Three-plate mold

A three-plate mold is used when part of the runner system is on a different plane to the injection location.

The runner system for a three-plate mold sits on a second parting plane parallel to the main parting plane. This second parting plane enables the runners and sprue to be ejected when the mold is opened. In the following diagram of a three-plate mold, the runners will be ejected separately to the cavities.

The runner system can be placed on a different plane from the injection location in the following situations:

- The mold contains multiple or family cavities
- A complex single cavity mold requires more than one injection location
- The injection location is in an awkward position
- Achieving a balanced flow requires the runner to be outside the parting plane

You can use a two-plate mold with hot runners to overcome these problems, but three-plate molds have some advantages over hot runner molds.

Advantages of three-plate molds

- Cheaper to build than a hot-runner mold
- Less likely to break down than a hot-runner mold
- Thermally sensitive materials are less likely to degrade

Disadvantages of three-plate molds
- Higher cycle time due to ejection of the runner system
- More material is wasted
- Greater injection pressure is required to fill the mold

Parent topic: Mold types
Tool Design - Mould Cooling System and Design Consideration...

*Mold cooling system overview*

**Importance of cooling system design:** Mold cooling accounts for more than two-thirds of the total cycle time in the production of injection molded thermoplastic parts. Figure 1 illustrates this point. An efficient cooling circuit design reduces the cooling time, which, in turn, increases overall productivity. Moreover, uniform cooling improves part quality by reducing residual stresses and maintaining dimensional accuracy and stability (see Figure 2).
FIGURE 1. Mold cooling accounts for more than two-thirds of the total cycle time.

FIGURE 2. Proper and efficient cooling improves part quality and productivity

Mold cooling system components

A mold cooling system typically consists of the following items:
- Temperature controlling unit
- Pump
- Supply manifold
- Hoses
- Cooling channels in the mold
- Collection manifold

The mold itself can be considered as a heat exchanger, with heat from the hot polymer melt taken away by the circulating coolant.

Figures 3 and 4 illustrate the components of a typical cooling system.
FIGURE 3. A typical cooling system for an injection molding machine.

FIGURE 4. A cooling channel assembly attached to the mold plates.

Cooling-channel configuration

Types of cooling channels
Cooling-channel configurations can be serial or parallel. Both configurations are illustrated in Figure 1 below.

**FIGURE 1. Cooling-channel configurations**

**Parallel cooling channels:**

Parallel cooling channels are drilled straight through from a supply manifold to a collection manifold. Due to the flow characteristics of the parallel design, the flow rate along various cooling channels may be different, depending on the flow resistance of each individual cooling channel. These varying flow rates in turn cause the heat transfer efficiency of the cooling channels to vary from one to another. As a result, cooling of the mold may not be uniform with a parallel cooling-channel configuration.

*Typically, the cavity and core sides of the mold each have their own system of parallel cooling channels. The number of cooling channels per system varies with the size and complexity of the mold.*

**Serial cooling channels**

Cooling channels connected in a single loop from the coolant inlet to its outlet are called serial cooling channels. This type of cooling-channel configuration is the most commonly recommended and used. By design, if the cooling channels are uniform in size, the coolant can maintain its (preferably) turbulent flow rate through its entire length. Turbulent flow enables heat to be transferred more effectively. Heat transfer of coolant flow discusses this more thoroughly. However, you should take care to minimize the temperature rise of the coolant, since the coolant will collect all the heat along the entire cooling-channel path. In general, the temperature difference of the coolant at the inlet and the exit should be within 5°C for general-purpose molds and 3°C for precision molds. For
large molds, more than one serial cooling channel may be required to assure uniform coolant temperature and thus uniform mold cooling.

**What do they do?**

Baffles and bubblers are sections of cooling lines that divert the coolant flow into areas that would normally lack cooling. Cooling channels are typically drilled through the mold cavity and core. The mold, however, may consist of areas too far away to accommodate regular cooling channels. Alternate methods for cooling these areas uniformly with the rest of the part involve the use of Baffles, Bubblers, or Thermal pins, as shown below.

**FIGURE 1. Baffle, bubbler, and thermal pin**

**Baffles**

A baffle is actually a cooling channel drilled perpendicular to a main cooling line, with a blade that separates one cooling passage into two semi-circular channels. The coolant flows in one side of the blade from the main cooling line, turns around the tip to the other side of the baffle, then flows back to the main cooling line.

This method provides maximum cross sections for the coolant, but it is difficult to mount the divider exactly in the center. The cooling effect and with it the temperature distribution on one side of the core may differ from that on the other side. This disadvantage of an otherwise economical solution, as far as manufacturing is concerned, can be eliminated if the metal sheet forming the baffle is twisted. For example, the helix baffle, as shown in Figure 2 below, conveys the coolant to the tip and back in the form of a helix. It is useful for diameters of 12 to 50 mm, and makes for a very homogeneous temperature distribution. Another logical development of baffles are single- or double-flight spiral cores, as shown in Figure 2 below.
Bubblers

A bubbler is similar to a baffle except that the blade is replaced with a small tube. The coolant flows into the bottom of the tube and “bubbles” out of the top, as does a fountain. The coolant then flows down around the outside of the tube to continue its flow through the cooling channels.

The most effective cooling of slender cores is achieved with bubblers. The diameter of both must be adjusted in such a way that the flow resistance in both cross sections is equal. The condition for this is:

\[
\frac{\text{Inner Diameter}}{\text{Outer Diameter}} = 0.707
\]

Bubblers are commercially available and are usually screwed into the core, as shown in Figure 3 below. Up to a diameter of 4 mm, the tubing should be beveled at the end to enlarge the cross section of the outlet; this technique is illustrated in Figure 3. Bubblers can be used not only for core cooling, but are also for cooling flat mold sections, which can’t be equipped with drilled or milled channels.
NOTE: Because both baffles and bubblers have narrowed flow areas, the flow resistance increases. Therefore, care should be taken in designing the size of these devices. The flow and heat transfer behavior for both baffles and bubblers can be readily modeled and analyzed by C-MOLD Cooling analysis.

**Thermal pins**

A thermal pin is an alternative to baffles and bubblers. It is a sealed cylinder filled with a fluid. The fluid vaporizes as it draws heat from the tool steel and condenses as it releases the heat to the coolant, as shown in Figure 4. The heat transfer efficiency of a thermal pin is almost ten times as great as a copper tube. For good heat conduction, avoid an air gap between the thermal pin and the mold, or fill it with a highly conductive sealant.
FIGURE 4. Thermal pin heat transfer efficiency

Cooling slender cores
If the diameter or width is very small (less than 3 mm), only air cooling is feasible. Air is blown at the cores from the outside during mold opening or flows through a central hole from inside, as shown in Figure 5. This procedure, of course, does not permit maintaining an exact mold temperature.

FIGURE 5. Air cooling of a slender core
Better cooling of slender cores (those measuring less than 5 mm) is accomplished by using inserts made of materials with high thermal conductivity, such as copper or beryllium-copper materials. This technique is illustrated in Figure 6 below. Such inserts are press-fitted into the core and extend with their base, which has a cross section as large as is feasible, into a cooling channel.

FIGURE 6. Using high thermal conductivity material to cool a slender core

Cooling large cores

For large core diameters (40 mm and larger), a positive transport of coolant must be ensured. This can be done with inserts in which the coolant reaches the tip of the core through a central bore and is led through a spiral to its circumference, and between core and insert helically to the outlet, as shown in Figure 7. This design weakens the core significantly.
Cooling cylinder cores

Cooling of cylinder cores and other round parts should be done with a double helix, as shown below. The coolant flows to the core tip in one helix and returns in another helix. For design reasons, the wall thickness of the core should be at least 3 mm in this case.

Cooling time

Theoretically, cooling time is proportional to the square of the heaviest part wall thickness or the power of 1.6 for the largest runner diameter. That is:
where the thermal diffusivity of polymer melt is defined as

\[
\text{Thermal diffusivity} = \frac{(\text{Thermal conductivity})}{(\text{Density}) (\text{Specific heat})}
\]

In other words, doubling the wall thickness quadruples the cooling time.

**Reynolds number and coolant flow**

Whether or not the coolant flow is turbulent can be determined by the Reynolds number \((\text{Re})\), as listed in Table 1. The Reynolds number is defined as:

\[
\text{Reynolds number (Re)} = \frac{\rho Ud}{\eta}
\]

where \(\rho\) is the density of the coolant, \(U\) is the averaged velocity of the coolant, \(d\) is the diameter of the cooling channel, and \(\eta\) is the dynamic viscosity of the coolant.

**TABLE 1. Coolant flow types and corresponding Reynolds number ranges**

<table>
<thead>
<tr>
<th>Reynolds Number (Re)</th>
<th>Type of Flow</th>
</tr>
</thead>
<tbody>
<tr>
<td>10,000 &lt; Re</td>
<td>Turbulent Flow</td>
</tr>
<tr>
<td>2,300 &lt; Re &lt; 10,000</td>
<td>Transition Flow</td>
</tr>
<tr>
<td>100 &lt; Re &lt; 2,300</td>
<td>Laminar Flow</td>
</tr>
<tr>
<td>Re &lt; 100</td>
<td>Stagnated Flow</td>
</tr>
</tbody>
</table>
Mold

Mold or die are the common terms used to describe the tooling used to produce plastic parts in molding.

Traditionally, molds have been expensive to manufacture. They were usually only used in mass production where thousands of parts were being produced. Molds are typically constructed from hardened steel, pre-hardened steel, aluminium, and/or beryllium-copper alloy. The choice of material to build a mold from is primarily one of economics, steel molds generally cost more to construct, but their longer lifespan will offset the higher initial cost over a higher number of parts made before wearing out. Pre-hardened steel molds are less wear resistant and are used for lower volume requirements or larger components. The steel hardness is typically 38-45 on the Rockwell-C scale. Hardened steel molds are heat treated after machining. These are by far the superior in terms of wear resistance and lifespan. Typical hardness ranges between 50 and 60 Rockwell-C (HRC). Aluminium molds can cost substantially less, and when designed and machined with modern computerized equipment, can be economical for molding tens or even hundreds of thousands of parts. Beryllium copper is used in areas of the mold which require fast heat removal or areas that see the most shear heat generated. The molds can be manufactured by either CNC machining or by using Electrical Discharge Machining processes.

<table>
<thead>
<tr>
<th>Injection molding die with side pulls</th>
</tr>
</thead>
<tbody>
<tr>
<td><img src="image1.png" alt="Image" /></td>
</tr>
</tbody>
</table>
Mold Design

Molds separate into two sides at a parting line, the A side, and the B side, to permit the part to be extracted. Plastic resin enters the mold through a sprue in the A plate, branches out between the two sides through channels called runners, and enters each part cavity through one or more specialized gates. Inside each cavity, the resin flows around protrusions (called cores) and conforms to the cavity geometry to form the desired part. This is similar to someone squeezing clay between their hands so that when it is removed, it matches the shape of the hollow of their cupped hands.

The amount of resin required to fill the sprue, runner and cavities of a mold is a shot. When a core shuts off against an opposing mold cavity or core, a hole results in the part. Air in the cavities when the mold closes escapes through very slight gaps between the plates and pins, into shallow plenums called vents. To permit removal of the part, its features must not overhang one another in the direction that the mold opens, unless parts of the mold are designed to move from between such overhangs when the mold opens (utilizing components called Lifters).

Sides of the part that appear parallel with the direction of draw (the direction in which the core and cavity separate from each other) are typically angled slightly with (draft) to ease release of the part from the mold, and examination of most plastic household objects will reveal this. Parts with bucket-like features tend to shrink onto the cores that form them while cooling, and cling to those cores when the cavity is pulled away. The mold is usually designed so that the molded part reliably remains on the ejector (B) side of the mold when it opens, and draws the runner and the sprue out of the (A) side along with the parts. The part then falls freely when ejected from the (B) side. Tunnel gates tunnel sharply below the parting surface of the B side at the tip of each runner so that the gate is sheared off of the part when both are ejected.

Ejector pins are the most popular method for removing the part from the B side core(s), but air ejection, and stripper plates can also be used depending on the application. Most ejection plates are found on the moving half of the tool, but they can be placed on the fixed half if spring loaded. For thermoplastics, coolant, usually water with corrosion inhibitors, circulates through passageways bored through the main plates on both sides of the mold to enable temperature control and rapid part solidification.

To ease maintenance and venting, cavities and cores are divided into pieces, called inserts, and sub-assemblies, also called inserts, blocks, or chase blocks.
By substituting interchangeable inserts, one mold may make several variations of the same part.

More complex parts are formed using more complex molds. These may have sections called slides, that move into a cavity perpendicular to the draw direction, to form overhanging part features. Slides are then withdrawn to allow the part to be released when the mold opens. Slides are typically guided and retained between rails called gibs, and are moved when the mold opens and closes by angled rods called horn pins and locked in place by locking blocks, both of which move cross the mold from the opposite side.

Some molds allow previously molded parts to be reinserted to allow a new plastic layer to form around the first part. This is often referred to as overmolding. This system can allow for production of one-piece tires and wheels.

2-shot or multi-shot molds are designed to "overmold" within a single molding cycle and must be processed on specialized injection molding machines with two or more injection units. This can be achieved by having pairs of identical cores and pairs of different cavities within the mold. After injection of the first material, the component is rotated on the core from the one cavity to another. The second cavity differs from the first in that the detail for the second material is included. The second material is then injected into the additional cavity detail before the completed part is ejected from the mold. Common applications include "soft-grip" toothbrushes and freelancer grab handles.

The core and cavity, along with injection and cooling hoses form the mold tool. While large tools are very heavy weighing hundreds and sometimes thousands of pounds, with the aid of a forklift or overhead crane, they can be hoisted into molding machines for production and removed when molding is complete or the tool needs repairing.

A mold can produce several copies of the same parts in a single "shot". The number of "impressions" in the mold of that part is often incorrectly referred to as cavitation. A tool with one impression will often be called a single cavity (impression) tool. A mold with 2 or more cavities of the same parts will likely be referred to as multiple cavity tooling. Some extremely high production volume molds (like those for bottle caps) can have over 128 cavities.

In some cases multiple cavity tooling will mold a series of different parts in the same tool. Some toolmakers call these molds family molds as all the parts.
Effects on the material properties

The mechanical properties of a part are usually little effected. Some parts can have internal stresses in them. This is one of the reasons why it's good to have uniform wall thickness when molding. One of the physical property changes is shrinkage. A permanent chemical property change is the material thermoset, which can't be remelted to be injected again.

The effects on work material properties is best illustrated by the table below from Manufacturing Processes Reference Guide:

<table>
<thead>
<tr>
<th>Work Material Properties</th>
<th>Effects of Injection Molding</th>
</tr>
</thead>
<tbody>
<tr>
<td>Mechanical</td>
<td>Plastic components may develop internal stress</td>
</tr>
<tr>
<td>Physical</td>
<td>Permanent shrinkage at elevated temperatures</td>
</tr>
<tr>
<td>Chemical</td>
<td>Thermoset materials cannot be remelted</td>
</tr>
</tbody>
</table>

Tool Materials

Tool steel or beryllium-copper are often used. Mild steel, aluminum, nickel or epoxy are only suitable for prototype or very short production runs.

Geometrical Possibilities

The most commonly used plastic molding process, injection molding, is used to create a large variety of products with different shapes and sizes. Most importantly, they can create products with complex geometry that many other processes cannot. There are a few precautions when designing something that will be made using this process to reduce the risk of weak spots. First, streamline your product or keep the thickness relatively uniform. Second, try and keep your product between 2 to 20 inches.

The size of a part will depend on a number of factors (material, wall thickness, shape, process etc). The initial raw material required may be measured in the form of granules, pellets or powders. Here are some ranges of the sizes.

<table>
<thead>
<tr>
<th>Method</th>
<th>Raw Materials</th>
<th>Maximum Size</th>
<th>Minimum Size</th>
</tr>
</thead>
<tbody>
<tr>
<td>Injection Molding (thermo-plastic)</td>
<td>Granules, Pellets, Powders</td>
<td>700 oz.</td>
<td>Less than 1 oz.</td>
</tr>
</tbody>
</table>
Machining

Molds are built through two main methods: standard machining and EDM. Standard Machining, in its conventional form, has historically been the method of building injection molds. With technological development, CNC machining became the predominant means of making more complex molds with more accurate mold details in less time than traditional methods.

The electrical discharge machining (EDM) or spark erosion process has become widely used in mold making. As well as allowing the formation of shapes which are difficult to machine, the process allows pre-hardened molds to be shaped so that no heat treatment is required. Changes to a hardened mold by conventional drilling and milling normally require annealing to soften the steel, followed by heat treatment to harden it again. EDM is a simple process in which a shaped electrode, usually made of copper or graphite, is very slowly lowered onto the mold surface (over a period of many hours), which is immersed in paraffin oil. A voltage applied between tool and mold causes spark erosion of the mold surface in the inverse shape of the electrode.

Cost

The cost of manufacturing molds depends on a very large set of factors ranging from number of cavities, size of the parts (and therefore the mold), complexity of the pieces, expected tool longevity, surface finishes and many others. The initial cost is great, however the piece part cost is low, so with greater quantities the overall price decreases.

Injection process

Small injection molder showing hopper, nozzle and die area
For detailed explanation refer below to "Injection Molding Cycle".

With Injection Molding, granular plastic is fed by gravity from a hopper into a heated barrel. As the granules are slowly moved forward by a screw-type plunger, the plastic is forced into a heated chamber, where it is melted. As the plunger advances, the melted plastic is forced through a nozzle that rests against the mold, allowing it to enter the mold cavity through a gate and runner system. The mold remains cold so the plastic solidifies almost as soon as the mold is filled.

**Injection molding cycle**

For the injection molding cycle to begin, four criteria must be met: mold open, ejector pins retracted, shot built, and carriage forward. When these criteria are met, the cycle begins with the mold closing. This is typically done as fast as possible with a slow down near the end of travel. Mold safety is low speed and low pressure mold closing. It usually begins just before the leader pins of the mold and must be set properly to prevent accidental mold damage. When the mold halves touch clamp tonnage is built. Next, molten plastic material is injected into the mold. The material travels into the mold via the sprue bushing, then the runner system delivers the material to the gate. The gate directs the material into the mold cavity to form the desired part. This injection usually occurs under velocity control.

When the part is nearly full, injection control is switched from velocity control to pressure control. This is referred to as the pack/hold phase of the cycle. Pressure must be maintained on the material until the gate solidifies to prevent material from flowing back out of the cavity. Cooling time is dependent primarily on the wall thickness of the part but also depends on the material being molded. Production molding usually requires faster cooling. Water is often channeled throughout the dies to produce faster cooling times. During the cooling portion of the cycle after the gate has solidified, plastication takes place.

Plastication is the process of melting material and preparing the next shot. The material begins in the hopper and enters the barrel through the feed throat. The feed throat must be cooled to prevent plastic pellets from fusing together from the barrel heat. The barrel contains a screw that primarily uses shear to melt the pellets and consists of three sections. The first section is the feed section which conveys the pellets forward and allows barrel heat to soften the pellets. The flight depth is uniform and deepest in this section. The next section is the transition section and is responsible for melting the material through shear. The flight depth continuously decreases in this section, compressing the
material. The final section is the metering section which features a shallow flight depth, improves the melt quality and color dispersion. At the front of the screw is the non-return valve which allows the screw to act as both an extruder and a plunger. When the screw is moving backwards to build a shot, the non-return assembly allows material to flow in front of the screw creating a melt pool or shot. During injection, the non-return assembly prevents the shot from flowing back into the screw sections.

Once the shot has been built and the cooling time has timed out, the mold opens. Mold opening must occur slow-fast-slow. The mold must be opened slowly to release the vacuum that is caused by the injection molding process and prevent the part from staying on the stationary mold half. This is undesirable because the ejection system is on the moving mold half. Then the mold is opened as far as needed, if robots are not being used, the mold only has to open far enough for the part to be removed. A slowdown near the end of travel must be utilized to compensate for the momentum of the mold. Without slowing down the machine cannot maintain accurate positions and may slam to a stop damaging the machine. Once the mold is open, the ejector pins are moved forward, ejecting the part. When the ejector pins retract, all criteria for a molding cycle have been met and the next cycle can begin.

The basic injection cycle is as follows: Mold close – injection carriage forward – inject plastic – metering – carriage retract – mold open – eject part(s) Some machines are run by electric motors instead of hydraulics or a combination of both. The water-cooling channels that assist in cooling the mold and the heated plastic solidifies into the part. Improper cooling can result in distorted molding. The cycle is completed when the mold opens and the part is ejected with the assistance of ejector pins within the mold.

The resin, or raw material for injection molding, is most commonly supplied in pellet or granule form. Resin pellets are poured into the feed hopper, a large open bottomed container, which is attached to the back end of a cylindrical, horizontal barrel. A screw within this barrel is rotated by a motor, feeding pellets up the screw's grooves. The depth of the screw flights decreases toward the end of the screw nearest the mold, compressing the heated plastic. As the screw rotates, the pellets are moved forward in the screw and they undergo extreme pressure and friction which generates most of the heat needed to melt the pellets. Electric heater bands attached to the outside of the barrel assist in the heating and temperature control during the melting process.

The channels through which the plastic flows toward the chamber will also solidify, forming an attached frame. This frame is composed of the sprue, which is the main channel from the reservoir of molten resin, parallel with the
direction of draw, and runners, which are perpendicular to the direction of draw, and are used to convey molten resin to the gate(s), or point(s) of injection. The sprue and runner system can be cut or twisted off and recycled, sometimes being granulated next to the mold machine. Some molds are designed so that the part is automatically stripped through action of the mold.

**Time Function**

The time it takes to make a product using injection molding can be calculated by adding:
- Twice the Mold Open/Close Time \( (2M) \)
- Injection Time \( (T) \)
- Cooling Time \( (C) \)
- Ejection Time \( (E) \)

Where \( T \) is found by dividing:
- Mold Size \( (S) \) / Flow Rate \( (F) \)

\[
\text{Total time} = 2M + T + C + E \nonumber
\]

\[
T = \frac{V}{R} \nonumber
\]

\[
V = \text{Mold cavity size (in}^3\text{)} \\
R = \text{Material flow rate (in}^3\text{/min)} 
onumber
\]

http://upload.wikimedia.org/wikipedia/commons/9/9e/Time_Calculation.jpg

**Molding trial**

When filling a new or unfamiliar mold for the first time, where shot size for that mold is unknown, a technician/tool setter usually starts with a small shot weight and fills gradually until the mold is 95 to 99% full. Once this is achieved a small amount of holding pressure will be applied and holding time increased until gate freeze off has occurred, then holding pressure is increased until the parts are free of sinks and part weight has been achieved. Once the parts are good enough and have passed any specific criteria, a setting sheet is produced for people to follow in the future.
Process optimization is done using the following methods. Injection speeds are usually determined by performing viscosity curves. Process windows are performed varying the melt temperatures and holding pressures. Pressure drop studies are done to check if the machine has enough pressure to move the screw at the set rate. Gate seal or gate freeze studies are done to optimize the holding time. A cooling time study is done to optimize the cooling time.

**Molding defects**

Injection molding is a complex technology with possible production problems. They can either be caused by defects in the molds or more often by part processing (molding)

<table>
<thead>
<tr>
<th>Molding Defects</th>
<th>Alternative name</th>
<th>Descriptions</th>
<th>Causes</th>
</tr>
</thead>
<tbody>
<tr>
<td>Blister</td>
<td>Blistering</td>
<td>Raised or layered zone on surface of the part</td>
<td>Tool or material is too hot, often caused by a lack of cooling around the tool or a faulty heater</td>
</tr>
<tr>
<td>Burn marks</td>
<td>Air Burn/Gas Burn</td>
<td>Black or brown burnt areas on the part located at furthest points from gate</td>
<td>Tool lacks venting, injection speed is too high</td>
</tr>
<tr>
<td><strong>Color streaks</strong> (US)</td>
<td>Colour streaks (UK)</td>
<td>Localized change of color/colour</td>
<td>Masterbatch isn't mixing properly, or the material has run out and it's starting to come through as natural only</td>
</tr>
<tr>
<td>Delamination</td>
<td></td>
<td>Thin mica like layers formed in part wall</td>
<td>Contamination of the material e.g. PP mixed with ABS, very dangerous if the part is being used for a safety critical application as the material has very little strength when delaminated as the materials cannot bond</td>
</tr>
<tr>
<td>Flash</td>
<td>Burrs</td>
<td>Excess material in thin layer</td>
<td>Tool damage, too much</td>
</tr>
<tr>
<td>Issue</td>
<td>Description</td>
<td>Cause</td>
<td></td>
</tr>
<tr>
<td>-----------------------</td>
<td>-----------------------------------------------------------------------------</td>
<td>-------------------------------------------------------------------------------------------</td>
<td></td>
</tr>
<tr>
<td>Exceeding normal part</td>
<td>Exceeding normal part geometry</td>
<td>Injection speed/material injected, clamping force too low. Can also be caused by dirt and contaminants around tooling surfaces.</td>
<td></td>
</tr>
<tr>
<td>Part geometry</td>
<td></td>
<td>Particles on the tool surface, contaminated material or foreign debris in the barrel, or too much shear heat burning the material prior to injection.</td>
<td></td>
</tr>
<tr>
<td>Embedded contaminates</td>
<td>Embedded particulates</td>
<td>Foreign particle (burnt material or other) embedded in the part</td>
<td></td>
</tr>
<tr>
<td>Flow marks</td>
<td>Flow lines</td>
<td>Directionally &quot;off tone&quot; wavy lines or patterns</td>
<td></td>
</tr>
<tr>
<td>Jetting</td>
<td>Deformed part by turbulent flow of material</td>
<td>Injection speeds too slow (the plastic has cooled down too much during injection, injection speeds must be set as fast as you can get away with at all times.</td>
<td></td>
</tr>
<tr>
<td>Poor tool design, gate position or runner. Injection speed set too high.</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>Polymer degradation</td>
<td>polymer breakdown from hydrolysis, oxidation etc</td>
<td>Excess water in the granules, excessive temperatures in barrel</td>
<td></td>
</tr>
<tr>
<td>Sink marks</td>
<td>Localized depression (In thicker zones)</td>
<td>Holding time/pressure too low, cooling time too short, with sprueless hot runners this can also be caused by the gate temperature being set too high.</td>
<td></td>
</tr>
<tr>
<td>Short shot</td>
<td>Non-fill / Short mold</td>
<td>Lack of material, injection speed or</td>
<td></td>
</tr>
<tr>
<td>Condition</td>
<td>Description</td>
<td>Cause</td>
<td></td>
</tr>
<tr>
<td>---------------</td>
<td>----------------------------------------------------------------</td>
<td>----------------------------------------------------------------------</td>
<td></td>
</tr>
<tr>
<td>Splay marks</td>
<td>Circular pattern around gate caused by hot gas</td>
<td>Moisture in the material, usually when hygroscopic resin is dried improperly</td>
<td></td>
</tr>
<tr>
<td>Stringiness</td>
<td>String like remain from previous shot transfer in new shot</td>
<td>Nozzle temperature too high. Gate hasn't frozen off</td>
<td></td>
</tr>
<tr>
<td>Voids</td>
<td>Empty space within part (Air pocket)</td>
<td>Lack of holding pressure (holding pressure is used to pack out the part during the holding time). Also mold may be out of registration (when the two halves don't center properly and part walls are not the same thickness)</td>
<td></td>
</tr>
<tr>
<td>Weld line</td>
<td>Discolored line where two flow fronts meet</td>
<td>Mold/material temperatures set too low (the material is cold when they meet, so they don't bond)</td>
<td></td>
</tr>
<tr>
<td>Warping</td>
<td>Distorted part</td>
<td>Cooling is too short, material is too hot, lack of cooling around the tool, incorrect water temperatures (the parts bow inwards towards the hot side of the tool)</td>
<td></td>
</tr>
</tbody>
</table>

**Tolerances and Surfaces**

Injection molding typically is capable of tolerances equivalent to an IT Grade of about 9–14. The possible tolerance of a thermoplastic or a thermoset is +/- 0.008 to +/-0.002 inches. Surface finishes of two to four microinches or better are can be obtained. Rough or pebbled surfaces are also possible.