

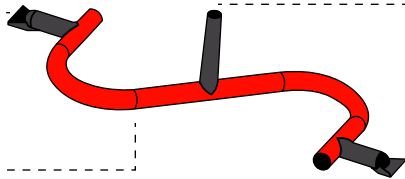
WHAT YOU NEED TO KNOW ABOUT GATES IN INJECTION MOLDING

The gates used in the production of your injection molded plastic parts can impact their cost, quality, and performance. This tip sheet gives you an overview of gating systems and tips on optimizing gate size and location to achieve a better final product.

GATING SYSTEM BASICS: WHAT ARE GATES, SPRUES, AND RUNNERS?

GATES: The narrow intersection between the main runner and the mold cavity. This is the point where the molten plastic enters the cavity.

- Gate location is critical to the performance and quality of the final molded part.
- Gate marks are visible from where they are cut off.



RUNNER: The channel that feeds molten plastic from the sprue to the cavity gate.

- Runner diameters should be as small and short as possible, yet large enough to fully pack the cavity.

SPRUE: A channel through which the molten plastic enters the mold.

- The sprue bushing is a component used within the sprue and works with runners and gates to inject molten plastic towards the cavity.
- They are manually cut from the piece or ejected from the mold once the part is ejected from the mold.
- Molders keep the sprue diameter as small as possible, yet large enough to pack the cavity to avoid waste.

TYPES OF GATES

EDGE	The most common gate type in injection molding. It is placed along the edge of the part, leaving a visible mark at the parting line. Suitable for flat parts, medium or thick sections.
TAB	Contains an auxiliary tab section, where shear stress can safely be absorbed without affecting quality. Suitable for thin and flat parts, and materials like PC, ABS, and acrylic.
FAN	A wide edge gate with variable thickness, which permits rapid and balanced filling of large parts through a large gate cross-section.
DIRECT/ SPRUE	The sprue carries material directly into the mold cavity. Easy to implement but produces high tensile stress around the gate. Suitable for single-cavity molds and non-aesthetic parts.
SUBMARINE/ TUNNEL	Used in two-plate molds. Uses an angled and tapered channel that meets the cavity near the parting line, filling the cavity from underneath the parting line. Suitable for hiding gate blemishes.
SPOKE/ CROSS	Typically has four injection gates to inject material into the mold. This configuration is used when making tube-type parts of different shapes.
DIAPHRAGM	Designed for cylindrical parts that are hollow in the middle. Gives the plastic some stability and produces parts with little to no blemishes since it is in the hollow part of the mold.
HOT RUNNER VALVE GATE	A type of hot runner gate that features a special pin or valve in the design. Similar in function to thermal gates, apart from the valve. The valve allows the flow of plastic into the mold to be turned on and off as needed.

RUNNER SYSTEMS

COLD RUNNER MOLDS	Usually consist of two- or three-plate molds that are held within the mold base. <ul style="list-style-type: none"> • More cost-effective than a hot runner. • Lower maintenance cost. • Allows for fast changes or adjustments. • More flexibility. • Handles a variety of commodity and engineering-grade resins.
HOT RUNNER MOLDS	Consist of 2 plates—with one containing a heated manifold system. <ul style="list-style-type: none"> • Faster cycle times. • Eliminates runner system. • Reduces post-processing of the part, like gate trimming. • Reduces waste from the runner system. • Increases overall efficiency of the molding process.

WHAT YOU NEED TO KNOW ABOUT GATES IN INJECTION MOLDING

GATE SIZE AND LOCATION

Gate size and location are the two most critical factors of the gating system that will directly impact your design decisions.

WHAT INFLUENCES GATE SIZE?

Material

Wall Thickness

Aesthetics

WHAT INFLUENCES GATE LOCATION?

Mechanical Loading

Fill Pattern

Aesthetics

Your injection molding partner will determine the optimal gating system, size, and location. They will base this decision on your design intent, performance and cosmetic expectations, and the material being molded.

HOW DOES MY MATERIAL SELECTION INFLUENCE THE GATING SYSTEM?

Plastics are chosen for characteristics such as tensile strength, durability, and mechanical properties based on the intended end use. Because of these unique properties, the rate at which plastics melt, flow, heat, and cool differ from one to the next. This is why mold designers must optimize the size of the gate to facilitate the flow of plastic without compromising its properties. Otherwise, the plastic could be degraded and produce a low-quality part.

Want tips for optimizing your part design to ensure proper gate location? Download our eBook [A Guide to Gating Systems](#).

GATE PLACEMENT TIPS

1	Keep away from cores and pins.	3	Minimize flow path length.
2	Allow space for de-gating.	4	Use multiple gates if necessary.



DID YOU KNOW?

Gate location can affect:

- Structural integrity, including the overall strength of the part.
- The ability to hold your desired roundness or flatness.
- Operational considerations like what injection pressure is needed.
- Aesthetics such as the appearance of sinks, voids, and flow lines.