Processing of Thermoplastics

McKelvey defined plastics processing as *"operations carried out on polymeric materials or systems to increase their utility"*. These types of operations produce flow, chemical change, and/or a permanent change in physical properties. Plastics processing techniques can be grouped into three categories:

1. Forming operations
2. Bonding operations
3. Modifications

Forming operations always involve flow, thermoplastic processes, such as extrusion, thermoplastic injection molding, thermoforming, and rotational molding, produce physical changes in the polymer whereas chemical change occurs in the casting of liquid monomers. Reactive extrusion and thermoset injection molding induce both chemical and physical change in the plastics materials.

Bonding operations join two or more materials by causing one or both joining surfaces to become molten or flow. The former occurs while laminating polyethylene to aluminum or paper, coating of polyvinyl chloride plastisols on fabric (to produce vinyl upholstery), and using adhesives to join materials.

Modifications include surface activation, mixing, and polymer modifications. Surface activation improves adhesion or printability of plastics materials.

Processing temperatures

Polymers are manufactured using two basic polymerization methods:

1. addition and
2. condensation.

Addition polymerization generally produces rapid chain growth, molecular weights greater than 100,000 daltons, and no by-products. In contrast, *condensation polymerization* provides for lower chain growth, typical molecular weights of 10,000 to 50,000 daltons, and by-products such as water. As a result, addition polymers are *less susceptible to water absorption*, and seldom *depolymerize* during processing. When these materials are dried prior to processing, it is usually to prevent foaming and surface defects such as splay. However, if poorly dried condensation polymers are melt processed, they tend to depolymerize. Since this reduces the molecular weight, material properties decrease. Consequently, *condensation polymers* are *always dried prior* to processing.

Polyethylene, polypropylene, polystyrene, impact-modified polystyrene, acrylonitrile-butadiene-styrene terpolymer, polymethylmethacrylate, poly(vinyl chloride), and polytetrafluoroethylene are addition polymers, whereas polyacetal, polycarbonate, polyamides, poly(ethylene terephthalate), poly(butylene terephthalate), polysulfones, polyetherimide, and polyetheretherketones (PEEK) are condensation polymers.

TABLE 5.1 Suggested Drying Conditions for Generic Resins[3,4]

Material	Water absorption, %	Maximum water, %	$T_{extrusion}$, °C	$T_{inj. molding}$, °C	T_{drying}, °C	t_{drying}, h
Acrylonitrile butadiene styrene (ABS)	0.25–0.40	0.20	225	260	88	3–4
Acetal	0.25	—	—	200	93	1–2
Acrylic	0.20–0.30	0.08	190	235	82	1–2
Polyamide-6 (nylon) (PA-6)	1.60	0.15	270	290	82	4–5
Polyamide-6, 6 (nylon) (PA-6,6)	1.50	0.15	265	265	82	4–5
Polycarbonate (PC)	0.20	0.02	290	300	120	3–4
Polybutylene terephthalate (PBT)	0.08	0.04	—	240	125	2–3
Polyethylene terephthalate (PET)	0.10	0.005	250	255	160	4–5
Polyetherimide (PEI)	0.25	—	—	370	155	4–5
High-density polyethylene (HDPE)	<0.01	—	210	250	—	—
Low-density polyethylene (LDPE)	<0.01	—	180	205	—	—
Linear low-density polyethylene (LLDPE)	<0.01	—	260	220	—	—
Polyphenylene oxide (PPO)	0.07	—	250	275	100	2–3
Polypropylene (PP)	<0.01	—	235	255	—	—
Polystyrene (PS)	0.03	—	210	220	—	—
High-impact polystyrene (HIPS)	0.10	—	235	230	—	—
Polyphenylene sulfide (PPS)			—	330	140	2–3
Polysulfone (PSU)	0.30	0.05	345	360	135	3–4
Polyurethane (PU)	0.10	0.03	205	205	82	2–3
PU (elastomers)	0.07	0.03	200	205	100	2–3
r-PVC (polyvinyl chloride)	0.10	0.07	185	195	—	—
p-PVC (polyvinyl chloride)	0.02	—	175	150	—	—
Styrene acrylonitrile (SAN)	0.03	0.02	215	245	82	3–4

Processing temperatures are associated with the transition temperatures of a polymer. In amorphous polymers, Tg is related to processing temperatures. As shown in figure 1.1, the modulus (stiffness) is relatively constant until the temperature rises above the Tg. The modulus then decreases gradually. When the polymer reaches its melt processing temperature, the polymer flows easily and can be extruded, injection molded, and extrusion blow molded.

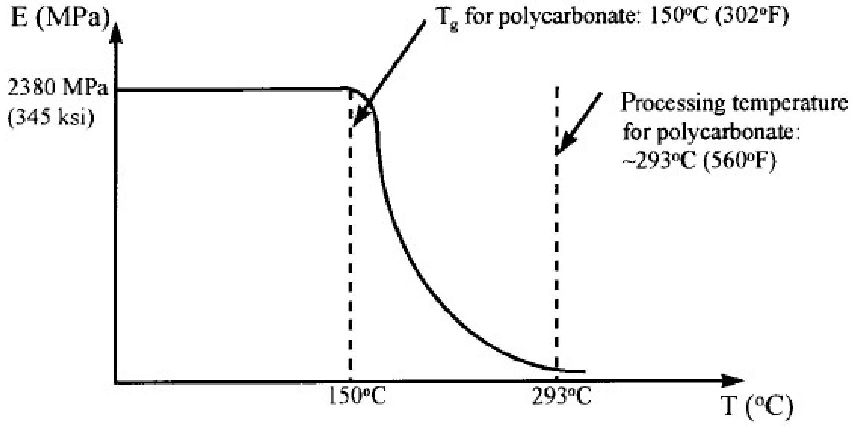


Fig 1.1 Modulus-temperature curves for polycarbonate

For polycarbonate, the difference between the softening temperature and processing temperature is about 140°C. Since this slow reduction in modulus over a wide temperature range facilitates stretching of the rubbery material, amorphous materials, such as polycarbonate, are easily thermoformed. Figure 1.2 shows the modulus-temperature curve of a semicrystalline polymer, polypropylene.

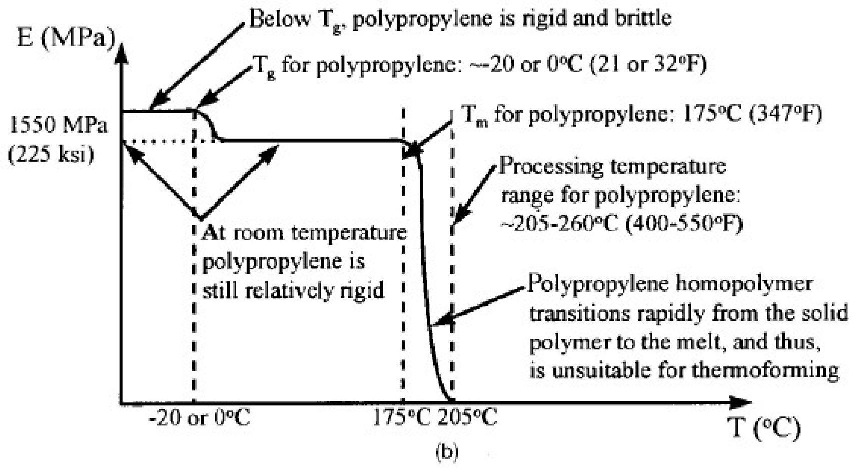


Fig 1.2 Modulus-temperature curves for polypropylene

It exhibits a glass transition and a melting transition (T_m). The modulus of polypropylene, like other polymers with high levels of crystallinity, does not decrease substantially when the temperature is raised above the glass transition temperature. Thus, polypropylene remains relatively rigid until it reaches its T_m. At that point the crystallites (highly ordered regions) in semicrystalline polymers break up and the polymer begins to flow. Since all polymers contain amorphous regions, they do not have well defined melting temperatures. Melt processing temperatures of semicrystalline polymers are usually less than 100°C above their melt temperatures. While thermoplastic polymers soften at T_g, and if semicrystalline, melt at T_m, cross-linked polymers do not melt and flow as shown in figure 1.3.

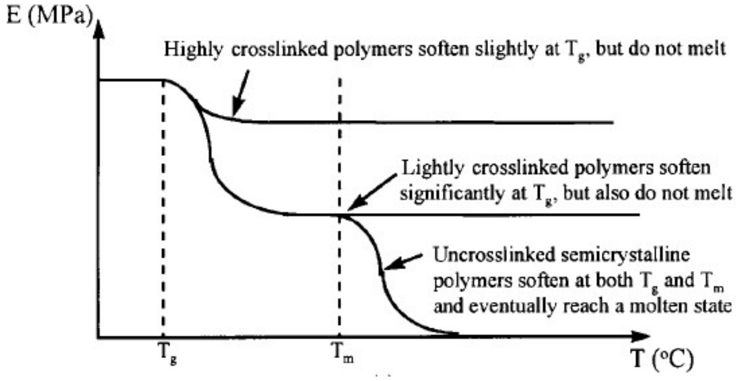


Fig 1.3 Modulus-temperature curves for generic thermoplastic and cross-linked materials

Lightly cross-linked polymers soften as the temperature exceeds *Tg,* but they remain rubbery solids until the polymer decomposes. Highly cross-linked polymers often do not even soften and retain a high modulus until reaching the decomposition temperature. Some thermoplastics will decompose before they melt and flow. Extremely long polymer chains combined with intermolecular attractions prevent conventional melt processing of ultrahigh-molecular weight polyethylene (UHMWPE) and polytetrafluoroethylene (PTFE). Thus, these materials are usually processed as powders or slurries.

Properties of polymer particles

For most thermoplastic processing methods, the resin is supplied as solid plastic particles. The flow of these particles affects the feeding of extruders and injection molding machines, as well as material flow in rotational molding. Spherical or cylindrical pellets with diameters averaging 1 to 5 mm tend to flow freely. Granules are smaller (sizes range from 0.1 to 1 mm) and may be free flowing or semifree flowing. Powders are very fine (0.1 to 100μm in size) and tend to be cohesive and trap air. Reground parts and film and fiber scrap provide large, irregular particles (typically greater than 5 mm in size) which tend to interlock and not flow. Particle flow is quantified by the measurements of particle size, shape, surface area, pore size, and volume, as well as the bulk densities and coefficients of friction of these materials [4].

Plastics processing

Plastics processing refers to the sequence of operations that are involved in the manufacturing of plastics by the transformation of polymer formulations or compounds to plastics products. A typical plastics processing system starts with a solid (and sometimes liquid) resin in granular or powder form (random configuration), and ends with the transformation of the resin into a solid product. The sequence of operations experienced by the material during this transformation involves solids transport, heating, melting, melt flow, shaping in a die or mold, and finally solidification, by cooling in the case of thermoplastics, or curing by heating in the case of thermosets [1].

Injection Moulding Process

During this process, molten plastic is forced (injected) into a mold and cooled until the melt solidifies. When the part is cooled sufficiently, the mold is opened, the part is ejected from the mold, and the mold is closed again to repeat the cycle. Thus, injection molding permits mass-production, high-precision, and three-dimensional virtual netshape manufacturing of plastic parts [4].

Types of Injection moulding machines (IMM)

The four types of injection ends are:

1. The straight plunger machine.
2. The two-stage plunger machine where one plunger plasticizes and the other shoots.
3. The reciprocating screw.
4. The two-stage screw-plunger machine where the screw plasticizes material, which is forced into the shooting cylinder.

Injection moulding machine (IMM)

IMM perform certain essential functions:

(1) **plasticizing:** heating and melting of the plastic in the plasticator,

(2) **injection:** injecting from the plasticator under pressure a controlled-volume shot of melt into a closed mold, with solidification of the plastics beginning on the mold's cavity wall,

(3) **after filling:** maintaining the injected material under pressure for a specified time to prevent back flow of melt and to compensate for the decrease in volume of melt during solidification,

(4) **cooling:** cooling the thermoplastic (TP) molded part in the mold until it is sufficiently rigid to be ejected, or heating: heating the thermoset (TS) molded part in the mold until it is sufficiently rigid to be ejected, and

(5) **molded-part release:** opening the mold, ejecting the part, and closing the mold so it is ready to start the next cycle with a shot of melt [2].

Reciprocating Screw Machine - Stages in injection molding process

For a reciprocating screw machine the process cycle can be split into five stages:

1. In stage 1, as shown in Figure 1.4, material is injected into the tool.

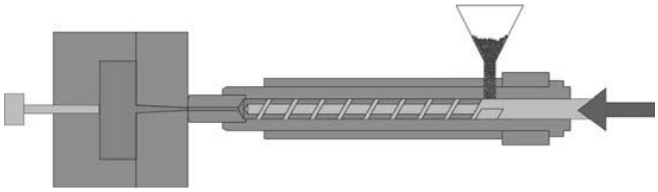


Figure 1.4 injection

2. In stage 2 (Figure 1.5), the screw begins to turn and retract, metering a specified weight of molten material for the next shot. The previous shot is now cooling in the closed tool.

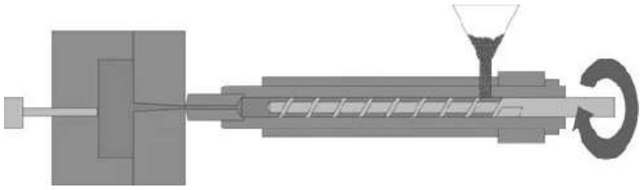


Figure 1.5 metering

3. In stage 3, the injection unit moves back from the clamping unit as shown in Figure 1.6.

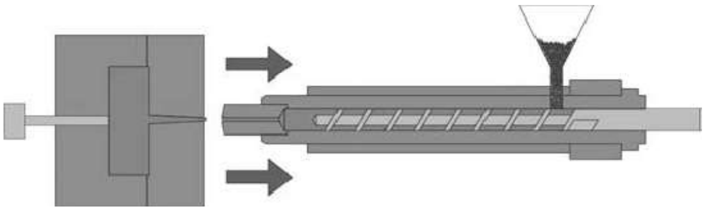


Figure 1.6 injection unit retracts

4. Stage 4 is shown in Figure 1.7 In this stage the tool opens to reveal a cooled injection moulded component.

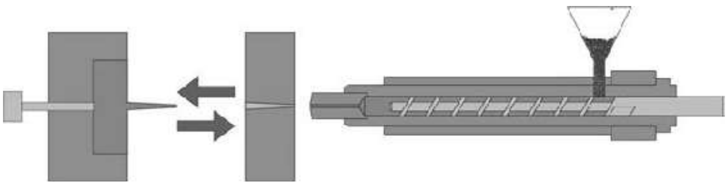


Figure 1.7 mould open

5. Stage 5 is ejection of the part as shown in Figure 1.8. The injection unit will then move forward to the clamp unit to start a fresh cycle as shown in stage 1.

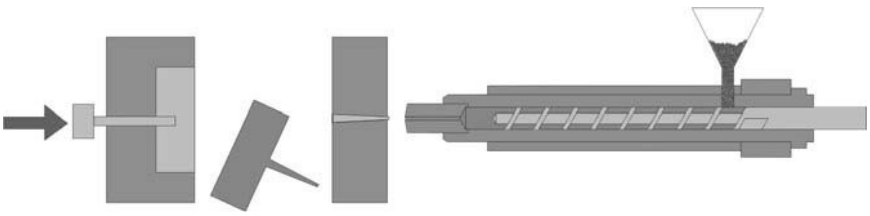


Figure 1.8 ejection

Thermoplastics as well as thermosets and classic elastomers can be processed with screw injection units. Thermoplastics are injected into a cold mould. The temperature of the mould must be sufficiently below the melting temperature of the material for it to solidify. This is because solidification is a physical process. Thermosets and classical elastomers are injected into a hot mould to make the crosslinking of the material possible. Crosslinking is a chemical process. Figure 1.9 shows the breakdown of injection molding cycle.

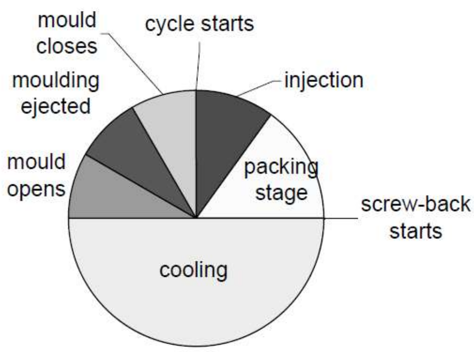


Figure 1.9 Breakdown of an injection moulding cycle

Phases of Injection Molding process

A plastic part's properties depend on how the part is molded. Two parts having identical dimensions and made from the same material but molded under different conditions will have different stress and shrinkage levels and will behave differently in the field, meaning that they are in practice two different parts. The way the plastic flows into the mold is of paramount importance in determining the quality of the part.

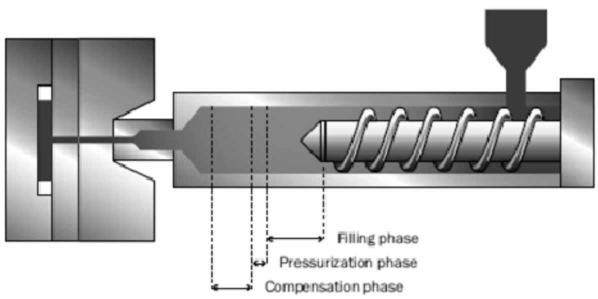
How Plastic Fills a Mold?

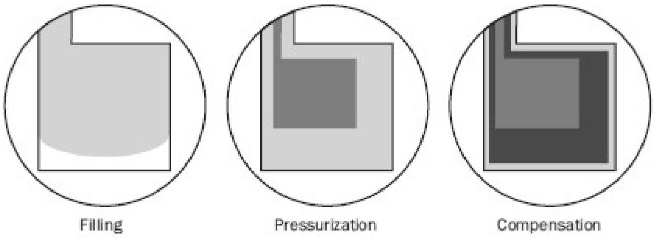
The injection molding process, could be divided into three phases

1. *filling phase*
2. *pressure phase*
3. *compensation phase*

1. Filling Phase

As the screw moves forward, it first moves at a steady speed as the plastic flows into the cavity. This is the filling phase. This phase lasts until the mold is just filled. See Figure 1.10.





1.10 Phases of injection molding

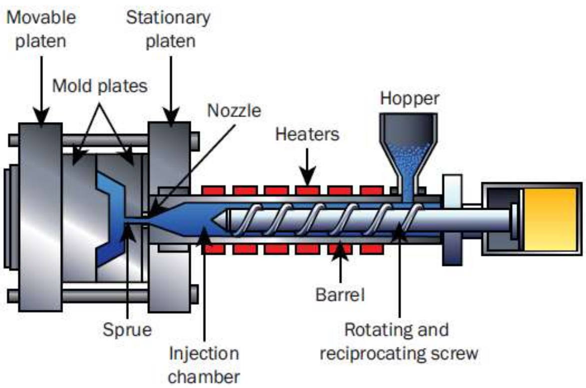
2. Pressurization Phase

The pressurization phase begins when the screw moves forward after the filling phase to bring the mold up to pressure. When the mold is filled, the screw will slow down, but it still moves quite some distance because plastics are very compressible materials. At injection molding pressure, an extra 15% volume of material can be forced into the cavity. (See above figures) Although fluids are usually assumed to be incompressible, molten plastics have to be considered to be more like a gas. *The compressibility of plastics can be observed by blocking off the nozzle and attempting to purge the barrel. The ram will jump forward when the pressure is applied, but will spring back when the pressure is released.*

3. Compensation Phase

After the pressurization phase, the screw still does not stop completely, continuing to creep forward for some time. Plastics have a very large volumetric change of about 25% from the melt to the solid. See above Figure 1.10. The screw moving forward to compensate for the volumetric change in the part is called the compensation phase. As the volumetric change is 25% and, at the most, only an extra 15% can be injected in the pressurization phase, there must always be some compensation phase [5].

Injection-molding machines - Reciprocating screw type



Injection-molding machines have three components:

1. the injection unit,
2. the clamping unit, and

3. the control system.

The injection unit plasticates and injects the polymer melt while the clamping unit supports the mold, opens and closes the mold, and contains the part ejection system whereas the control system controls the sequence of operations of the machine.

Injection Unit

The injection unit has two major components:

1. the plasticating unit
2. the sled

The injection unit brings the nozzle into contact with the sprue bushing of the mold, generates contact pressure between the nozzle and sprue bushing, melts the plastic material, injects the molten material into the mold, and builds up packing and holding pressure. In single-stage reciprocating-screw injection units the screw rotates to plasticate the polymer and moves linearly to inject the melt. The injection unit consists of a hopper, feed throat, barrel, screw, screw drive motor, and nozzle.

The Hopper

Thermoplastic material is supplied to molders in the form of small pellets. The hopper on the injection molding machine holds these pellets. The pellets are gravity-fed from the hopper through the feed throat into the barrel and screw assembly. The feed throat is water cooled to prevent the granulated plastic from melting (bridging) in the feed throat.

The Barrel

The barrel of the injection molding machine supports the reciprocating plasticizing screw. It is heated by the electric heater bands. A motor rotates the screw, conveying and melting the plastic as it travels through the barrel. Injection-molding barrels (Fig.1.11) are shorter than extruder barrels. The discharge end of the barrel fastens directly to an end cap or nozzle adapter, the counterbore at the end of the barrel centers the end cap. Since the barrel sees pressures in excess "bell ends" and/or a high-pressure sleeve are located at discharge end of the barrel. This sleeve is a "stronger heat-treated alloy steel" that is shrunk over the barrel. The typical *L/D* ratio is 18:1 to 24:1 for conventional machines, 22:1 to 26:1 for "fast running machines," and 28:1 for vented barrel injection-molding machines.

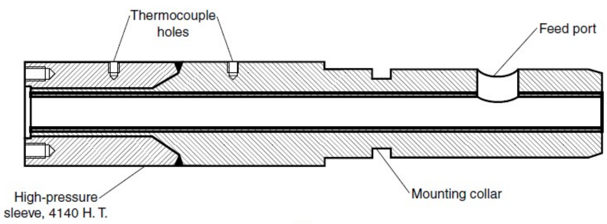


Fig.1.11 Components of an injection unit - barrel

The Injection Screw

The screw plays the vital role in the process of achieving a homogeneous melt. The screw provides shear heat to assist in the melting process along with the required mixing and homogenizing of the melt. It also helps in accurately measuring the volume of the shot to be injected into the mold. The most common screw is called a general purpose screw or the GP screw. The design of a GP screw is shown in Fig. 1.11.

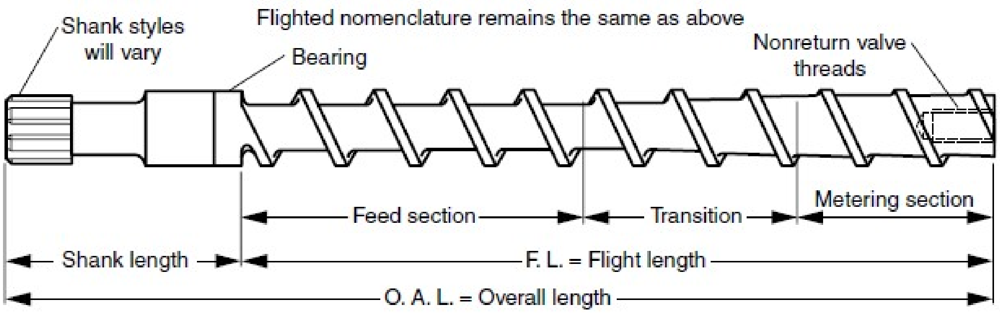


Fig. 1.11 injection screw

Screw terminologies

Outside Diameter: This is the diameter of an imaginary cylinder that is created by joining the outside area of the flights. The outside diameter is constant and is slightly smaller than the internal diameter of the barrel.

Root Diameter: This is the diameter of the shank. The root diameter changes from the back to the front of the screw, depending on the section of the screw.

Channel Depth: The difference between the outside diameter and the root diameter is the feed depth. Since the root diameter changes, the feed depth also changes from the back to the front of the screw.

The GP screw has three main sections (or zones), each of which serves a special purpose.

1. Feed zone
2. Transition Zone (Compression Zone)
3. Metering Zone

Feed Zone: This is the section of the screw that picks up the material from the feed opening (base of the hopper) and begins to soften the material as it is being conveyed. The root diameter is the smallest here and is constant. Since the root diameter is constant, the channel depths are also constant; they are also called the feed depth. In the feed zone, the material is picked up and is softened as it is conveyed by the rotation of the screw. The material must never be completely molten in this section because that would prohibit the picking up of additional material. A term commonly used to describe this phenomenon is **screw slipping**, where the melt rotates with the screw and prohibits the screw from moving back to pick up more material and build the next injection shot.

Transition Zone (Compression Zone): In this section, the root diameter increases gradually, resulting in the decrease of the channel depth. At the start of this section, the root diameter is the same as the root diameter of the feed section, where it gradually increases until the section ends. This causes the feed depth to steadily decrease. As the screw rotates, the softened pellets begin to get compressed and the air and any other volatiles are forced out from between them because the feed depth is decreasing and the plastic is being conveyed. With the help of heat from the external heater bands and the shear from the rotation of the screw, the plastic begins to melt. As the feed depth reduces because of dispersive and distributive mixing, the plastic ends up as a homogeneous melt by the time it reaches the end of the transition zone.

Metering Zone: The metering zone is the last zone and is the closest to the nozzle of the machine. The depth of the channels in this section is minimal compared to the other two sections. The root diameter stays constant and therefore the channel depth is also constant. Since the shot is built by moving the screw back until it reaches a set linear position (shot size), the metering depth must be as minimum as possible to reduce the variation in the amount of melt for each consecutive shot. With a larger metering depth, the amount of material that is fed in front of the screw can vary, leading to inconsistencies. However, as the depth reduces, the shear increases and therefore the risk of material degradation is also increasing, especially for shear sensitive materials such as PVC. A compromise must be found and special screw designs are therefore necessary for certain types of materials. Figure 1.12 shows the melting progression of the plastic as it travels through each of these sections. In the feed zone, the pellets have softened and begin to adhere to each other. When they travel to the transition zone, there is a combination of melted and un-melted plastic. There is still evidence of plastic pellets that have been compressed together. The metering zone shows a ribbon of completely molten plastic. In a GP screw, the length of the metering and transition zones are the same and the feed zone is usually twice the length of any of one these sections. In custom designed screws, these lengths can be altered. Longer feed zones increase the throughputs, longer transitions decrease shear, and a longer metering section will output a more homogeneous melt but will create more shear.

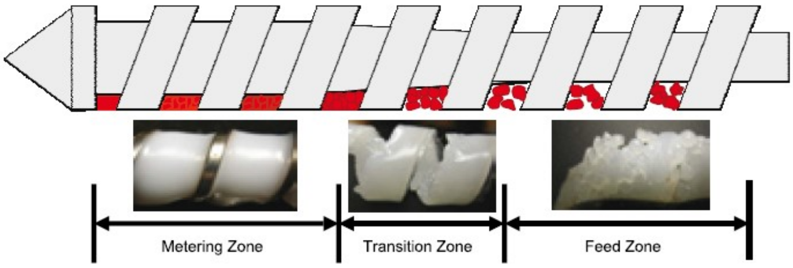


Figure 1.12 Melting progression of the plastic as it travels through the sections of the screw

Compression Ratio: This is the ratio of the feed section channel depth to the metering section channel depth. It defines the amount of compression to which the material has been subjected. The higher the compression ratio, the better is the melt homogeneity, but also the higher is the shear. The depth of the channels also contributes to the amount of shear heat, melt homogeneity, and the throughput. Typical compression ratios are mentioned below:

1. Low compression ratio: 1.5 : 1 to 2.5 : 1 used for shear sensitive materials such as PVC
2. Medium compression ratio: 2.5 : 1 to 3.0 : 1 used in general purpose materials
3. High compression ratios: 3.0 : 1 to 5.0 : 1 used for crystalline materials such as nylons

L/D Ratio: The L/D ratio is the working length of the screw flight to the outside diameter of the screw. Most injection molding screws have L/D ratios of 20:1. Greater L/D ratios allow more exposure of the plastic to heat and shear, improving the melt homogeneity and therefore increasing throughput at the desired processing temperature [6].

An injection-molding screw is rotated using an electric drive motor coupled with a reducer or gear box or a direct hydraulic drive. Electric drive motors are usually employed with larger hydraulic machines [clamp force > 15,000 kN (1700 tons)] and in all-electric molding machines. The injection unit (or sled) retracts away from the mold along its rails. This permits purging of material from the barrel when changing the resin type used in the machine or getting rid of contaminated or degraded material. The injection unit's position is also adjustable to accommodate proper sprue bushing seat position for different mold bases and for different nozzles. The contact force of the injection unit prevents melt from leaking at the open interface between the nozzle and sprue bushing.

Shot size

During injection, the screw is driven forward by a hydraulic piston or electric motor. Melt is forced through the nozzle into the mold. After injection the screw travels back (refill) to a predetermined distance until a limit switch is reached, and this triggers a signal that stops the screw's motor. This gives the desired shot size. The plastic metering or shotsize indicator indicates the linear position of the screw.

This is used to set shot size.

The Check Valve

Screw tip has a counterbore that accepts a smear tip or nonreturn valve. During injection and holding, the nonreturn valve prevents melt from flowing back along the screw. There are two types of nonreturn valves:

1. **sliding rings and**
2. **ball check valves**

A sliding ring valve (Fig.1.12) is pushed forward during plastication and is forced backward as injection begins. This provides more streamlined flow, is less restrictive in terms of materials, and produces a smaller pressure drop. However, the movement of the sliding ring causes wear and permits material to leak, particularly when glass fibers become stuck under the ring. Sliding ring valves are used for ***high-viscosity materials and filled compounds and with vented injection molding.***

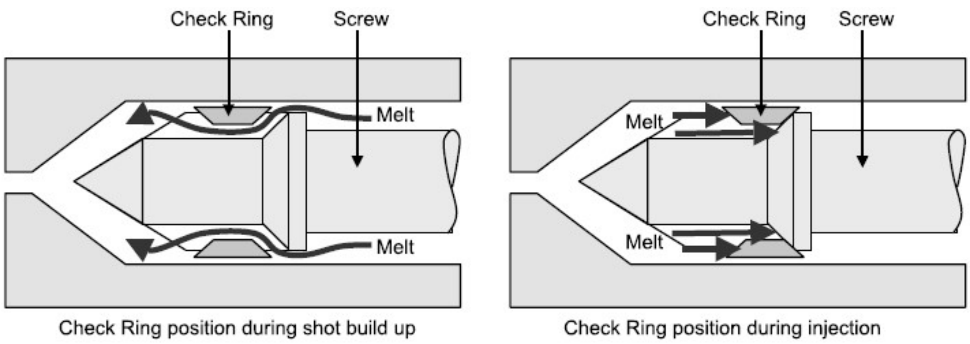


Figure 1.12 Working of a check ring

In a ball check valve (Fig. 1.13), a sphere is forced forward during plastication and backward for injection. While this produces more positive shutoff and better shot control, it is more restrictive to flow, provides a greater pressure drop, and causes more barrel wear than sliding rings. Ball check valves are available in front discharge and side charge designs. The front discharge valves are more difficult to clean and have a front angle that does not match the angle of the end cap or nozzle adapter. Side discharge valves eliminate these problems. Ball check valves are typically employed with **unfilled, low-viscosity materials.**

When no nonreturn valve is used, a smear tip (Fig.1.14) is added to the end of the screw. Although these do not restrict flow, they permit back flow. Smear tips are used with **high-viscosity and heat-sensitive materials such as rigid PVC.**

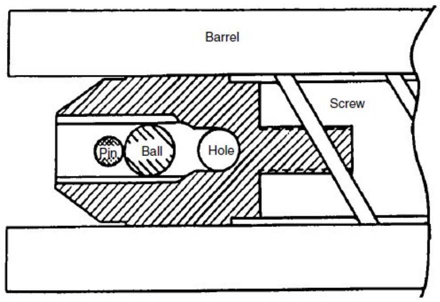


Figure 1.13 Nonreturn valves: ball check valve

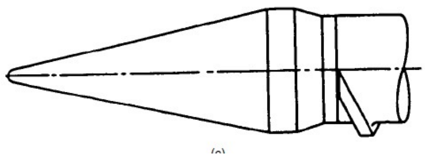
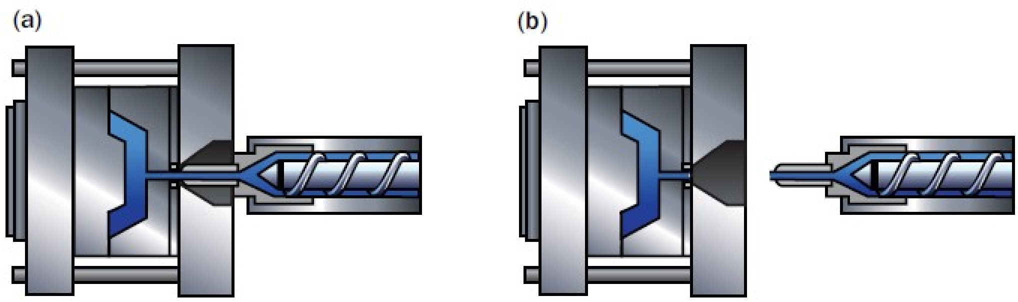


Figure 1.14 Nonreturn valves: smear tip

Nozzle is the tip of the plasticating unit and provides a leakproof connection from the barrel to the injection mold with minimum pressure loss. The radiused tip aligns the nozzle and the sprue bushing (smaller than the sprue bushing) of the mold. Nozzles are also long enough to have heater bands and require their own heating zone(s).



(a) Nozzle with barrel in processing position (b) Nozzle with barrel backed out for purging

There are three types of nozzles:

1. open channels,
2. internally actuated shutoff nozzles, and
3. externally actuated shutoff nozzles.

In the common design (open channel), no mechanical valve is placed between the barrel and mold. This permits the shortest nozzle and unimpeded flow of the polymer melt. With highly fluid plastics, such as polyamide-6,6, the nozzle diameter becomes smaller and then is enlarged again before reaching the sprue bushing; this prevents drooling (of melt through the nozzle).

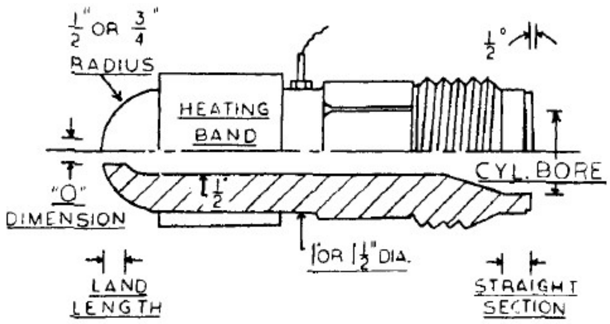


Fig. 1.15 Standard nozzle

Internally actuated shutoff nozzles are held closed by either an internal or an external spring. They are opened by the plastic injection pressure. Externally actuated shutoff nozzles are operated by external sources such as hydraulic or pneumatic pistons. While both types of shutoff nozzles are longer than open-channel nozzles, they eliminate drooling and permit plastication when the nozzle is not in contact with the sprue bushing. Special cut-off nozzles are also used for injection molding of foams and for gas-assisted injection molding [4].

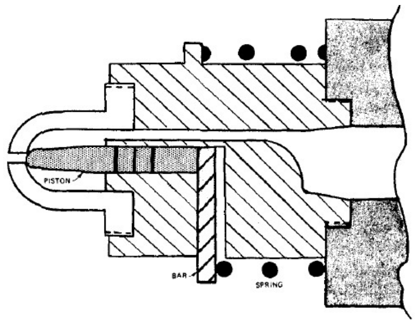


Fig. 1.16 Needle-type shutoff nozzle

Specification for injection units

Injection units are specified by

1. shot size,
2. maximum injection pressure,
3. plasticating capacity and recovery rate,
4. maximum injection velocity.

The **shot size** is the maximum weight or volume of plastic that can be injected in one shot. Injection unit sizes are specified by the maximum amount of plastic material they can dispense with one forward movement of the injection screw. For U.S. machines, the shot size is rated in ounces of general-purpose polystyrene (GPPS), whereas in European machines, the shot capacity is based on the volume (in cubic centimeters3) displaced with 100-MPa injection pressure. The shot volume (size) should be at least 10 to 15 percent of the maximum and no more than 75 to 90 percent of the maximum shot size. Smaller shot volumes provide longer residence times, thereby producing greater variations in melt viscosity and the amount of material delivered to the mold.

The **maximum injection pressure** is the maximum available pressure for injection. In hydraulic machines, the injection cylinders are rated for a maximum hydraulic pressure, typically 14 to 21 MPa (2000 to 3000 lb/in^2).

The **maximum injection velocity** is the maximum injection rate available in the machine; its units are in inches per second (millimeters per second). Standard machines typically have maximum injection velocities of about 150 to 250 mm/s (6 to 10 in/s), whereas thin-wall machines can reach 1500 mm/s (59 in/s).

The clamping unit

The clamping unit supports the mold, holds the mold closed during injection, opens and closes the mold as rapidly as possible, provides for part ejection, and provides mold close protection. The four types of clamps are: hydraulic, hydraulically actuated toggle (mechanical), electrically actuated toggle, and hydromechanical. All clamps have a stationary and moving (or movable) platen. Since the stationary platen supports the core or A side of the mold, it contains a hole through which the nozzle contacts the sprue bushing. Typically, the sprue bushing is surrounded by a locating ring that aligns the mold with the nozzle. The stationary platen is often water cooled when the mold has a hot manifold. While the moving platen supports the B side of the mold, it moves horizontally to open and close the mold, applies clamping force to the mold, and houses the ejector system. The two platens align the two halves of the mold, thereby minimizing wear on contacting surfaces. These platens are usually supported and aligned by four tie bars. The moving platen is also guided by these tie bars when it travels to open or close the mold. Since the forces that hold the mold closed also stretch the tie bars, tie-bar adjustments are used to periodically realign the platens. In some machines, the four tie bars are replaced with slides or two tie bars (bottom). This supports and aligns the platens from the bottom only and facilitates mold changes and part removal.

Hydraulic clamp unit

A conventional hydraulic clamp (Fig.1.19) has one large cylinder in the center of the movable platen with no mechanical advantage applied. Thus, hydraulic fluid and pressure open and close the clamp. To move the clamp forward, hydraulic fluid is directed to a booster tube or cylinder and the prefill valve is opened. As the booster cylinder moves the clamp forward, a slight vacuum in the main clamp cylinder pulls oil from the tank, through the prefill valve, and into chamber of the main clamp cylinder. Once the clamp is closed, the prefill valve is closed. This traps oil in the main cylinder area. When the clamp is pressurized, high-pressure fluid moves the main cylinder forward, thereby compressing the oil in the main cylinder. To open the clamp, hydraulic fluid is directed to the pull-back side of the booster cylinder while the prefill valve is open. The backward motion of the cylinder forces the fluid through the prefill valve and back into the tank. In hydraulic clamps, the clamp force is controlled by the pressure in the main cylinder. As a result, clamp force can be varied during the molding cycle. Typically, a higher clamp force is used during mold filling and packing while the force is reduced during cooling. The clamp force can be built up by increasing clamp pressure and maintaining maximum pressure using a high-pressure pump or a pressure pump intensifier or the clamp pressure can be raised with a high-pressure pump or a pressure intensifier and then maintained using a check valve in the return line. Die-height adjustment (setting the distance between the stationary and moving platens) occurs through travel of the main clamp cylinder.

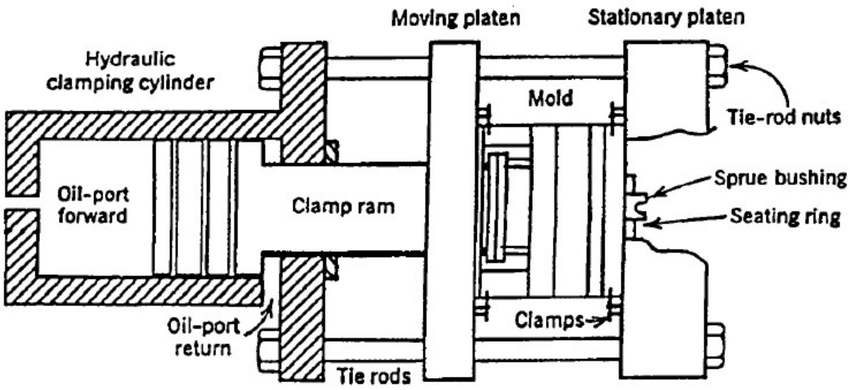


Figure 1.19 Hydraulic clamping unit

Toggle clamp unit

While a hydraulic clamp machine requires a very large cylinder in the center of the platen to apply full clamp tonnage with hydraulic pressure, hydraulically actuated toggle clamps (Fig.1.20 a.) use a small cylinder and a mechanical toggle. To close the clamp, the cylinder moves forward, extending the toggle links (Fig. 1.20 b). Clamp movement is rapid until the B side of the mold approaches the A side. At a predetermined distance, the clamp speed slows and continues its forward travel until the mold halves are joined. The low speed prevents damage to the mold, while the fully extended toggle links (Fig.1.20 b.) not the hydraulic cylinder, provide the clamping pressure. Retraction of the hydraulic cylinder opens the clamp. The opening stoke also starts slowly and then speeds up when the B side of the mold has cleared the leader (guide) pins on the A side of the mold. Hydraulically actuated toggle clamps are the most common type of clamp. Single toggles are used for small machines [clamp force ≤ 500 kN (55 tons)]. These require little space because the actuating cylinder is connected directly to the toggle with cross-links or it is pivoted at the tail stock platen (platen at the clamp cylinder end of the clamp unit) or machine support. Double toggles are employed when the clamp tonnage is about 1000 to 50,000 kN (~ 110 to 5620 tons). As shown in Fig. 1.20 most double toggles have one centrally located hydraulic cylinder. While double toggle clamps can have four- or five-point toggles, the five-point toggles are more common because they provide longer opening strokes and require less floor space. These clamps provide relatively rapid and constant platen speed for most of the platen travel, whereas the platen speed slows as the moving platen approaches the stationary platen. In toggle clamps, die-height adjustment employs a clamp-adjusting ring gear. This motor-driven gear rotates all four tie-bar adjusting nuts simultaneously, thereby producing linear movement of the moving platen. While procedures for die-height adjustment vary, the moving platen is typically backed away from the stationary platen during mold changes. Thus, when the clamp toggles are fully extended, the moving platen does not touch the mold. The ring gear is then rotated until the moving platen contacts with the mold. After the B side of the mold is attached to the moving platen, the clamp toggle is retracted and the mold opens. The ring gear is then adjusted until the desired clamp force is achieved when the toggles are locked. Since toggle clamping units provide a 50:1 mechanical advantage, they are activated by a smaller cylinder. As a result, they are faster by 10 to 20 percent, are less expensive to build and operate, and require less floor space and less energy. In hydraulic clamping units, the force is easily determined, and so clamp force is easily monitored and controlled. This also provides a constant clamping force, permits self-compensation for mold expansion, prevents platen deflection, and facilitates mold-close protection and die-height adjustment. However, hydraulic clamping units produce somewhat slower platen movement, are more expensive to manufacture and operate, and require more floor space. Toggles are subject to wear (around the toggle). Moreover, since the extended toggle determines the clamp force, it is not self-compensating for mold and tie-bar expansion and cannot correct platen deflection. It is also difficult to determine the exact clamping force; auxiliary equipment, such as strain gauges placed on the tie bars, dial indicators on the ends of the bars, and a strain gauge inside the tie bars, have been used to measure the exact clamp force.

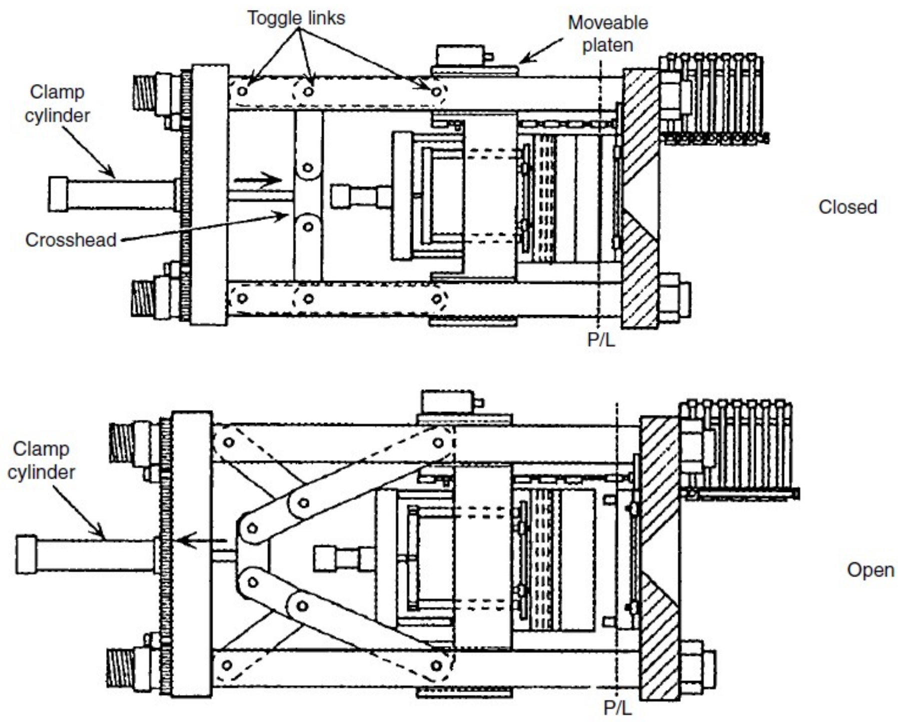


Figure 1.22 Toggle clamping unit: (a) closed and (b) open.

Hydromechanical clamping units

In hydromechanical clamping units, toggles are combined with hydraulic cylinders. The toggle is used to open and close the clamp, but the hydraulic piston builds the clamp pressure. Since this requires small hydraulic cylinders, clamp movement is faster and the clamping units are smaller than comparable hydraulic units. However, hydraulic clamping provides better control of the clamp force.

Electric clamp unit

An all-electric machine has no hydraulic pumps. The toggles in the clamping unit are extended and retracted by a servomotor and a reduction drive gear is used to achieve the required forces. The clamps are much more stable since they have no hydraulic system to generate heat and the servomotors provide extremely accurate movement of the machine components. It is also much cleaner to operate than the other types of molding machines. All-electric machines are the machine of choice for most medical products.

Ejection system

At the end of the molding cycle, the mold opens and cooled parts are ejected from the injection mold. This requires the ejection system shown in Fig. 1.21. When the mold opens, the plastic part typically stays with the B side of the mold. Once the mold is opened, the hydraulic ejection cylinder extends, forcing the ejection platen forward (in all-electric machines, the cylinders are replaced by servomotors). Since the platen is usually tied to the mold's ejector plate by ejector rods, the ejector plate also moves forward. This forces the ejector pins (in the mold) forward and they, in turn, push the plastic part out of the mold.

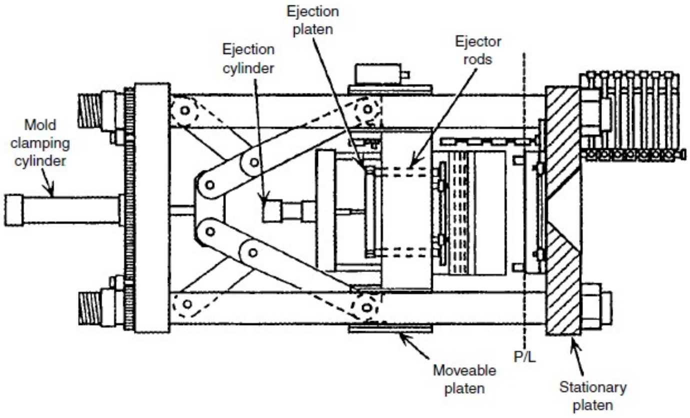


Figure 1.21 Ejection system

Clamp specification

The primary specification for a clamping unit is the *maximum clamp force,* the force needed to keep the mold closed during injection and packing. If the clamp force is not greater than the injection and packing forces, the mold can be forced open during injection or packing. This results in leakage of the molten polymer or flash. Clamp force Fc can be estimated from

$$Fc = f\eta A$$

where $f\eta$ is an empirical pressure or viscosity factor and A is the projected area of the part (i.e., the area perpendicular to the sprue). For easy flow materials, $f\eta$ is 2 to 3 tons/in^2, while for viscous polymers, $f\eta$ is 4 to 5 tons/in^2. The viscosity factor can be replaced by the actual or estimated peak cavity pressure, Pmax, to provide another estimate of clamp force:

$$Fc = P_{\max}A$$

Other clamp specifications (Fig. 1.22) include the maximum daylight, clamp stroke, maximum and minimum mold height, distance between tie rods, tie-rod diameter, platen size, clamp speeds, ejector stroke, and ejector force.

The **maximum daylight** is the maximum distance between the stationary and moving platens when the clamp is fully retracted. In hydraulic clamping units, this distance, minus the distance required for part ejection, determines the maximum depth for the mold. The mold opening must be at least twice the height of the core pin or the part cannot be ejected. With toggle machines, the extension of the toggle provides the clamp stroke (stroke is not specified for hydraulic clamps). Since the stroke limits the depth of the mold that can be mounted in the clamp, toggle machines specify a maximum mold height.

The **minimum closed daylight, shut height,** or **mold height** is the minimum distance between platens when the clamp is closed. This dictates the shortest mold that may be run in the injection-molding machine. When the mold height is smaller than the minimum closed daylight, a spacer bar mounted to the platen provides a shorter minimum shut height. The distance between tie rods and tie-rod diameter limits the size of the mold that may be mounted in the clamp unit. While molds are typically hung vertically between the tie bars, small molds may be mounted horizontally. This restriction to mold size does not occur with tie-bar-less machines, but the mold size is limited by the platen size. If a mold is too small, the platens will bend, thereby damaging the clamping unit. Thus, the minimum mold size is typically one-half the distance between the tie rods, both vertically (*V*) and horizontally (*H*). Although clamp opening and closing speeds contribute to overall cycle time, they are more important for thin-wall and other high-speed molding [4].

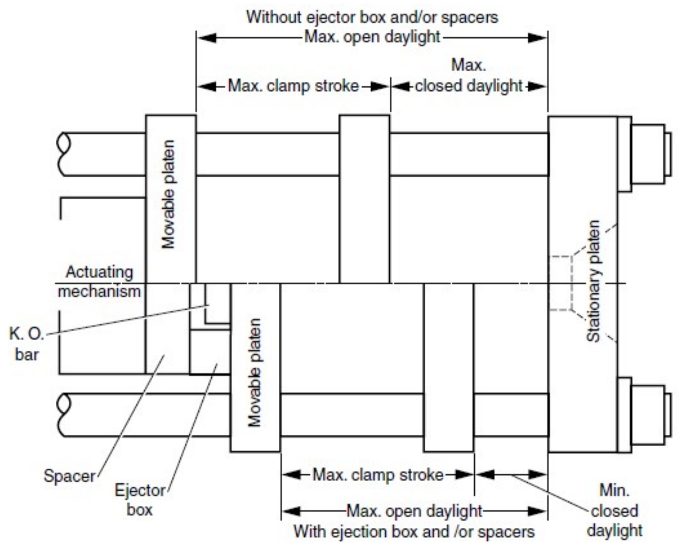


Figure 1.22 Specifications of a clamping unit

Orientation in injection moulding and its effect

Orientation effects are very important. The term orientation means the alignment of the molecule and molecular segments in the direction of flow. The strength in the direction of flow is greater than perpendicular to the line of flow. Thus plastic that is oriented will be stronger in the direction of flow than perpendicular to it. As no material orients completely; but the greater the orientation is, the closer the material. The second major implication is that the oriented plastic will shrink more in the direction of flow than it will perpendicular to it. Shrinkage is a result of two factors-a normal decrease in volume due to temperature change and relaxation of the stretching caused by carbon-carbon linkages. As there are more carbon-carbon linkages in the direction of the oriented flow than perpendicular to it, this phenomenon occurs. Plastics do not all exhibit orientation to the same degree. Consider molding a rectangular plaque of clear polystyrene 2 inches wide and 6 inches long, 0.090 inch thick and gated on the 2-inch end. If the molding were held between crossed Polaroid filters, a colored pattern would be seen. This property is called birefringence and is used to measure orientation. The material front that flows past the gate is randomized, and freezes as such on the walls of the cavity. This section is totally unoriented. However, one end of the molecule is anchored to the wall, and the flow of other material past it pulls the other end of the molecule in its direction, giving a maximum amount of orientation. As the part cools, the orientation is frozen at the walls. The center of the section remains warm for the longest time, allowing Brownian motion to disorient many of its molecules. Therefore, the center section is the least oriented. This is shown by birefringence patterns. This behavior can be easily demonstrated by milling off one-third (0.030 inch) of the thickness. In effect, one section is highly oriented, and the center section, which has been exposed by the milling, is less oriented. If the milled piece is heated, the stretched carbon-carbon linkages should return to their normal position. Because the oriented section has the carbon-carbon linkages lined up more in one direction than they are in the less oriented sections, that part should shrink more. In effect, then, it would be acting as a bimetallic unit, one side shrinking more than the other, and the piece should bend over. This is what happens. As the amount of orientation depends on the flow and on the forces that aid or prevent the motion of the molecular segments, it is easy to see what conditions can affect orientation. Anything that increases the mobility of the segments decreases orientation. Therefore, higher material temperatures, higher mold temperatures, and slower cooling would decrease orientation. Pressure on the material would limit mobility. Thus, low injection pressures and a short ram forward time decrease orientation. The use of a thicker part would decrease orientation because a longer time is needed for the center portion to cool with increasing thickness [3].

Processing parameters and their effect on product quality - Importance of Process Conditions

The quality of the molded part is greatly influenced by the conditions under which it is processed. For example, the process window shown in Figure 1.23. As the temperature is lower, higher pressure is needed to deliver the polymer melt into the cavity. If the temperature is too high, there is a risk of causing material degradation. If the injection pressure is too low, a short shot could result. If the pressure is too high, there will be flash in the molded compounds.

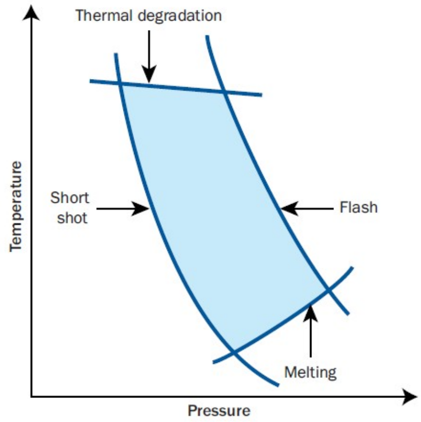


Fig 1.23 Process window showing the influence of pressure versus temperature

Setting Machine Process Conditions

Before setting process conditions, you should make certain the molding machine is in proper working order, and that the mold you plan to use was designed for the particular machine you plan to use. Follow the step-by-step procedure provided below to control the settings on your machine.

Step 1— Set the Melt Temperature

Melt temperature is one of the most important factors in molding plastic parts. If it is too low, the resin might not be completely melted or it might be too sticky to flow. If the melt temperature is too high, the resin could degrade, especially if the resin is POM or PVC. Suggested melt and mold temperatures for specific materials are shown in the below table.

Setting Heater Band Temperatures: Most melting of the resin occurs because of the frictional heating from the screw rotation inside the barrel. The barrel heater bands serve mainly to keep the resin at the appropriate temperature. Typically there are three to five temperature zones or heater bands on the cylinder. The rules for setting the heater band temperatures are as follows:

- The last temperature zone, nearest the hopper, should be about 40 to 50ºC lower than the calculated melt temperature to give better transport of plastic pellets during plasticization.
- The heater band at the nozzle zone should be set to the calculated melt temperature, and should keep the temperature uniform. Improper heater band temperature settings may cause drooling at the nozzle, and degradation or color change, especially for PA materials.

Table 1: Typical melt and mold temperatures for various generic classes of resins

Generic Name	Melt Temperature (C/F)			Mold Temperature (C/F)			Ejection Temp (C/F)
	Min.	Rec.	Max.	Min.	Rec.	Max.	Rec.
ABS	200/392	230/446	280/536	25/77	50/122	80/176	88/190
PA 12	230/446	255/491	300/572	30/86	80/176	110/230	135/275
PA 6	230/446	255/491	300/572	70/158	85/185	110/230	133/271
PA 66	260/500	280/536	320/608	70/158	80/176	110/230	158/316
PBT	220/428	250/482	280/536	15/60	60/140	80/176	125/257
PC	260/500	305/581	340/644	70/158	95/203	120/248	127/261
PC/ABS	230/446	265/509	300/572	50/122	75/167	100/212	117/243
PC/PBT	250/482	265/509	280/536	40/104	60/140	85/185	125/257
HDPE	180/356	220/428	280/536	20/68	40/104	95/203	100/212
LDPE	180/356	220/428	280/536	20/68	40/104	70/158	80/176
PEI	340/644	400/752	440/824	70/158	140/284	175/347	191/376
PET	265/509	270/518	290/554	80/176	100/212	120/248	150/302
PETG	220/428	255/491	290/554	10/50	15/60	30/86	59/137
PMMA	240/464	250/482	280/536	35/90	60/140	80/176	85/185
POM	180/356	210/410	235/455	50/122	70/158	105/221	118/244
PP	200/392	230/446	280/536	20/68	50/122	80/176	93/199
PPE/PPO	240/464	280/536	320/608	60/140	80/176	110/230	128/262
PS	180/356	230/446	280/536	20/68	50/122	70/158	80/176
PVC	160/320	190/374	220/428	20/68	40/104	70/158	75/167
SAN	200/392	230/446	270/518	40/104	60/140	80/176	5/185

Air-shot Temperature

The actual melt temperature, or air-shot temperature, is usually higher than the heater band controller setting. This difference is because of the influence of back pressure and screw rotation on frictional heating and the melt temperature, as mentioned above. (You can measure the actual melt temperature by quickly sticking a probe thermometer into an air shot with the nozzle backed away from the mold.)

Step 2—Set the Mold Temperature

Suggested melt and mold temperatures for specific materials are available from the resin supplier. Appropriate melt and mold temperatures for many generic, base resins are listed in Table 1. The mold temperature can be measured by using a thermometer. As illustrated in Figure 1.24, the average cavity surface temperature will be higher than the temperature of the coolant during production. Thus, you should set the coolant temperature to be 10 to 20ºC lower than the required mold temperature. If the mold temperature is 40 to 50ºC or more, consider insulation plates between the mold and the clamping plates for energy savings and process stabilization.

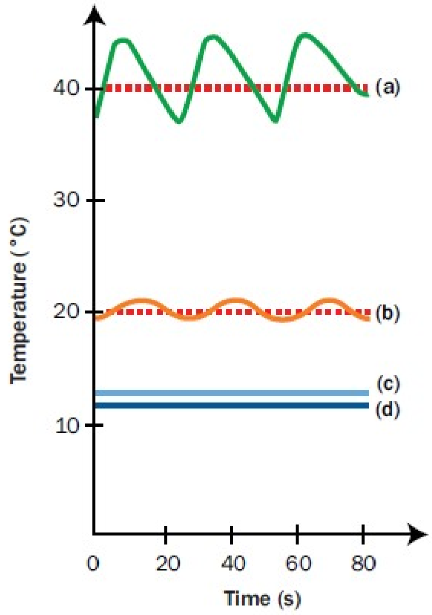


Fig 1.24 Temperature-time curve at various locations in the mold (a) Mold cavity surface (b) Cooling channel wall (c) Cooling channel outlet (d) Cooling channel inlet

Use the lowest temperature setting to achieve the shortest cycle time. However, you might try using higher temperatures to improve the appearance of the part. A higher mold temperature produces a higher gloss and more crystallization.

Considering Temperature Difference: For parts with a deep core, a lower coolant temperature is needed for the core (moving plate) in order to minimize the temperature difference between the mold surfaces on the core and cavity. A lower surface temperature difference will produce parts with higher quality, at a lower cost. By a rule of thumb, the coolant temperature for fixed and moving plates should not differ by more than 20ºC. This is related to thermal expansion, which can be determined only by the user. A large temperature difference results in differential mold plate thermal expansion, which may cause alignment problems in guide pins, especially in large molds. The mold will sometimes lock up for this reason. The cycle time can be increased to reduce the required coolant temperature difference.

Step 3—Set the Switch-over Position

The switch-over position is the ram position where the velocity controlled filling (injection) stage switches to the pressure controlled packing phase. Once the ram is under pressure control, the ram continues to move forward to pressurize the cavity and to compensate for shrinkage. The cushion is the distance from the ram position at the end of the packing phase to the farthest position that the end of the screw can reach, as shown in Figure 1.25. The typical cushion distance is about 3 to 10 mm (1/8 to 3/8 in). At this step, set the switch-over position to fill about two-thirds of the mold. This prevents damage to the press or the mold.

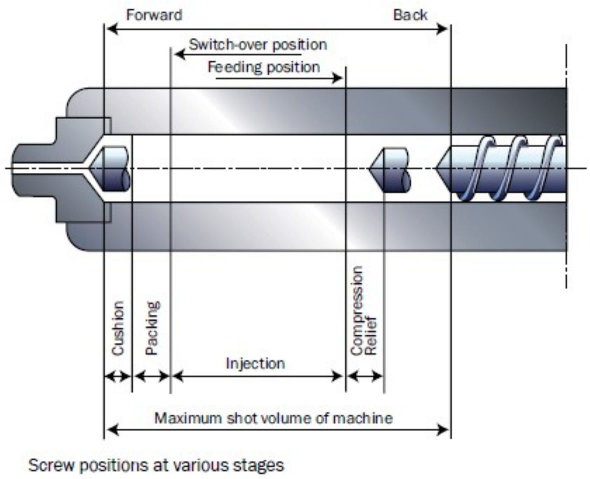


Fig. 1.25 Screw positions at various stages

Step 4—Set the Screw Rotation Speed

Set the screw rotation speed to the level required to plasticize the resin. Plasticizing should not prolong the cycle time. If it does, increase the speed. The ideal speed causes plasticizing to complete at the latest possible point in the cycle without exceeding cooling time and prolonging the cycle time. Resin vendors supply the suggested screw rotation speed for specific resins.

Step 5—Set the Back Pressure

Back pressure is the amount of pressure exerted on the material volume ahead of the screw as the screw is pushed back in preparation for the next shot. The recommended back pressure is about 5 to 10 MPa. Back pressure that is too low can result in inconsistent parts. Increasing the back pressure will increase the frictional contribution to the melt temperature and decrease the plasticization time. To speed up plasticization, use a higher back pressure to achieve a shot volume that is a larger percentage of the injection machine's capacity. Use a lower back pressure for a smaller percentage shot volume because the material will remain in the barrel longer (for many cycles) before it reaches the screw head.

Step 6—Set the Injection Pressure to the Machine Maximum

The injection pressure is the pressure of the melt in front of the screw. The injection pressure should be as low as possible to reduce part internal stress. On the machine, set the injection pressure to nearly the machine maximum. The purpose is to completely exploit the injection velocity of the machine, so that the pressure setting valve does not limit the velocity. Because the switch-over to packing pressure occurs before the mold is completely filled, no damage will be done to the mold.

Step 7—Set the Packing Pressure at 0 Mpa

For now, set the packing pressure at 0 MPa, so the screw will stop when it reaches the switchover position. This will prevent mold or press damage. In later stage the packing pressure is increased to its final setting.

Step 8—Set the Injection Velocity to the Machine Maximum

Injection Speed

The injection (or ram) speed is the forward speed of the screw during its injection operation. Setting the Injection Speed For most engineering resins, the ram speed should be set to the fastest setting that the part design and process will allow for technical and economic reasons. However, slower injection speed at the beginning of injection may be necessary to avoid turbulent flow and jetting, as material passes through the restrictive areas (e.g. gates). The injection speed should be reduced again toward the end of injection to avoid flashing at the end of stroke, and to enhance the formation of homogenous weld lines after a divided flow. With the highest possible injection velocity within shear rate limits, you can expect less flow resistance, longer flow length, and improved strength in weld lines. However, you may need to create additional vents once you do this.

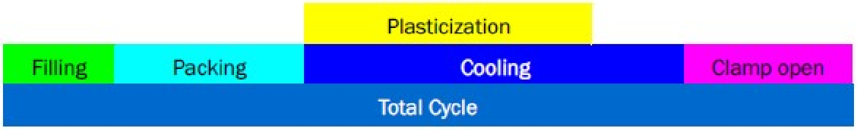
Proper Venting Minimizes Defects: Insufficient venting causes compression of air trapped in the cavity. This results in very high temperatures and pressures in the cavity, causing burn marks, material degradation, and short shots. You should design a venting system to avoid or minimize the defects caused by trapped air in the mold. Moldflow shows you where weld lines, meld lines, and air trap locations will occur: use these predictions to improve your design. Remember that it is necessary to clean the mold surface and venting system regularly, especially for PVC or ABS/PVC materials.

Step 9—Set the Packing Time

The ideal packing time setting is the gate freezing (sealing) time or the part freezing time, whichever is shorter. The gate and part freezing times can be calculated or estimated. The calculated values for the packing time are based on packing analysis results when the frozen layer fraction is 1.0 for the gate. Without packing analysis results, the packing time is estimated to be 10 times the filling time.

Step 10 — Set Ample Cooling Time

Cooling time can be calculated or estimated. The cooling time is after packing time, as shown in Figure 1.26. During cooling the part continues to solidify so it can be ejected, and material for the next shot is prepared. The calculated value of cooling time is from a cooling or packing analysis. Without Moldflow results, the cooling time can be 10 times the filling time. For example, if the predicted filling time is 0.85 seconds, the initial cooling time would be 8.5 seconds. The combination of packing time (if estimated would be 8.5 seconds) and the cooling time should be a high estimate to ensure the part and runner system will be sufficiently solid for ejection.



1.26 Cycle time and its components

Step 11 — Set the Mold Opening Stroke

The mold opening stroke is comprised of the core height, the part height, and the capsize space, as shown in Figure 1.27. You should minimize the mold opening stroke. The mold opening speed should be slow at the very beginning, then accelerate, then slow down again at the end of the stroke. The sequence of the mold closing speed is similar to the mold opening speed: slow, fast, slow.

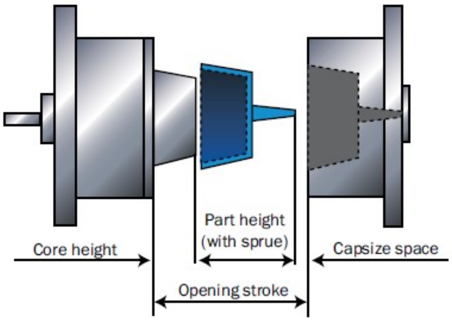


Fig 1.27 Required mold opening

Step 12 — Set the Ejector Stroke, Start Position, and Velocity

Relieve any slides first. The ejector travel should be, at a maximum, the core height. If the machine is equipped with a hydraulic ejector, set the start position at the point where the part is clear of stationary mold parts. (When the ejector velocity is equal to the opening speed, the part remains where it was in relation to the stationary mold part.)

Step 13—Set the Mold Open Time

The mold open time is usually set at 2 to 5 seconds. This includes mold opening, ejection of parts from the mold, then mold closing, as shown in Figure 1.26. The cycle time is the sum of the filling time, cooling time, and mold open time.

Step 14—Mold a Short-shot Series by Increasing Injection Volume

Moldflow provides the part weight and sprue/runner/gate weight. From this information, along with the screw diameter or barrel inner diameter, the total injection volume and the feeding position (see Figure 1.25) can be estimated for each shot. For now, fill only two-thirds of the mold. The packing pressure should already be set at 0 MPa, so that mold filling stops when the screw reaches the switch-over position, thus protecting the mold structure and the press. Next, increase the volume in increments of 5 to 10%, up to 95% of mold filling. In order to prevent material from escaping from the open nozzle, relieve the back pressure created during plasticizing by drawing back the screw a few millimeters, immediately after the rotation has stopped.

Step 15—Switch to Automatic Operation

The purpose of an automatic operation is to obtain stability in the process.

Step 16—Set the Injection Volume to 99% Mold Filled

When the process has stabilized (when the same parts are produced each time), adjust the switch-over position to 99% of filling. This will exploit the maximum injection speed in as large a part of the injection as possible.

Step 17—Increase the Packing Pressure in Steps

Increase the packing pressure in steps of approximately 10 MPa in the melt. If the first step does not fill the mold completely, increase the injection volume. De-mold and remove the part. Write the packing pressure on it. This packing pressure series forms a good basis for a more thorough examination. You can then discuss the possibilities and limitations with the customer. Choose the lowest acceptable packing pressure, as this minimizes the internal stresses in the part and saves material, as well as operating costs. A high packing pressure can cause excessive residual stresses that could warp the part. Molded-in residual stresses can be released somewhat by annealing at around 10ºC below the heat deflection temperature. If the material cushion is completely used (see Figure 1.25), the last part of the packing pressure time will not be effective. This calls for a change in the injection stroke position, in order to increase the injection volume.

Calculating Injection Pressure: The hydraulic pressure in the injection cylinder can be read on the machine manometer. However, the injection pressure in front of the screw is more important. To calculate the injection pressure you will need to multiply the Intensification ratio (hydraulic pressure by the resin/hydraulic pressure ratio). This ratio is usually found on the molding machine near the injection unit or in the instruction manual for the machine. The ratio is usually in the range of 7 to 15, as shown in Figure 1.27.

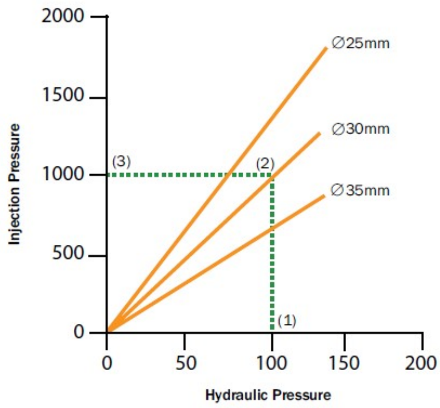


Fig 1.27 Intensification ratio for a 30 mm screw is 11.1

Step 18—Minimize the Packing Time

If consistent part dimensions are essential, the following method of packing time determiniatin is more accurate. The gate seal time must be determined. The gate seal time can be determined experimentally on the molding machine with a gate seal study. This involves initially starting with a long packing time and reducing the pack time until the part weights begin to change. This is an indication that the gate is open. When the packing time is decreased, the cooling time should increase the same amount to maintain the same cycle time. For example, Figure 1.28 shows that the packing pressure does not influence the part weight after 9 seconds. This is your minimum packing time.

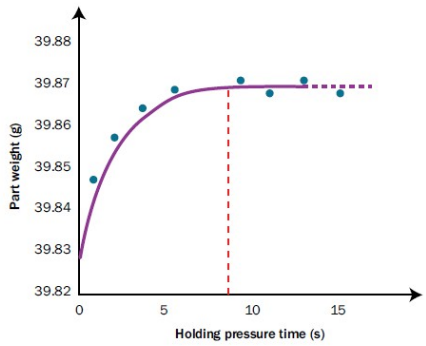


Fig 1.28 Determination of the gate/part freezing time by weighing parts manufactured at various packing times.

Step 19—Minimize the Remaining Cooling Time

Reduce the remaining cooling time until the maximum surface temperature of the part reaches the heat deflection temperature of the material. The heat deflection temperature can be provided by the resin supplier [5].

Control system

The control system controls the sequence of operation of the molding cycle (sequence control) and maintains the process temperature, time, pressure, and speed at the required values (process control). The three generations of control systems are

- electrical relays,
- programmable logic controllers (PLC)
- computer control

Electrical relays

In an electrical relay control system, an electrical relay starts the next operation based on the state of limit switches and timers. Limit switches are actuated by cylinder movement. Pressure, speed and time are set manually at pressure relief valves, flow control valves and analog timers respectively. Temperature is controlled by individual analog temperature controllers.

PLC

In a PLC control system, limit switches are replaced by the more reliable proximity sensors which are magnetically activated as cylinders move. They feed into the inputs of the PLC, the outputs of which are controlled by a program. Such programs are usually written in a language called ladder diagram. Inputs regarding pressure, speed and time are set digitally, switches, or the more advanced (and simpler) membrane switches (like the keys of some calculators) with LED outputs, and latest by touch screen. Hydraulic pressure and cylinder speed are controlled by proportional pressure and flow valves, which in turn are controlled by the PLC through amplifiers. Time control is integral to the PLC. Temperature is controlled by individual digital temperature controllers. Digital settings make for good repeatability.

Computer control

A computer control system contains a CPU, which is usually based on the open design compatible with that found in a personal computer. Like a personal computer, the display is either a cathode ray tube (CRT) or a liquid crystal display (LCD). Display of all input settings and outputs are done by flipping through screens. Magnetic card and/or diskette are used to retain the settings of a particular job, facilitating repeat of the same job at a later time. In a computer control system, potentiometers or encoders are used to measure cylinder movements. This eliminates the multitude of proximity switches and their inherent manual settings on sliders. Settings are now done through input at the keyboard. Temperature, time, pressure and speed control are done by the computer.

Temperature control

Temperature controllers are classified by the complexity of their control algorithm, which ranges from the simplest on-off control, through

1. proportional control,
2. PD (proportional plus derivative) control,
3. PID (proportional plus integral plus derivative) control and more recently the
4. PID fuzzy logic control.

The more sophisticated the algorithm, the faster and the closer the actual temperature to the desired value will be attained. Common to all types of controllers is the measurement of temperature by thermocouples. Thermocouple of type K is widely used since it measures a temperature range of up to 400°C which covers most of the thermoplastic melting temperatures. Temperature controllers are either analog or digital. In an analog temperature controller, the set temperature is input via a dial, and the temperature deviation is shown via a meter. In a digital temperature controller, both inputs and outputs are done digitally. A/D (analog to digital) converter converts the amplified analog thermocouple signal before it is processed.

Cooling control

The mould, the barrel throat and the hydraulic oil need to be cooled. The simple method is by running water through coils and controlling the flow rate using flow control valves. Flow meters are available for monitoring. The running water is in turn cooled at a water tower and is re-circulated. The mould is cooled so the melt in the cavity solidifies in a reasonable amount of time. In applications where cycle time is critical, the cooling period could be shortened by running chilled water through the mould. The temperature controller for the chiller is in a separate unit from the injection molding machine. The barrel throat is cooled so plastic pellets remains solid to be conveyed forward in the feeding section of the screw. Molten plastic sticks to the screw and does not move forward. Hydraulic oil needs to be kept below 46°C to reduce leakage at the rings, and to avoid deterioration of the oil and the rings. Injection molding machines capable of handling thermosetting plastics need to have active cooling control on top of active heating control at the barrel. The cooling avoids the plastic from reaching thermosetting temperature within the barrel, destroying it from further functioning.

Closed loop control

Nowadays, temperature control in injection molding machines is closed loop. It means the actual temperature is measured, compared to the desired temperature and the appropriate control action is taken to remove any deviation in an optimal fashion. Closed loop control removes the effects of all disturbances to the target variable attaining the desired value. As a result, the molded part is repeatable from one shot to the next. Mould position and screw position are measured by limit/proximity switches or by potentiometers or encoders. As a result, position measurement is also closed loop. However, more often than not, pressure and speed control are open loop for cost reasons. Three types of pressures could be measured. In increasing order of difficulties are hydraulic pressure, melt pressure at the nozzle and cavity pressure.

If hydraulic pressure is measured, the pressure sensor is located close to the injection cylinders as accurate injection pressure is to be controlled during injection. As hydraulic oil is about 40°C and the pressure 140 bars, the environment is not demanding.

Nozzle pressure sensor works in a more demanding environment. The melt is from 200 to 400°C and pressure is often at 1400 bars. Furthermore the sensor surface must be flush with the nozzle inner surface to avoid stagnant materials from degrading through prolonged heating. Sometimes, a temperature sensor is also incorporated in the same housing. Together, temperature and pressure give the state of the melt. (Conventional nozzle temperature measurement measures the temperature of the nozzle, not the melt at the nozzle).

Cavity pressure sensor works in a similarly harsh environment. It is located inside a mould to detect when the cavity is filled so the injection speed could switch to holding pressure. The sensor actually touches the melt but must not leave a noticeable mark on the article. Such sensor is often the very expensive quartz type which needs an expensive charge amplifier for conditioning before the signal could be read by a computer. Furthermore, being part of the mould, such sensor is usually not removed for reuse in another mould. With a sensor on each tiebar, **tiebar tension measurement** can indicate the tiebars are not evenly strained due to unparallel mould faces and the non-symmetrical cavity. A more economical setup has sensors on two diagonal tiebars. Tiebar tension measurement prevents tiebar and platen breakage which is costly in terms of downtime.

In a toggle clamp machine, tiebar tension control sets and maintains the clamping force of the machine when increase in mould temperature expands the mould and increase the clamping force. Mould height adjustment is called upon to reduce the clamping force to its set value. With tiebar tension control, the clamping tonnage could be set to below its maximum. Assuming the tiebars are evenly stressed, this could be accomplished using a sensor on only one tiebar.

The **speed of the screw** could be measured and fedback during injection. This is usually done by measuring the displacement of the screw in fixed intervals of time. The **back pressure** could be measured by a hydraulic oil pressure sensor at the back of the injection cylinder. **Screw rotational speed** is measured by a rotary encoder and converted to screw surface speed, which is what needs to be controlled.

Closed loop control requires proportional valves with fast response to take the control action. Such valves are called servovalves. Not only are they much more expensive, they also need a much cleaner oil to work with. This translates to higher filtration requirements.

Trouble shooting in injection moulding process

Attempting trouble shooting

1. When troubleshooting, it is important to remember that the first step is to identify the “true” issue.
2. When making processing changes, make one change at a time and allow two cycles, at a minimum, before moving on.
3. If the change made does not fix the problem, then you should put that setting back to its original setting before making the next change.

SHORT SHOT - Causes

1. Insufficiently-sized gates, runners, and thin walls.
2. Low melt and/or mold-wall temperatures.
3. Insufficient machine injection pressure (resulting from high melt resistance and a restricted flow path), volume, and/or ram speed.
4. Machine defects such as a worn non-return (check) valve that causes loss of injection pressure or leakage of injection volume.
5. Premature solidification of the polymer melt due to poor filling pattern, or prolonged injection time.

6. A lack of vents to bleed the air trapped inside the cavity.

REMEDIES

Increase the,

1. Dosing stroke
2. Injection speed
3. Injection pressure
4. Mould temperature
5. Melt temperature
6. Holding time/pressure

Check

1. Switchover Point
2. Venting
3. Ring Plunger set

Flash

Flash is a defect where excessive material is found at locations where the mold separates, notably the parting surface, movable core, vents, or venting ejector pins.

Causes

1. Low clamp force

If the clamp force of the injection machine is too weak to hold the mold plates together during the molding process, flash will occur.

2. Gap within the mold

Flash will occur if the parting surface does not contact completely, due to a deformed mold structure, parting surface defect, improper machine and mold set up, or flash or foreign material stuck on the parting surface.

3. Molding conditions

Improper molding conditions, such as a high melt temperature (which makes a thinner melt) or high injection pressure, will cause flash.

4. Improper venting

An improperly designed venting system, a very poor venting system, or a venting system that is too deep, will cause flash.

Remedies

Increase Clamp Force

Decrease

1. Holding pressure
2. Melt temperature
3. Injection speed / pressure

Check

1. Regrind ratio
2. Platen parallelism
3. Mould parting line/Parallelism
4. Conditions at Switchover point
5. Material grade

Sink marks and Voids

A *sink mark* is a local surface depression that typically occurs in moldings with thicker sections, or at locations above ribs, bosses, and internal fillets.

A *void* is a vacuum bubble in the core.

After the material on the outside has cooled and solidified, the core material starts to cool. Its shrinkage pulls the surface of the main wall inward, causing a sink mark.

If the skin is rigid enough, as in engineering resins, deformation of the skin may be replaced by formation of a void in the core

Causes

1. Low injection and packing pressure.
2. Short hold time or cooling time.
3. High melt temperature or mold temperature.
4. Localized geometric features.

Remedies

Increase

1. Holdon Pressure
2. Holdon time
3. Gate size
4. Back pressure

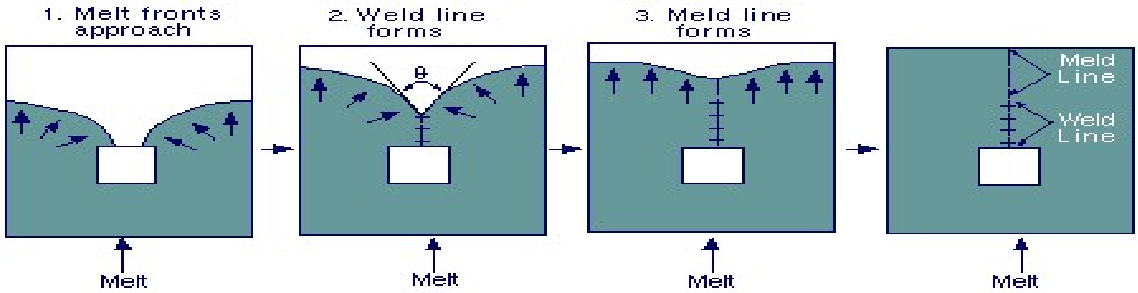
Decrease

1. Melt temperature
2. Mould temperature

Weld lines

A *weld line* (also called a weld mark or a knit line) is formed when separate melt fronts traveling in opposite directions meet.

A *meld line* occurs if two emerging melt fronts flow parallel to each other and create a bond between them. Weld and meld lines can be caused by holes or inserts in the part or due to multiple gates

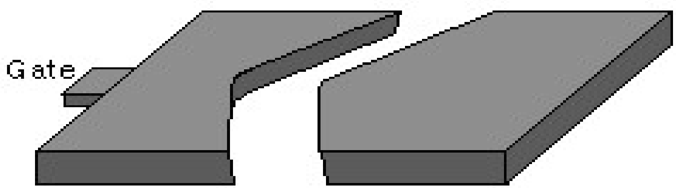


Increase

1. Injection speed
2. Mould temperature
3. Melt temperature
4. Holding pressure
5. Venting
6. Switchover point

Brittleness

A brittle molded part has a tendency to break or crack. Brittleness results from material degradation leading to shorter molecular chain length (thus lower molecular weight). As a result, the physical integrity of the part is substantially less than the specification.

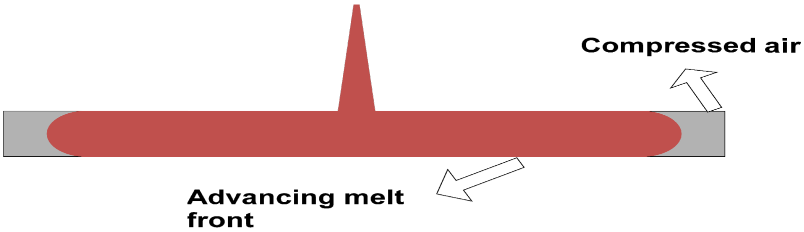


Causes

1. Melt temperature
2. Regrind ratio
3. Moisture
4. Screw speeds/ back pressures

5. Frozen-in stresses
6. Jetting
7. High injection speed at the gate point
8. Melt/mould too cold
9. Incorrect gate location

Burn Marks



Causes

High injection speeds

Poor venting

Black Specks

1. Dirty plasticising unit
2. Wear of screw / barrel
3. Granule / regrind impurities
4. Nozzle not matching with the mould

Delamination

Peeling off of surface layers

1. High injection speeds
2. Sharp corners
3. Contamination with other polymers

Flow lines

1. Low injection speed
2. Low injection pressure
3. Mould too cold

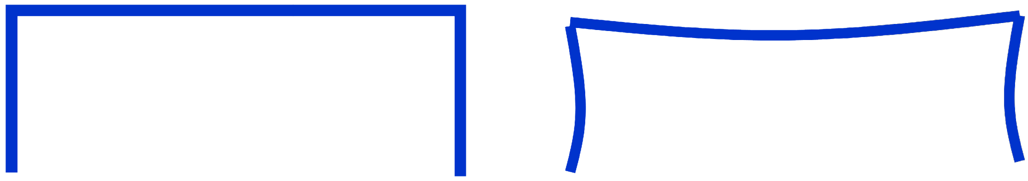
Low melt temperature

Silver streaks

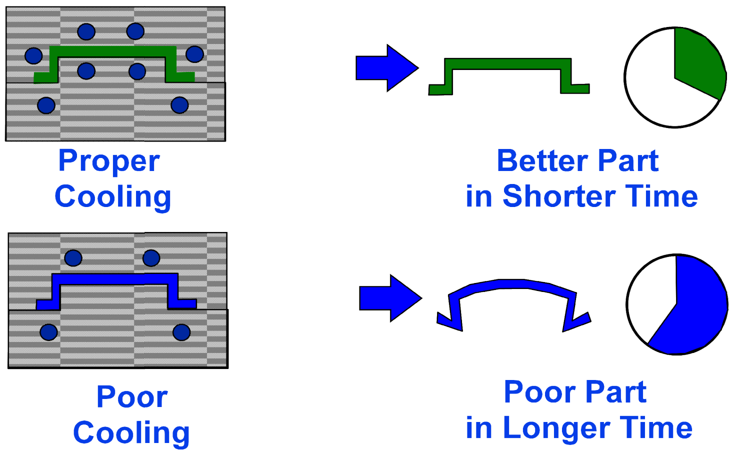
Streaky silvery appearance of the moulding nearly always radiating from the gate area

1. Moisture in the pellets
2. Volatiles due to over heating
3. Low back pressure
4. Under sized gate

Warpage



Warpage is the deformation of the part after ejection due to high internal stresses



1. Unbalanced cooling
2. Cavity pressure differences
3. High ejection temperatures
4. Low Holding time/pressure
5. Radial gating

Blisters

1. Low back pressure
2. Low packing pressure
3. Low cooling time

References

1. Musa R. Kamal, "Injection Molding Technology and Fundamentals", Hanser Publishers, Munich, 2009.
2. Dominick V. Rosato, Donald V. Rosato, Marlene G. Rosato, "Injection molding handbook - 3rd ed.", Springer Science+Business Media New York, 2000.

3. M.L.Berins “Plastic Engineering Handbook”, Society of Plastic Industries, Chapman & Hall NY 1991.
4. Charles A. Harper," Modern Plastics Handbook", McGraw-Hill, 2000.
5. Jay Shoemaker, "Moldflow Design Guide A Resource for Plastics Engineers", Moldflow Corporation Framingham, Massachusetts, U.S.A, 2006.
6. Suhas Kulkarni, “Robust Process Development and Scientific Molding", Hanser Publications, 2010.