Lubrizol LifeSciences

POLYMER SOLUTIONS

Injection Molding Guide

Lubrizol

Lubrizol LifeSciences

Injection Molding Guide

Contents:

Introduction

LifeScience LifeSciences is a business within The Lubrizol Corporation that provides innovative polymer solutions for healthcare products. Among the chemical technologies employed by LifeSciences are thermoplastic polyurethane elastomers (TPUs). LifeSciences TPUs are grouped into several families of products with varying chemical and physical characteristics. They can be readily converted to durable, tough, impact-resistant articles such medical device housings, tubing connectors, etc., on conventional thermoplastic injection-molding equipment.

This brochure is intended to provide general guidelines for equipment, procedures and molding machine conditions that will help the customer obtain the best possible performance from the the Lubrizol LifeSciences of TPU molding grades. Additional information can be obtained by consulting individual product Technical Data Sheets or by contacting your Lubrizol technical service or sales representative.

Materials

LifeSciences TPUs are available in both aromatic and aliphatic types. The choice of polyol raw material during manufacture leads to the families of polyester, polyether and polycarbonate grades. Separate literature is available describing detailed properties of the various molding grades. All of these TPU types have common characteristics as far as basic molding processing behavior and practices. This guide covers those general characteristics.

TPUs are generally not considered to be crystalline polymers that would be represented by a product like low-density polyethylene (PE). Some of the harder TPU grades (~> 90 Shore A) could be considered semicrystalline. Softer grades 62A–85A can be nearly amorphous — they have no definite melting point, but soften gradually on heating. Upon heating and shear mixing within the barrel of the molding machine, they achieve a viscous fluid state characteristic of a polymer much like low-density polyethylene.

Theoretically, TPUs can be injection-molded; but from a practical standpoint, many, particularly the softer grades (<70A), are not viable for practical, commercial use. Many of the softer types are nearly amorphous; consequently, molding would require long setup times. They could be very tacky and stick to the metal mold surfaces.

Melt viscosities suitable for molding will generally be lower than for typical extrusion processing. Temperatures and shear forces in the molding machine barrel will be higher than for extrusion. Some of the harder grades will have melt temperatures approaching 450°F/232°C. In general, 450°F/232°C is the onset of TPU thermal degradation. All LifeSciences TPU grades can be injection-molded below this temperature; individual Technical Data Sheets should be consulted and the recommendations followed. Products should be selected from the LifeSciences molding product sheets. If a specific set of end-use physical properties is found on on the LifeSciences TPU Extrusion list, it is very likely that the grade can be successfully injection-molded.
Molding conditions can be recommended by Lubrizol technical representatives.

The harder thermoplastic polyurethane TPU compounds are specifically designed for fast, easy injection-molding of large and small parts. By the nature of their chemical structure, the softer TPU compounds, in the Shore hardness range of 70A to 90A, will take longer to set-up (longer cycle times). Many of the grades are formulated with small amounts of additives to facilitate mold release and prevent blocking of molded parts. All TPU materials produce a very low-viscosity melt compared to other thermoplastics such as polyethylene. Polyethylene PE is a crystalline polymer that still has a relatively high melt viscosity when the crystalline structure “melts” it. This is due to the high molecular weight (length of polymer chain) of PE. PE can have molecular weights above 1,000,000. In contrast, when a harder (e.g.g. 50D) TPU melts, it goes to a low melt viscosity rapidly. This is due to the relatively low molecular weight of TPU. TPU will typically have molecular weights in the 100,000 to 200,000 range. Once melted, increasing the melt temperature generally does not reduce the melt viscosity significantly.

TPU melt viscosity easily fills the most complex mold cavities under low injection pressure. The low viscosity melt permits the material to flow through small gates and into thin wall sections under low injection pressure. Low-viscosity melt combined with low injection pressure minimizes the possibility of producing highly stressed parts.

Equipment Requirements

Type of Machine

Although all types of machines have been successfully used, a reciprocating-screw machine is preferred for molding LifeSciences TPU compounds. A reciprocating-screw machine is capable of producing the most uniform melt, is the most easily controlled, and is capable of the fastest cycles.

Machine Size

Barrel capacity

To obtain the widest processing latitude and optimum physical properties, an appropriate match of shot size, (i.e., volume of cavities, runners and sprue) to barrel capacity is very desirable.
A shot weight of 60% to 75% of barrel capacity is recommended.

This minimizes melt residence time in the barrel, enabling processing at higher stock temperatures with optimum melt flow while avoiding degradation.

Since the optimum match of barrel capacity is not always practical due to clamp requirements or machine availability, shot sizes as low as 30% to 35% may be used with the understanding that the processing latitude of the material may be significantly reduced. Lower stock temperatures mean higher melt viscosity and more resistance to flow. Greater injection pressures will be needed to fill the part, and molded-in stresses may result. It is likely that these molded-in stresses could adversely affect impact, dimensional stability and other properties of the finished part.

When calculating optimum barrel capacity, always consider the specific gravity of the TPU compounds versus the specific gravity of the material for which the machine was rated. Most machines are normally rated for kilograms (ounces), a unit of weight of general-purpose polystyrene.

Example: Given that the specific gravities of TPU compounds and general purpose polystyrene are 1.20 and 1.05, respectively, a 1.7 kg (60 oz.) barrel rated in general-purpose polystyrene will deliver 1.9 kg (67 oz.)* of TPU.

$$*1.7 \text{ kg} \times \frac{1.2}{1.05} = 1.9 \text{ kg (67 oz.)}$$

A targeted TPU shot weight, including sprue, runners and parts, would then be 1.4 kg (50 oz.) on this machine.(1.9 kg x 75% capacity = 1.4 kg or 50 oz.)

Clamp Capacity

A new machine having a minimum clamp force of 300 kg/cm2 to 400 kg/cm2 (2 to 3 tons/square inch) of projected part area, including runners, is recommended. The area of runners in a three-plate mold should be included.

Screw and Tip

The screw and tip designs that are designated "general purpose" are best. The compression ratio of most of the general-purpose screws falls between 2:1 and 3:1; this range is the most desirable for melting and homogenizing TPU compounds.

The general-purpose tip usually has a 60° included angle and a free-flow mechanism of either ball check or sliding ring type. It is recommended that an antibackflow valve, in good working order, be used when TPU compounds are molded. The low viscosity of their melts makes satisfactory packing-out of the mold cavity very difficult unless an antibackflow device is used.

Nozzles

A straight, open nozzle with a full internal taper tip is recommended. A positive-shut-off, antidrool nozzle can be helpful but is not a necessity. If the particular machine or mold design requires a long nozzle or a nozzle extension, it should be well insulated with heater bands so that there are no cold spots. The harder TPU compounds, because of their sharp melting point, can crystallize (set up) if a cold spot exists. The next shot will result in cold slugs of material carried into the cavity along with the hot melt. The longer the cycle required by the particular part, the more troublesome this can become; therefore, the nozzle must be well insulated with heater bands. The bands must extend as close to the tip as possible.

Molds

Any type of mold that incorporates good thermoplastic design principles is satisfactory for TPU compounds. Two-plate, three-plate and hot-runner molds have all been used successfully for a variety of large and small parts. For hot-runner molds, as with nozzles, it is important that heaters provide full coverage of the runner system so that cold spots do not exist; the TPU can be maintained in its fluid melt state. The mold must also be adequately cored for cooling because TPU normally requires a relatively cool mold [(10°C to 44°C) (50°F to 110°F)] to produce optimum cycle. The nozzle tips within the hot-runner mold must be well-insulated from the cold side of the mold. Poor insulation and the quick setup characteristics of TPU compounds (primarily the harder grades) can lead to plugged nozzles. A sheet of transite separating the hot and cold sections of the mold is recommended.

Mold and Part Design

Sprue Bushing

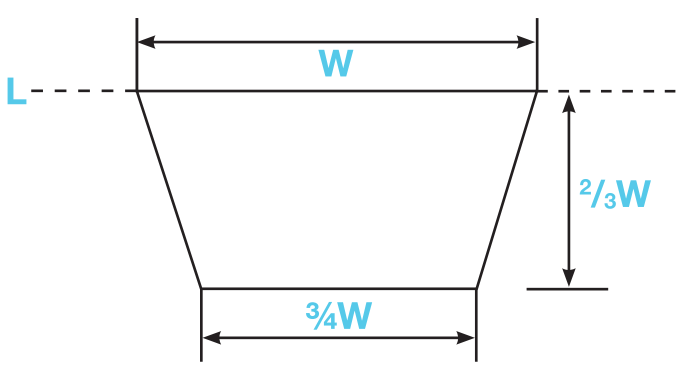
A sprue bushing with a standard 2½° included angle, approximately 42mm taper per meter (0.5 in. taper per foot) should be used. The entrance diameter of the bushing should always be slightly larger than the nozzle exit orifice. To promote a balanced pressure to the runners and cavities, the exit diameter of the sprue bushing should be larger than the diameter of the main runner. Z-pin-type pullers are preferred for easy removal of the sprue.

Runners

In a two-plate mold, full-round runners are preferred because they provide the highest volume-to-surface ratio, the least pressure drop and are the easiest to eject from the mold. Depending on the part size and weight, typical full-round runner diameters are 0.6 cm to 1.0 cm (0.25 in. to 0.4 in.). Because of excessive flow restriction, small diameter runners, less than 0.6 cm (0.25 in.) diameter, should be avoided. Excessively large-diameter runners offer little advantage and contribute to longer cycle times and greater material usage. If a three-plate mold is being used, full-round runners are still preferred, but trapezoidal runners can be used.

Figure 1 shows typical relative dimensions of a trapezoidal cross-section runner. The flow through a trapezoidal runner is equivalent to that of the largest circular runner whose cross-section can be inscribed within the trapezoid.

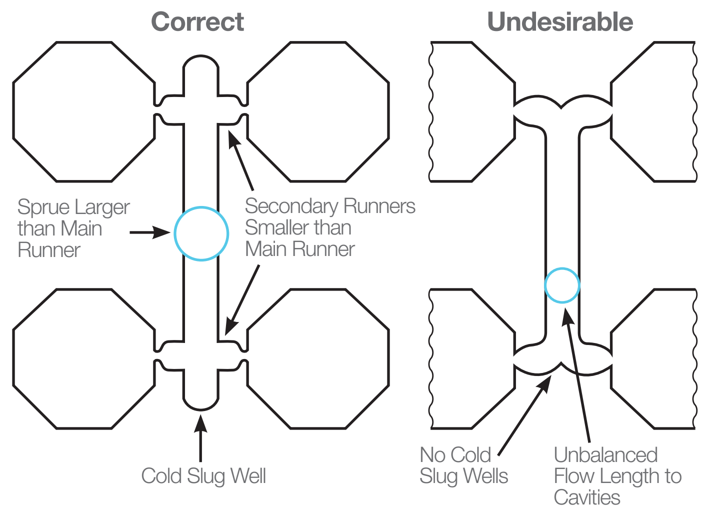
Figure 1 Relative Dimensions of a Trapezoidal Runner for Use in a Three-Plate Mold.



To maintain pressure and balanced flow during injection into a multiple cavity or multigated mold, the secondary runners should be slightly smaller in cross section than the main runner.

Secondary runners should be perpendicular to the main runner, and the runner junction should be vapor-honed to remove burrs and sharp edges, and contain a cold slug well at every turn of direction. **Figure 2** shows a properly sized runner system.

Figure 2 Proper Runner Sizing.



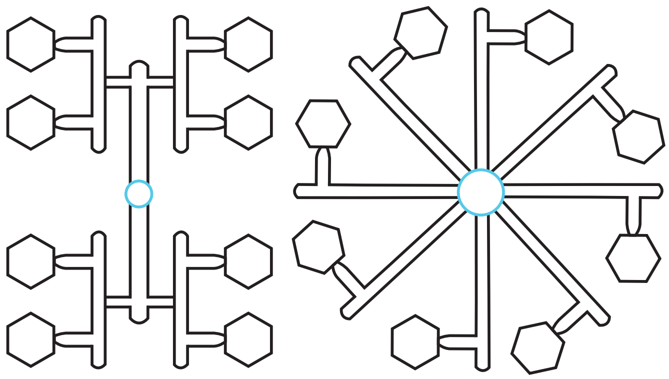
In addition to proper runner sizing, the layout of the mold is also an important consideration. A runner system should be designed to give balanced flow to all gates, ideally designed so that the melt reaches all of the gates simultaneously.

TPU compounds have been molded successfully in hot-runner systems. In many cases, hot-runner mold design and temperature control have become the most complex part of the molding process. Hot-runner molds have usually been designed originally for plastics other than TPU. In general, those created for PVC would offer the best characteristics for success with TPU.

Cold Slug Wells

During injection, the initial surge of material is generally cool since it has remained dormant in the nozzle while the previous shot was being ejected from the mold. To prevent this cold material from entering the cavity and causing a visual defect, cold slug wells or runoffs should be incorporated into the runner system before material is allowed to enter the cavities. Properly sized runner systems designed for balanced flow which incorporate cold slug wells are shown in **Figure 3.**

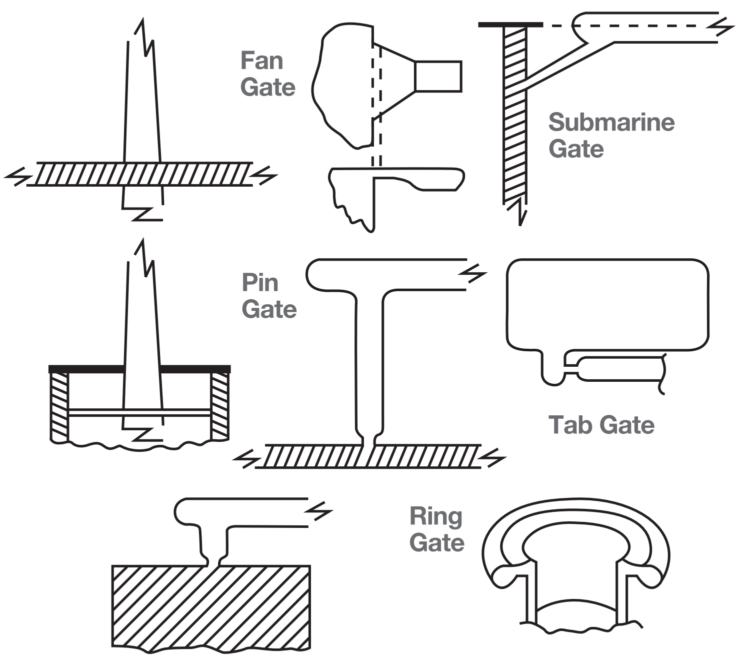
Figure 3 Runner Systems with Balanced-Flow Cavity Layouts and Cold Slug Wells.



Gates

TPU compounds have been molded satisfactorily through awide variety of gate designs including fan, lab, edge, submarine and sprue. In general, the gates should have a generous cross-sectional area to allow the material to flow freely with a minimum of pressure loss. The gates should be vapor-honed with all rough edges and sharp corners removed. **Figure 4** illustrates several acceptable gate designs with rounded corners for minimum restriction.

Figure 4 Gate Designs.



Tab gates are strongly recommended for the softer TPU grades. They eliminate the distortion in the gate area that commonly occurs with very flexible materials. The use of pinpoint gates and tunnel gates should be restricted to very small parts of a few ounces or less in weight where the flow length from the gate is less than two inches. The land length for gates should always be as short as possible. A good rule of thumb for determining the proper land length is that it should be no greater than one-half the gate thickness.

In multigated cavities, the gate location and number of gates are very important in relation to the appearance and performance of the molded part. Since gate areas are almost always more highly stressed due to orientation, gates should be located in noncritical sections of the part. Gating in thick sections of the part and allowing the material to flow to the thinner sections keep sink marks to a minimum. When gating into a thick section, the flow should be directed toward a cavity wall or deflector pin to break up the melt entering the cavity and to prevent a condition called "worming." Worming is a random pattern of weld lines opposite the gate caused by the rapid cooling of the injection melt. If the design of the part requires a split in the flow front coming from the gate, a weld line will usually result when the flow fronts meet. Care should be taken in designing parts to keep the number of gates to a minimum to minimize weld lines. Multiple weld lines could detract from the surface appearance and may affect performance.

Mold Shrinkage

Mold design, part design and operating conditions all affect the mold shrinkage value of any thermoplastic material. In cases where very close tolerance must be maintained, it is suggested that a prototype tool be made before building the production tool. Where standard or coarse tolerances are all that is required,the standard mold shrinkage allowance for the particular TPU compounds should be used. It should be noted that post annealing or exposing parts to a point over temperature will increase the mold shrinkage from what is normally expected. This data is presented in Technical Data Sheets of Lubrizol LifeSciences TPU molding compounds.

Ejection of the Part

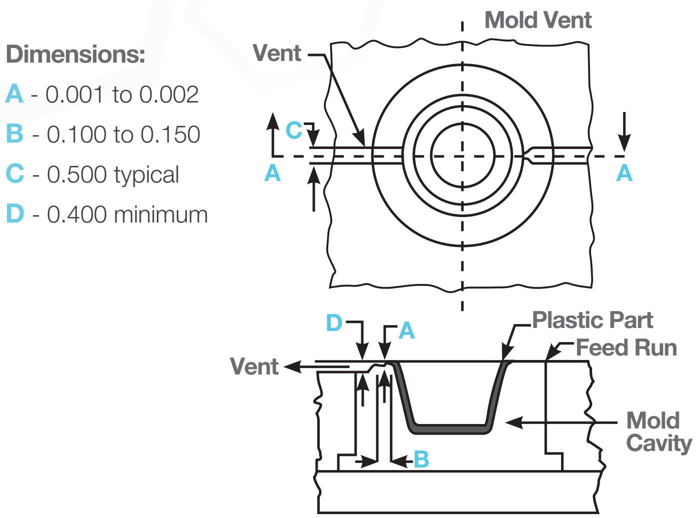
TPU compounds release easily from properly prepared mold surfaces. Highly polished, chrome-plated surfaces should be avoided except for simple flat parts. However, for high-clarity parts, a glossy surface is required. Because most parts are more complex, a vapor-honed matte surface is recommended to provide the easiest, most troublefree release. An electrical discharge machining (EDM) surface can also provide a suitable, easy release. Ejector pins should have as large a surface area as possible, especially those located at thick part sections where the interior may still be very soft at the time of ejection.

Stripper plates and air ejection systems may also be used with TPU compounds.

Vents

Because of the low viscosity of the melt, normal vent depths of .05 mm (.002 inches) and greater will allow TPU compounds to flash. Therefore, vents should be cut only after initial trials on the new tool have indicated necessary locations. A vent channel 6.4 mm to 12.7 mm (.25 in. to .50 in.) wide by .03 mm (.001 inch) deep is usually sufficient. **See Figure 5.**

Figure 5 Mold Venting.



The Molding Operation

Processing Parameters

Successful processing of LifeSciences TPU compounds by injection molding is very dependent upon a wide range of variables such as machine size, shot size, screw geometry and mold design. Due to these factors, exact machine conditions for optimum processing have to be determined by the processor for the system chosen. The sections that follow will outline the conditions the molder should strive to achieve. The best processing latitude and ultimate properties in the molded part will then be realized. Finally, startup and shutdown procedures are summarized in the Processing Guide section. This section and the Troubleshooting Guide section should be readily available to setup personnel and machine operators.

Dryers

TPU compounds are fully reacted thermoplastic polyurethane materials and will not react when exposed to atmospheric moisture. However, as with all polyurethane compounds and many other thermoplastic materials (e.g.g. nylon), LifeSciences TPU compounds are hygroscopic (absorb water from the atmosphere). Although TPU is dry when packaged, storage time and conditions and transportation will allow moisture to enter. The amount and rate of absorption will depend on the type of urethane as well as the type of storage container, temperature and humidity of the air to which it is exposed. Excessive moisture can also cause splay, voids and parts sticking to the mold. Even if the part appearance is not being affected, a severe reduction of the service life of the part can be result.

When moisture reacts with the TPU, the result is shorter polymer chains (lower molecular weight). Even though physical properties at the time of molding may be satisfactory, the reduced molecular weight may be the equivalent to years of use. For optimum appearance and part longevity, TPUs must be dry when molded.

For a small volume of material, an oven dryer is satisfactory. The TPU compounds should be spread in the trays one-inch deep and dried for two hours at 105°C (220°F).

When larger volumes of material make oven drying impractical, a dehumidifying hopper dryer is recommended. A -40°C (-40°F) maximum inlet air dewpoint is recommended. A 0.02% or less moisture content should be obtained before the material is molded.

Polyurethanes are hygroscopic and will absorb moisture when the containers are opened to the atmosphere.

Purging

The barrel should be thoroughly purged before and after molding TPU. Since TPU compounds purge readily from the machine, the use of purging compounds may not be necessary. When needed, suitable purging materials are reground polystyrene, general-purpose ABS, and acrylic molding compounds and even PE. Purging should be done immediately after the production run while the material is still molten. If the machine is allowed to cool to room temperature, purging can be more difficult because the cold material will stick tenaciously to the screw and, on reheating, will be difficult to remove. Slowing cooling TPU in a barrel can also crystallize within the barrel by an annealing process.

Mold Temperature

Molds should be provided with good temperature control to obtain optimum appearance and production rates. The set temperature depends on the particular design, operation and the TPU grade being used. Polyether-type polyurethane materials tend to release better and set up faster when parts are cooled at a relatively slow rate. Thin-walled parts [less than 3mm (.118 in.)] require higher temperatures; thicker-walled parts require lower mold temperatures.

shots and will not cause any degradation from overheating. Care should be taken so that there are no cold spots in the nozzle area. The full length of the nozzle should be covered with heater bands as close to the tip as possible. Inadequate coverage by the heater bands can produce cold spots and allow some of the material to set up between shots. The result will be that the parts will contain lumps of material that were carried into the cavity by the melt stream.

Stock Temperature

The stock temperature can be controlled by a proper combination of the heater band settings, screw backpressure and screw RPM. To develop ultimate physical properties, it is important that recommendations for stock temperature be followed.

To measure stock temperature, use an accurately calibrated needle-probe pyrometer. When making a temperature measurement with a needle pyrometer, the molten material should be injected directly from the nozzle onto a piece of heavy cardboard or some other insulating material that will not absorb heat from the plastic. The injection pressure, injection speed and backpressure are normally at a lower setting for taking these airshots than when at normal cycle; therefore, a stock temperature of approximately 5°C to 10°C (9°F to 18°F) lower than the recommended range is a good objective when starting. As with any thermoplastic resin, airshots should be conducted with adequate ventilation to dissipate hot vapors. Precautions should be taken to avoid being struck by hot melt. The needle should be jabbed into the molten plastic successively four to five times in different locations before the actual reading is taken.

Occasional wiping of the needle probe with some mold release agent will help prevent "freezing" of plastic on the probe during the initial portion of the reading. If material "freezes" to the probe on the first insertion, it acts as an insulator on the probe's surface and erroneously low values for stock temperature will be obtained.

If gassing or bubbling of the hot plastic is observed during the airshot, it generally indicates a higher-than-recommended stock temperature is being achieved and/or excessive moisture is present. Stock temperature and moisture content should be rechecked. The molten plastic rope should appear smooth and reasonably glossy if the stock temperature is near optimum.

Heater Band Settings

To achieve a given stock temperature, heater band settings depend greatly on machine size, screw design and other settings such as backpressure and screw RPM. Large machines typically yield stock temperatures higher than the heater band settings.

For the initial trial of TPU, an ascending barrel temperature profile from rear to front zones is recommended. These settings should be adjusted to achieve an airshot, stock temperature 5°C to 10°C (9°F to 18°F) less than the final desired temperature (more heat will be generated once the machine is cycling continuously). Since heat is being generated by the screw within the material, it is quite normal for the middle and front barrel temperature zones to override the setpoint. As long as the machine is cycling regularly, these setpoints do not need adjustments. Carefully monitor stock temperature during initial startup and after any condition changes.

Nozzle Temperature

The nozzle should be controlled to the same temperature as that of the melt. This will prevent material from setting up between shots and will not cause any degradation from overheating. Care should be taken so that there are no cold spots in the nozzle area. The full length of the nozzle should be covered with heater bands as close to the tip as possible. Inadequate coverage by the heater bands can produce cold spots and allow some of the material to set up between shots. The result will be that the parts will contain lumps of material that were carried into the cavity by the melt stream.

Lubrizol LifeSciences

Injection Speed

A slow-to-moderate injection speed should be used at the start of the molding run and increased to the point where the part fills and no signs of weld lines or sinks exist. If the injection speed is too fast, excessive frictional heat buildup can result in velocity burning as the material flows through restrictions or over sharp edges.

This frictional heat can result in surface appearance problems, or even degradation of the material. Injection speeds for airshots should be relatively slow since there is very little resistance to the material flow.

A good rule of thumb for the injection speed is to use a time of two seconds per inch of ram travel.

Screw Backpressure

The proper value for screw backpressure will vary from machine to machine, but generally the backpressure should be in the low end of the 0.3 MPa to 0.7 MPa (50 psi to 100 psi) range. Low- compression ratio screws could require backpressures to 1.4 MPa (200 psi).

Screw RPM

For a screw of recommended geometry, a rotating speed of 40 RPM to 50 RPM should be satisfactory; higher speeds to 75 RPM have been successful in some applications. Large machines generally require less RPM at optimum conditions. Due to increased diameter, a larger screw has a greater circumferential velocity than a smaller screw at a given RPM. The greater velocity promotes more shear heating of the molding compound.

Injection and Holding Pressures

Scientific Molding or Decoupled Molding techniques based on a two stage volumetric part filling process are used most often for TPU molding. However in some circumstances filling and packing the part completely on the first stage filling process is used.

The amount of first-stage injection pressure (booster pressure) is required to fill the mold cavity will depend on the stock temperature, injection speed, mold temperature and mold design. Generally, velocities/pressures in the range of 50% to 75% maximum available offer the best consistency and processing latitude. It is advisable to start with lower pressures and increase to the desired pressure to avoid flashing the mold. The position for the first-stage injection velocity/pressure should be set to switch to the second stage holding pressure just as the part is reaching full. This should coincide with the moment that the screw completes its relatively fast-forward travel, leaving a 0.3 cm to 0.6 cm (0.125 in. to 0.25 in.) cushion.

The second-stage injection pressure (holding pressure) should be just enough to maintain a full part as the part cools and shrinks in the cavity. This pressure is typically one-half to two-thirds of the first-stage injection pressure. Parts having thicker sections usually require greater holding pressure.

Overpacking the part with excessive holding pressure or time on the first-stage injection pressure can increase molded-in stress. This can be detrimental to properties. Generally, sink marks opposite the gate indicate that more injection pressure/time is needed. Once it is apparent that gates are frozen off, hold pressure can be reduced to save on energy consumption.

A small cushion must be maintained ahead of the screw to compensate for part shrinkage as it cools under holding pressure, thus preventing sink marks. Ideally, the screw should only reach full forward position when material movement has ceased.

Cooling Time

Except for parts with very thick sections, over 9.5 mm (3/8 in.), the time required to retract the screw after the holding pressure is released is generally sufficient for cooling harder (~>90A) TPU compounds. The mold can be opened immediately after the screw stops. In general, the harder the compound, the faster the setup time. The cooling time required for a 85 Shore A TPU compound, for example, may be two to three times as long as that required for a 65 Shore D TPU compound.

Colorants

LifeSciences TPUs may be colored by adding dry inorganic pigments or color concentrates. The chemistry of the carrier resin to the type of TPU being used in the process may need to be the same. Since most TPUs commercially molded are polyester grades, off-the-shelf color concentrates may usually be based on a polyester TPU rather than a polyether type. Having a specialty concentrate made could be required.

Unless there is a protocol requiring that concentrates match the identity of the resin being molded, making a special concentrate with the same grade being used is not necessary for successful coloring. It is best if the TPU concentrate has been made from a softer TPU than that being molded. In cases where the TPU being molded is a hard grade (e.g.g. ~50D or higher), making a color concentrate can result in a masterbatch that can be difficult to melt in the machine.

Summary

In summary, to develop the ultimate physical and appearance properties for LifeSciences TPU thermoplastic polyurethane compounds, the material should be at the maximum allowable stock temperature in a fully dried state. It should be injected at a moderate speed, packed at the minimum pressure required to fill out the mold details and allowed to relax during the cooling stage.

Startup Procedure

Thoroughly clean the injection unit by either dismantling and mechanical cleaning or by an approved purge.

Set temperature controllers and reduce injection pressure settings, back pressure setting and screw RPM to the lower end of their operating ranges.

After temperature zones have stabilized, introduce the TPU pellets into the machine.

Take air-shot stock temperatures and make adjustments to temperature settings and screw RPM to approach the desired stock temperature. Observe the appearance of the molten plastic very carefully at this stage. A smooth, glossy surface is indicative of a good homogeneous melt, while a bumpy rope and/or matte surface indicate nonhomogeneity and low melt temperature. A smoking or frothy melt suggests that the stock temperature is too high or has excessive moisture. Another evidence of good melt temperature is the ability to draw down the hot rope into a thin monofilament. A brittle break indicates a low melt temperature. Backpressure should be set to achieve adequate mixing and optimum melt temperature.

Start molding ports in the semiautomatic mode of operation while adjusting screw travel (feed) injection pressures and injection speed to obtain a full part.

Consult the Troubleshooting Guide for correcting any defects in the molded part.

Troubleshooting Guide

Here are some typical molding problems and several possible causes for each.

Molding Problems	Possible Causes
Sink Marks or Dimples	• Shot too small • Holding pressure too low • Holding lime too shod • Gates too small or in wrong location • Injection speed too fast • Stock temperature too high • Mold temperature too high • Cooling time too short • Sprues, runners or gates too small
Short Shots	• Insufficient material • Injection pressure too low • Injection speed too slow • Cylinder temperature too low • Mold temperature too low • Insufficient venting • Sprues, runners or gates too small
Splay	• Material not dried properly • Melt too hot • Injection too fast • Gates too small
Unfluxed Granules in Part	• Barrel temperature profile wrong. Too cold in the rear and middle zones • Cold spots in the nozzle or nozzle adopter • Cold spots in the hot runner
Foamy Appearance in Part	• Melt too hot • Excessive moisture
Swelling or Ballooning of Thick Sections	• Melt too hot • Excessive moisture • Cooling time too short • Mold temperature too high

Global Presence

Our global presence means we have the application and development laboratories (with technical service ability) already in place, as well as worldwide production capabilities and renowned customer service. We are also a speciality producer of performance coatings and other unique products for apparel. Teaming with Lubrizol simplifies a complex global supply chain.

Global Locations

Headquarters · Regional Office · Manufacturing · Technical Center

Lubrizol LifeSciences

For more information, visit **Lubrizol.com/LifeSciences** email LifeSciences@Lubrizol.com

LSP-MD-MOLD-GDE
18-0120389
APR 2018