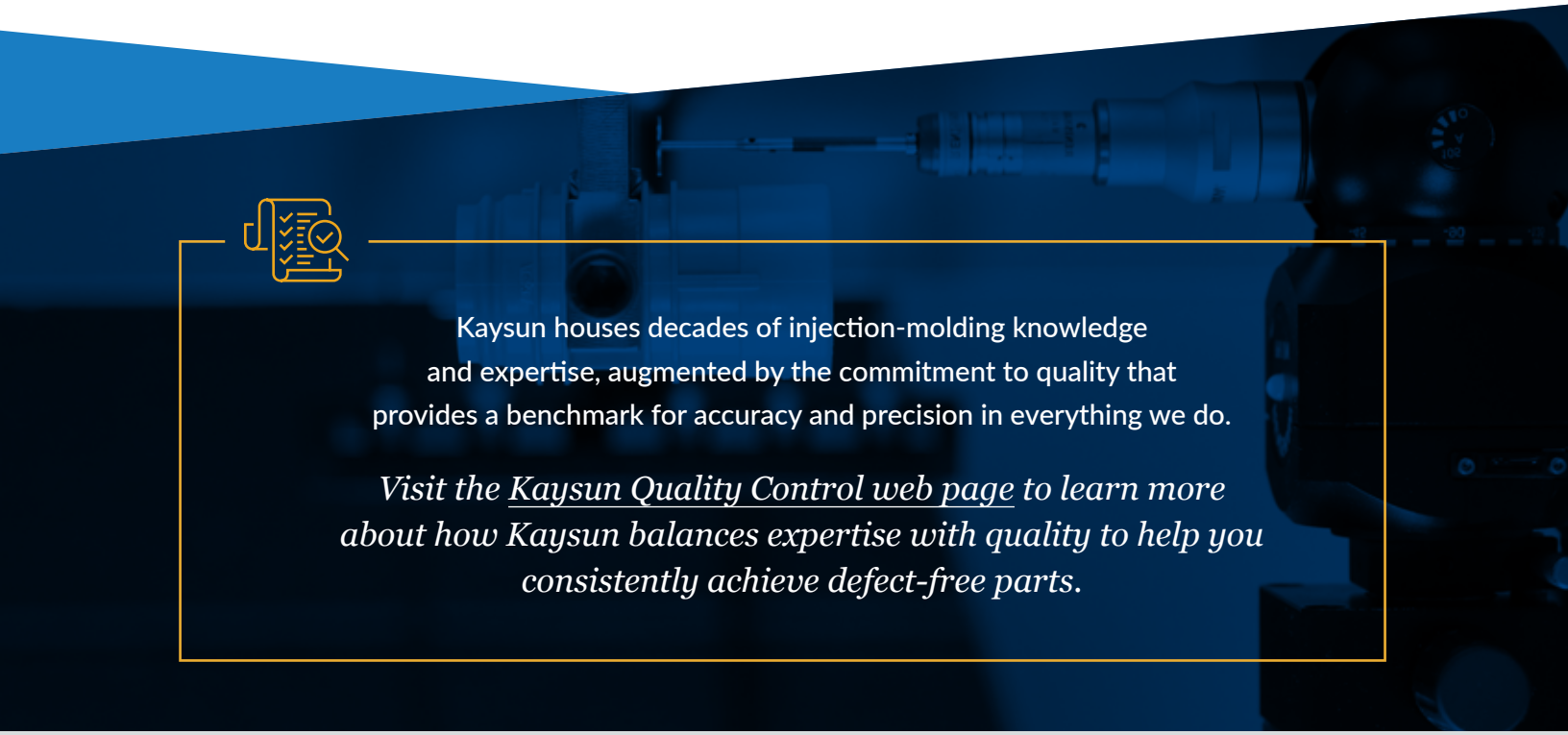
The ideal molder has the depth of experience to efficiently detect and correct potential issues, and a passion for quality that drives results.

KAYSUN IS THAT IDEAL MOLDER

The Kaysun Quality Lab brings cutting-edge technology, testing equipment, and molding experts together under one roof to facilitate quality control throughout all project phases. The in-house lab serves as a hub for:

- cross-functional team collaboration
- comprehensive testing during development and production
- advanced IQMS quality system-monitored parts and processes
- visual and structural quality control checks throughout and post-production
- real-time adjustments by specialized QA technicians to ensure all parts are in-spec and aligned with customer requirements

The Kaysun Quality Lab provides our team with capabilities that few molders can offer. It also demonstrates the uncompromising standards to which we hold ourselves and the work we do for our customers.



Kaysun houses decades of injection-molding knowledge and expertise, augmented by the commitment to quality that provides a benchmark for accuracy and precision in everything we do.

Visit the Kaysun Quality Control web page to learn more about how Kaysun balances expertise with quality to help you consistently achieve defect-free parts.

KAYSUN
INJECTION MOLDING & ENGINEERING SOLUTIONS

www.kaysun.com | 920-686-5800