Research of Injection Molding Processes of Different Types of Material Using Computer Simulation

Emil Velev

Faculty of Physics and Technology, Paisii Hilendarski University of Plovdiv, 24 Tsar Assen str., 4000 Plovdiv, Bulgaria

Abstract. In recent decades, polymers have widely entered all spheres of life. A prerequisite for this is the ability to produce details of different types, complexity and color. This is done in injection molds, which can be single-well or multi-well depending on the production program. With the constant appearance of new plastics and modifications of existing ones, new and new challenges appear before the designers of injection molds, namely, even before a given tooling is produced, what is the possibility of producing parts from different types of plastic with the same plastic injection mold. This problem without the use of computer technology is practically impossible.

Keywords: simulation, plastics, molding

1. INTRODUCTION

The continuous pursuit of the highest possible automation and mechanization of injecttion processes and the ever-increasing prices of plastics have forced design bureaus to develop injection systems with minimal requirements for additional processing of the castings and with minimal input residue. As a result, forms appear without casting residues, i. e. injection molding without technological waste.

The use of modern software products facilitates and improves the construction of the tool for the production of plastic products from different types and characteristics of plastic. Without the use of programs to perform different types of simulations, it would be expensive and time-consuming to redesign tooling that was not designed to use different types of plastics. This is due to their chemical composition, which largely determines the type of hot nozzles used for their production. Computer programs have widely entered the engineering activity, thereby reducing the time required for designing and manufacturing details of various types and complexity [1].

The correct choice of materials for making the injection mold largely predetermines the durability of the tool [2].

2. THEORETICAL BASIS

Injection molds are the tools for producing plastic parts. These tools are divided into:

- forms with insulating distribution channels of common design;

- molds with a heated inlet system.

The first have distribution blocks operating at temperatures lower than the melting point of the processed plastic material. Distribution channels k (Fig. 1) leading to the individual forming cavities of the mold have a large cross-section (up to over 50 mm^2).

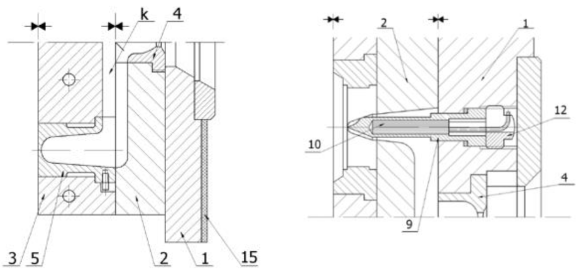

Fig. 1 Forms with an insulating distribution channel.

Advantages of this design of the forming parts is:

- relatively low production costs;

- lower energy consumption (without additional heating);

- not subject to noticeable effects of expansion (does not affect function);

- very good hermeticity against melt leakage;

- does not need regulating devices to control the mold temperature;

- quick and easy transition to another type or color of plastic.

However, the system also has a number of disadvantages, for example:

- time-limited injection cycles - the time between two injections should not be more than 20 s;

- requires continuous operation - with a longer cycle (over 20 s) or with a short interruption, the partially solidified melt can enter the mold and negatively affect the quality;

- with a longer interruption (only after a few minutes), the melt solidifies in the entire cross-section of the distribution channel. Therefore, the molds must be designed so that they can be easily and quickly started in the event of a break and the hardened material can be easily removed;

- injection molded plastic must have a wide range of processing temperatures. For plastics with a higher melting point, it indicates insufficient insulation function;

- with greater fluctuations in the injection temperatures in the injection block of the machine, there may be a longer downtime. Again, there is the possibility of a negative impact on the quality of the part.

Molds with a heated inlet system are:

- incoming systems with external heating distribution;

- inlet systems with an open inlet.

In incoming systems with external heating distribution, the most common solution is to store the distribution duct in a heated distribution block or panel.

Nozzles with internal heating The most famous manufacturer is INCOE Corp. (USA). This company offers a wide range of standardized die components, including a wide range of complete nozzles for heat runner systems and control devices.

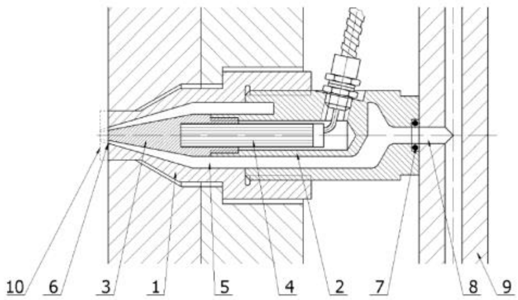

Fig. 2 Nozzle with internal heating.

Melt flowing from the distribution block into the nozzle flows around the heated torpedo and is injected through the open inlet opening into the forming cavity of the mold (Fig. 2). The cylindrical heating element has a built-in temperature sensor so that the inlet mouth temperature can be well adjusted. The front tip has an insert to allow connection to the die cavity. Due to the insulating effect of the plastic in the nozzle body, the inflow does not "freeze". The tip can be used for single molds without a manifold or with a manifold design common to multiple molds. The distribution block is heated by cylindrical heating elements. A stainless steel sealing ring is placed between the distributor block and the nozzles to seal against melt leakage. It is stored in a groove between the nozzle body and the distributor block with some overlap. In the first injection, the melt enters through the small holes on the inside of the hollow ring, fills and expands inside it. This seals the surfaces between the nozzle and the distributor block very well.

Externally heated nozzles HASCO supplies the so-called “high performance nozzles” shown in Fig. 3.

Plastics is the module used to implement the simulations developed by SolidWorks. With its help, it is possible to simulate the manufacturing processes of a given plastic part even before the tool is manufactured. The following simulations can be performed:

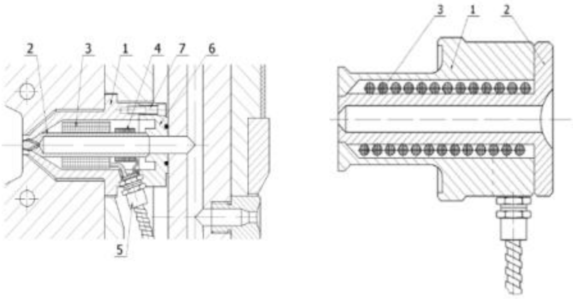

Fig. 3 Externally heated nozzles HASCO.

- Fill Time: The fill time graph shows the profile of the plastic melt as it flows through the mold cavity during the fill stage of the injection molding process. The blue areas indicate the start or beginning of the flow front. The red areas show one of the following: The front position of the flow at any given time interval during the fill step animation. The end of filling when the flow has stopped, even if the software detects no filling;

- Pressure at End of Fill: During the filling stage, the forward injection speed of the reciprocating screw is controlled, resulting in the required pressure to fill the cavity at that speed. Injection pressure propagates through the molten plastic and results in a pressure drop distributed along the length of the stream. The pressure at the end of the fill is a very good indication of how evenly the cavity is filled.

- Temperature Growth at End of Fill: At the end of filling, the plastic in contact with the cavity wall freezes into a very thin frozen layer that has cooled to the temperature of the mold. The thickness of this frozen layer is independent of the wall thickness of the part. The thickness depends on the temperature difference of the melt and the matrix and the thermal conductivity of the material. On the outer surface of the part, the end-of-fill temperature result shows temperature values taken from the center of the dense mesh cell closest to the surface. Since this location is slightly inboard of the part, the temperatures will be higher than the mold wall temperature. This difference will decrease as you create a finer dense mesh with more cells across the thickness of the part;

- Volumetric shrinkage at End of the Fill: Volumetric shrinkage at the end of the fill can indicate areas of potential problems. High levels of shrinkage will occur in thick sections of plastic part that do not go through a sufficient packing stage during the molding process;

- Cooling time: The cooling stage is designed to reduce the temperature of the material to the deformation temperature under bending load, the extrusion temperature. Typically, the cooling time is 70 % of the total cycle time;

- Ease of fill: You can use the ease of filling chart to determine if the cavity is filling successfully. Green areas indicate areas that can be filled at normal injection pressure. Yellow areas indicate areas where the injection pressure exceeds 70 percent of the machine's maximum injection pressure. Red areas indicate areas where the injection pressure exceeds 85 percent of the machine's maximum injection pressure [3].

Simulations were performed on the "Box" part (Fig. 4). The part is executed according to the client's order.

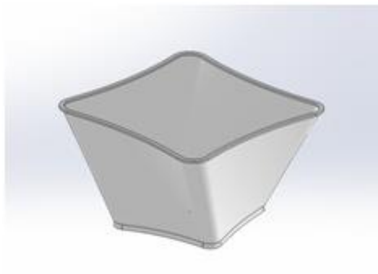

Fig. 4 The part "Box".

3. EXPERIMENTAL RESULTS

Computer simulations of injection molding of two types of plastic material approved for use in the food industry were performed.

The following simulations were carried out for the thus selected part: the influence of the gate diameter for two types of material was investigated. The materials are polypropylene and polystyrene.

From Fig. 6 it can be seen that at the four edges of the part, there are problem areas to fill.

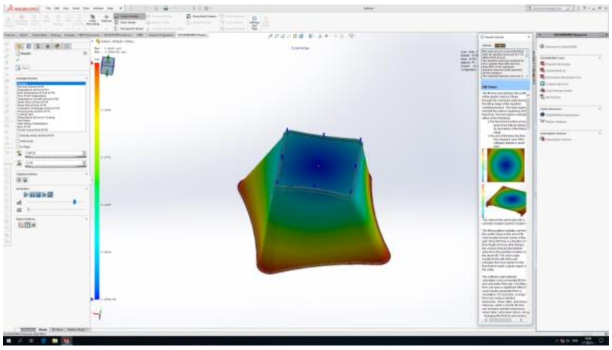

Fig. 5 Filling time with PP material and gate diameter 0.5 mm.

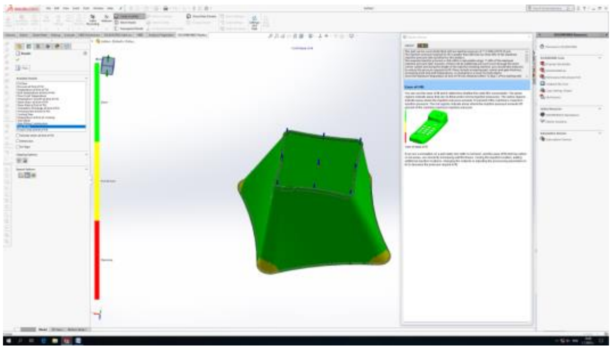

Fig. 6 Easy filling with PP material and gate diameter 0.5 mm.

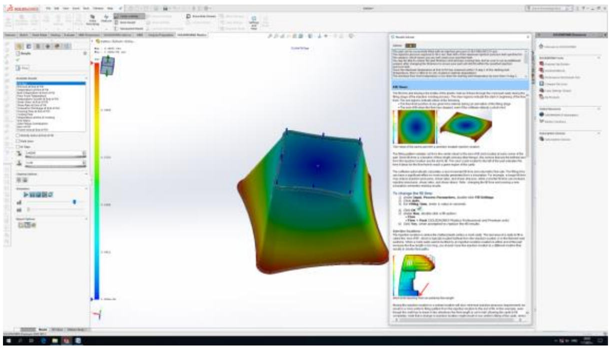

Fig. 7 Filling time with PP material and gate diameter1.0 mm.

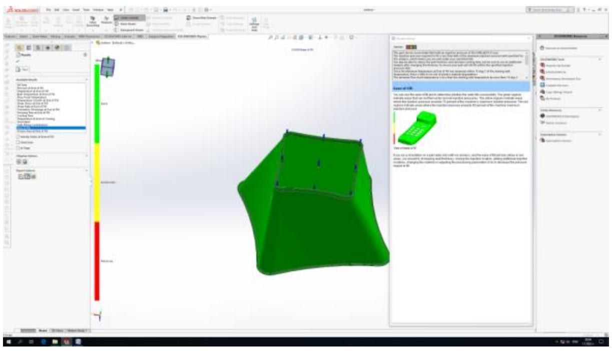

Fig. 8 Easy filling with PP material and gate diameter 1.0 mm

After increasing the diameter of the gate, there are no more problem areas.

By increasing the diameter of the gate, the injection conditions are improved and the injecttion effort is also reduced. In this way, the entire process is improved and the unit operates at lower pressures.

The same simulations with a change in the diameter of the gate were made when the part was made of polystyrene. These are shown in the figures below (Fig. 9 - 13).

From Fig. 9 it is clear that with a diameter of the gate of 0.5 mm the material cannot fill an entire cavity.

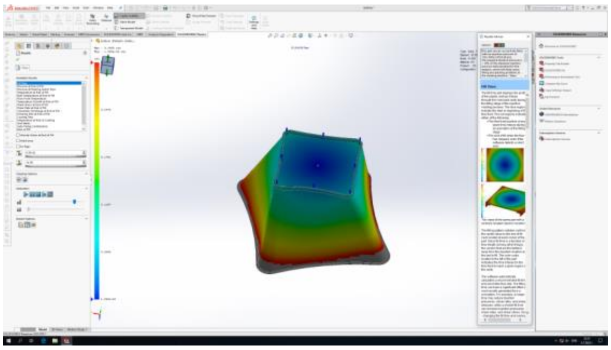

Fig. 9 Filling time with PS material and gate diameter 0.5 mm.

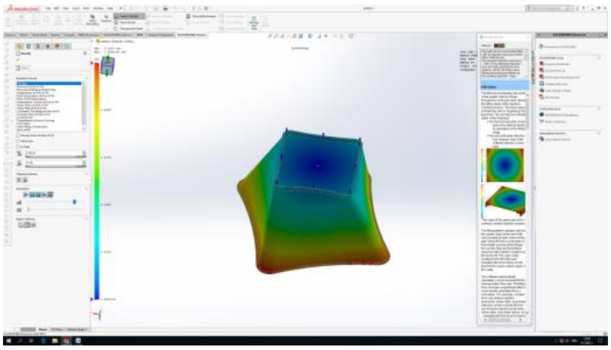

Fig. 10. Filling time with PS material and gate diameter 1 mm.

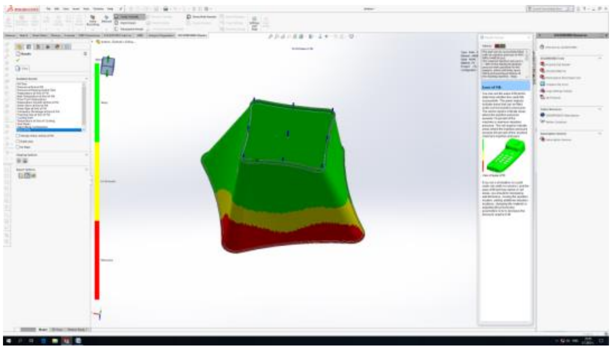

Fig. 11 Easy filling with PS material and gate diameter 1 mm.

As the diameter of the gate increases, injection conditions improve, but there are still problem areas. (Fig. 11)

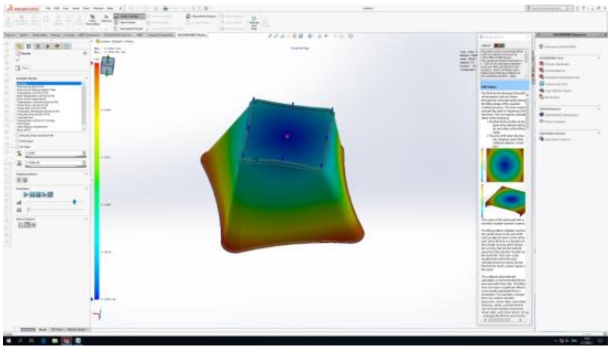

Fig. 12 Filling time with PS material and gate diameter 2.0 mm.

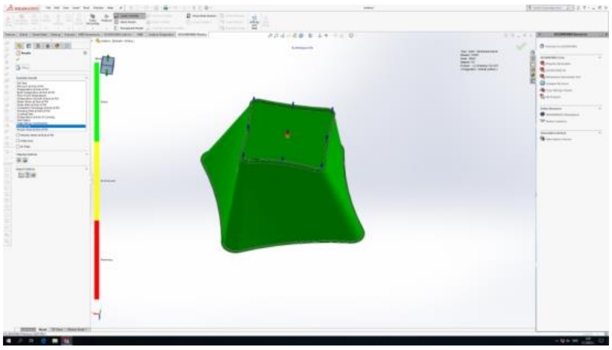

Fig. 13 Easy filling with PS material and gate diameter 2.0 mm

It is necessary to increase the diameter of the gate to 2.0 mm so that the problem areas disappear and material fills the entire cavity (Fig. 12-13).

4. CONCLUSIONS

The following conclusions can be drawn from the simulations shown:

1. For the production of the plastic part from polypropylene, the minimum diameter of the gate should be 1.00 mm;

2. For the production of the plastic part from polypropylene, the minimum diameter of the gate should be 2.0 mm;

ACKNOWLEDGEMENTS

The research was also supported by the Operational Program “Science and Education for Smart Growth”, project number BG05M2OP001-2.016-0026.

REFERENCES

V. Raydovska (2020). Building a laboratory for simulation and experimental research in electrical engineering, ICERI2020 Proceedings, *pp. 2687-2697.*

Mishev G., Rupetzov V., Simov M. (2014). Possibilities for Increasingthe Resource of Injection Molds, Tribological Journal BULTRIB Vol. 4, 2014, ISSN 1313-9878, pp. 69-74

SolidWorks 2019