KAYSUN
INJECTION MOLDING & ENGINEERING SOLUTIONS

INJECTION MOLDING GLOSSARY
for Product Development Engineers

Developing new products and the manufacturing systems behind them takes input and expertise of engineers from interconnected disciplines: design, packaging, quality and process — especially when tackling critical-use applications. To do so successfully takes a willingness to dig into all aspects of the project, including posing questions to suppliers about design, performance, materials and manufacturability.

Injection molders are often called on to produce plastic components and parts for critical-use applications for a number of industries, making them a valuable resource for better understanding how designs align with manufacturability. Involving your injection molding partner early in the design phase benefits you, them and the project by preventing the common mistakes that can lead to rework, decreased ROI, inconsistent product quality and unsuccessful product sell-in.

To make the most of your partnership, it's important that product development engineers and injection molders are speaking the same language. That's why we created this Injection Molding Glossary for Product Development Engineers. Organized into categories based on common injection molding topics, finding terms in this valuable reference tool is fast, easy and a surefire way to convey and understand project expectations, needs, and outcomes.

Analysis

COMPONENT FAILURE ANALYSIS: The process of collecting and analyzing data to determine the cause of a failure, with the goal of determining corrective actions.

DESIGN OF EXPERIMENTS (DOE): A facet of Scientific Molding, DOE is a highly precise injection molding practice that improves end results and ROI for complex, critical-use plastic parts and products. Specifically, DOE is a branch of applied statistics that deals with variables used in controlled tests as input and output to determine certain values, like failure probability in injection-molded components.

DESIGN FOR MANUFACTURABILITY (DFM): In broadest terms, Design for Manufacturability (DfM) — also known as Design for Manufacturing — is the process of consciously and proactively designing products to optimize all facets of manufacturing, including injection molding. DfM simultaneously helps ensure cost and time efficiencies, superior quality, regulatory compliance and end user satisfaction. Since manufacturing processes vary, there are set guidelines for DfM practices that define tolerances, rules and best practices.

DIFFERENTIAL SCANNING CALORIMETRY (DSC): A thermal analysis technique that looks at how a material's heat capacity (Cp) is changed by temperature. A sample of known mass is heated or cooled and the changes in its heat capacity are tracked as changes in the heat flow.

FINITE ELEMENT ANALYSIS (FEA): A computerized method for predicting how a product reacts to real-world forces, vibration, heat, fluid flow and other physical effects. FEA indicates whether a product will break, wear out or work the way it was designed.

FOURIER-TRANSFORM INFRARED SPECTROSCOPY (FTIR) ANALYSIS: A technique used to obtain an infrared spectrum of absorption or emission of a solid, liquid or gas. An FTIR spectrometer simultaneously collects high-spectral-resolution data over a wide spectral range.

GEOMETRIC DIMENSIONING AND TOLERANCING (GD&T): A symbolic language that's used on engineering drawings and computer-generated models to communicate geometric dimensions and allowable tolerances for various parts.

IMPACT STRENGTH ANALYSIS: Measurement of the capability of a plastic to withstand a suddenly applied load as expressed in terms of energy.

MELT FLOW ANALYSIS: Measurement of the extrusion rate of molten resin through a die of specified length and diameter.

MODULUS OF ELASTICITY: The ratio of stress to corresponding strain below the proportional limit of a plastic in tensile testing.

MOLDFLOW ANALYSIS (AKA MOLD-FILLING ANALYSIS): The process of using specialized software to simulate an injection molding cycle with a particular plastic and analyzing the results as they relate to the design of a particular injection molded component. Moldflow analysis should occur during the design phase, well before the injection molding process begins.

REVERSE ENGINEERING: The process by which an injection molded component is deconstructed to reveal its designs and/or architecture, or to extract knowledge from the object.

THERMOGRAVIMETRIC ANALYSIS (TGA): A method of thermal analysis in which the mass of a sample is measured over time as the temperature changes.

Certifications, Classifications and Regulatory Agencies

ANSI: American National Standards Institute, a private non-profit organization that oversees the development of voluntary consensus standards for products, services, processes, systems and personnel in the United States.

ASTM INTERNATIONAL (FKA AMERICAN SOCIETY FOR TESTING AND MATERIALS): An international standards organization that develops and publishes voluntary consensus technical standards for a wide range of materials, products, systems and services.

CLEANROOM: A controlled environment used in manufacturing or scientific research that has a low level of pollutants such as dust, airborne microbes, aerosol particles and chemical vapors. Contamination is strictly controlled and specified by the number and size of particles per cubic meter.

ISO: International Organization for Standardization, an international consortium of representatives from various national standards organizations that promotes worldwide proprietary, industrial and commercial standards.

ISO 9000: The ISO 9000 family addresses various aspects of quality management that provide guidance and tools for companies and organizations that want to ensure their products and services consistently meet customers' requirements, and that quality is consistently improved.

ITAR: International Traffic in Arms Regulations, export controls regulated by the federal government that restrict and control the export of defense and military-related technologies to safeguard U.S. national security and further U.S. foreign policy objectives.

SPI MOLD CLASSIFICATIONS (CLASS 101-105): Standard tools are defined by the Society of Plastics Industry (SPI) and categorized into five classifications to guide quotes and orders into uniform tool types.

Design Considerations and Molding Challenges

ASPECT RATIO: Ratio of total flow length to average wall thickness.

BACKFLOW: Molten resin that flows back out of the tool, returning to the runners.

BLOOM (AKA MIGRATION): An undesirable cloudy effect or whitish powdery deposit on the surface of an injection molded component caused by the migration of a lubricant, stabilizer pigment, plasticizer or other non-bonded component.

COLD FLOW LINES: Imperfections within the part wall caused by thickening or solidification of resin prior to full cavity fill.

CRAZING: Distinct surface cracks or minute frost-like internal cracks in plastic resulting from stresses that exceed the material's tensile strength.

CREEP: The change in shape that occurs in a molded part under stress.

CYCLE TIME: The time required by an injection molding system to mold a part and return to its original position/state.

DEFORMATION: Any changes in the shape or size of a plastic part due to some type of applied force or change in temperature.

DELAMINATION: The separation of a plastic material along the plane of its layers due to a loss of bond between laminate plies.

DIMENSIONAL STABILITY: Retention of the precise shape of the part.

DRAG MARKS: Deep scratches on the surface of a plastic component usually caused during ejection from the tool.

DRAFT: The degree of taper of a side wall or the angle of clearance designed to facilitate removal of parts from a tool.

FLASH: Any excess material that is formed with and attached to the plastic component along a seam or tool parting line.

GATING: The feature in an injection molding tool that allows the material runners to interface with the tool cavity. Gating location and design are critical to the quality of the molded part.

HAZE: The degree of cloudiness in a resin.

KNIT LINES: The point where melted material flows together during molding to form a visible line or lines on a finished part that may cause weakening or breaking of the component.

MOLDABILITY: The characteristics of a product design or a resin that allows it to be easy to mold.

MULTI-SHOT: A process where two or more resins are injected into the tool to form one part.

OVERPACKING: Melted plastic following and continuing to fill the easiest flow path in a tool while material reaches the other areas, causing unbalanced flow and warping.

PACKING: The portion of a molding cycle when pressure is applied to material in a tool cavity until the gate freezes.

PART GEOMETRY: A three-dimensional representation of the part shape.

RIB: A reinforcing member of a fabricated or molded part.

SHOT: The yield from one complete molding cycle, including cull, runner and flash.

SHOT CAPACITY: The maximum weight of material that a machine can produce from one forward motion of the plunger or screw, usually measured in ounces.

SHORT SHOT: Failure to completely fill the cavities of the tool.

SHRINKAGE: Contraction of an injection molded part upon full cooling.

SINK MARK: A depression or dimple on the surface of an injection molded part due to collapsing of the surface following local internal shrinkage after the gate seals.

STRESS: The force producing or tending to produce deformation in a body measured by the force applied per unit area.

STRIATION: Marks on the surface of a molded part that indicates melt flow direction or impingement.

SURFACE TREATING: Any method of altering the surface of a plastic part to make it receptive to inks, paints, lacquers and/or adhesives.

Injection Molding Specifics

COMPLEX INJECTION MOLDING: Translating precise, intricate part designs to injection molded parts and components often used in critical-use applications. Complex injection molding relies on multiple scientific methods, analytical tools and molding process technologies to meet the precision and performance requirements.

CUSTOM INJECTION MOLDING: Applying the processes and methodologies of complex injection molding to a unique project or application. Customization often occurs at varying degrees throughout the process and requires collaboration between the injection molder and the customer on tooling, engineering and design support, scientific methods, analytical tools, plastic part design and molding process technologies.

EXTRUSION: The process of forming continuous shapes by forcing a molten plastic material through a die.

HAND LOADS: Tool inserts that are physically placed into a tool and then removed when the part is finished.

INJECTION MOLDING AUTOMATION: Using machines to perform certain tasks within the injection molding process to maximize production and manage costs.

INSERT MOLDING: The process of adding a metal or plastic component into a larger plastic part by placing the insert into the tool cavity and injecting plastic around and over it to create the final combined part.

LIGHTWEIGHTING: A concept, especially prevalent in the auto and defense industries, that involves using injection molded plastic components to reduce product weight and improve durability.

METAL-TO-PLASTIC CONVERSION: Replacing metal components with ones molded out of plastic without compromising product function.

MULTI-MATERIAL INJECTION MOLDING: Process that provides opportunities to introduce several disparate materials into one injection molded part based upon injection molder materials expertise, specialized engineering teams and advanced molding capabilities.

OVERMOLDING: An injection molding process in which one material (usually a TPE) is molded onto a second material (typically a rigid plastic) and bonded without use of primers or adhesives to improve product safety, performance and ergonomics.

SCIENTIFIC MOLDING: A systematic approach to developing the injection molding process that uses advanced sensors and sophisticated software to monitor each phase, making sure critical variables such as temperature, flow rate, fill rate and cooling temperature remain consistent. Scientific molding engineers are specially trained to understand what happens to plastics during injection molding, down to the molecular level, and know how to quickly correct any variations that might occur to ensure greater process repeatability and control versus traditional molding methods.

Materials-Related Terminology

ALLOY: A physical modification of an existing plastic to achieve higher performance and or functionality, such as ABS.

AMORPHOUS: Plastic devoid of crystallinity or stratification, typically achieved at processing temperatures.

BINDER: A resin, glue, gum or casein used to hold particles together and reinforce certain plastics to improve strength and ensure uniform consistency, solidification and/or adhesion to a surface coating.

COMPOSITE: The combination of a thermoset or thermoplastic resin and a strengthening agent.

CORROSION RESISTANCE: The ability of plastics to resist degradation in many environments, usually due to oxidation.

CRYSTALLINITY: A state of molecular structure in some resins that denotes uniformity and compactness of the molecular chains forming the polymer.

ELASTICITY: The ability of a plastic to quickly recover its original dimensions after removal of a load that has caused deformation.

ELASTOMER: A rubber-like material that at room temperature can be stretched repeatedly to at least twice its original length and, upon immediate release of the stress, returns with force to its approximate original length.

Secondary Processes

ADHESIVE ASSEMBLY: The process of joining two or more plastic parts by means of an adhesive.

HEAT STAKING: The process of connecting two components by creating friction fit between two plastic pieces — one with a hole, the other with a boss — then using heat to deform the plastic boss.

IN-MOLD LABELING AND IDENTIFICATION: A fully automated decorating process in which a printed film or paper label is placed in the tool container or cavity before molding. The label fuses to the plastic part as it's being molded, eliminating the need for post-mold decoration or identification.

PAD PRINTING: A printing process that can transfer a 2-dimensional image onto a 3-dimensional object.

PLASTIC MACHINING: Shaping blocks of solid plastic using precision software and machinery to achieve the desired part dimension. This process differs from injection molding, wherein melted plastic pellets are injected into a tool cavity and cooled so they take on the shape of the tool.

POTTING: The process of sealing an enclosed component with a plastic compound or material that protects it from moisture, corrosion, shock and/or vibration.

SNAP-FIT CONNECTIONS: The method used to attach flexible plastic parts to form the final product by pushing the parts' interlocking components together.

SPIN WELDING: A process of fusing two objects by forcing them together while one of the pair is spinning, until frictional heat melts the interface. Spinning is then stopped and pressure held until they are frozen together.

ULTRASONIC WELDING: A process of joining complex plastic injection molded parts using high-frequency sound waves and pressure instead of fasteners or adhesives.

Tooling

CAVITY: A depression in an injection tool which forms the outer surfaces of the plastic component.

COOLING CHANNEL: A channel located within the body of a tool through which a cooling medium is circulated to control the tool surface temperature.

EJECTOR PIN: A rod, pin or sleeve that pushes a molded part off of a core or out of a cavity of a tool.

FAMILY MOLD: An injection mold tool with more than one cavity. Each cavity has a different geometry.

FILL: The portion of the molding cycle when molten material flows into the cavity or cavities of the tool as required to complete a part.

FILL PATTERN: The contours of the advance of the plastic as the tool cavity fills.

FILL PRESSURE: The pressure required to fill the tool cavity.

FILL TIME: Time required to fill the tool cavity.

GATE: The feature in an injection molding tool that allows the material runners to interface with the tool cavity. Gating location and design are critical to the quality of the molded part.

HOT RUNNER MOLD: A thermoplastic injection mold in which the runners are insulated from the chilled cavities and remain hot so that the center of the runner never cools in normal cycle operation. Runners are not, as is usually the case, ejected with the molded pieces.

MULTI-CAVITY MOLD: A tool with multiple cavities of the same part that is typically used to reduce piece-part pricing for higher volume parts.

PARTING LINE: Mark on the injection molded part indicating where the two halves of the tool met in closing.

PLATENS: The mounting plates of a press on which the tool halves are attached.

PROTOTYPE TOOL: A preliminary tool used for testing, design adjustments and creation of the final production tool.

RECOVERY TIME: The length of time for the molding machine screw to rotate, create a shot and return to original position.

RUNNER: The feed channel, usually of circular cross section, which connects the sprue with the cavity gate. The term is also used for the plastic piece formed in this channel.

SCRAP: Any output of a tool that is not usable as the primary injection molded part.

SCREW TRAVEL: The distance the screw travels forward when filling the tool cavity.

SIDE-DRAW PINS: Projections used to core a hole in a direction other than the line of closing of a tool and which must be withdrawn before the part is ejected from the tool. These are also commonly referred to as cam pins or angle pins.

SPRUE: The feed opening provided in injection molding between the nozzle and cavity or runner system.

TIE BARS: Bars that provide structural rigidity to the clamping mechanism of a press often used to guide platen movement.

TONNAGE: The measure by which injection molding presses are typically categorized, representing the clamping force of the press.

VENT: A shallow channel or opening cut in the cavity to allow air or gases to escape as the melt fills the cavity.

Product development engineers are increasingly relying upon injection molding processes and injection molder expertise to create dynamic, high-performance components for critical-use applications. Aligning needs and methods early in the design process ensures consistent, cost-effective outcomes and faster time to market.

Learn how Kaysun can bene it your current or upcoming project by reaching out the the Kaysun team at 920-686-5800.

KAYSUN
INJECTION MOLDING & ENGINEERING SOLUTIONS

Follow us on: