

Glossary of Injection Molding Terms

In most industries, a specialized vocabulary has developed over time to facilitate clear discussion of the unique equipment and processes involved. The injection molding industry is no exception; many specialized terms are used to differentiate types of equipment, parts, and processes.

Some injection molding terminology is common to other industries. “Die,” for example, which refers to the primary shaping mechanism of the injection molding process, is used in several other industrial manufacturing processes. But there are also many terms unique to the injection molding field. These industry-specific terms can be confusing — especially for those who are working with an injection molder for the first time — so it’s important to start out with a solid understanding of key phrases.

To help you navigate injection molding terminology and get your next project off to a quick, smooth start, we’ve put together a glossary of some of the most commonly used industry terms.

B

Boss A boss is any round protrusion on a mold or part.

C

Cavity A cavity is the interior of an injection molding die, specifically its top half. It is usually, but not always, the “show surface” of the finished part and is most commonly concave in shape.

Core The bottom half of the die, the core is the side of the die into which plastic is injected.

Core-outs A core-out is the gutted portion of a completed injection molded part. This gutting is done to create uniform wall thickness (see “wall thickness” definition below), reduce the part weight, and/or reduce warping (see “warp” definition below).

D

Draft Referring to a section of an injection molded part that has been designed with a taper, drafts make it easier to remove the part from the mold. With very few exceptions, injection molded parts are generally designed with drafts.

G

The gate is the point at which molten plastic enters the cavity of the die. There are two types of gates:

Gate

- **Automatically trimmed** — Automatically trimmed gates have a specialized feature that automatically breaks or cuts the gate as the molding tool opens to eject the part.
- **Manually trimmed** — Parts with manually trimmed gates require a human operator to remove the gate runner from the part.

Gibbs

This refers to the section of a custom injection mold that holds the slide in place so that it can be actuated by the cam.

H

Hand Load

A hand load is a feature in a mold — usually made from steel or aluminum — that creates undercuts (see “undercuts” definition below) in a molded part. Hand loads must be manually removed from a completed part after ejection.

Heel

The heel on a custom injection mold holds the slide in a forward position when the molding machine is closed on the mold.

Horn Pin

The horn pin (or cam pin) activates the slide in an automatic injection molding process.

L

Line of Draw

The line of draw is the direction in which the mold’s halves are separated after injection. The line of draw must be free of obstructions, such as hand loads.

R

Ribs

Usually thin blade-like structures, ribs are used to provide additional wall or boss strength, or to help minimize warping.

Runner

The runner is a channel cut into an injection mold, connecting the sprue (see “sprue” definition below) with the gate and ultimately the cavity, through which molten plastic travels during injection.

S

Shear

Shear refers to speed- and pressure-induced friction inside of a mold that maintains the melt of the plastic while the cavity fills. Too little shear can lead to material freeze-off and subsequent short shot (see “short shot” definition below), while too much shear can burn the plastic.

Short Shot

Commonly caused by low shear, short shot refers to an injection that fails to completely and properly fill a mold cavity.

S

Shrink Rate	Shrink rate is the measure of how much a plastic will shrink as it cools. Every plastic has a different shrink rate, which can be as little as 0.001 inches or as great as 0.060 inches, with an average between 0.004 inches and 0.021 inches. Shrink rate is generally accounted for during the design phase and added to the size of the mold so that completed parts are the proper size after shrinking.
Side Action	Side action refers to any slide or hand pull used during the tooling of an injection molding die.
Sink Marks	Sink marks are any section of a completed plastic injection molded part that has become sunken in appearance. Sink marks are generally caused by nonuniform walls, extra-thick walls, or poor rib or boss ratios.
Slide	The slide is the area of an injection molding die used to create undercuts. This is a required feature in automatic injection molding scenarios.
Sprue	The sprue of an injection molding die is the passage that connects the nozzle of the molding machine with the runner of the die.
Steel Safe	Steel safe is any amount of extra metal intentionally left on a mold. This allows small dimensional tweaks to be made to correct unanticipated issues, such as poor flow or excessive shrinking, without retooling the entire mold.
Thin wall molding	Thin wall molding is the molding of any plastic injection part that has one or more notably thin walls, between 0.005 inches and 0.060 inches.
Undercut	An undercut is any part design feature — such as a clip, window, or hole — that requires a hand pull, slide, or hand load to create.
Vestige	Vestige refers to any extra material extending from a molded part at the area of the gate after gate runner removal (such as by an automatically trimmed gate). Vestiges generally require manual removal.
Wall Thickness	Wall thickness is a measure of the thickness of any given cross section of a plastic part.
Warp	Warp describes any area of a completed injection molding part that has been distorted in some way, whether during injection or cooling. Warp is often, but not exclusively, caused by lack of uniformity in wall sections of a part design.

T

U

V

W