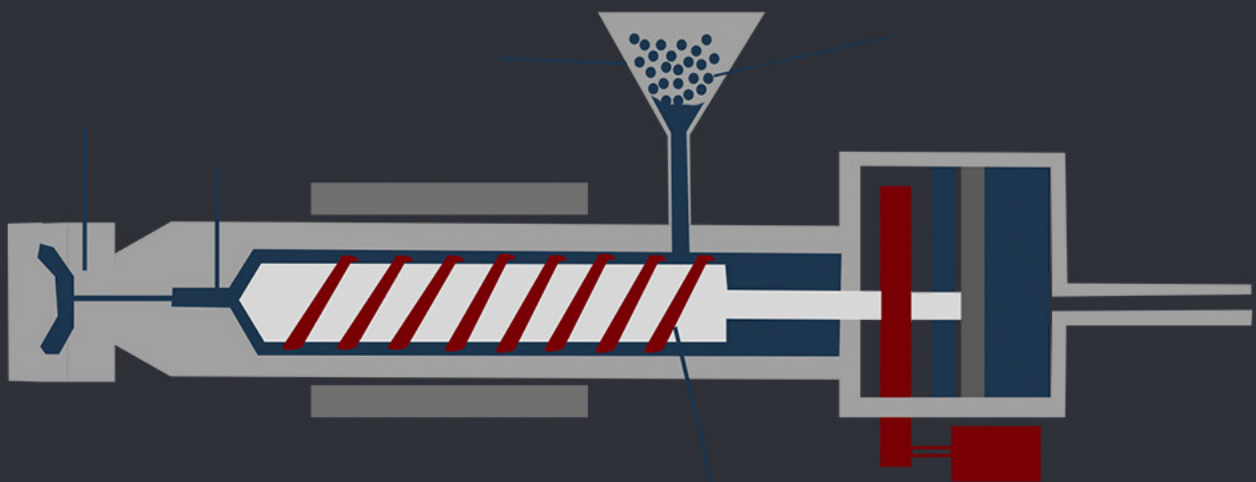


Plastic Injection Molding

FUNDAMENTALS

For Product Designers **and** Engineers



STAR RAPID 

Contents

1	Introduction	37	Chapter Eight Anatomy of a Plastic Injection Mold Tool
2	Chapter One Plastic Resin	41	Chapter Nine The Plastic Injection Molding Machine
6	Chapter Two Surface Finishing for Mold Tools	44	Conclusion
11	Chapter Three Coloring for Plastic Resin	45	Glossary of Terms
15	Chapter Four Preparing Design Drawings		
18	Chapter Five Understanding the Molding Process		
21	Chapter Six Common Defects and Design Optimization		
31	Chapter Seven Gates and Runners		

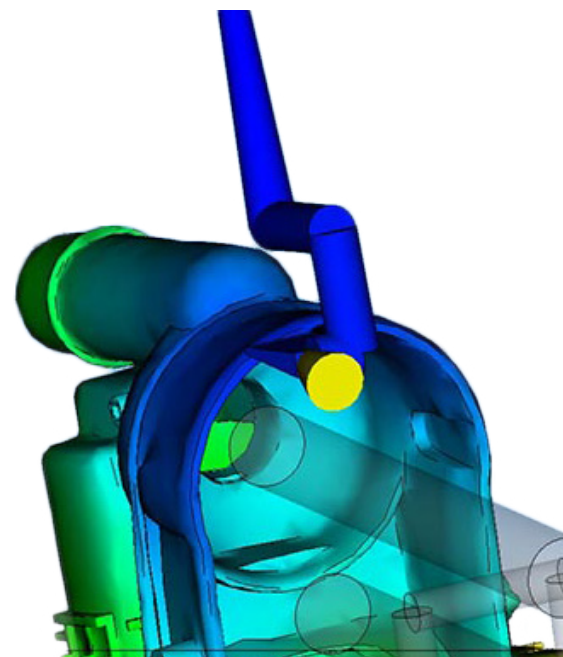


Introduction

What does a product engineer or designer really need to know when preparing a project for plastic injection molding? There are libraries of technical information for injection molding, but not all of them may be directly relevant to a product designer's special needs.

This eBook aims to provide you with relevant content that you can use to choose the right material for any project, to plan production goals realistically, and optimize design drawings to reduce product defects.

The information offered here is meant to address the majority of concerns that product developers and engineers come across every day. Since every project is unique, you will need to work closely with your supplier to get exactly the support you need.





Chapter One

Plastic Resin



Currently there are more than 20,000 commercially available thermoforming resins on the market. There are resins to meet a wide array of demanding applications, so no doubt there are many choices available to suit your needs.

To help you understand these choices, here is a list of some of the most common plastic molding materials and their uses. These thirteen resins, in various formulations, cover the majority of commercial and industrial products. They are stable, available from stockists around the world, and their characteristics are well understood to allow for predictable and reliable process set-ups.



Nylon

Nylon produces strong mechanical parts. Bushings, gears and bearings are some common automotive parts made with nylon. It's tough, has a high melting temperature, good wear and chemical resistance.



Acrylic

Acrylic is used to make transparent parts such as windows, clear pipes and various lighting equipment. It's often used as an alternative to glass due to high tensile strength and weather and scratch resistance.

Polycarbonate

Polycarbonate has excellent optical properties and is extremely durable. Precise dimensional control can be maintained as it has predictable and uniform shrinkage properties.

PC + ABS

Combining the best qualities of both materials, this resin, in various formulae, forms a rigid and dimensionally stable plastic often used for electronic enclosures and in automotive applications. Fire-retardant additives are used to enhance its safety rating and versatility.

PCTG

This is a tough, clear resin with excellent chemical resistance and durability in harsh environments. It can be used for medical devices as well as food-safe



containers and clear water bottles. It recycles well and is becoming more popular as a filament for 3D plastic FDM printing.

POM

Polyoxymethylene (POM) is a type of acetal resin used as a substitute for metal. This engineering-grade plastic is very strong, tough and rigid. It's used for gears, fasteners, knife handles and ball bearings.

ABS

This is an opaque engineering plastic. It's tough, has good dimensional stability, high impact strength and resists scratching, breaking and tearing. It has a low melting temperature and is relatively inexpensive. You will find ABS in sporting goods, recreational items and electronic switches.

Polypropylene

This thermoplastic is found in the food storage and packaging industry as it does not chemically mix with food products. Polypropylene can be washed in hot water without degrading, and it has high chemical and moisture resistance. It has incredible impact strength, elasticity and toughness.

TPE

This thermoplastic elastomer combines the properties of thermoforming resins and rubbers. It can be molded in different Shore hardnesses to make soft-touch buttons and grippable overmoldings, especially on PP. It is often found in toys, sporting goods and kitchen utensils.





TPU

Thermoplastic polyurethane, because of its soft and elastic nature and great tensile and tear strength, is often used for parts that demand a rubber-like elasticity. TPU performs well at high temperatures and is commonly used in power tools, cable insulation and sporting goods.

TPR

A combination of plastic and rubber with attributes similar to both that's easy to mold. It has outstanding chemical and weather resistance and high impact strength. It's used for medical catheters, suspension bushings and headphone cables.

TPV

Another type of elastomer, this class of resins is even more like fully vulcanized rubber but which can be processed like an injection molding resin. Both RoHS and REACH compliant for environmental safety, it's used for rubber-like grommets, seals, clips and gaskets.

PEI

A semi-transparent to amber colored plastic similar in quality to high-strength PEEK engineering plastic. It has good impact strength and excellent high-temperature resistance and flame retardancy. Can be used in steam autoclaves for medical equipment and takes post-machining well.





Chapter Two

Surface Finishing for Mold Tools

Making a surface texture on a finished part requires careful preparation of the mold tool core and cavity. These textures can in turn affect other aspects of mold tool design.

For example, rough textures require larger draft angles to allow the part to be released from the mold. Rough surfaces also alter the heat transfer properties during production and can therefore impact cycle times. In contrast, glossy and smooth surfaces can also cause a part to stick in the mold and may therefore require more ejector pins in strategic locations.

We follow the finishing standards created by the Plastics Industry Association (formerly SPI). These are the standards we recommend you adhere to when quantifying a surface finish on your plastic parts. This information is usually recorded on your 2D drawings.

Below are the standard SPI surface finish categories and how they are typically applied.



Surface Finish	Guide
SPI-A1	Grade #3 Diamond
SPI-A2	Grade #6 Diamond
SPI-A3	Grade #15 Diamond
SPI-B1	600 Grit paper
SPI-B2	400 Grit paper
SPI-B3	320 Grit paper
SPI-C1	600 Stone
SPI-C2	400 Stone
SPI-C3	320 Stone
SPI-D1	Dry Blast Glass Bead
SPI-D2	Dry Blast # 240 Oxide
SPI-D3	Dry Blast # 24 Oxide



Hand Polishing for the mold

Finishes Created by Polishing

Category A is made using diamond buffing paste. This is the highest grade. A rotary tool polishes the surface in a random, non-linear pattern, scattering or reflecting light without indicating a clear texture. There's no direction to the waviness of the surface, so it looks the same no matter the viewing angle.

There are three levels in the A category, representing different degrees of polish, and within these levels an Ra value is specified for each. As an example, A3 corresponds to Ra 2~3. Surface flatness is a subject unto itself, but this essentially means it's very flat.

Finishes Made By Sandpaper

A category B finish is made with sandpaper. It's applied in a back-and-forth motion so it leaves behind characteristic linear scratches producing a very obvious pattern. It's possible to have an SPI surface finish for B1 which also measures a flatness of Ra 2~3. But this finish doesn't look the same as A3, nor would the part made from such a tool.

What is The Plastics Industry Association?

The PIA is an independent trade organization, educational resource and regulatory body based in the United States. They've created standards for surface finishes on injection mold tooling. These are used for measuring the typical kinds of finishes that can be applied to mold tools and hence they affect the resulting parts made from those tools.

What are the Surface Finish Categories?

In the PIA classification system there are four major grades: A, B, C and D. Group A finishes are made via diamond buffing, B with grit sandpaper, C with grit sanding stones, and D with pressure blasting using glass beads or aluminum oxide grains.

How Do Different Finishes Affect Product Cost?

The higher you go in the classification system, the more time and effort it takes to produce the finish. In fact, each step of refinement requires first completing the step preceding it. There are no shortcuts here so the costs are additive.



A B-grade finish is considered semi-gloss. It's fine enough to hide mold defects, tooling and machining marks while being less expensive to produce than A-grade. These finishes are applied with sanding grit ranging from 600 grit for B1, 400 for B2 and 320 grit for B3.

Finishes Made with Grit Stone

The C-grade finishes are made with grit sanding stones. As with B-grade finishes the grits of the stones also range from 600 to 320, though they leave behind a surface that is rougher and less flat because the stones are more aggressive in their cutting action. Such a process is used to quickly erase tooling and mold marks and will leave behind a matte finish on subsequent plastic parts.

Finishes Made By Blasting

Rougher finishes are made by abrasive blasting with sand or glass beads. These correspond to PIA class D. Because of the random nature of the spraying, the resulting finish is uniform and non-directional. This classification is used to produce dull or flat finishes.

It is possible to use surface flatness gauges to measure finishes up to the B4 grade. Beyond that, direct physical measurements for the purpose of quality assurance are not advised since these surfaces are rough enough to damage the very sensitive stylus used by the testing equipment.

Finishes Made via EDM

In addition to these, there are similar results that can be produced using electronic discharge machining (EDM) or spark erosion.

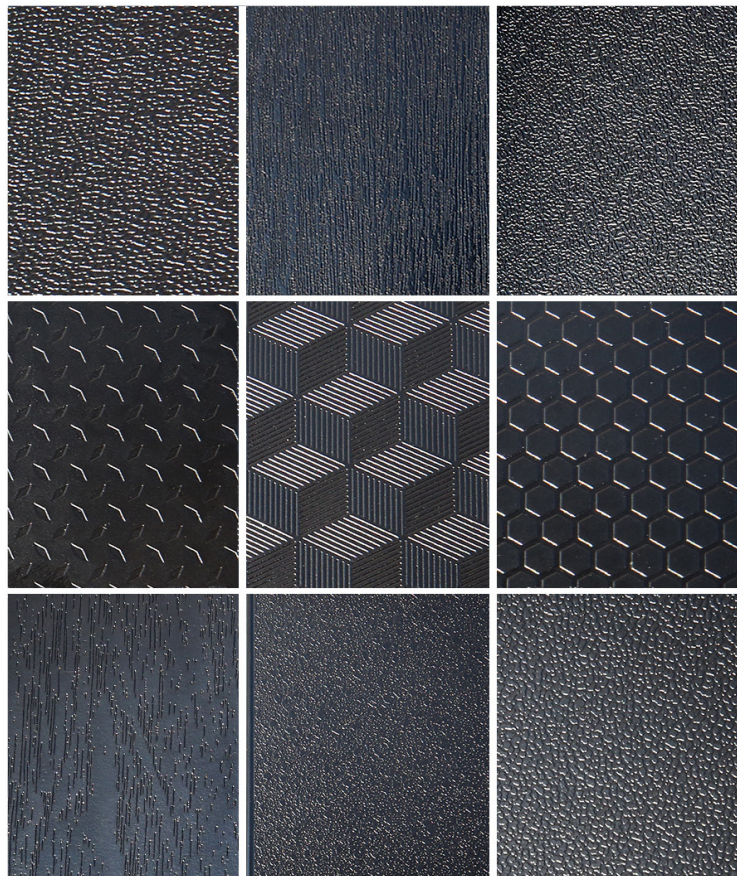
EDM finishes are most often expressed using the VDI system, with each number also corresponding to an Ra value. This system is the standard of the Society of German Engineers but is commonly referenced worldwide.

The chart below shows VDI-Ra comparisons along with recommended draft angles for common materials.



Engraved Finishes

Several companies offer proprietary surface texturing services using chemical texturing and engraving. Mold-Tech© is one such well-known example. They have a standard catalogue of stock finishes that simulate stone, leather, wood grain and more. Custom patterns are also available upon request. Such surface treatments should only be applied by licensed dealers.





Understanding Gloss

Surface finishes can look different to the naked eye even if the measured Ra values from one sample to another are similar. To further specify the look you want on a finished part you will need to specify the gloss, which is defined as the reflectance of a surface compared to a polished black glass standard.

A gloss meter is used to bounce a light off the surface of a sample, at different angles depending on the Ra value being tested. The measurement values are called gloss units or GU. The black glass standard is rated as 100, so measured gloss units from a target part are compared to this reading.

From these gloss readings it's then possible to describe a surface as high-gloss, semi-gloss, matte or flat. Taken together with a PIA mold surface specification and/or a Mold-Tech© pattern number, this is the information that manufacturers need to create a surface treatment on mold tools.

Because plastic is easy to colorize, almost any cosmetic affect can be achieved but you must first know exactly what color you want.



Measuring Gloss Percentage



Chapter Three

Coloring for Plastic Resin



A spectrophotometer is used to test color samples and to qualify production parts against a standard. Product developers should understand how this is done so that you can clearly communicate your color preferences with your supplier using standard terminology.



What Is A Spectrophotometer?

A spectrophotometer is used to measure the reflected wavelength of energy from a target source and compare this to a known standard. Each wavelength represents a unique color in the LCH color space. LCH means light, chroma and hue. These are defined as numerical values, so it's very useful when communicating color information objectively.

The spectrophotometer has a powerful light source with a known energy level. In operation, the unit shines this light off the target and then reads the wavelength reflected from the surface. This reading is compared to a numerical value supplied by you (a Pantone® or RAL number, for example) or derived from your physical sample.

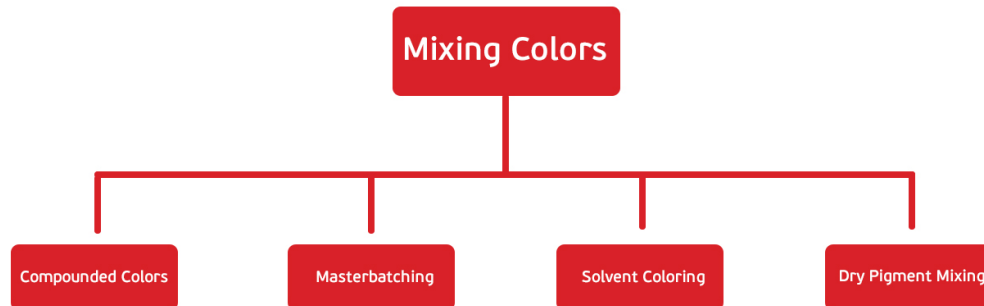
Because color is very subjective and can differ according to the light source, it's best to use an unambiguous and quantitative method to specify the color that you want with your supplier. Once this reference color has been established, the next step is to mix the correct color pigments or dyes with the raw plastic resin to make a color match for your parts.





Mixing Colors

There are four ways that color is mixed in a production environment, and you should know about them because each method has an effect on product cost and production volume.



Compounded Colors

The first method is called “compounded colors” or sometimes “pre-mixed”. Compounded colors are made by the resin supplier, where pigment and raw material have already been blended together to produce colored plastic pellets that are now ready to mold as-is.

For well-known Pantone® or RAL colors this is probably the cheapest and easiest way to go. Manufacturers can obtain common colors quickly and they may even have your color in stock. If you want a custom color, this too can be compounded by a resin supplier but they will usually require a minimum order of one ton, so it’s not cost-effective to make a custom color for a small batch of finished parts.

Masterbatching

Masterbatch colors come directly from the resin manufacturer. These are heavily pigmented pellets that are meant to be diluted by volume at a 2% ratio of colored pellets to uncolored resin. Because of this ratio, a minimum order of one ton of masterbatch – which is required by most suppliers – represents 20 tons of



moldable plastic. Thus, custom colors requiring masterbatch pigments only make financial sense when producing large volumes of finished parts.

Solvent Coloring

Here a pigmented oil is mixed with a solvent. This solvent is then sprayed over uncolored pellets, thoroughly coating them just prior to injection. Using this method it's difficult to precisely control the ratio of color to plastic so exact color matching is impractical.

The advantage to solvent coloring is that there is no minimum order for the pigmented oil, so it's a way to save money on smaller volumes if an exact match is not required.

Dry Pigment Mixing

The last method is similar, but uses dry pigment powder directly mixed with plastic pellets in a tumbler, again with a ratio of 2% pigment by volume.

For many plastic resins the pellets must be dried first before being transferred to a tumbler for mixing. This mixing in the tumbler is also an approximation – most of the dry pigment will stick to the plastic pellets (which is what you want) but some will also stick to the walls of the tumbler and related plumbing, thereby diluting the concentration of color.



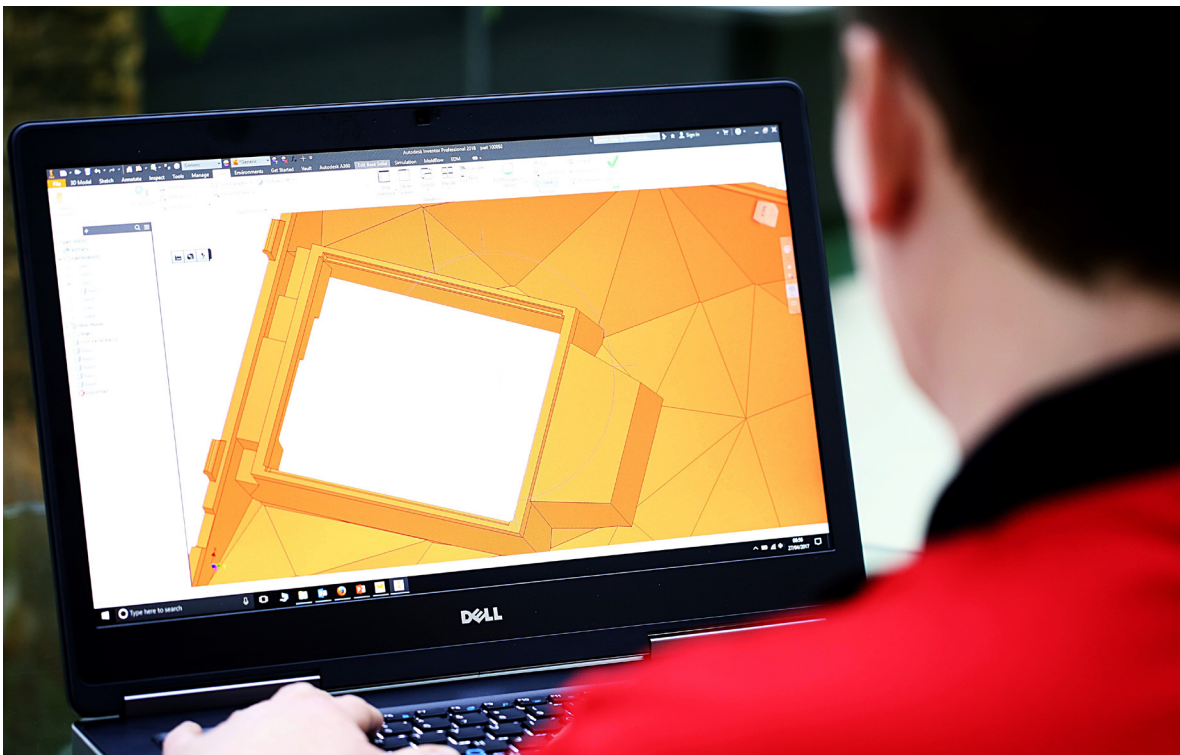
All told, this method is expedient if there is a small volume of a custom color – too small to have it compounded for you. But the price of expediency is that there are too many variables to provide assurance you'll get exactly the color you want.

If exact color consistency from one order to another is a priority for you on a project, it is highly advised that you purchase and mix all colored resin at one time. This ensures there is no variation in color between subsequent production runs.



Chapter Four

Preparing Design Drawings



Modern computers and CAD software make both 2D plane views and 3D solid models easy to create. These two types of engineering drawings, 2D and 3D, serve different but related purposes, so here are some tips to keep in mind when preparing your design drawings for manufacturing.



What Is A 3D Drawing?

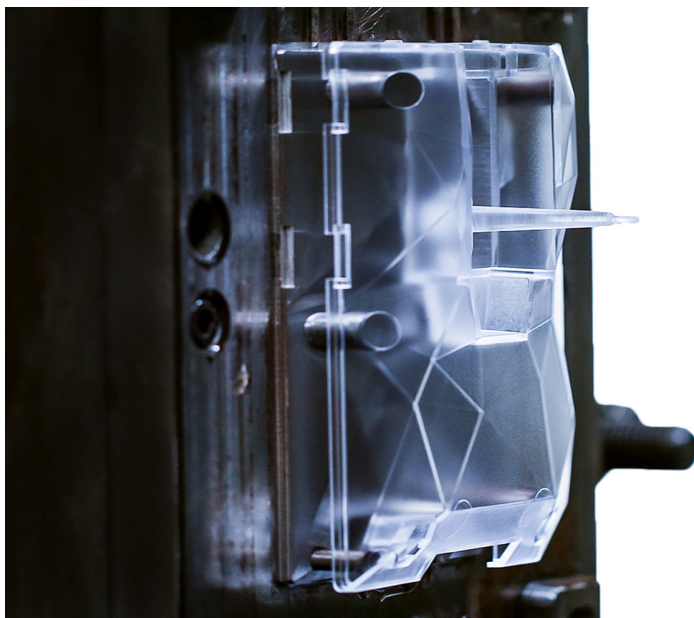
3D CAD software is used to make a model of your design. That model contains all of the dimensional data for the part. What you see on the screen may seem like a solid object, something you can rotate and look at from any angle. Remember that it's a mathematical model, composed purely of data.

What Is The 3D Model For?

First, it helps the supplier get a sense of the part's geometry and how it might fit together with some other assembly. This can be very important both in how to optimize making it and how to measure it.

Secondly, a solid model helps to determine the volume of the part. From this is derived the size of the "shot" that needs to be used for each plastic injection cycle, and that in turn can effect other variables such as determining the capacity of the injection molding machine needed to make it.

The third main function is that the 3D data is used to generate the CNC program for milling and cutting the tool. The 3D model is translated into G-code, and G-code tells the CNC machine what path it will take to cut the shape out of a block of metal.





What's A 2D Drawing?

2D or plane views are derived directly with the click of a button from a 3D model.

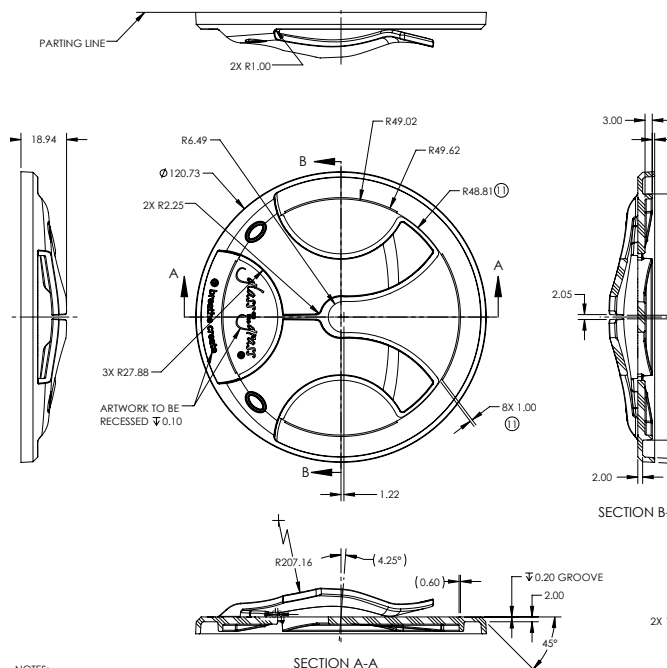
The 2D file shows something like a snapshot of the part, from one perspective or one angle at a time.

What Is A 2D Drawing For?

Unlike a 3D solid model, the 2D drawing does not automatically contain any data. You, the designer, need to specify everything about the part that you determine is important for its look, feel and function.

2D drawings are most often used to determine surface finish treatments and how the part is to be measured. Remember, for CNC machining the 3D file is used to make the part, and the 2D is used to measure it, since only you know which dimensions and features are most important for your application.

Both 3D and 2D drawing files should be supplied with every project. This gives the most complete picture of the total part, which is helpful for generating an accurate quotation and also identifying any areas which might need to be modified for manufacturability.



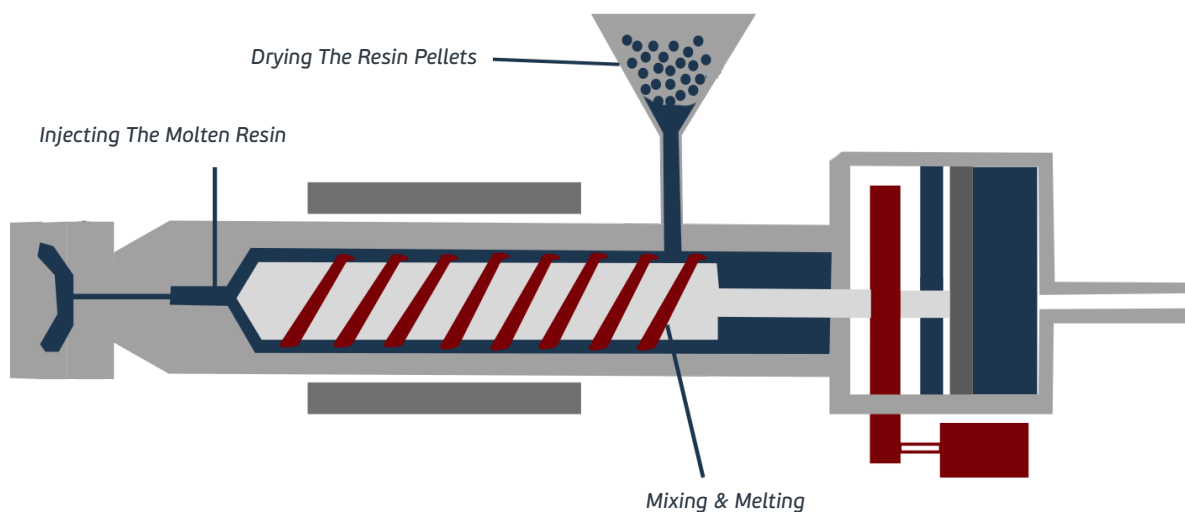
Drawing file should include, at minimum, the following:

- » Physical dimensions and tolerances
- » Hole sizes, locations, and type of threads if any
- » Surface Flatness
- » Surface finish and color
- » Material type
- » Corner radii
- » Critical dimensions



Chapter Five

Understanding the Molding Process



Successful plastic injection molding relies on several discrete steps that work together: drying, mixing, melting, injection, packing, cooling and ejection. In Chapter Eight we will discuss the specific machine components, but here is the process flow for most jobs using standard injection molding techniques.



Drying

Plastic resin pellets must be dried according to the manufacturer's specification, and kept dry during the entire production process. Hoppers and driers should be sealed to prevent contamination by moisture or debris.



Drier

Melting

The hopper feeds the pellets into the barrel of the machine, which is surrounded by electric heating bands that begin to warm the resin. Inside the barrel, a reciprocating screw has flutes that pull the material forward towards the injection nozzle. These flutes shear the plastic pellets, heating and mixing them thoroughly until the resin is a uniform liquid.



Hopper

Injection

This liquid resin is now ready to be injected into the cavity of the mold. The two halves of the mold are held together by powerful clamps to withstand the necessary injection pressure. The pressure of injection is determined by the viscosity of the resin, the size of the cavity and the architecture of the gate and runner system.



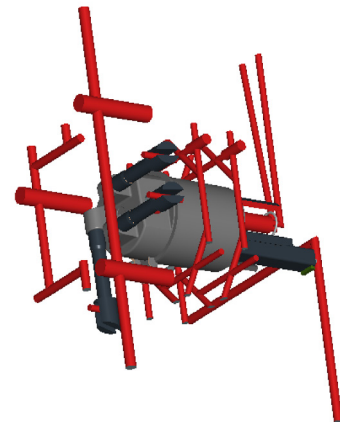
Packing

Once the cavity has been filled with a pre-determined shot of resin, packing pressure is maintained for an additional period of time to ensure the part achieves a uniform maximum density. The resin will be forced into contact with the exterior walls of the mold tool where it begins to cool. The packing stage is useful only to the point where the gate has frozen off, after which no more material can enter the cavity.

Cooling

The temperature of the tool walls, and hence of the resin, is moderated by a liquid flowing through a series of channels that surround the core and cavity.

This liquid distributes heat uniformly around the part and then lowers the temperature in a controlled fashion to avoid warping and other defects.

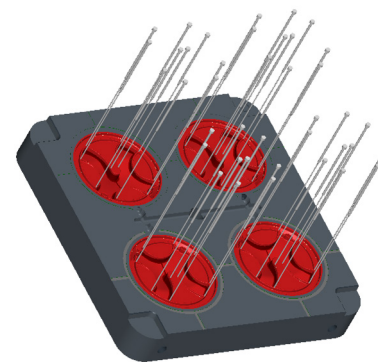


Cooling Channels (in red)

Ejection

When finished the molding machine opens automatically. Usually the part will be on the core or B-side of the tool. Ejector pins are used to push the part free of the mold. The die can then be clamped shut again and the process repeated for the next part.

This is a typical scenario. If anything goes wrong in any of these steps it can result in a variety of potential part defects. The secret to preventing such defects is by controlling the effects of heat and employing sound design fundamentals.



Ejector Pins



Chapter Six

Common Defects and Design Optimization



Product Defects and the Importance of Good Design

To improve the results of your design, it's important to understand the role of heat and how liquid resin behaves during the molding and curing process.

Shrinkage

All plastic resins exhibit differing amounts of thermal expansion and shrinkage which must be accounted for in the design of the part and the mold tool. This deviation in dimension or tolerance away from the nominal value of the design will vary according to the type of feature: hole, cutout, flat surface, thread, etc.



To minimize the negative effects of such shrinkage, good engineering practices have been standardized across the industry. They include:

- » Keep wall thicknesses to a minimum for your application.
- » Keep adjacent wall surfaces equal. When this is not possible, create a transition from one wall to another using a gradient of 3:1.
- » When possible, core out thick sections of walls or other features to reduce thermal mass while maintaining the nominal wall thickness.
- » Orient thicker wall sections closer to gates and runners, so that these areas fill first before thinner sections.
- » Plastic in contact with the tool wall will cool first and shrink, so areas touching the tool wall should be as uniform in thickness as possible.

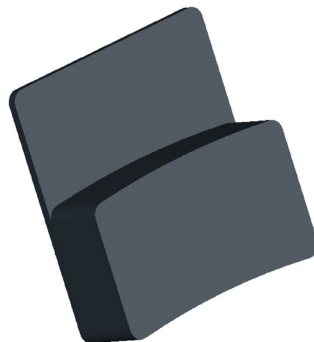
Sink Marks

Areas in your design with higher thermal mass will cool more quickly, and as they cool they shrink. This non-uniform rate of shrinkage creates the characteristic dimple or crater of a sink mark. This is the direct result of the thermal stresses that happen inside the mold.

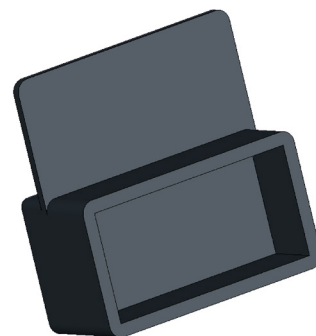
To control sink marks it is necessary to control shrinkage in a uniform fashion. In many cases sink marks can be minimized somewhat through design optimization.



Original Design



Potential Sink Mark



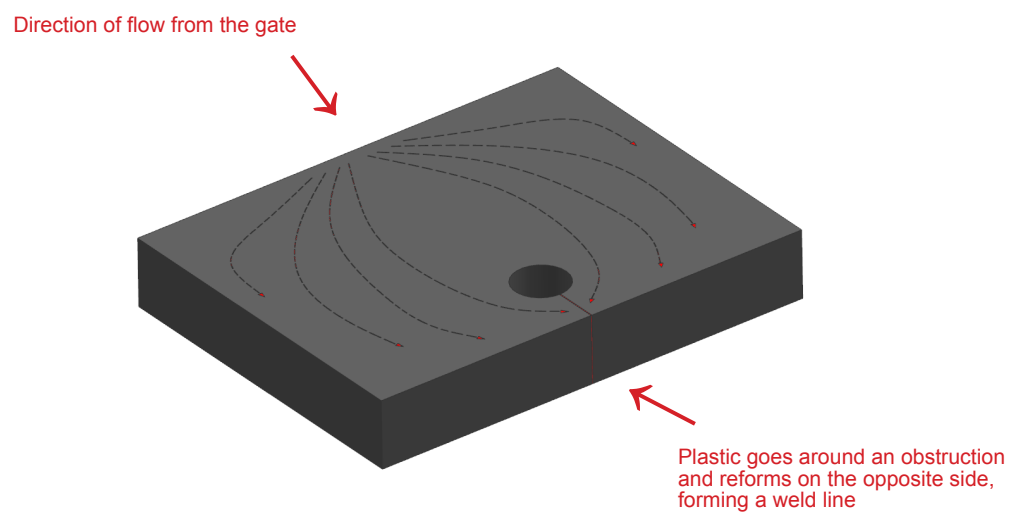
Core Out Thick Sections
to Reduce Sink Mark



Weld Lines

A weld line is created where two separate channels of resin flow around an obstruction, meet and then solidify. This partial solidification at the point of contact makes a characteristic mark on the finished part surface.

Weld lines can vary greatly, from nearly invisible to the naked eye to highly visible areas representing weak part formation.



Weld lines may be unavoidable in many cases but there are some ways to optimize the results using the following best practices:

- » Create more cooling circuits at the area where weld lines form.
- » Camouflage weld lines within the design of the part.
- » Move gate locations.
- » Rougher surface textures hide weld lines better than high-gloss surfaces.
- » Lighter colors hide weld lines better than darker ones.

It's also possible to use substitute plastic resins to achieve better cooling results.

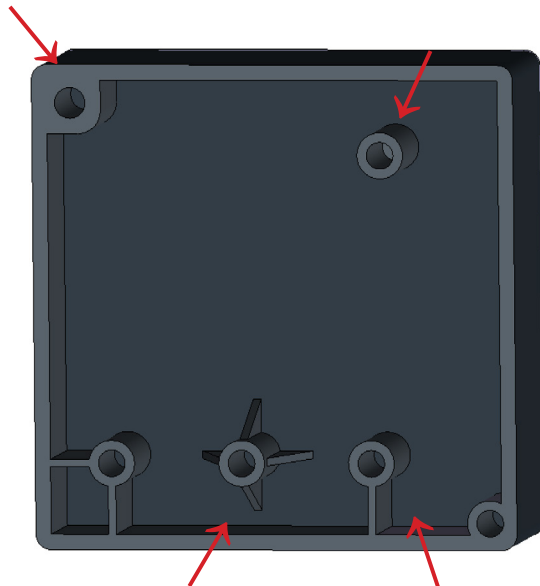


Bosses

A boss is a stressed point of engagement, shaped like a post or pillar. It's usually the place where a fastener like a screw will connect one piece to another.

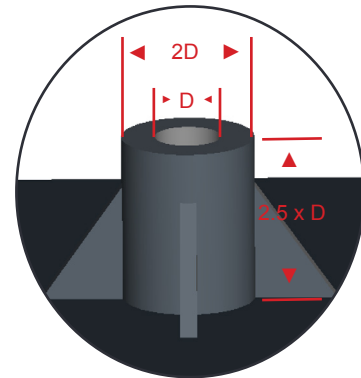
The walls are too thick

This boss is unsupported



Correct design of boss with gusset

Correct design of boss with rib



Bosses represent significant additional thermal mass in an injection molding so good design practices are needed to prevent sink marks and other defects when using these features.

- » The boss should be radiused at the base, with a radius between 25-50% of the wall thickness.
- » Add a fillet to strengthen the boss without adding mass.
- » Anchor a boss to an adjacent wall with a rib.
- » The boss should be no taller than 2.5X the outside diameter.
- » A minimum draft angle on the outside should be $1/2^\circ$. A smaller draft is possible on the inside, if the tool and core have been well-polished.
- » Two or more bosses should be located no nearer than twice the adjacent wall thickness.

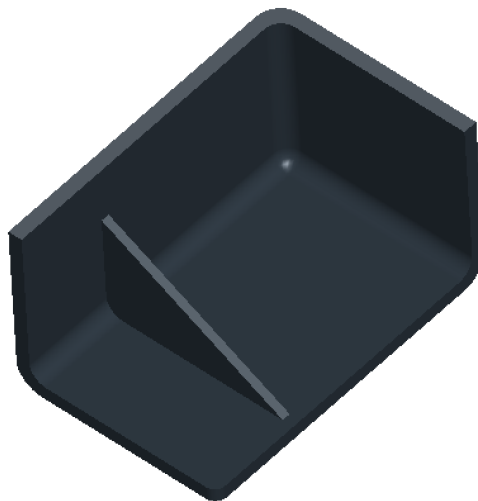


Ribs and Gussets

Ribs are supports that stiffen wall sections without adding much additional mass. They do have an effect on distribution of thermal stress so good design rules should be employed with their use.

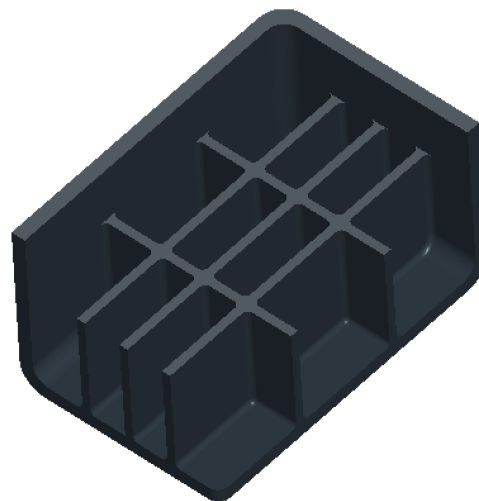
- » Where a rib meets a wall, the rib should be no thicker than 50% of the wall thickness.
- » The base of the rib where it meets the wall should be radiused, between 25-50% of wall thickness.
- » Rib height should be no more than 3X wall thickness.
- » Draft angles on ribs should be increased to allow for easier release from adjacent walls.

Gussets are similar to ribs but are smaller and are used for strengthening the base of a feature. The same design rules apply for gussets as for ribs.



Support walls with gussets and radiused corners

Reinforce high-stress areas with multiple ribs

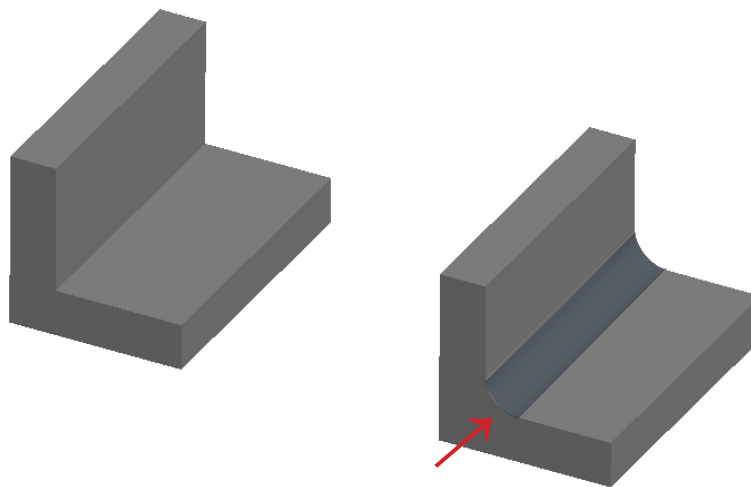




Corner Radii

Sharp corners, both inside and outside, should be avoided because they concentrate too much stress at the joint. Radiused corners are stronger and are easier to fill. You should design corner radii according to the following guidelines:

- » The inside radius should be at least 50% of the wall thickness.
- » The outside radius of a wall section should equal the inside radius + wall thickness.



This corner radius is much stronger and easier to mold

Draft Angles

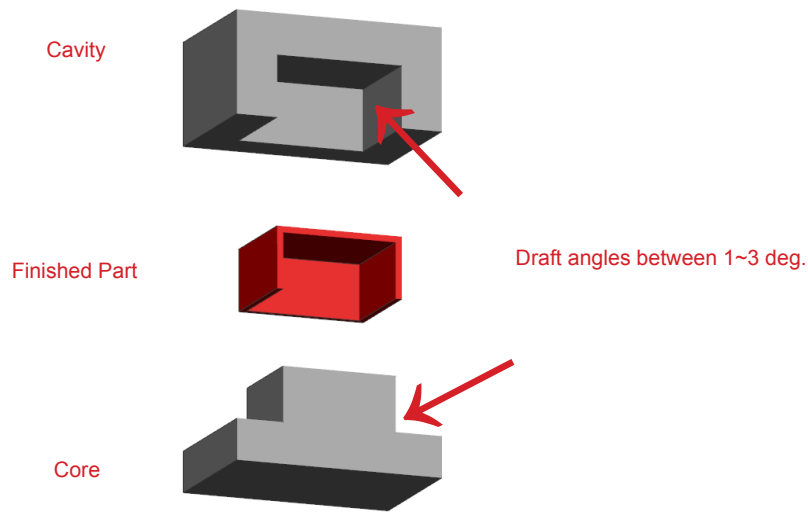
Draft angles create a separation between the face of a part feature and the adjacent wall of the mold. Draft angles are calculated in the line of draw, or the direction in which the tool opens. This also applies to the direction in which a slider moves into and out of a part feature.

There is no one recommended draft angle for all applications but there are some general guidelines:

- » 1~2° is the preferred minimum in most cases.
- » Increase draft angles for rough surface textures.



- » Larger angles make part removal easier but may compromise some design features. Larger drafts can lead to features becoming unnecessarily large or small over the length/height of the feature.
- » Draft angles also depend on part geometry, wall thickness and material type. Zero draft is possible on some features if using a perpendicular slider.



Ejector Pin Position

Ejector pins are metal rods placed at strategic locations inside a mold tool. When an injection cycle is done and the tool opens, the pins protrude from the mold and help to push the now completed part free of the tool.

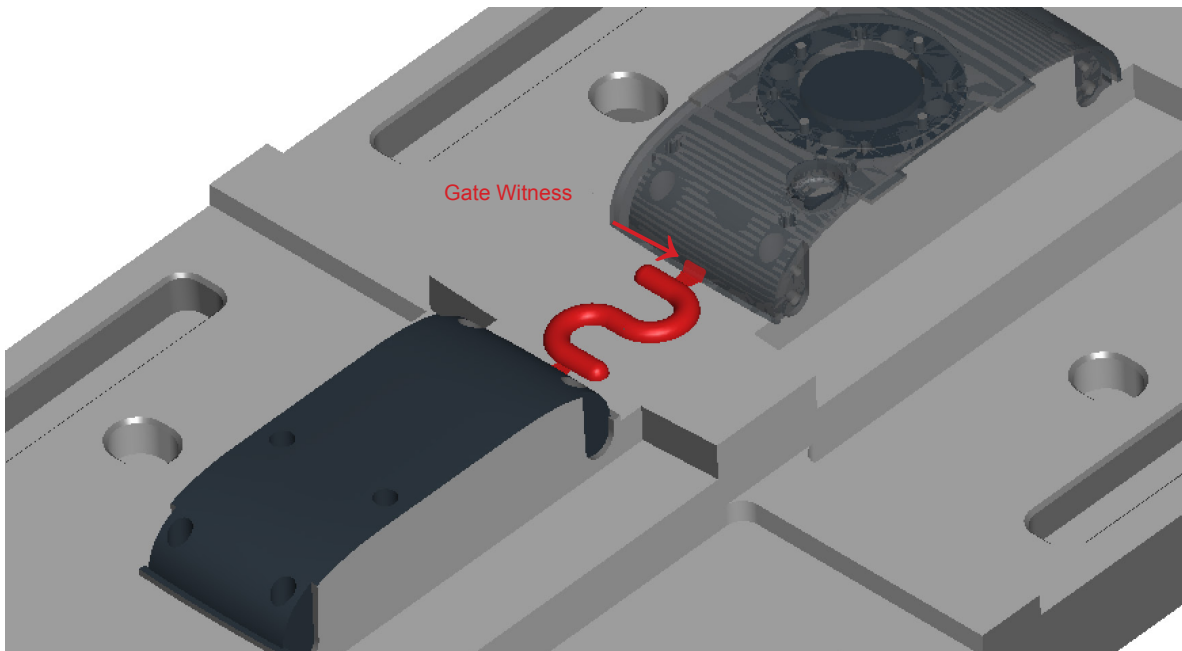
They're a necessary part of mold tool design but they will leave a corresponding shallow, circular mark on the soft plastic as it's molded. To optimize pin marks:

- » Design pin locations so the mark is unobtrusive.
- » Incorporate the ejector pin mark into the part design if possible.
- » For thin-walled areas or narrow ledges it is preferable to use a stripper plate rather than an ejector pin to distribute the pulling force over a larger surface area.
- » Ejection of parts with core pins can be accomplished using sleeve ejectors.
- » More pins are needed around the gate of the tool.



Gate Witness

The gate is where the liquid resin is injected into a mold tool cavity from the runner or feed system. This area leaves a mark on the finished part. It's always best practice to locate this mark in a place where it won't show or can be easily removed.



Please note the following points for good gate design strategy:

- » Gates should be located so they fill thicker sections before thinner ones.
- » When using multiple gates, arrange them to distribute material evenly to allow for balanced filling of the cavity.
- » Different gate and runner systems can increase mold tooling complexity and therefore cost.
- » Different gate styles will leave different witness marks. Edge gates need to be trimmed, leaving a visible scar. Other gate types, such as sub gates (submarine gates) are removed automatically leaving only a circular or oval scar.



Holes And Depressions

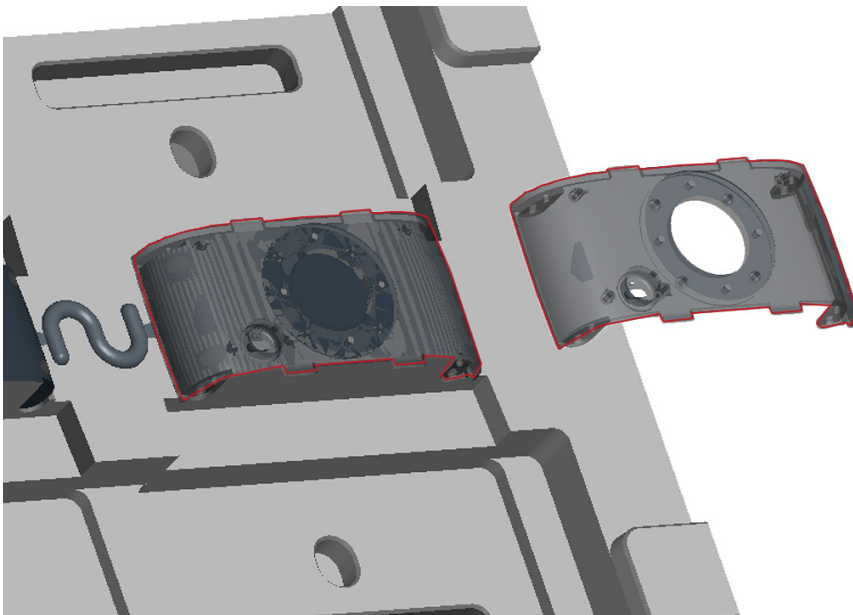
Holes are created with the help of core pins, which not only define the size and shape of the hole but also provide cooling.

Under high injection pressures and temperatures holes can become deformed, so designers should note the following rules:

- » For blind holes, depth should be no more than 2X the hole diameter. This ratio can increase to 3:1 for holes larger than 5mm in diameter.
- » For through-holes, the length-to-width ratio can increase to 4. For holes above 5mm in diameter, the maximum ratio is 6.
- » The distance between adjacent holes, or a hole and the part edge, should be 2X the wall thickness or 2X the hole diameter, whichever is greater.

Parting Line or Split Line

This is the line where the two halves of a mold tool separate to release the part from a single large cavity. A corresponding mark may be left behind on the part which is often unavoidable.





Here are some tips for good design:

- » Incorporate the parting line into the part design.
- » If there are recessed holes near the parting line, exploit the opening action of the tool to create the hole.
- » Deep ribs, bosses and other standout features may require “knockouts”, or flat spots that can be used for ejector pins to push the part free.
- » Split line location is most often determined by the draft angle designed into the part.

Texture

Heavily textured molds may grip the surface of the corresponding part as it cools. This can cause scratching or even part breakage when trying to remove the part from the mold.

Thus it is good design practice to:

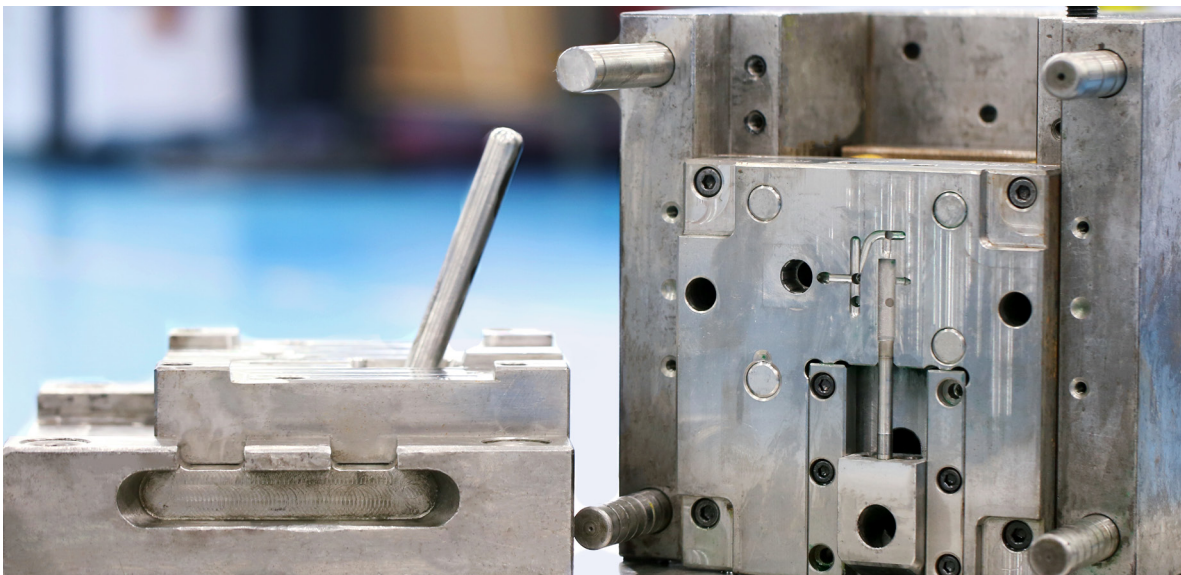
- » Increase the draft angle with the increased depth of texture.
- » Taper the texture at a wall's edge to provide additional release.





Chapter Seven

Gates and Runners



The runner and gate system directs resin from the nozzle of the molding machine into the cavity of the tool. The gate architecture affects cycle times, tooling costs and lead times, the location of witness marks and many other potential defects that are common on molded parts.

For these reasons, designing the proper type of gate and runner is important and should be planned carefully. Here are some questions to consider.



Where is the Best Gate Location?

These considerations affect gate location:

- » A gate placed near an obstruction causes the resin to flow around the pin and make a weld mark. However, the resin at this point is in its most liquid state, thus creating the optimal weld conditions.
- » Gates should be placed closest to thick walled areas to ensure complete packing.
- » Improper gate location can cause part distortion if the build volume of the entire part is filled unevenly.
- » Gates are areas of high stress. Try to locate gates away from places that will be affected by stress marks or plastic degradation.
- » Gates need to be located in areas that will be easy to de-gate, either manually or automatically. Note that some plastics are susceptible to high sheer forces and therefore can only be de-gated manually.
- » Thin walled parts might need flow channels or additional gates to provide a sufficient volume of plastic in a minimum cycle time.
- » Moldflow or other simulation software can help determine ideal gate locations and allow you to simulate effects such as weld lines and sink marks.

What Potential Defects Can Be Caused Via Gates?

All gates force molten plastic under pressure into the cavity which accelerates and heats the plastic as it does so. This creates a variety of effects which may need a change to gate design strategy.

These are some of the defects caused by gates and gate location:

Jetting

Jetting happens when molten resin sprays from the gate opening instead of flowing smoothly. This can be reduced by lowering the injection pressure, enlarging the gate or angling the gate against an adjacent wall.

Overheating



High injection speed through the gate produces heat due to friction. Too much heat at this point may cause the resin to decay by destroying molecular bonds.

Slowing the injection speed can reduce overheating. However, there is a risk of creating weaker weld lines and reducing the number of parts per hour.

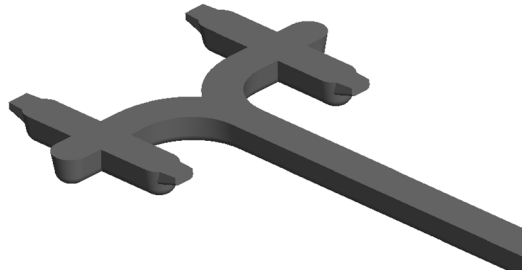
It may be also be possible to reduce overheating by dividing the total pressure over more gates, but that increases complexity and cost.

Which Gate Type is Best For Your Project?

There are different sizes and shapes of gates to consider, and each has trade-offs. Here are the most common ones:

Edge Gates

Edge gates are the most common. They are machined into the mold at the parting line and fill the cavity from the side of the part.



Advantages

- Inexpensive to design and manufacture.
- Easy to enlarge if necessary without removing the tool from the machine.

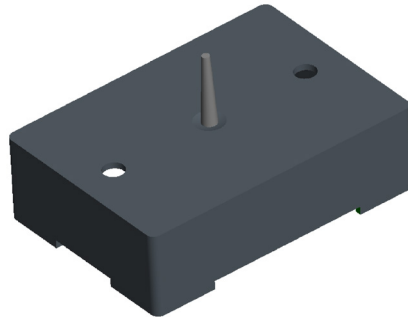
Disadvantages

- Location of the witness mark may not be ideal.
- Plastic flowing from a single gate will create noticeable weld lines as it flows around obstructions in the mold.
- High injection pressures and velocity at the small gate opening can cause the plastic to degrade.



Direct or Sprue Gate

A gate built into the sprue directly where it enters the fixed or A-side of the tool.



Advantages

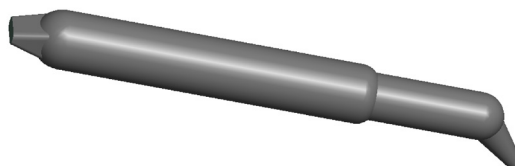
- Easiest type of gate to make.
- Able to inject large volumes of plastic quickly.
- Ideal for round or cylindrical parts where concentricity is important.
- Good for thick-walled, glass-filled parts with no cosmetic requirements.

Disadvantages

- Leaves behind a large witness mark with the potential for a sink mark on the opposite side.
- Must be de-gated manually.

Submarine Gate

The runner directs plastic to the edge of the cavity at the parting line, but then the gate drops below the parting line and tunnels upward to fill the part from below. This design is only possible on two-plate mold construction.





Advantages

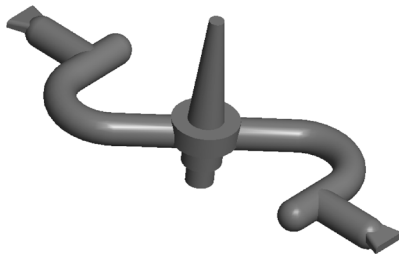
- Moves the witness mark to a more desirable location.
- Excess vent gasses can be pushed up and out of the mold from below.
- De-gates automatically as the mold opens or when the part is ejected.

Disadvantages

- More complex, expensive and time-consuming to manufacture.
- Not ideal for glass-filled materials as edge will wear faster.

Fan Gate

An alternative type of edge gate, the fan maintains a consistent thickness but spreads out to increase the volume over a larger area. Recommended for polycarbonate plastics.



Advantages

- Good for increasing the flow volume for thin-edged parts.
- Reduces injection pressure for a given volume.

Disadvantages

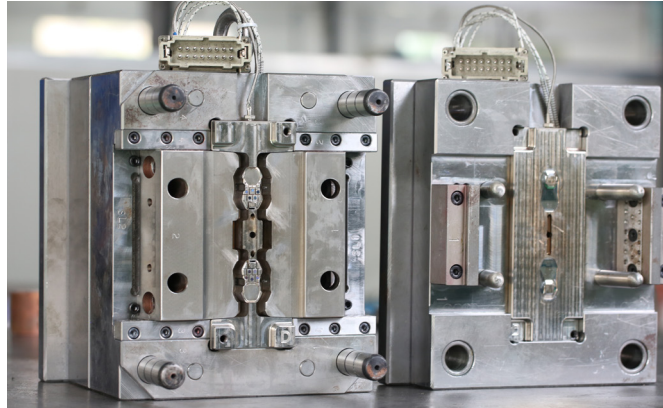
- May leave a large, tab-like witness mark that must be post-processed.



Hot Vs. Cold Runner Systems

Hot Runner

A hot gate/runner system uses an electrically heated cylinder to deliver pre-heated plastic into the mold cavity. The runner is built into the mold in the form of a manifold plate and a number of hot runner drops.



Advantages

- Precise temperature control for better finished appearance.
- Eliminates waste in multi-cavity tools.
- Decreases cycle times.

Disadvantages

- More complex and expensive.
- Color changes from one production part to another take longer to ensure adequate cleaning of the hot runner system.
- Once the gate has been placed it is very expensive to move its position.

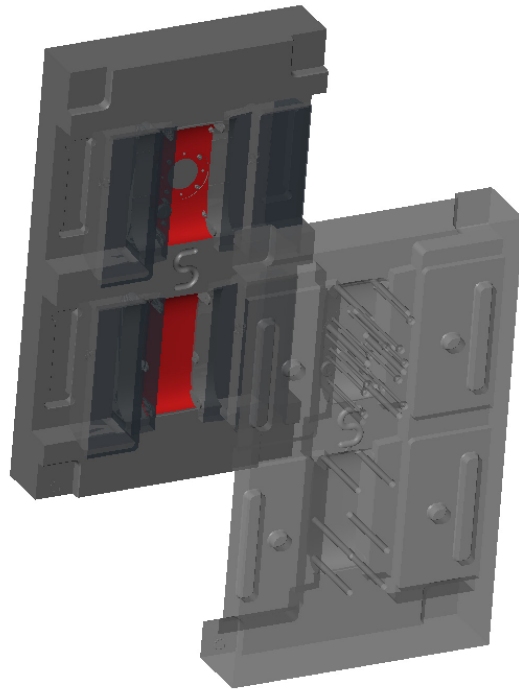
Cold Runners

Cold runner systems are less expensive than hot runners, but the length of the runner represents plastic that will be wasted for each cycle, which in turn increases costs.



Chapter Eight

Anatomy of a Plastic Injection Mold Tool



Plastic injection mold tools can vary from simple hollow cavities to multi-cavity tools with automatic and semi-automatic sub-systems composed of hundreds of discrete components.

All of these systems must work together in a controlled orchestration to make a quality finished part. All mold tools, however, share a few basic parts that product developers should understand so that they can communicate clearly and effectively with the mold maker.



Mold Base

A rectangular block of metal made to fit within the platens of the injection molding machine. The two halves of the block, the core and cavity, are held in alignment with guide rods and springs.

Core and Cavity

In most molding operations, the cavity side of the mold is called the A-side, and is held stationary against the unmoving platen of the injection molding machine. It is also this side that contains the gate and runner system, from which the molten resin is injected into the cavity when the mold is closed.

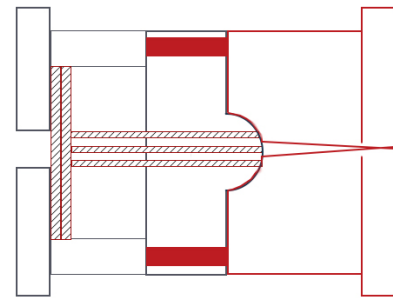
The B-side is the core, and is mounted to the moving platen which provides the clamping pressure. It is this side of the tool that normally contains the ejector pins to release the part from the core after molding.

Cooling Channels

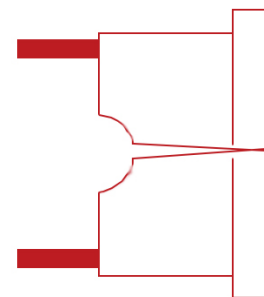
Cooling channels transport water or some other fluid in a circuit around the core and cavity of the mold tool.

The primary purpose is not to remove heat as quickly as possible. Rather, it's a temperature control system, ensuring that excess heat in the resin is removed in a controlled and predictable manner.

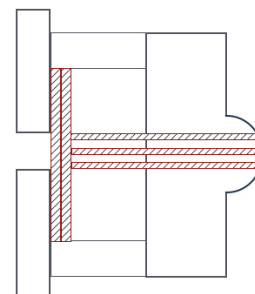
Since cooling makes up the largest percentage of overall cycle time, optimizing the cooling parameters has the most beneficial effect on improving total process efficiency.



Core & Cavity



A-Side



B-Side



The design of the cooling circuit is also critical for preventing many heat-related defects in plastic parts. In conventional mold tools, these cooling channels must follow the straight paths made by standard machining techniques. Introducing cross-drilled channels, plugs, baffles, tubes or bubblers create more complicated cooling paths that are necessary in some applications.

More complex, non-linear cooling geometries require additive manufacturing techniques like 3D metal printing. This offers an advantage by making cooling channels that conform to any surface geometry, thereby greatly enhancing cooling efficiency.



Conformal Cooling Channels made with 3D Printing

Ejector Pins and Stripper Plates

Ejector pins and stripper plates are mechanical aids that help to push, and in some cases pull, finished parts out a mold tool cavity.

Pins are typically mounted in the cavity or B-side of a mold tool. After a molding cycle is completed, the finished part remains on the core when the mold opens. Pins are used to push the part free of the core and then retract back into the tool for the next cycle.

Stripper plates help to pull a part free from the core by acting over a large surface area. They are normally used on thin-walled sections or narrow edges of parts where pins would be impractical.

The location, quantity and sizes of ejector pins should be optimized so the ejection forces are balanced to avoid stressing or deforming the part. Part designers should be mindful of this and help the toolmaker by specifying ejector *go* or *no-go* areas for pin location.

Runners and Gates

Runners and gates are channels made along the split line of the mold tool. They connect to the injection nozzle of the molding machine, and carry liquid resin into the cavity of the tool. Runners can be cut with various cross sections, of which



full-round, half-round and trapezoidal are the most common. Full-round is the most desirable cross section and is used in most cases.

Sliders and Handloads

Sliders are forming tools used to create various geometries on the finished part, usually perpendicular to the direction of the tool opening, or its *draw*. As the name suggests, they must slide out of the way once a part is molded, allowing a finished part to be released.

Sliders are often necessary because many complex shapes like holes, indentations, undercuts or protrusions must be made at angles other than the line of draw.

Handloads perform a similar function but they are separate components that are removed by hand and later replaced for each molding cycle. Handloads are typically used in place of sliders for low-volume molding of only a few hundred parts.

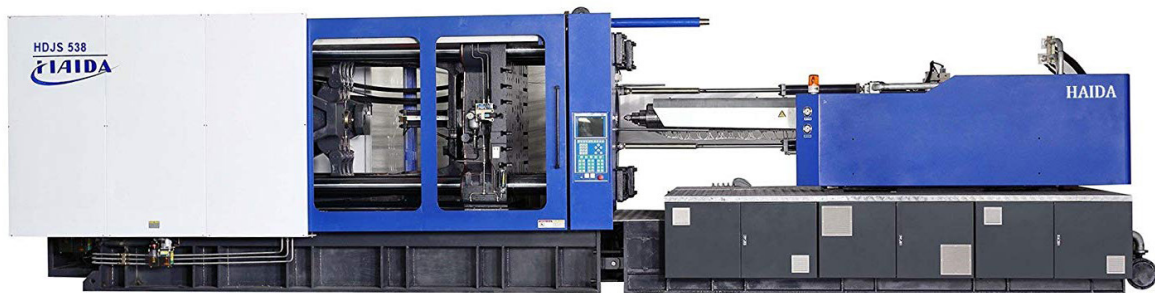


← Sliders move perpendicular to the line of draw →



Chapter Nine

The Plastic Injection Molding Machine



The function of a plastic injection molding machine plays a large role in determining the kinds of parts that can be made with it, and therefore influences the design of the mold tools. Machines may vary a little but here are the basics that every designer should know to help them optimize their mold designs for the best results.



Barrel and Heaters

The barrel guides the pellets as they're transferred from the hopper to the injection nozzle. The exterior of the barrel is surrounded by heating bands that help to melt the pellets and regulate their temperature. Inside the barrel is the reciprocating screw.

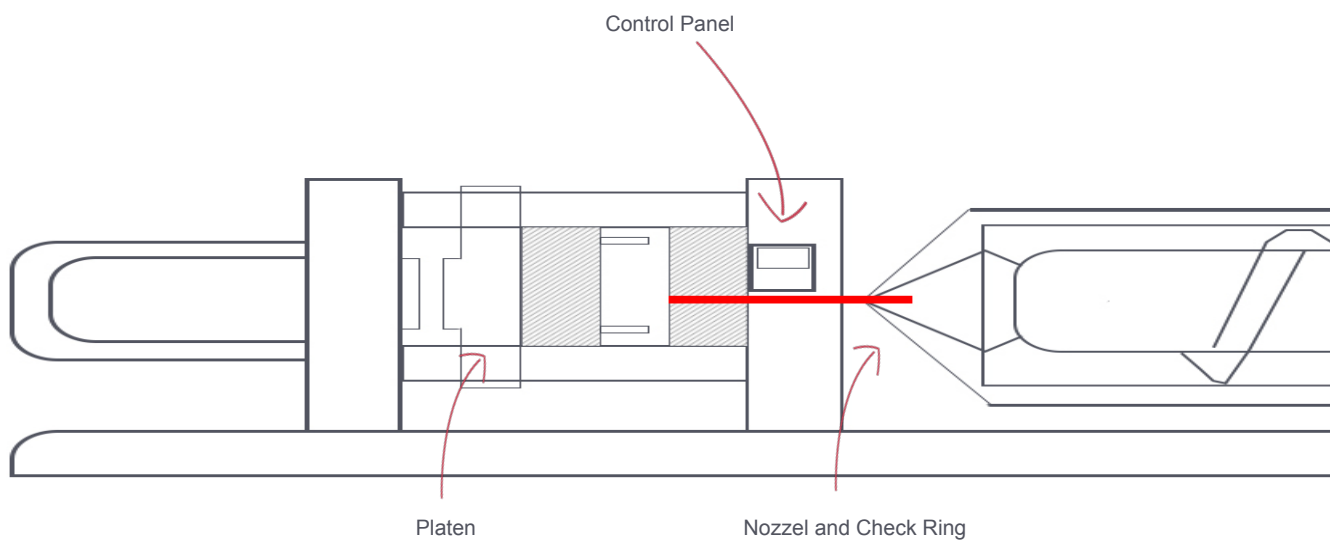
Reciprocating Screw

The reciprocating screw is shaped something like an augur, with spiral flutes along its length. As this screw rotates, these flutes pull the resin down the length of the barrel and towards the nozzle. The shearing action of the flutes helps to mix the resin and heat it at the same time.

The other purpose of the screw is to push forward against the gate of the tool, creating the packing pressure that helps fill the mold cavity to maximum density.

Nozzle and Check Ring

The nozzle is the aperture through which the plastic is pressurized into the mold. The molten plastic is injected into the sprues and runners of the mold, which is held inside the platens of the machine.





Just behind the nozzle is a check ring, which acts as a stop or non-return valve, preventing plastic from being forced back into the barrel. Eventually plastic at the tip of the nozzle will cool and solidify, “freezing off” and preventing further injection.

Platens

The platens hold the two halves of the mold tools. One platen is fixed, usually to the A-side of the mold, while the other, holding the B-side, is moved back and forth by hydraulics or motors and locked into place with a toggle.

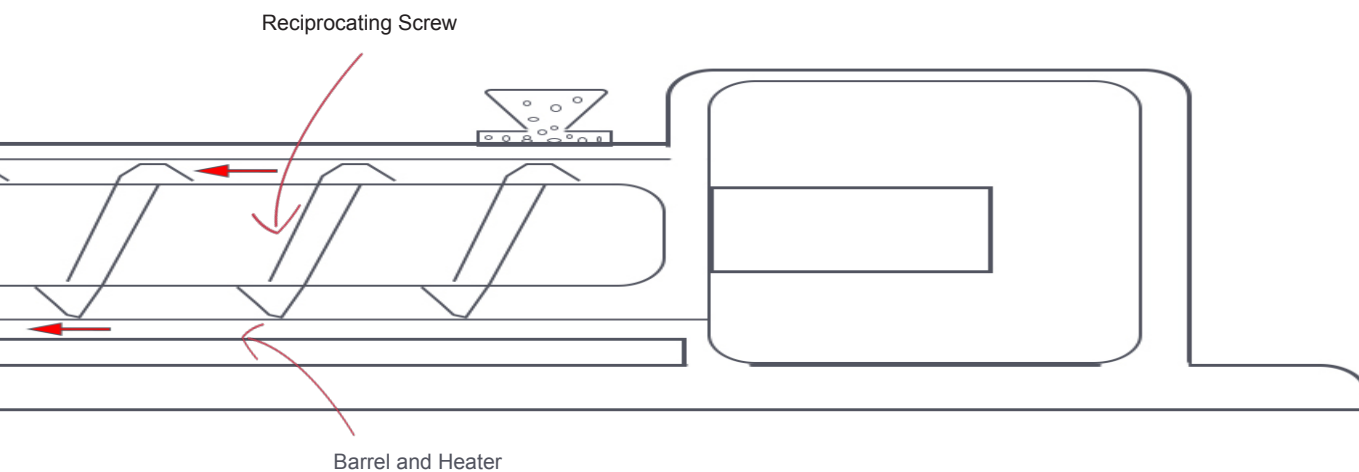
There are also hose fittings on both platens, through which coolant is fed into the molds.

Clamp

A hydraulic cylinder at the rear of the machine drives the clamp forward and locks it in place, holding the two mold halves tightly together until they can be completely filled. The clamp must be able to withstand tons of pressure as the mold is packed. Clamp holding pressure, time and other variables are adjusted by the operator at the control panel.

Control Panel

A skilled operator uses the functions of the control panel to adjust many injection molding parameters including barrel temperature, packing time and pressure, holding time, shot volume, etc.





Conclusion



We hope that you have enjoyed this primer on the essentials of plastic injection molding for product development. We believe that using this guide will help you when it comes time to choosing the right plastic resin and preparing your design drawings for the best results.

This information is necessarily very general and will not apply to all applications. Every job is unique, so you should prepare your data carefully in advance before talking with a manufacturing professional such as Star Rapid. Our engineers and material specialists can give you specific advice about design for manufacturing optimization and alternate material options to help you achieve superior results for rapid prototyping and low volume manufacturing. We look forward to hearing from you today.



Glossary of Terms

A	B	C	D	E	F	G	H	I
J	K	L	M	N	O	P	Q	R
S	T	U	V	W	X	Y	Z	

Term	Meaning
2K Molding	A type of injection molding in which two separate resins with unique properties are injected at the same time.
3D Printing	Also known as additive manufacturing, this refers to the production of a solid object by sequentially building up 2D layers of printing substrate.
A	
A-Side	The A-Side is often the cosmetic side of the part. This side of the part is usually formed by the cavity or A-plate of the mold tool.
Angle Pin	A strategically placed inclined pin, designed to retract a sliding core when the mold opens.
B	
B-Side	The B-side is often the inside or non-cosmetic side of a molded part. This side is usually formed by the core or B-plate of the mold tool.
Baffle	Flat or corkscrew shaped blades of metal inserted into the coolant channels of a mold tool to create a return coolant circuit within that channel.
Barrel (Molding Machine)	Long cylinder that houses the screw and check ring. This part of the molding machine takes the plastic pellets from the hopper and transports them to the machine nozzle, heating and melting the plastic into a continuous moldable consistency along the way.
Bosses	Projections or protrusions in cast or injected molded parts. They're intended for hole location, part alignment and strengthening, while also avoiding the buildup of thermal stresses in adjacent part walls.



Term	Meaning
Bubblers	Long hollow tubes that are fitted into drilled holes to supply coolant in narrow or hard to reach areas of a mold tool.
Bump Off	A shallow undercut on a molded part that is formed without lifters, sliders or handloads. The flexibility of the plastic allows it to release from the mold, preserving the feature.
C	
CAD	Computer Aided Design. The use of computer software to create 2D and 3D digital models of an object in preparation for manufacturing.
Cascade Gating	A type of sequential gating using hot runners, in which a series of gates are automatically activated in succession to fill a mold cavity.
Cavity	Depression in a die casting or plastic injection mold tool, usually on the fixed or non-moving side of the mold. Typically forms the outside or convex side of the part. The cavity is most often the cosmetic face or A surface of the part.
Check Ring	Fitted on the end of the screw inside the barrel of a plastic injection molding machine, the check ring acts as a non-return valve allowing correct holding pressures to be reached and maintained.
Clamping Force	The amount of pressure applied between the 2 platens of a molding machine. Suitable clamping force ensures that the mold is closed properly and held closed under high injection and packing pressures.
Cold Slug	Semi solidified plastic material formed at the nozzle opening, blocking the flow of material.
Cold Slug Well	A trap placed opposite the sprue or extension to the runner to stop the cold slug from entering the mold cavity or blocking the gate.
Conformal Cooling	The cooling of a mold tool using internal channels created via 3D printing. Such channels are able to conform to complex geometries, greatly enhancing cooling efficiency and finished part quality relative to conventional, line-of-sight cooling channels.
Cooling Time	A waiting period for die casting or plastic injection molding, allowing the part to solidify within the mold so that it can be safely ejected from the tool without deformation.



Term	Meaning
Copolymer	A polymer derived from two or more monomers.
Core	Upstanding protrusion in the mold, usually on the moving half of the mold. Typically forms the inside or concave side of the part.
Core Pin	A separate, removable insert in a plastic injection mold tool. Used to form a hole, aperture or other feature requiring high precision, or in areas requiring molded features difficult to access by other means.
Cycle Time	Production time required to complete one finished part in a molding operation and prepare for the next injection.
Cyclic Cooling	An approach to mold temperature control that allows rapid switching between hot and cold circuits. This approach can reduce cycle times and improve part quality.
D	
Date Marking	Indexable inserts used to mark the date an item was molded. Useful for long production runs where similar items are produced over long periods of time.
Delamination	This is when the surface of a molded part splits or separates as a result of processing conditions or contamination.
Diaphragm Gate	A variant of the edge gate, a diaphragm gate is used to gate around the full circumference of an internal hole on a part.
Draft Angle	An angle applied to the design of a surface or side wall of a molded part, allowing it to be removed from a mold easily and without any cosmetic damage due to scuffing against the surface of the mold tool.
Dryer	Heated chamber used to remove excessive moisture from plastic pellets of raw material before injection molding.
E	
Edge Gate	A gate that feeds a molded part from the split line. This usually has a rectangular cross section and is trimmed flush after molding.
EDM	A machining process for metal, using a charged electrode or wire to erode away the surface of the workpiece using electrical pulses. Especially useful for creating deep pockets, square corners, thin ribs and other features difficult or impossible to achieve through mechanical machining.



Term	Meaning
Ejection	Removing a finished piece from a mold via the use of retractable pins.
Ejector Blade	Square or rectangular pins of different dimensions strategically placed in the mold to push out or eject the formed part. Blades are typically used on ribs or areas not convenient for round pins.
Ejector Pin	Round pins of different diameters strategically placed in the mold to push out or eject the formed part.
Ejector Plates	Comprising two plates, the front plate has pin clearance holes and counterbores for the ejector pin heads, the back plate is fixed against the front plate to hold the ejector pins in place. The back plate has a high flatness and thickness tolerance requirement to ensure all pins sit flush with the core and have minimal forward and backwards movement.
F	
Family Mold	A multi-cavity mold that creates 2 or more different parts, each of which form a partial of a complete assembly
Fan Gate	A type of edge gate at the parting line that fans out to a wide gate no deeper than a normal edge gate.
Flash	In plastic injection molding, a thin residue of waste plastic extruded from between the halves of a mold tool along the parting line. Flash can be caused because there is damage to the cavity edges, mismatched shutoff heights, the two mold halves not closing properly or the mold being pushed open during injection due to inadequate clamping force.
Flow Front	The leading edge of molten plastic as it works its way through the mold cavity during injection. The front will be the first part to contact the solid walls of the cavity and hence to cool and solidify first
Food Safe	Plastic which has been approved for use in food storage or preparation, and hence will not outgas or leech harmful bi-products.
Forced Ejection	Some undercut features with suitable geometry can be pushed out of the mold using standard ejection, relying on the elasticity of the plastic. Also known as bumping off.



Term	Meaning
G	
Gas trap	As plastic is injected into the mold the plastic flow pushes out the air inside the cavity. If the mold is not properly vented this air is trapped and compressed. This can lead to burn marks or short shots.
Gate	An opening in a mold tool allowing raw material to flow from the runner into the cavity. Different types, shapes and sizes are available to suit the part, process or material requirement.
Gate Freeze Off Time	During a plastic injection molding cycle, the amount of time typically available for molten resin to fill a cavity before it begins to cool and harden. Hardening at the gate blocks additional resin from being injected into the cavity.
H	
Handload	A manually placed and removed component to allow the molding of undercuts.
Heel Block	An angled face that engages with a matching angle on the back of a sliding core. This pushes the core and locks it into place to oppose high injection forces.
Hopper	Drum or cylinder into which is loaded plastic pellets. The open throat at the bottom of the hopper allows the material to be fed into the screw of an injection molding machine.
Hot Runner/ Manifold	This is a heated system of channels used to distribute molten resin from the machine nozzle to hot tip locations within a mold. Used to feed multiple cavities in a mold or when two or more gate locations are required on a part.
Hot Sprue	A heated, tapered hole that allows material to flow directly from the molding machine nozzle into the mold cavity.
Hot Tip	Heated units that allow the flow of material directly from the molding machine barrel directly onto the surface of the molded part or to two or more gate locations via a hot runner system.
Hydraulic Cores	In order to mold long cores against the line of draw it is necessary to use hydraulic actuators. These work in a similar fashion to traditional sliding cores but hydraulics allow for a longer pull than heel blocks and angle pins will allow.



Term	Meaning
I	
Injection	Molten thermoplastic or thermoforming resin forced into a mold cavity under pressure.
J	
Jetting	An injection molding defect that causes snake like lines on the surface of a molding. Often caused when material leaves a smaller gate at high velocity directly into a thick section of the part or cavity.
K	
Knock Backs	Sometimes known as push backs or return pins. These pins are usually set in the corners of the mold away from the core and contact the ejector plate. When the mold closes these pins are pushed back by the closing split line causing the ejector plate to be returned to its home position.
L	
Line of Draw	This describes the orientation of the part that best allows both halves of the mold to separate cleanly. A hole through a side wall could be described as being 'not in line of draw'.
M	
Masterbatch	Heavily pigmented plastic resin created by a raw material compounder. It is mixed by the injection molder with uncolored plastic to create the finished part color, usually in a ratio of 1:20.
Melt Temperature	To maximize mold performance and efficiency it is necessary to control the mold temperature as recommended by the material manufacturer and maintain this within as small a margin of error as possible.
Molding Cycle	A molding cycle is composed of the following steps: Close, Inject, Pack, Cool, Open, Eject and Close. A single cavity mold with a 20 second cycle time produces one component every 20 seconds. A four cavity mold with a cycle time of 20 seconds produces one part every 5 seconds
MUD	Master Unit Die. A mold base of a standard size which simplifies the installation and removal of mold tools for plastic injection molding.



Term	Meaning
Multi Cavity	A mold tool that can create multiple finished parts per cycle using separate cavities.
N	
Nozzle	Fitted to the end of the barrel of the injection molding machine and designed to contact the sprue bush of the molds. The molten plastic exits the barrel through the nozzle into the mold.
O	
Overmolding	An injection molding technique that allows a second injection cycle to mold plastic on top of a substrate. Examples could include molding a soft TPE seal around the edge of an ABS case. A 2K machine can be used for this purpose.
P	
Packing	After the initial injection phase, a packing phase is required to push further material into the mold to enable full part density, to minimize sink marks and help with the repeatability of dimensional features.
Parting Line	The parting line is the physical break or split between the two halves of the mold. This parting line is often visible on the molded part. Good product design can ensure that the parting line is in an area that will be hidden, forms a part of the finished design, or is otherwise in an area not affecting final part quality.
Pigment	Coloring agent used in plastics, paints, and other raw materials.
PIM	Plastic Injection Molding
Pin Push	This is a molding defect that usually presents as stress marks on the A-surface of the part directly opposite the ejector pins. This can indicate a number of potential problems such as Insufficient ejection or over packing of the mold.
Platen	The large fixed and moving plates in an injection molding machine. The mold halves are clamped to these plates
Process Control	Controlling the parameters of the plastic injection molding cycle to ensure optimal and repeatable part quality.



Term	Meaning
Q	
R	
Resin	Thermoforming or thermosetting plastic raw material used for injection molding.
Ribs	Support structures in molded parts, usually placed at right angles to walls or bosses. The use of ribs can provide strength without creating thick wall sections, a common source of warping and distortion.
Runner	Feed channels cut into the faces of injection molds, directing the plastic from the sprue to the gate. Usually with a circular cross section and machined equally into both halves of the mold.
S	
Screw (Molding Machine)	Fit inside the barrel of a plastic injection molding machine, the screw has a helical thread of varying diameter and pitch. The screw action draws fresh material from the resin hopper, moving it through the barrel to be heated and plasticized before being compressed and injected through the nozzle into the mold cavity.
Short Shot	An injection molding defect where material has failed to completely fill the cavity. This can be caused by gas traps or poor process settings leading to insufficient shot volume.
Shot	Total volume of material injected into a mold cavity during one cycle.
Shot Weight	The total weight of all parts, sprues, runners and gates produced in a single molding cycle
Shrink	Decrease in volume of material when it transitions from a molten to a solid state. Varies according to the material type and part geometry. Fillers such as glass fiber can also affect shrinkage.
Sleeve Ejector	A cylindrical ejector pin with a precision bore cut through the center allowing the sleeve to pass over a fixed core pin.
Slider/Sliding Core	An automatic mechanism built into the mold to form undercuts or holes on the part. The opening and closing action of the mold pushes and retracts the undercut detail using angled pins and wedges.



Term	Meaning
Split line	Mark left on a molded part indicating where the two halves of the mold separate or the edges of a sliding core form an under cut.
Sprue	A tapered hole that feeds molten plastic from the injection molding machine nozzle into the mold runner system.
Sprue Bush	Designed to contact the nozzle of the injection molding machine and transport the injected plastic into the mold.
Sprue Puller	An undercut on the moving half of the mold opposite the sprue to pull the sprue clear of the sprue bush and aiding the ejection of the sprue and runner system so the feed system is clear for the next cycle.
Stress Marks	Visual marks on an injected molded part where unequal heating and cooling during manufacture caused local distortion.
Stripper Plate	Moveable plate used to push molded parts off of a mold core after injection. Usually used with thin walled parts where ejector pins would not be effective.
Submarine Gate	Type of gate that fills a mold cavity from beneath the level of the parting line.
T	
Thermoplastics	One group of moldable plastic resins which become molten when heated. Once set, thermoplastics can be re-melted and remolded with the addition of heat, so their physical state is said to be reversible.
Thermoset	One group of moldable plastic compounds which are comprised of two or more constituents undergoing a chemical change during processing and molding. Once molded, the process is irreversible.
Tolerance	Allowable deviation away from a nominal value.
U	
Undercut	A hole or feature in a part design that is not in line of draw. Typically an undercut will trap the part in the mold if undercut release mechanisms aren't used.
UV Stable	Plastic molding resin which is formulated to resist the deteriorating effects of UV light.



Term	Meaning
V	
Valve Gate	A mechanical valve used in a hot runner injection system which closes off the gate between injection cycles. This prevents leaking of plastic at the gate between cycles.
Vent	Opening in a plastic injection mold to allow air to escape as the cavity is filled.
W	
Wall	Vertical structures in molded parts, usually defining the exterior volume of a part cavity.
Warp	Distortion of molded part caused by unequal cooling rates after molding.
Water Channels	Channels in the cavity and core allowing a liquid media to pass through and control the mold temperature.
Weld Line	Visible mark on a molded part's surface, usually occurring around an obstruction such as a hole or cutout. Formed as a consequence of two flow fronts meeting during the injection cycle and cooling at differing rates.
X	
Y	
Z	

Thank you

Get more information to Click [here](#)

