During the injection molding process, hot melted resin is forced under pressure through a cavity mold and then cooled to form a shape. During that cooling time of the process, the resin shrinks a small amount. Most of the shrinkage occurs within the mold, but until the shape has fully cooled to its core it will continue to shrink.

At SEA-LECT Plastics, we use a variety of engineered resins and materials during molding to increase durability and strength of every part we mold or assemble. Plastic injection molding part shrinkage units are expressed as thousandths of an inch per linear inch (0.00x /in/in). Typical shrink rates vary between 0.001/in/in and .020/in/in, with the average shrink rate around 0.006/in/in. For parts designed by our team of engineers, we incorporate the appropriate shrink rate into the part design and tooling in the early stages based on the material chosen for the molded part. Thickness of the part will also have an effect on the part, which will also be designed in accordingly.

This table shows the shrink rate of many commonly used engineered resins in the industry:

Plastic Material Name	**Shrinkage Ratio (%)**	**Plastic Material Name**	**Shrinkage Ratio (%)**
Acrylonitrile Butadiene Styrene (ABS)	0.4〜0.9	Acrylonitrile styrene (AS)	0.2〜0.7
Polystyrene (PS)	0.4〜0.7	Ethylene vinyl acetate (EVA)	0.7〜1.2
Poly Propylene (PP)	1.0〜2.5	Poly Propylene (with 40% glass fibers)	0.2〜0.8
High-Density Polyethylene (HDPE)	2.0〜6.0	Methacrylic Acid Methyl Ester (acrylic) PMMA	0.1〜0.4
Polyamide (Nylon 6)	0.5〜1.5	Polyamide (Nylon 66)	0.8〜1.5
Poly Acetal (POM)	2.0〜2.5	Poly Butylenes Terephthalate (PBT with 30% glass fibers)	0.2〜0.8
Polycarbonate (PC)	0.5〜0.7	Poly Phenylene Sulfide (PPS with 40% glass fibers)	0.2〜0.4

Liquid Crystal Polymer (LCP with 40% glass fibers)	0.2～0.8	Modified Polyphenylene oxide (Modified PPO)	0.1～0.5
Poly Sulfone (PSF)	0.7～0.8	Polyether Sulfone (PES)	0.6～0.8
Poly Ethylene Terephthalate (PET)	0.2～0.4	Polyether Ether Ketone (PEEK)	0.7～1.9

The Two Classifications of Resins

There are two main classifications of resins that you need to understand the differences between: Amorphous and Crystalline.

Amorphous - The common plastic resins that are in this classification are Polystyrene, Polycarbonate, PMMA (Acrylic), ABS, and PVC. This group of resins generally does not have an organized polymer structure and the shrinkage occurs due to thermal contraction in the mold during cooling. The shrinkage is usually more uniform (the same form all directions) and usually a small percentage. A major difference to crystalline plastic resins are that amorphous plastic resins typically have a very small dimensional change after molding.

Crystalline - The common plastic resins that are in this classification are Acetal, Nylons, Polyester, Polypropylene, and Polyethylene. Crystalline materials usually have a more structured molecular organization than the amorphous classification. The shrinkage percentage is greater due to a process of thermal contraction during cooling and re-crystallization. These resins should shrink .010-inch to .030-inch per inch of part length. One additional note is that crystalline resins have a tendency to shrink at a larger percentage rate in a transverse direction to the flow of the material in the mold (as opposed to the direction of flow). Crystalline materials continue shrinking beyond molding and in fact roughly 90% of the shrinkage occurs within the first hours after molding. The last 10% can continue until 48 hours after molding of the final part.

How Can Shrinkage Be Affected During Molding?

During the molding process, adjustments can be made to effect on the shrinkage of the end part. Typically the options are to adjust the density and increase or decrease the shrink rate.

Density – The density of the material can be fine-tuned during molding. The part can be allowed to cool longer in the mold or the pressure can be increased to pack the material more. Each tool and material should be tested to optimize the settings in order to achieve the desired part.

Increasing shrinkage **–** There are a handful of ways to increase shrinkage of a component during molding: a shorter cooling time in injection mold, a higher mold surface temperature, a larger thickness, a higher melt temperature during injection, additional plasticizer in base resin, decreasing injection speed, and decreasing packing density or holding pressure. Each of these can increase the shrinkage individually and also used together to fine-tune the molded component result.

Decreasing shrinkage **–** If you desire a decreased shrink rate, add cooling time in the injection mold, lower the mold surface temperature, decrease component wall thickness, decrease the resin melt temperature during injection, increase resin injection speed, and increase packing density or holding pressure. Each of these can decrease the shrink rate individually or in combination. Fillers such at talc, glass beads, and glass fibers may be added to change the shrink rate if the end component can accommodate their inclusion.

Does Shrinkage Occur Equally?

The resin being used will have a great effect on the shrink rate, but also the direction of the shrinkage occurring. Resins with polymer fillers will shrink more in the transverse (cross section) than the longitudinal (flow direction of the material) within the tool. It creates a challenge for every tooling design to accommodate a resin that doesn't shrink symmetrical. How to accommodate such a design is to only use it with non-critical tolerances. If the end part has critical tolerances, such as datum holes for assembly, they should be added after the part has cooled to prevent assembly issues.

Materials can play a factor in the shrink rate of every molded part, but part design also has a large effect on the molding process. One design consideration that needs to be addressed is ensuring a uniform wall thickness. Thicker sections will have a slightly varied shrink rate than thin section, which can affect the overall part dimensions.

How Can Tooling Design Affect Injection Molding?

Plastic resin flow through the mold can be affected by the part design. One way to ensure material reaches every section of the molding tool cavity is to place an entry gate where the material can flow easily before it starts to cool inside the tool. Uneven cooling can affect the overall part thickness and dimensions. The cooling rate and running mold temperature significantly affect the end parts, especially for crystalline materials. Over 75% of the molding process is the cooling portion of the material, and having a tool capable of ensuring uniform cooling will reduce the chance of non-uniform cooling and poorly molded parts.

Injection speed, packing density, and mold temperatures can affect every facet of the end component. Injection molding is a complex process and a key to success is careful design and planning. We have experts at every step in the process.