NOVATEUR PUBLICATIONS
JournalNX- A Multidisciplinary Peer Reviewed Journal
ISSN No: 2581 - 4230
VOLUME 7, ISSUE 2, Feb. -2021

RESEARCH SPECIAL INJECTION MOLDINGPROCESSES OF POLYMERS

GOFURJONOV DURBEK GOFURJON OGLI
Andijan Machine-Building Institute, Assistant
Tel: +998 97 338-08-36

ABSTRACT:

This article examines the mold produced various methods of producing rotating molding process by melting polymers under pressure are described. The change in thermal conductivity curves according to the type and amount of fillers added to thermoplastic polymers is explained by diagrams.

KEYWORDS: Polymer, core, thermoplastic, mold, temperature, melt spinning.

INTRODUCTION:

In today's fast-paced time, synthetic raw materials are replacing natural materials. in particular, polymerized polymer materials have become the main raw material for the manufacturing industry. Polyethylene, polypropylene, phenol-formaldehyde and other types of polymers are widely used in industry. The advantage of thermoplastic polymers is that they can be recycled. we can also see this process in the production of film and fiber.

METHODS AND OBJECTS OF RESEARCH:

There are several variations of injection molding processes, many of which are still under development. Furthermore, due to the diversified nature of these special injection molding processes, there is no unique method to categorize them. Fig.1. attempts to schematically categorize special injection molding processes for thermoplastics. The most common special injection molding processes are multi-component injection molding, co-injection molding, gas assisted injection molding, injection-compression molding, reaction injection molding, and injection molding of liquid silicone rubber.

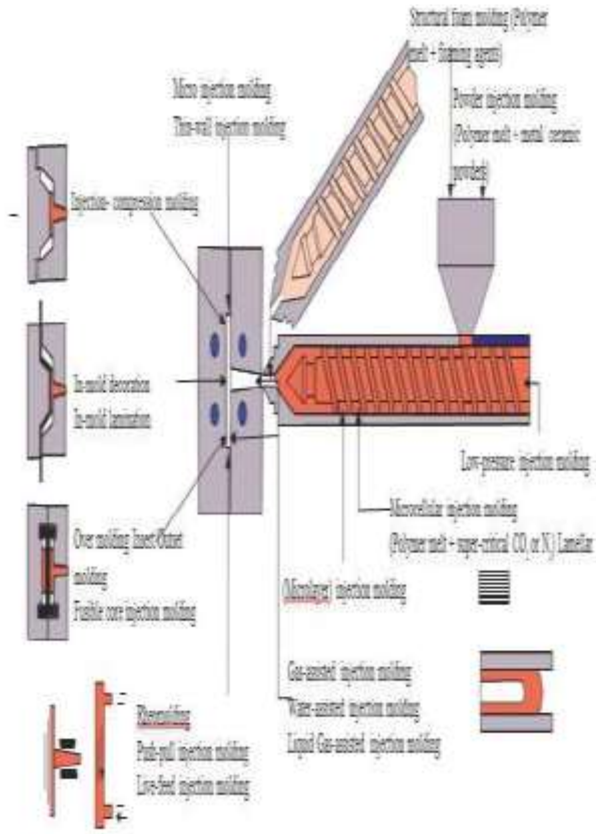

Figure 1.Schematic classification of special injection molding processes for thermoplastics.

Multi-component (or multi-color) injection molding is used to inject two or more components through different runner and gate systems at different stages during the molding process. Each component is injected using its own plasticating unit. The molds are often located on a turntable. Multi-color automotive stop lights are molded this way. In multi-component processes, often two incompatible materials are molded or one component is cooled sufficiently so that the two components do not adhere to each other. For example, to mold a ball and socket system, either the ball or

NOVATEUR PUBLICATIONS
JournalNX- A Multidisciplinary Peer Reviewed Journal
ISSN No: 2581 - 4230
VOLUME 7, ISSUE 2, Feb. -2021

the socket of the linkage is molded first. The component that is injected first is allowed to cool somewhat before the second component is molded in. This results in a perfectly movable system; if the socket is injected first, the assembly will be loose and if the ball is molded first, the assembly will be tight, as the socket shrinks over the ball. This type of injection molding process is used to replace tedious assembling tasks and is becoming popular in countries where labor costs are high. Hence, this type of process is referred to as assembly injection molding. A commonly used method of multi-component injection molding employs a rotating mold and multiple injection units, as shown in Fig.2. Once the insert is molded, a hydraulic or electric servo drive rotates the core and the part by 180 degrees (or 120 degrees for a three-shot part), allowing alternating polymers to be injected. This is the fastest and most common method because two or more parts can be molded in every cycle. Another variation of multi-component injection molding cores or slides while the insert is still in the mold. This process is called core- pull or core-back, as shown in Fig. 3. To be specific, the core retracts after the insert has solidified to create open volume to be filled by the second material within the same mold.[1].

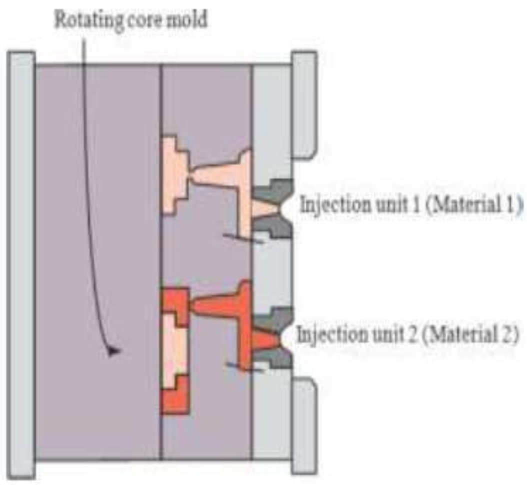

Figure 2. Schematic diagram of a rotating mold used to produce multi-component injection molded parts

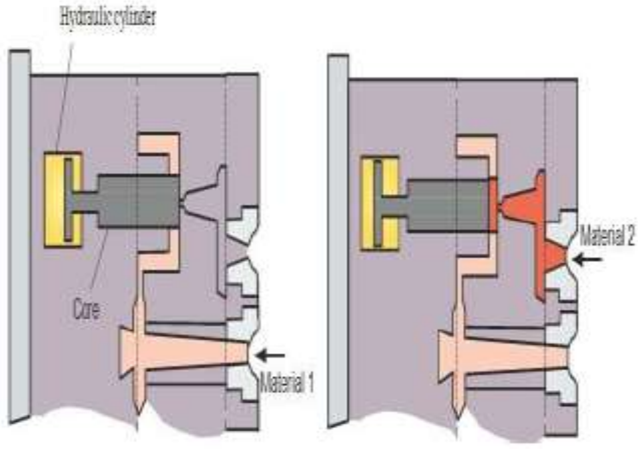

Figure 3. Schematic diagram of multi-component injection molding using a core pull or a core back technique.

CO-INJECTIONMOLDING:

In contrast to multi-color or multi-component injection molding, co-injection molding uses the same gate and runner system. Here, the component that will from the outer skin of the part is injected first, followed by the core component. Fig.4. illustrate the typical sequences of the co-injection molding process using the one channel technique and the resulting flow of skin and core materials inside the cavity. This is accomplished with the use of a machine that has two separate, individually controllable injection units and a common injection nozzle block with a switching head. Due to the flow behavior of the polymer melts and the solidification of skin material, a frozen layer of polymer starts to grow from the colder mold walls. The polymer flowing in the center of the cavity remains molten. As the core material is injected, it flows within the frozen skin layers, pushing the molten skin material at the hot core to the extremities of the cavity. Because of the fountain flow effect at the advancing melt front, the skin material at the melt front will show up at the region adjacent to the mold walls. This process continues until the cavity is nearly filled, with skin material appearing on the surface and the end of the part. Finally, a small additional amount ofskin material is injected again to purge the core material away from the spruce so that it will not appear on the part surface in the next shot. When not enough skin material is injected prior to the injection of core material, the skin material may sometimes be depleted during the filling process and the core material will show up on portions of the surface and the end of the part that is last filled. This is referred to as core surfacing or core breakthrough. There are other variations to the sequential (namely, skin-core-skin, or A-B-

A) co-injection molding process. It particular, one can start to inject the core material while the skin material is being injected (i.e., A-AB-B-A). That is a majority of skin material is injected into a cavity, followed by a combination of both skin and core materials flowing into the same cavity, and then followed by the balance of the core material to fill the cavity. Again, an additional small amount of skin injection will cap the end of the sequence, as described previously. In addition to the one channel technique configuration, two and three- channel techniques have been developed that use nozzles with concentric flow channels to allow simultaneous injection of skin and core materials.[2].

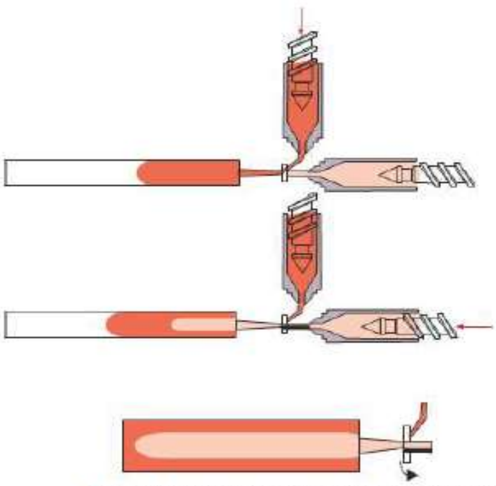

Figure 4. Sequential co-injection molding process.

GAS-ASSISTED INJECTION MOLDING (GAIM):

The gas-assisted injection molding (GAIM) process begins with a partial or nearly full injection of polymer melt into the mold cavity, followed by injection of an inert gas (typically nitrogen) into the core of the polymer melt through the nozzle, spruce, runner, of directly into the cavity. The compressed gas takes the path of least resistance, flowing toward the melt front, where the pressure is lowest. As a result, the gas penetrates and hollows out a network of predesigned, thick-sectioned gas channels, displacing molten polymer at the hot core to fill and pack out the entire cavity. As depicted in Fig.5. gas assisted injection, as well as other fluid assisted injection molding technologies, work based on several variations of two principles. The first principle is based on partially filling a mold cavity and completing the mold filling by displacing the melt with a pressurized fluid. Fig.6. presents the gas-assisted injection molding process cycle based on this principle. With the second principle, the cavity is nearly or completely filled and the molten core is evacuated into a secondary cavity. This secondary cavity can be either a side cavity that will be scrapped after de-molding, a side cavity that will result in an actual part, or the melt shot cavity in front of the screw in the plasticating unit of the injection molding machine. In the latter, the melt is reused in the next molding cycle. In the so-called gas-pressure control process, the compressed gas is injected with a regulated gas pressure profile, either constant, ramped, or stepped. In the gas-volume control process, gas is initially metered into a compression cylinder at preset volume and pressure; then, it is injected under pressure generated from reducing the gas volume by movement of the plunger. Conventional injection molding machines with precise shot volume control can be adapted for gas- assisted injection molding with add-on conversion equipment, a gas source, and a control device for gas injection, as schematically depicted in Fig.7. Gas assisted injection molding, however, requires a different approach to product, tool, and process design due to the need for control of additional gas injection and the layout and sizing of gas channels to guide the gas penetration in a desirable fashion.

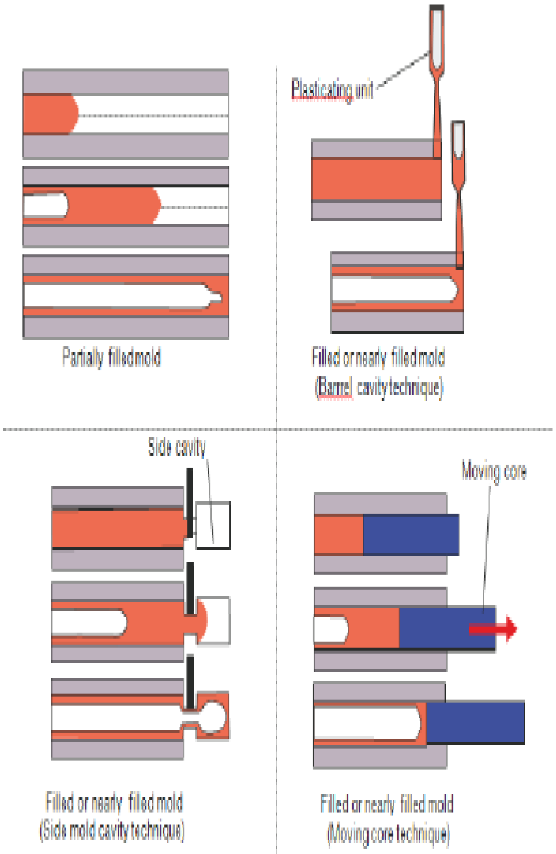

Figure 5.Schematic classification of fluid-assisted injection molding processes

The gas-assisted injection molding process is a special form of a more general category of fluid- assisted injection molding. Another process that falls under this category is water-assisted injection molding. The main difference of this latter process is that water is incompressible and has a much higher thermal conductivity and heat capacity than air. Consequently, this leads to significant reductions in cycle time. [3].

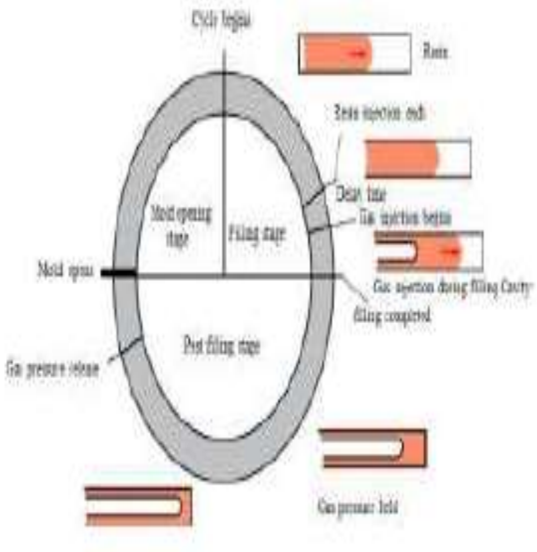

Figure 6. Gas-assistedinjection molding cycle.

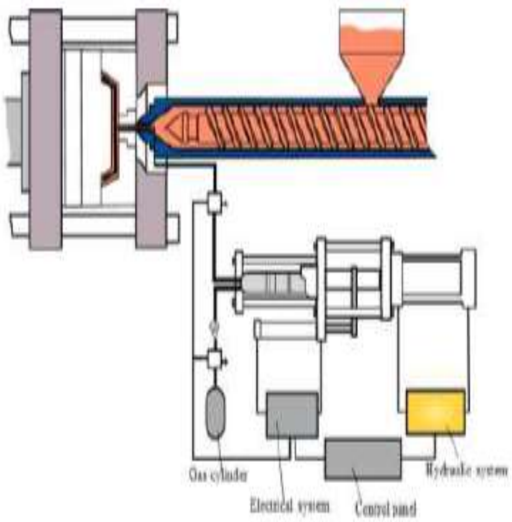

Figure 7.Schematic diagram of a typical injection molding machine adaptedfor gas-assisted injection molding with anadd-on gas-compression cylinder and accessory equipment.

CONCLUSION:

In conclusion this project was useful in exploring and understanding how different control factors have different effects on a desired characteristic for a part. Through the use of design expert it was shown how some control variables have a strong effect on a specified characteristic, while others have little to no effect. Through design expert it was found that the variables affecting the final weight of the plastic mold injected part the most, were pressure and flow weight. Each of these factors had a larger impact than the nozzle temperature and dwell time combined.

REFERENCES:

1) Стручкова (Ючюгяева) Т.С. Триботехнические материалы на основе ПТФЭ и углеродных наполнителей // Тр. XIII межд. научной конф. студентов, аспирантов и мол.ученых "Ломоносов". Москва. 2006. С.482–483.
2) Tim A.Osswald and Georg Menges “Material Science of Polymers for Engineers” 3rd
3) Edition. Carl HanserVerlag, Munich 2012.
4) History of Plastic Injection Moulding& Rotational Moulding.|| Web 20 Apr. 2010.