

In addition to the manufacturing of parts, plastic injection molding involves a variety of related fields, including mold building, engineering, part design, quality assurance and inspection, machining, secondary operations, decoration, assembly, and more. As such, there are quite a few specialized concepts used by industry professionals. Accordingly, we compiled this glossary to act as a simple guide and a quick reference tool for all those interested. If there is a topic you do not see here or would like to know more about, we would be happy to provide you with whatever information you need.

## **ADDITIVES**

Material additives are combined with different plastics to enhance certain characteristics, such as to provide UV protection, strength, antimicrobial protection, and flame retardance.

## **ALTERNATIVE RESINS**

Alternative resins are replacement materials that could lower costs, improve mold performance, and/or enhance part quality.

## **AMORPHOUS**

See **Crystallinity**.

## **ASSEMBLY**

Assembly related to plastic injection molding can include everything from attaching two parts to combining a part and other components into a portion of or a complete product, some of which often can be done by a machine operator while the machine is producing parts.

## **BRIDGE TOOLING**

Bridge tooling refers to building low-volume molds to transition a plastic injection molding program from the R&D or prototyping phase to fabricating high-volume, multiple cavity molds.

## **CAVITY**

In a plastic injection mold, the cavity is an impression in the front, stationary half of the mold.

## **CAVITY BALANCE**

Multi-cavity plastic injection molds should have cavity balance, meaning the number of cavities or flow groups (collections of cavities) on both sides of the sprue should be equal, and there should be a geometrically balanced runner system.

## **CLAMPING FORCE**

Clamping force is the amount of pressure applied by an injection molding machine to a mold to keep it closed, which is in opposition to the fluid pressure of the compressed resin within the mold cavity and runner system.

## **COLD RUNNER**

In a plastic injection mold with a cold runner, molten thermoplastic is injected from the molding machine through a sprue into a series of runners leading into the mold cavities, which form the parts. (See also **Sprue** and **Hot Runner**.)

## **COLORANT**

Plastic colorants are chemical compounds, which are added to the base material of a plastic injection molded part to achieve the desired hue, and they come in the form of dyes and pigments. (See also **Let-down Ratio**.)

## **CORE**

The core is the movable, rear half of the mold that forms the part when the mold is closed.

## CRITICAL DIMENSIONS

Critical dimensions are identified locations on an injection molded part where specific tolerances must be met to ensure proper form, fit and function of the part.

## CRYSTALLINITY

The structure of a polymer is defined in terms of crystallinity or the degree of order in the molecules' arrangement. A well-ordered polymer is considered crystalline, which is what gives them strength and rigidity. The opposite is called amorphous, which has greater flexibility and elasticity. Most polymers have some of both structures.

## CYCLE COUNTERS

Cycle counters monitor the opening and closing of plastic injection molds and cannot be reset.

## CYCLE TIME

Cycle time is the total time required to complete all stages of the injection molding cycle, including Fill (Injection), Packing/Holding (Pressure), Cooling (Mold Closed), and Mold Open (Ejection).

## DECORATION

Plastic injection molded parts can be decorated using a variety of methods, the most common of which include pad printing, hot stamping, thermal transfer, and in-mold decoration.

## DENSITY

Density is the mass per unit volume of a material (Density = Mass / Volume). (See also **Mass** and **Specific Gravity**.)

## DESIGN FOR MANUFACTURABILITY (DFM)

A Design for Manufacturability (DFM) study analyzes the design of an injection molded part with the intent of optimizing the quality of the part and the efficiency of the manufacturing/molding process.

## DRAFT

Draft refers to the angle, slant or taper of a part surface or face, which facilitates ejection of the part from the mold.

## EJECTION

Part ejection in a plastic injection mold commonly is achieved by using ejector pins, sleeves, blades, stripper rings, pusher plates, compressed air, or a rotating core (for internal threads).

## FAMILY MOLD

A family mold has cavities to make two or more different parts. They often have runner shutoffs, so parts can be produced independently, even using different colors and plastics.

## FIRST ARTICLE INSPECTIONS (FAI)

First article inspections evaluate every aspect of an initial sample in relation to governing specifications, which can include dimensional, material and aesthetic requirements.

## GATE

The gate is where molten plastic enters the cavity in a mold and forms the part.

## HOLES

Through holes (or thru holes) go completely through a part, whereas blind holes extend to a certain depth and do not break through to the other side.

## HOT RUNNER

In a plastic injection mold with a hot runner, plastic is injected directly into the cavities, thereby eliminating the series of runners.

## HYGROSCOPIC

Hygroscopic resins absorb moisture into their molecular structure if exposed to humid air, whereas in non-hygroscopic plastics, absorbed moisture only collects on the surface.

## IMPACT MODIFIERS

Impact modifiers are additives included in a polymer compound to compensate for brittleness by absorbing the energy of an impact or by dissipating it and usually are elastomeric or rubbery in nature.

## INSERT MOLDING

The process of insert molding involves molding plastic around a preformed insert, which usually is metal. The inserts are placed in the mold during the injection molding process, thereby eliminating the need to install the inserts after the parts are manufactured, thus often reducing costs.

## INTERCHANGEABLE INSERTS

A part with different variations sometimes can be made using interchangeable inserts in a mold instead of building separate tooling for each version of the part.

## INTERNAL THREADS

The most common methods for creating internal threads in a molded part are with rotating cores, collapsible cores, hand-loaded inserts, bump-off ejection, or by machining as a secondary operation.

## LAY

The lay is the general direction of the pattern of a surface. (See also **Surface Finish**, **Surface Roughness**, **Waviness**.)

## LET-DOWN RATIO

The amount or percentage of color concentrate used when molding a colored part is called the let-down ratio, which typically is between 1% to 5%, and the percentage used depends on how much colorant is needed to achieve a color match using the base resin. (See also **Colorant**.)

## LIVING HINGES

Living hinges are thin, flexible sections of a plastic part that can bend and fold repeatedly and often are used to reduce the number of components in an assembly.

## MASS

Mass is the amount of matter in an object (as opposed to weight, which depends on gravity). (See also **Density** and **Specific Gravity**.)

## **MATERIAL CERTIFICATION**

While different names and definitions exist, a Certificate of Compliance or a Certificate of Conformance (or Conformity) usually certifies (a) that the material supplied or used in the manufacturing of goods contained within a shipment conforms to the requirements; (b) that parts have been processed in accordance with applicable instructions and specifications; or (c) that parts conform to the applicable specifications. Information related to testing and traceability often is included.

## **MATERIAL DATA SHEETS**

Material data sheets for plastics are provided by the polymer suppliers and describe the properties of a specific grade, which can include physical, mechanical, thermal and processing information.

## **MESH MODEL**

Mesh model part files describe triangulated or tessellated surfaces containing geometric shapes and are used by 3D printers. (See also **Solid Model**.)

## **MOLD CLASSIFICATIONS**

Commonly-accepted plastic injection mold classifications range from Class 105 (Prototype/Up to 500 cycles) to Class 101 (High Volume/1 Million+ cycles) and consider mold design, construction, materials and components.

## **MOLD QUALIFICATION**

Mold qualification looks to see if part dimensions are to specification and if the part meets aesthetic or surface requirements.

## **MOLD STACK HEIGHT**

The height of a plastic injection mold refers to the distance between the two outside support plates (or when the plates are “stacked” on top of each other), although that dimension is horizontal when placed in most molding machines.

## **OVERMOLDING**

Overmolding is a process where one type of thermoplastic, (often an elastomer or a urethane), is molded over a previously-molded part, which acts as a substrate, thereby producing a single part with two layers of plastic.

## **PARTING LINE**

The parting line is a seam on a finished part where the two mold halves met when they were closed.

## **PLASTIC INJECTION MOLDING**

Plastic injection molding is a process where parts are produced by a machine that injects molten plastic into a mold.

## **PREVENTATIVE MAINTENANCE**

Preventative maintenance of a plastic injection mold includes cleaning and repairs and should be performed regularly over the life of the mold to ensure mold performance, tool longevity, and part quality.

## **PROCESS VALIDATION**

In plastic injection molding, process validation establishes the optimal process parameters for a particular mold, so that it consistently produces parts that conform to the specifications.

## **QUALITY ASSURANCE**

Quality assurance is the part of a quality management system that focuses on ensuring that quality requirements are met.

## **QUALITY CONTROL**

Quality control is the part of a quality management system that focuses on fulfilling quality requirements.

## **QUALITY INSPECTION**

Quality inspection includes measuring, examining, testing and analyzing the product of a manufacturing process to determine conformity with quality requirements.

## **REGRIND**

Regrind refers to plastic from sprues, runners and rejected parts, which has been reclaimed, granulated and usually mixed with virgin material in a predetermined percentage to mold parts. (See also **Virgin Material**.)

## **RELATIVE THERMAL INDEX**

Relative thermal index is a measurement of the maximum temperature a material can be exposed to over time without significant loss of properties.

## **REQUEST FOR QUOTE (RFQ)**

A request for quote (or quotation) is a process where a company asks for a quote (or proposal) from a supplier for a product or service.

## **RUNNER**

A runner is a channel in a plastic injection mold that feeds molten resin from the nozzle or sprue into the gate.

## **RUNNER SHUTOFFS**

Runner shutoffs block molten resin from entering certain cavities in a plastic injection mold and generally are used in family molds, where part demand can be unequal.

## **SECONDARY OPERATIONS**

Secondary operation performed by plastic injection molders can include machining, decorating, assembly, ultrasonic welding, and other finishing services.

## **SHORE HARDNESS**

Shore hardness of a plastic, elastomer or rubber is an indication of its resistance to indentation and is measured with a durometer.

## **SHOT SIZE**

Shot size is the amount of resin required to fill the sprue, runner and cavities of a plastic injection mold and can be calculated in terms of volume or mass.

## **SHRINKAGE**

Shrinkage is the reduction in size of an injection molded plastic part as it cools after injection, and all polymers have different shrink rates.

## **SIDE ACTIONS**

Side actions are mechanical components in a plastic injection mold that are used to create undercuts or internal threads in parts by operating and moving independently from the two mold halves, examples of which include slides, lifters, collapsible cores, and unscrewing (or unwinding) mechanisms.

## **SOLID MODEL**

Solid model files are three dimensional representations of parts containing geometric and topological data, which are used to build plastic injection molds. (See also **Mesh Model**.)

## **SPECIFIC GRAVITY**

Specific gravity is the ratio of the density of a material to the density of water (which is 1 gram per cubic cm).

*Quick Conversion:* Density / 1000 = Specific Gravity *or* Specific Gravity x 1000 = Density. (See also **Density** and **Mass**.)

## **SPRUE**

The sprue is where molten plastic flows from the injection molding machine into the mold.

## **STEEL SAFE**

Plastic injection molds often are built steel safe in certain critical areas, leaving extra metal in those spots that can be machined away for any necessary adjustments.

## **SURFACE FINISH**

Surface finish of a plastic part refers to its texture or topography (i.e., the surface shapes and features or the three-dimensional quality of the surface), which generally is achieved by polishing the mold and can range from high glossy to rough. Texture can be characterized by three things: The lay, surface roughness, and waviness of the surface. (See also **Lay**, **Surface Roughness**, **Waviness**.)

## **SURFACE ROUGHNESS**

Roughness refers to the average of vertical deviations from the surface over a specified length of surface. (See also **Lay**, **Surface Finish**, **Waviness**.)

## **TECHNOPOLYMER**

A technopolymer is a base resin, often with a reinforcing additive, used to produce a part requiring high strength and impactability, sometimes replacing metal as the part's material.

## **THERMOPLASTIC ELASTOMERS**

Thermoplastic elastomers have both plastic and rubber-like qualities and can be injection molded.

## **THERMOPLASTICS**

A thermoplastic is a polymer that can be melted and reshaped repeatedly. (See also **Thermosets**.)

## **THERMOSETS**

A thermoset is a resin that is irreversibly hardened by curing from a soft solid or viscous liquid. (See also **Thermoplastics**.)

## **TOLERANCES**

Tolerances for specified dimensions of a molded part are ranges of acceptable deviation from too small to too large.

## **TONNAGE**

Tonnage represents the clamping force of a plastic injection molding machine.

## **TOOL**

In the plastic injection molding industry, the term “tool” often is used to refer to a mold.

## **UNDERCUTS**

Undercuts are protrusions or indentations that impede ejection of a part and usually require slides in the mold.

## **UV RESISTANCE**

UV resistance for plastic parts is created by including an additive with the polymer to inhibit degradation, discoloration or deterioration, which is applied in different concentrations depending on the protection time desired.

## **VENT**

Vents are shallow grooves or slots machined into the cavity blocks of an injection mold, which allow air and gases to escape from the cavity as it is being filled with molten plastic.

## **VIRGIN MATERIAL**

Virgin material refers to plastic that has not been used, processed or mixed with regrind or another material. (See also **Regrind**.)

## **WAVINESS**

Waviness is the measure of surface irregularities with a spacing greater than that of the surface roughness. (See also **Lay**, **Surface Finish**, **Surface Roughness**.)



Precision Molded Plastics is an ISO certified, vertically integrated, plastic injection molding company that builds custom molds and tooling, manufactures parts and components, and performs a variety of value-added services all under one roof.

## **QUESTIONS?**

Contact our support team > 909-981-9662