COMPUTATIONAL MECHANICS 2022
37th conference with international participation
Srní
November 7 - 9, 2022

Analysis of production parameters of automotive components by injection moulding technology

A. Sapietová, V. Dekýš, R., Z. Šavrnoch

Faculty of Mechanical Engineering, University of Zilina, Univerzitná 8215/1, 010 26 Žilina, Slovakia

The article will present selected requirements necessary for the manufacturability of a plastic component by injection molding technology. With the help of simulation tools, manufacturability will be assessed on the selected plastic component of the car seat. The simulation of the injection molding process will be analyzed using Autodesk Moldflow software. The analysis will take into account the conditions of production of the polymer part by injection with regard to the requirements for its functionality defined by the production drawing.

If we take a closer look at the technology of plastic injection molding, we will discover significant advantages such as automation, reproducibility, precision and low production costs. This technology also has its limitations and specifics, which should be paid attention to. Today, we have simulation tools available that help either directly in product development or tool design [1]. By simulating flow, we can predict the properties of the behavior of the hot-melt in the mold and assume the influence of important parameters such as the choice of the location of the injection point or the deformation .

The component in Fig. 1 (Lockshift) is part of the locking mechanism, which, after pressing the button, allows the movement of the headrest in the vertical direction.

When choosing the material, the strength limit and its impact strength are taken into account. A disproportionate increase in wall thickness is not recommended, as it leads to increased weight, material usage and lengthening of the production cycle due to the extension of the cooling time.

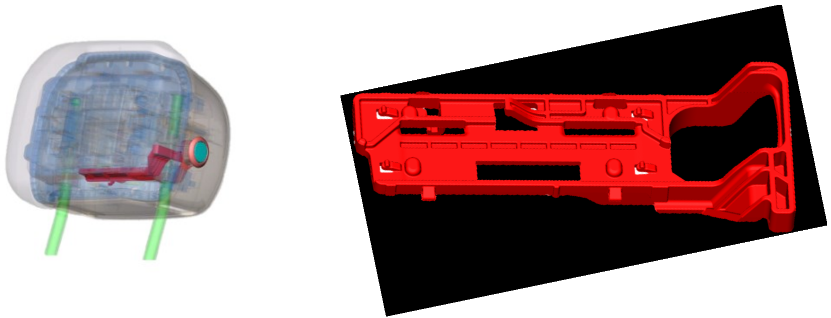

Fig. 1. Headrest (left) and Lockshift (right)

Polymers have become popular due to their properties. They are relatively cheap, have a low specific density and often good electrical insulating properties. They can be combined with other materials to create composites [4]. POM (polyoxymethylene) material was chosen for Lockshift. Emissions of volatile substances are a disadvantage of this material, as it is produced by polymerization of formaldehyde and this is excreted in small quantities during the life cycle. Thanks to its other positive properties (very good sliding and abrasion properties, it is stable in shape and size and has a relatively high strength), it is difficult to replace it.

The injection point is the connection between the inlet system and the molding. Its cross-section should be as small as possible, as it is visible on the molding, but it must also be large enough so that the values of shear stress and pressure loss are not exceeded. The cross-section and position of the injection point have a significant effect on the injection process and are a very common cause of injection problems. Editing it already in its finished form is often difficult and expensive. Therefore, every designer must choose the concept of the inlet system very carefully when designing the part and the mold.

The designer of the molding should already think about the chosen concept of the mold and, therefore, in which direction the molding will be unmolded. The strength of the machine's closing unit depends on the projection of the molding surface in the main direction and the material used, as the injection pressure is projected onto the molding surface. Injection pressures depend on the material and the nature of the product [5]. The simulation program can predict the required closing force based on the viscosity, flow index and shape of the model. When the necessary closing force is estimated, the parts manufacturer can calculate how strong an aggregate will be needed for production. Subsequently, it is necessary to design the part with regard to the release angles. The Lockshift part does not have any elements with negative angles.

The material and its parameters are defined in the simulation program libraries. The simulation program Autodesk Moldflow perform the analysis by using the finite element method (FEM). The simulation results presented in Fig. 2 show that the injection point is not optimally positioned, as the injection time is too long (5.6 s) and the melt junction point is in the weak point of the component. The pressure during switching is 1190 bar (119.2 MPa).

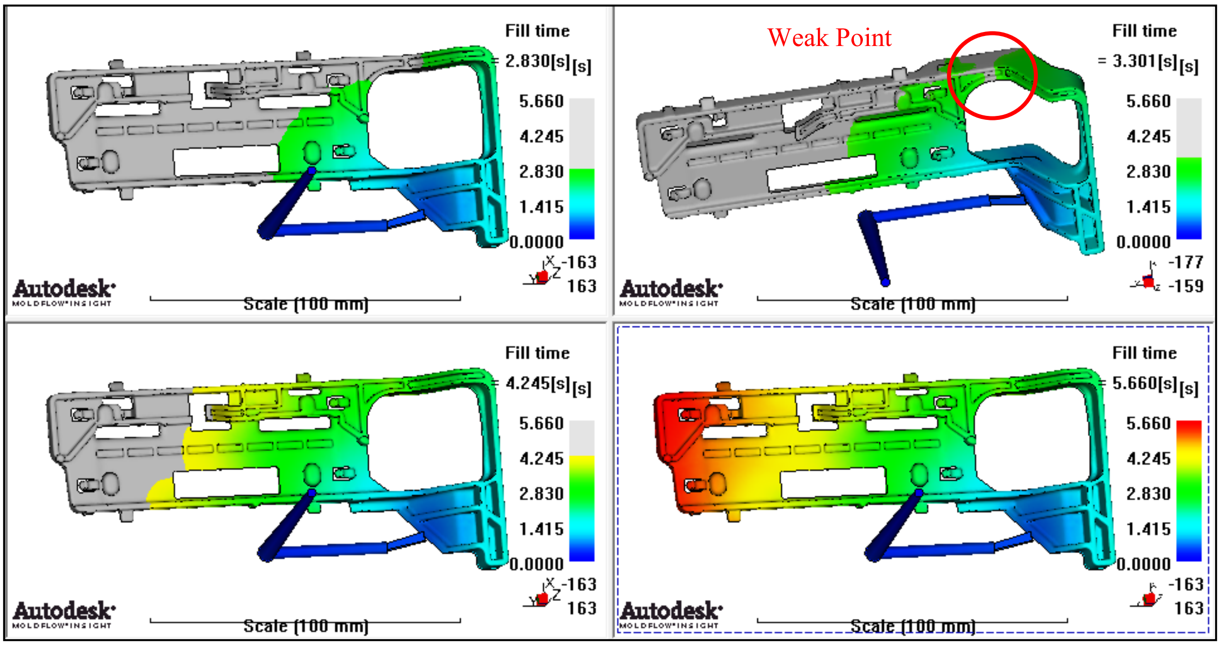

Fig. 2. Flow rate with the originally positioned injection point

From the filling simulation in Fig. 2 and Fig. 3, it takes 5.66 s to fill the part and the switching pressure is approximately 1120 bar. It can be seen from the simulation that the melt flow is not optimal (Fig. 2). The melt has to unnecessarily pass narrow places in order to subsequently reach the open space and also joins in the narrow place. Here it creates a joint line that has lower strength and may break during dynamic tests. Our effort is to make the connection line as long as possible. For a part that weights 21 g, the filling time is relatively long, which is due to how the part is complicated to fill. The filling pressure for POM material should be in the range of

1000 to 1500 bar [2, 3]. The currently predicted injection pressure is within this range, but if the filling point were better chosen, the pressure could be lower. Higher injection pressures mean higher shear stress for the material.

For the reasons discussed above, we changed the location of the injection point and subjected the component to a new analysis in Moldflow software.

From the visualization of filling the cavity with the new injection point in Fig. 4, it is clearly seen that the filling time has been reduced to about 2.8 s. The fusion joint has moved to a more robust part of the part and thus the risk of cracking under dynamic stress has been reduced.

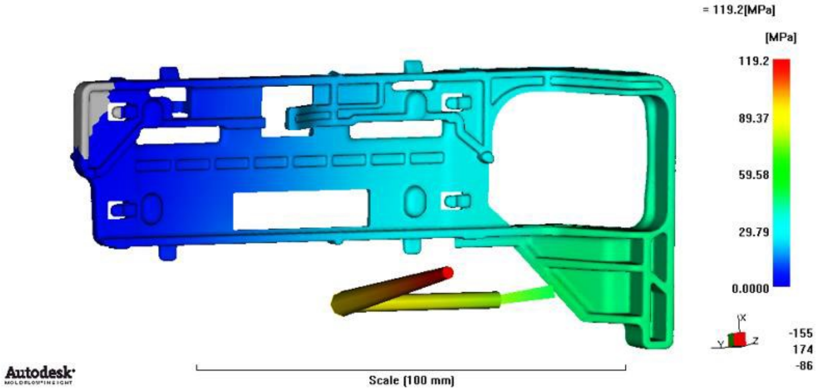

Fig. 3. Mold pressure

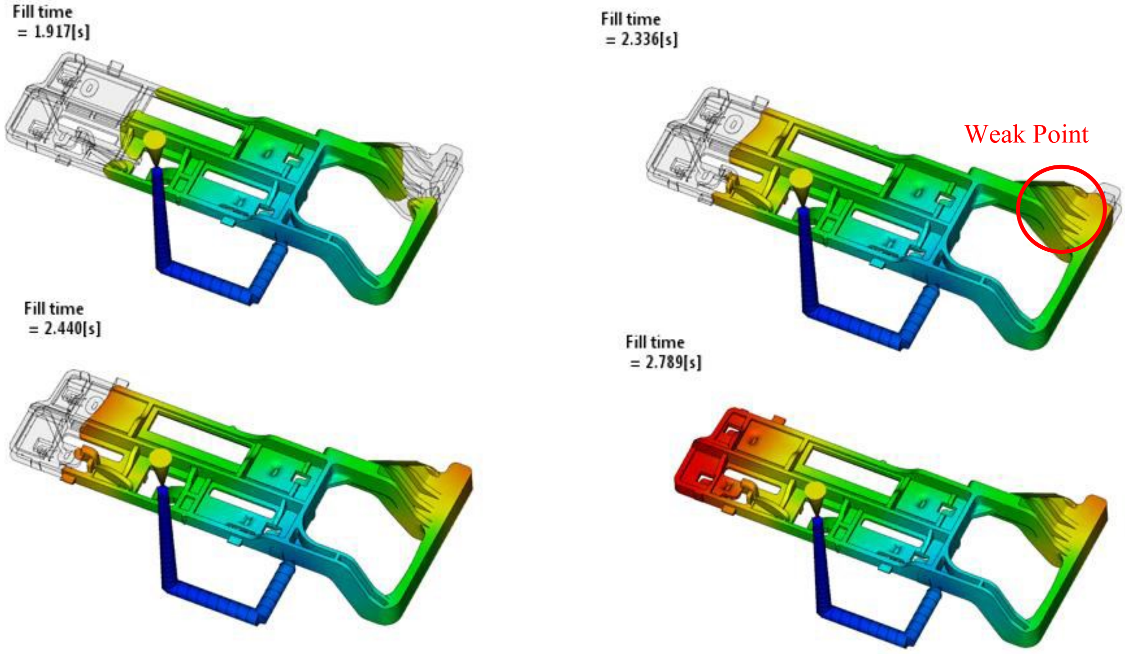

Fig. 4. Flow rate with repositioned injection point

Another improvement is the injection pressure dropped significantly, down to a value of 573 bar. This represents a significantly reduced stress on the material and the reduction in injection time will represent a shortening of the injection cycle by almost 3 seconds. If we further compare the visualization of the deformation from Fig. 5 before and Fig. 6 after adjusting the location of the injection point, we can see that here too we have achieved a significant improvement.

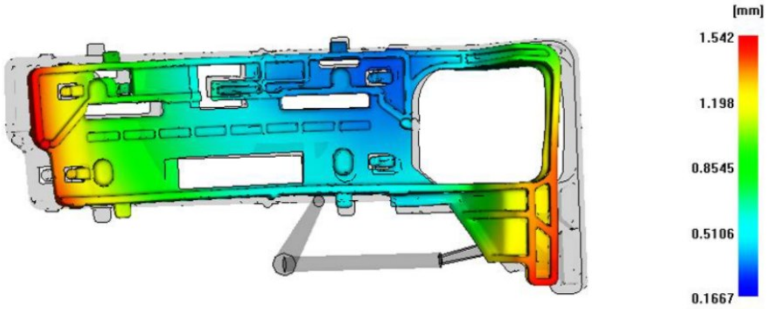

Fig. 5. Deformation with originally positioned injection point

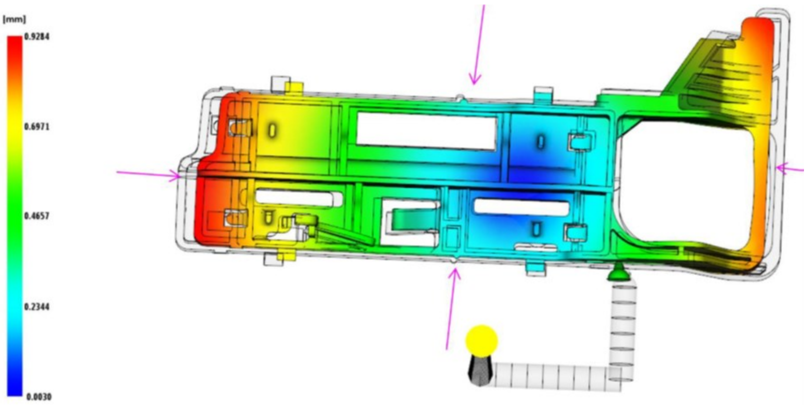

Fig. 6. Deformation after the reposition of injection point

Acknowledgements

This article has been possible with the support of the VEGA 1/0510/20 and VEGA 1/0141/20 projects.

References

[1] Bikas, A., Pantelelis, N., Kanarachos, A., Computational tools for the optimal design of the injection moulding process, Journal of Materials Processing Technology 122 (1) (2002) 112-126.
[2] Kerkstra, R. Brammer, S., Injection molding advanced troubleshooting guide, Hanser Publications, Ohio, 2021.
[3] Lucyshyn, T., D'avernas L.V.D., Holzer, C., Influence of the mold material on the injection molding cycle time and warpage depending on the polymer processed, Polymers 13 (18) (2021) 3196.
[4] Majko, J., Vasko, M., Handrik, M., Saga, M., Tensile properties of additively manufactured thermoplastic composites reinforced with chopped carbon fibre, Materials 15 (12) (2022) 4224-4224.
[5] Yang, W. Zhiwei, J., Injection moulding of polymers, Advances in Polymer Processing: From Macro to Nano Scales, 2009, pp. 175-203.