



Key Terms To Know For Plastic Injection Molding

K&B
Molded Products



Boss: Raised rounded stud feature on plastic parts and molds.

Cavity: The space between both sides of the mold. This is mainly concave and the upper part of the mold, which is filled to create the injection-molded part—sometimes referred to as the “A side” or “A part” of the mold and usually the surface side of the finished product.

Core: Generally the half of the mold where the plastic part is ejected from. Usually at the bottom and sometimes referred to as the “B side” or “B part” of the mold.

Core Outs: Area of the part that is removed to achieve uniform thickness.

Draft: Angle and taper built into walls and ribs of the part that facilitate removal from the mold.

Flash: Excess material that goes outside the intended geometry of the part, usually appearing on parting lines.

Gate: The point where plastic enters the mold through the cavity. There are automatically trimmed gates, which shear when the part is ejected, and manually trimmed gates, which require a secondary operation by an operator in order to be separated from the mold.

Hand Load: A metal feature used to create details like undercuts in molded parts. They are removed manually during or after ejection.

Heel or Lock: This is the piece of a custom automatic injection mold that keeps the slide positioned forward when the mold is closed.

Horn Pin or Cam Pin: Pins that are used to activate the slide on an automatic injection mold.

Line of Draw: The direction in which the two halves of the injection mold separate from the part, thereby allowing it to be ejected without metal obstructions that cause undercuts.

Ribs: Thin features that strengthen wall sections and bosses while minimizing warp.

Runner: A canal cut into custom injection molds, where the plastic travels from the injection-molding machine, through the sprue, runner, and the gate until it eventually fills the mold to make the part.

Shear: The force created between layers of resin as they glide against one another during various phases of the injection molding process. This can cause heat to develop.

Short Shot: When a plastic part doesn't fill the mold completely. This results in features being absent from the part.

Shrink Rate: The rate of how much plastic will shrink when cooled, ranging from .001 per inch to as much as .060 per inch. Most shrinkage rates fall in between .004" and .021."

Side Action: Phrase used to describe the hand pulls or slides commonly utilized in injection molding.

Sink Marks: Depressed areas of a molded part due to thickness ratios being off in walls and ribs.

Slide: Feature required in automatic injection molds for creating undercuts.

Sprue: The canal that connects the injection molding machine nozzle to the runner.

Steel Safe: The amount of metal left on the mold in order to fine-tune a dimension. An example would be if you have an inside diameter that needs to be .300" you may leave the mold at .305; in case of shrinkage.

Thin Wall Molding: When molded parts have walls between .005" to .060" thick.

Undercuts: Area of the designed component where a slide or hand pull is necessary to make clips, windows, or holes that are not in the line of draw.

Vestige: Excess material that protrudes from the gate and usually needs to be trimmed by the molding machine operator after the gate runner has been removed from the injection molded part.

Wall Thickness: The thickness of a cross section in a plastic part.

Warp: A distorted area of an injection molded part. This usually happens after molding or during cooling and is often caused by residual stresses in the part resulting in differential shrinkage.