OPTIMIZATION OF THE INJECTION MOLD RUNNER SYSTEM OF THE TRANSPORT MEANS PLASTIC PARTS

Damián Čechmánek*, Róbert Kohár, František Brumerčík, Michal Lukáč, Jakub Fiačan

Department of Design and Mechanical Elements, Faculty of Mechanical Engineering, University of Zilina, Zilina, Slovakia

*E-mail of corresponding author: damian.cechmanek@fstroj.uniza.sk

Damián Čechmánek 0000-0003-2143-6009, Róbert Kohár 0000-0001-7872-2829, František Brumerčík 0000-0001-7475-3724, Michal Lukáč 0000-0003-3390-5319, Jakub Fiačan 0000-0001-5171-8467

Resume

This paper describes the optimization of dimensions of the individual injection mold runner system of the plastic part used as the frame for various devices in transport means. Nowadays, the great emphasis is placed on the production time and finances associated with design and production. As a result, simulations are being employed increasingly. This paper aims to describe the design the optimum size and shape of the sprues and runners required for proper filling and to simulate the injection molding process to avoid problems and defects that tend to be caused by an incorrectly sized runner system. As a result, some adjustments were made to the size of the runner system to eliminate problems with underfilling, overfilling, air traps and weld lines. The difficulties with the original design were fixed by the optimized solution, which improved the monitored parameters. Additionally, the technological parameters of the injection process were modified, resulting in lower stress on the injection mold in the injection process.

Article info

Received 24 March 2023
Accepted 2 May 2023
Online 16 May 2023

Keywords:

runner system
injection molding
injection mold
constructional design
analysis

Available online: https://doi.org/10.26552/com.C.2023.046

ISSN 1335-4205 (print version)
ISSN 2585-7878 (online version)

1 Introduction

The plastic injection molding is a highly efficient way of producing a wide range of thermoplastic products and offers many advantages. Injection molding ensures the production of complex and intricate shapes of plastic products in short production cycles with excellent surface quality, good dimensional and shape accuracy and high reproducibility of mechanical and physical properties [1]. Injection molding can be used to manufacture plastic products by injecting molten polymer under high pressure into a designed mold [2]. The injection mold must be well designed and manufactured before any products can be molded [3].

In addition to the proper design of the mold cavity, the proper design of the runner system is very important, as well. The runner system ensures the passage of molten plastic from the injection molding machine nozzle to the mold cavity [4]. The cold runner system consists of the sprue, the main runner, the runner branch, the cold slug and the gates [5]. An important factor is the location of the gates, which can affect the direction of flow and solidification of the melt during and after the filling. The design of the runner system, especially the gate system, is critical to eliminate underfill, overfill and weld lines [6].

In recent years, many programs for computer simulation of injection molding processes have been developed and among the most widely used programs is Autodesk Inventor [7]. This software is often used to optimize the injection molding process. The development of computer simulation techniques that mimic the injection molding process, such as CAE, has the potential to replace the costly trial-and-error experiments that have been associated with the process [8]. These experiments were used at a time when the mentioned CAE software did not yet exist. They consisted in designing a distribution system of a certain shape and dimensions according to general design rules and procedures and then modifying it to achieve an optimal solution.

In this paper, the dimensions of the runner system are optimized using the Autodesk Inventor software. The parameters monitored were the injection process parameters, simulation of filling time, plastic flow, confidence of filling, quality prediction, air traps and weld lines. Finally, the results of the analysis were compared between the two variations of the runner system's size.

The proper design of the mold runner system is based on the use of various progressive systems, such as design CAD systems [9-11] or calculation FEM systems [12-15], which support the design process of the molded part significantly.

2 Analysis of mold filling

The mold-filling analysis was created using the Autodesk Inventor Mold Design. The basis of the analysis is the design of the injection mold. The first step of the design is to insert a 3D model of the plastic part and adjust its orientation, concerning the opening direction of the injection mold. This is followed by selection of a suitable type of material. For this analysis, SCHULAMID 6 GF 30 plastic from manufacturer A. Schulman GMBH was chosen. Next, the mold design proceeds with creation of the cavity and the core. It consists of defining the position of the gate location and workpiece setting, creating the patch surfaces and creating the runoff surface. Subsequently, the main parts of the injection mold are created using the "Generate core and cavity" command. The design then continues by sketching the runner and selecting its shape and dimensions. According to Fattori [16] the runner should not be too big or too small. It requires much more force to fill the cavities if it is too small and there is a danger that they will freeze off before

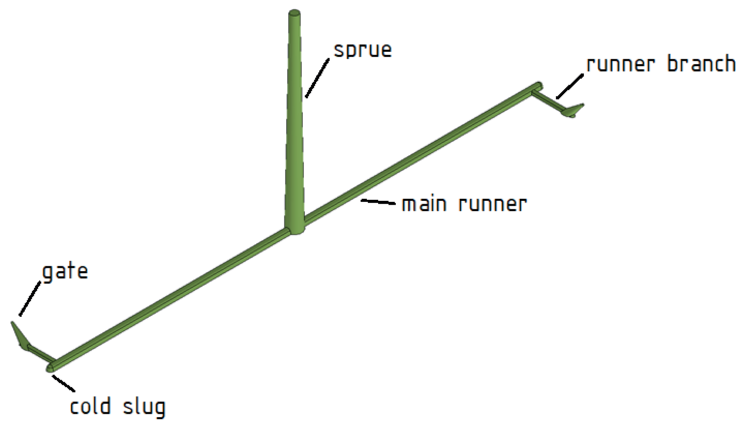

Figure 1 *Runner system of the original design*

Figure 2 *Elements of the runner system of the original design*
a) Main runner b) Runner branch

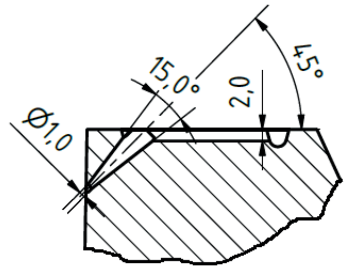

Figure 3 *Submarine-type gate of the original design*

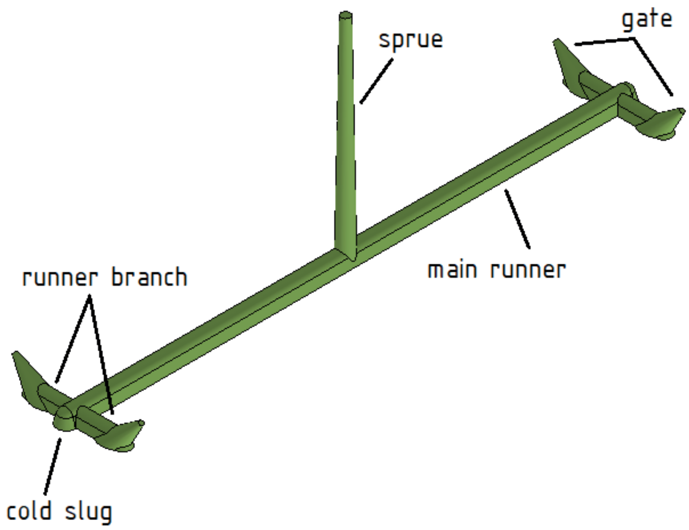

Figure 4 *Runner system of the optimized design*

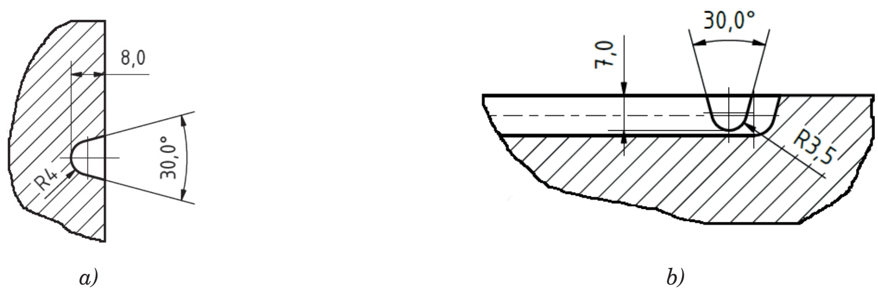

Figure 5 *Elements of the runner system of the optimized design*
a) Main runner b) Runner branch

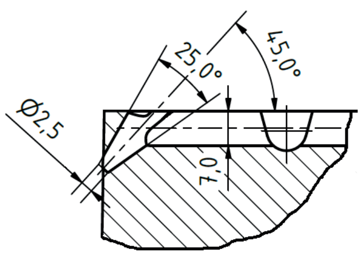

Figure 6 *Submarine-type gate of the optimized design*

the part is completely packed out. Even if reground material is allowed, a large runner should still be avoided because it can increase cycle time. Determining the size of a runner, just like a gate, is a balancing act. To properly size a runner, one has to start at the gate end and work the way back toward the sprue. The shape and corresponding dimensions of the gate are defined using the command "Gate". For the ejection system to function correctly, the injection molded product must remain on the side of the cavity after the mold has been opened. This is ensured using the cold wells, which were defined in the next design step. The next step was to create the inserts for the holes that contain the product. The last step is creation of the mold base. The type of the mold base, its manufacturer and dimensions are selected; next, the position and type of ejectors, sprue bushing and locating ring. Once the mold has been successfully designed, it is still necessary to define the mold processing setup input parameters, such as mold temperature, melt temperature, the maximum injection pressure of the machine and the opening time of the machine fixture. In both variants of the runner system design, the input parameters were the same. The change of parameters does not depend on the shape and dimensions of the system, but are based on the product data sheet that is supplied with the plastic. Then, the analysis can be started using the "Mould filling analysis" command.

Original design

The original design of the runner system, Figure 1, consists of a sprue, which is formed by the sprue bushing and a main runner, which is U-shaped of a depth of 3 mm, Figure 2a. It is further formed by a runner branch, which has the same shape and a depth of 2 mm, Figure 2b. The termination of the runner branch is formed by the gates. A submarine-type gate with a diameter of 1 mm was chosen, Figure 3.

Optimized design

The optimized design of the runner system, Figure 4, consists of a sprue, which is formed by the sprue bushing and a main runner, which is U-shaped of a depth of 8 mm, Figure 5a. It is further formed by a runner branch having the same shape and a depth of 7 mm, Figure 5b. The termination of the runner branches is formed by the gates. A submarine-type gate with a diameter of 2.5 mm was chosen, Figure 6.

2.1 Parameters of the injection molding process

The resulting values of the technological parameters of the analysis for the original design can be seen in Table 1 and for the optimized design in Table 2. The actual filling time represents the filling time of the injection mold cavities. The simulation of the filling time for both designs can be seen in Figures 7 and 8. The main role of injection pressure, in conjunction with injection velocity, is to ensure the volumetric filling of

***Table 1** Parameters of the injection molding process of the original design*

Parameter	Value
Actual filling time	1.50 (s)
Actual injection pressure	180.0 (MPa)
Clamped force area	21397.15 (mm^2)
Max. clamp force during filling	59.38 (t)
Velocity/pressure switch-over at % volume	99.1 (%)
Velocity/pressure switch-over at time	1.48 (s)
Estimated cycle time	15.49 (s)

***Table 2** Parameters of the injection molding process of the optimized design*

Parameter	Value
Actual filling time	1.71 (s)
Actual injection pressure	94.202 (MPa)
Clamped force area	23973.13 (mm^2)
Max. clamp force during filling	54.527 (t)
Velocity/pressure switch-over at % volume	99.02 (%)
Velocity/pressure switch-over at time	1.69 (s)
Estimated cycle time	15.15 (s)

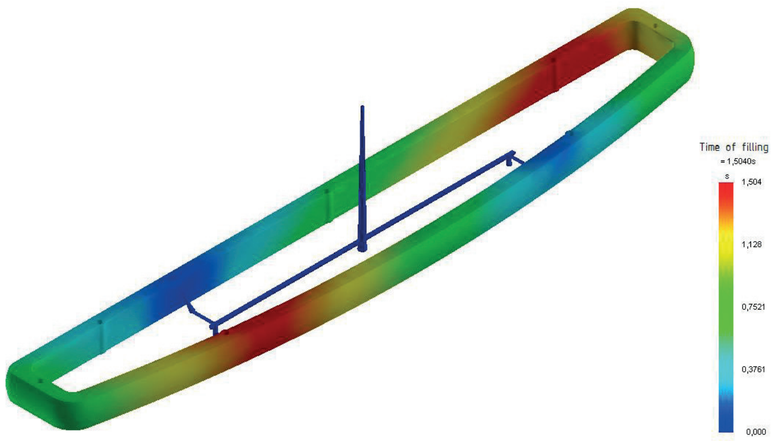

***Figure 7** Actual filling time of the original design*

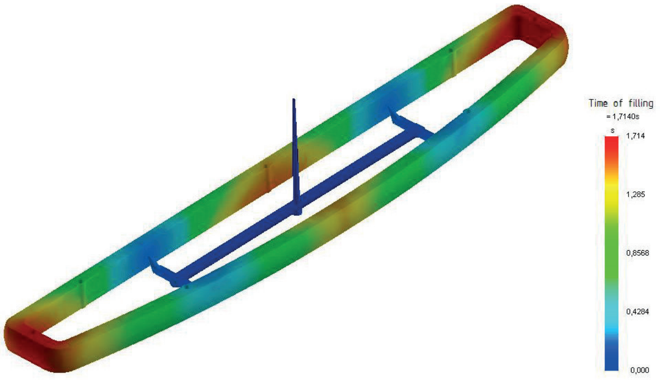

Figure 8 *Actual filling time of the optimized design*

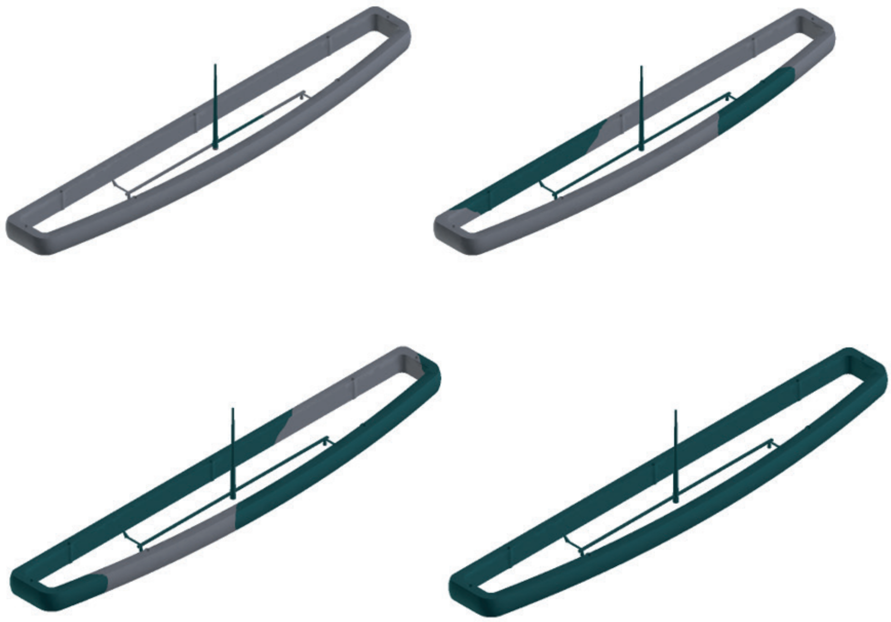

Figure 9 *Simulation of the plastic flow of the original design*

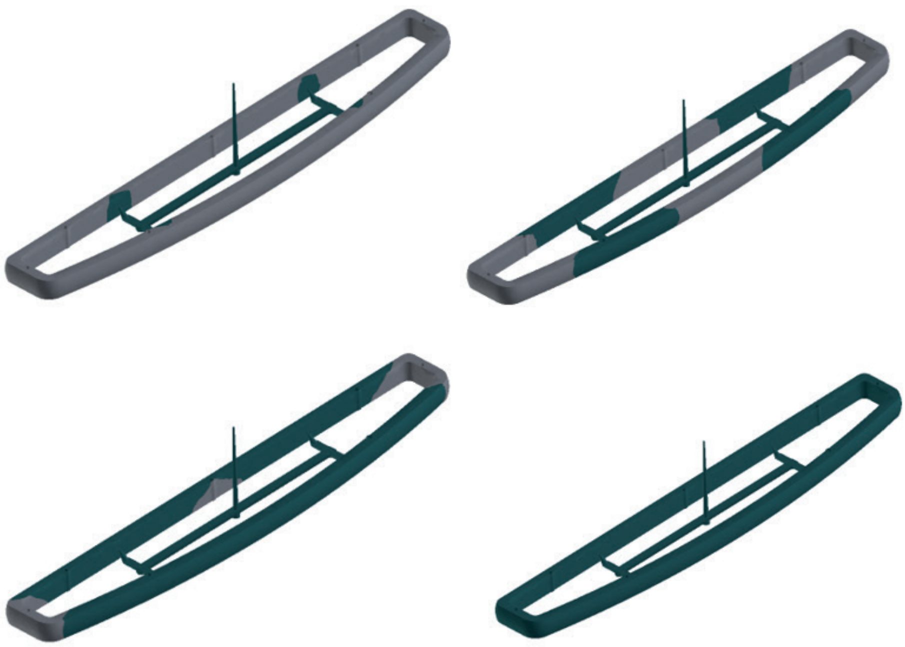

Figure 10 *Simulation of the plastic flow of the optimized design*

the mold cavity with the least possible melt shear stress, which is also related to a reduction in the strength stress of the injection mold. The clamping force is a reaction against the injection pressure and its magnitude must ensure the production of an injection molded product without burrs and overshoots. When switching from the injection pressure to the overpressure phase, there is a change in the control of the injection process. In the injection phase, the flow rate and injection velocity are regulated and in the overpressure phase, the pressure is regulated [4]. The last parameter of the analysis is the estimated filling time. This is composed of five main periods, namely the clamping time, injection time, dwelling time, cooling time, mold opening time and ejection time.

2.2 Simulation of the plastic flow

Figure 9 shows a simulation of the plastic flow in the mold cavity of the original design, Figure 10 represents the flow in the optimized design. The melt advances from the injection unit through the nozzle tip into the sprue and then into the main runner. Next, it passes through the runner branch and flows into the forming cavity through the gate. Simulation of the flow shows that there are no problems and thus the entire cavity space is filled with the melt at the end of the cycle.

2.3 Confidence of fill

Another output of the simulation is the confidence of fill, which shows the probability of filling the cavity area with plastic under the normal injection molding conditions. The simulation result is derived from the pressure and temperature results. If the filling process does not fill the molding cavity and results in a short shot, the design must be modified, the gate location changed, the plastic type reconsidered and the process conditions modified. Appropriate adjustments must also be made if the quality of the injected product is not satisfactory. Figure 11, shows confidence in filling in the original design, Figure 12, shows confidence in filling in the optimized design [17].

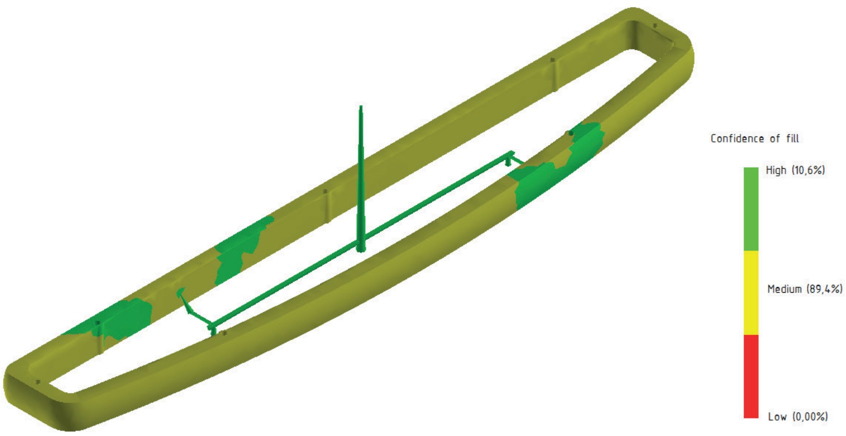

Figure 11 *Confidence of fill of the original design*

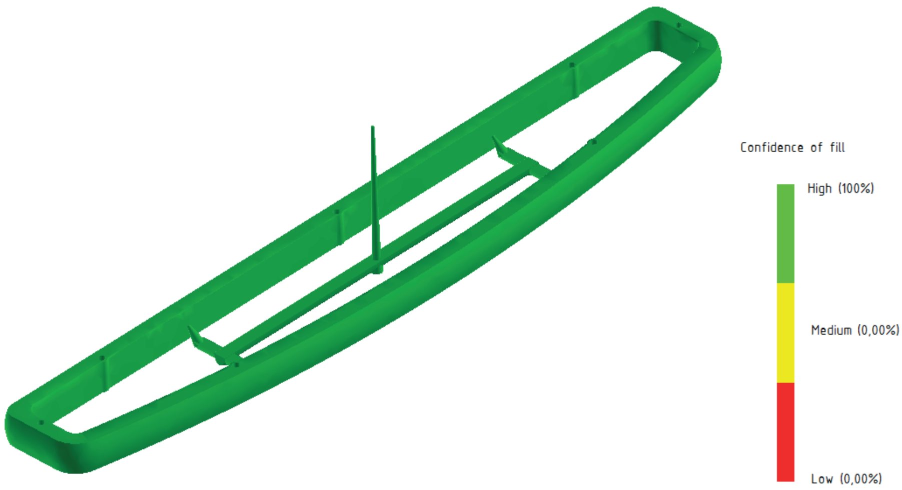

Figure 12 *Confidence of fill of the optimized design*

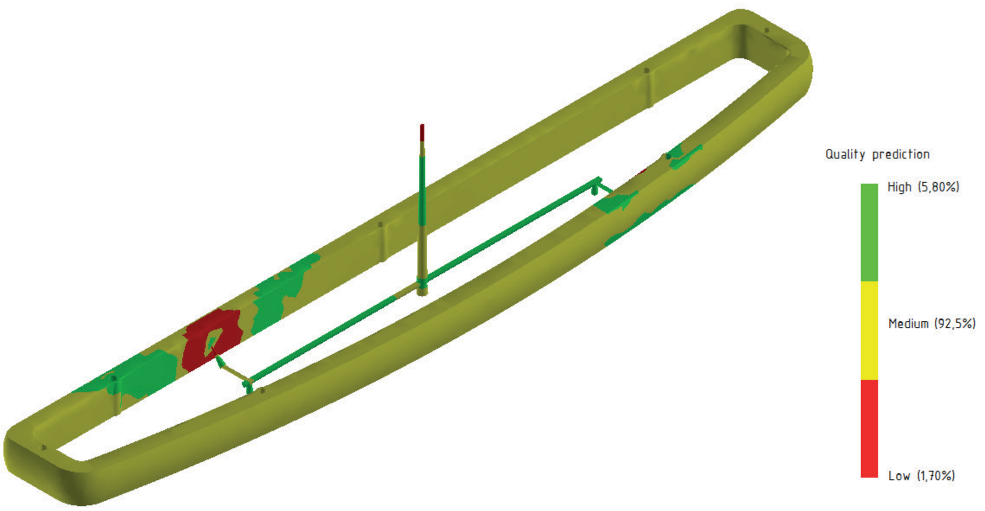

Figure 13 *Quality prediction of the original design*

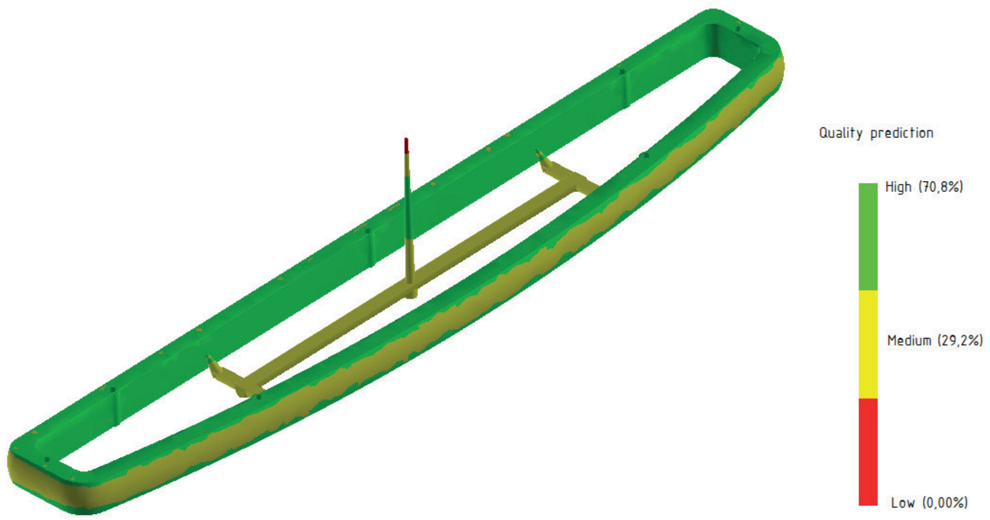

Figure 14 *Quality prediction of the optimized design*

2.4 Quality prediction

The task of the quality prediction simulation is to estimate the quality of the mechanical properties and appearance of the injected product. The basis for this analysis is the functional filling of the mold cavity. The simulation result is derived from pressure, temperature and other results. Figure 13 represents the quality of the injected product of the original design and Figure 14 represents the quality of the injected product of the optimized design. The quality of the parts of the injected product shown in green should be high, the yellow color represents the parts with limited quality and the red color represents the quality problems [18].

2.5 Air traps

When the mold is closed in the injection molding process, the air in the mold cavity is sealed. After a batch of melt is injected, the enclosed air is pushed in front of the melting face, enclosed by it, compressed and excessively heated, which can lead to its expansion. The inability of the air to escape leads to several problems. Expansion is responsible for the local degradation of the melt and as it burns it turns black and its surface contains residues from combustion. Another problem can be insufficient melt flow, resulting in an untouched spot at the edge or in the middle of the shot area or shape. This leads to apparent defects and technical problems such as:

- Insufficient melt flowability - solidification of the melting face,
- Air sealing - appearance of bubbles in the walls of the injected product,
- Increased probability of weld lines leading to reduced local strength,
- Internal stresses,
- Increase in anisotropy of product properties and
  The greater probability of mold corrosion due to

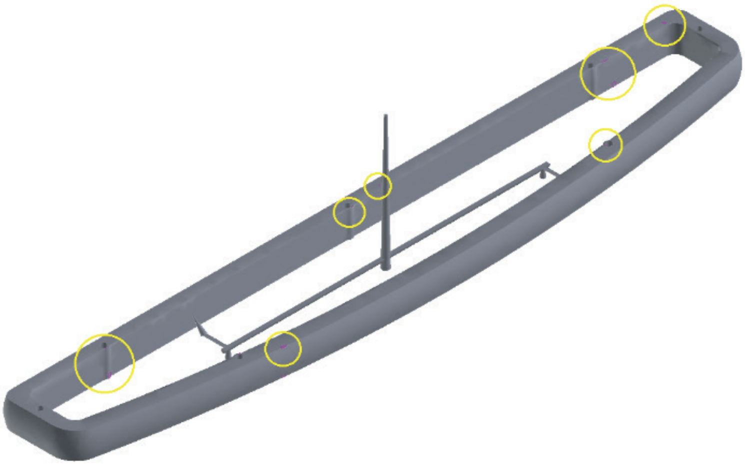

Figure 15 *Air traps of the original design*

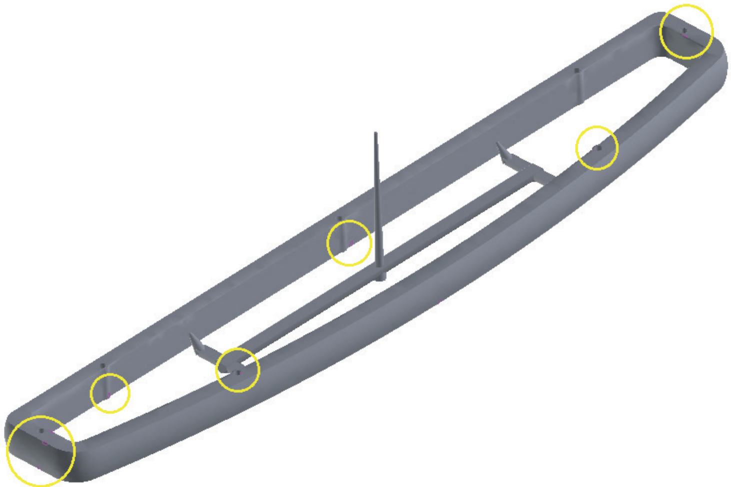

Figure 16 *Air traps of the optimized design*

an increase in temperature and pressure, leading to the release of gases from the injected material that can chemically react with the mold material.

There are several methods to prevent the air trap problems. The first method uses simulation programs output of which is a prediction of the occurrence of air traps. The second method uses the knowledge and experience of the designer, who already considers the method of venting in the mold design process. The last way is to address venting after the mold has been tested, if necessary [4].

In this mold design, the venting of the molding cavity was already considered during the design process. The venting is implemented through the following design modifications. The core is designed as a divided part and consists of a core plate and a core insert. There is a gap between the two to ensure the air leakage. A further modification are the gaps between the ejector holes and the ejectors themselves and the gaps for false cores, which provide sufficient space for air leakage. Figure 15 shows the predicted locations of the air traps for the original design, Figure 16 for the optimized design. The air traps are located close to the holes for the ejectors and false cores, which was also required for the proper function of the injection mold.

2.6 Weld lines

Occurrence of the weld lines is directly related to filling of the molded cavity. A weld line occurs when the main melt stream is split into two or more streams by the mold elements in the mold cavity, or when the melt streams from multiple gates meet. Weld lines virtually always occur on the opposite side of the bypassed shape from the gate. The melt faces of the split flow cold and then join frontally, laterally, or tangentially. The weld lines adversely affect the properties of the product, specifically strength and appearance. Simulation programs can be used to predict the occurrence of weld lines, which would reveal potential problems before the actual mold production [4]. As can be seen in Figure 17 (original design) and Figure 18 (optimized design), the occurrence of weld lines is minimal and they are located at the meeting points of the melt faces.

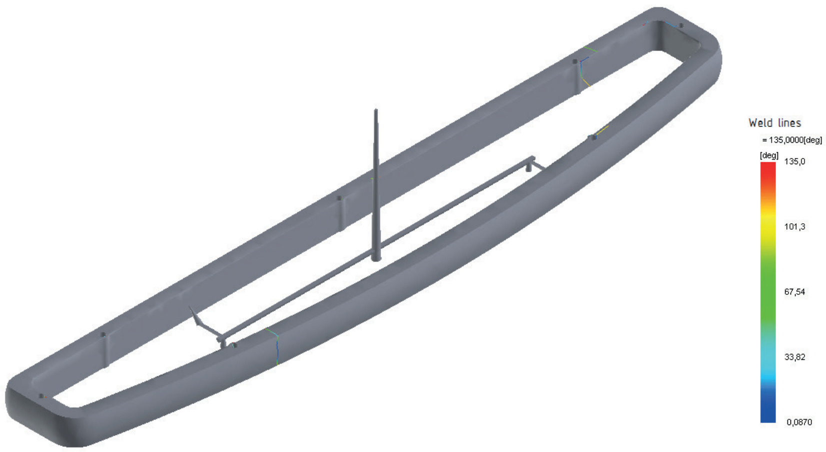

Figure 17 *Weld lines of the original design*

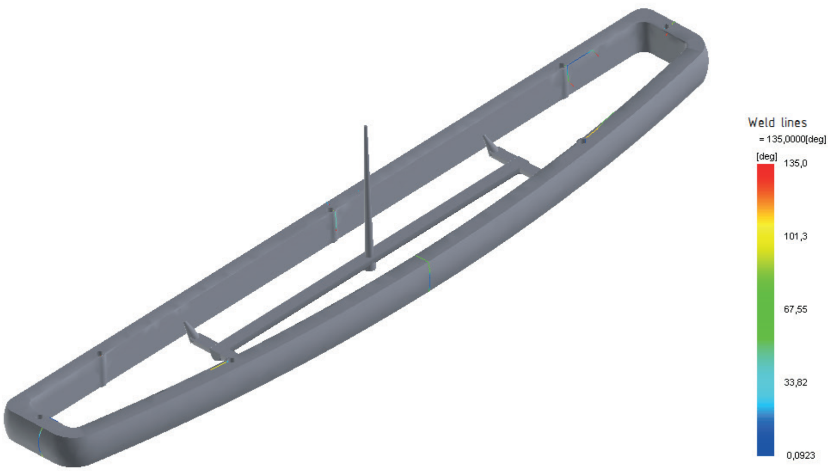

Figure 18 *Weld lines of the optimized design*

3 Conclusions

When designing an injection mold, it is necessary to follow a set of rules and recommendations that are generally known. The mold design process is very complex and consists of several areas. The selection of the type and proper design of the runner system is an essential part of this process. The runner system ensures the transfer of the melt from the injection molding machine to the injection mold. Its function is very important and the overall functionality of the injection process and product characteristics depend on it.

In the field of injection mold design, simulation programs are of great help. They have the advantage of providing the design results before the actual production of the injection mold. This results in a reduction in design and production time, as well as in cost savings associated with the production of prototypes. Autodesk Inventor software program was used for this optimization.

The original design of the runner system was a simple shape with certain dimensions. Based on the analysis carried out, it was found that the dimensions of the runner system do not provide the melt with sufficient space for the proper filling and may be responsible for development of defects. By evaluating the results of this analysis and its technological parameters, it was concluded that the dimensions of the runner system of the original design needed to be optimized. The optimization of the runner system design consisted in modifying the dimensions of the runner system and increasing the number of gates. This solution improved the melt flow into the mold cavity, ensured higher filling confidence and better product quality and eliminated the problems of the original design. In addition, the technological parameters of the injection process were changed, resulting in lower stress on the injection mold in the injection process. The advantage of lower stress is to ensure a longer lifetime of the injection mold.

The result of the runner system optimization process is a solution that meets the requirements for its function. This is proven by the results of the analysis, which are an excellent basis for the subsequent production of the injection mold.

Acknowledgement

The research was supported by the Cultural and Educational Grant Agency of the Ministry of Education, Science, Research and Sport of the Slovak Republic under the project No. 045ZU- 4/2021.

Conflicts of interest

The authors declare that they have no known competing financial interests or personal relationships that could have appeared to influence the work reported in this paper.

References

[1] LENFELD, P. Technologie II. [online] [accessed 2023-02-06]. 2009. Available from: http://www.ksp.tul.cz/cz/kpt/obsah/vyuka/skripta_tkp/sekce_plasty/04.htm

[2] BOBCIK, L. *Moulds for plastics processing: injection molding of thermoplastics. Part 1/Formy pro zpracovani plastu: vstrikovani termoplastu. Cast 1* (in Czech). 2. ed. Brno: Uniplast, 1999.

[3] GRESKOVIC, F., SPISAK, E., DULEBOVA, L., VARGA, J., FABIAN, M. *Plastic processing tools - injection moulds/Nastroje na spracovanie plastov - vstrekovacie formy* (in Slovak). 1. ed. Kosice: Technical University of Kosice, Faculty of Mechanical Engineering, 2010. ISBN 978-80-553-0350-5.

[4] ZEMAN, L. *Injection molding of plastics/Vstrikovani plastu* (in Czech). 1. ed. Prague: Grada Publishing, a.s., 2018. ISBN 978-80-271-0614-1.

[5] Gates and runners - Xcentric Mold [online] [accessed 2023-02-14]. Available from: https://www.xcentricmold.com/gates-and-runners/

[6] ZEMAN, L. *Injection molding of plastics 2/Vstrikovani plastu 2* (in Czech). 1. ed. Prague: Grada Publishing, a.s., 2021. ISBN 978-80-271-1294-4.

[7] Choosing a 3D CAD program for molding design - protolabs [online] [accessed 2023-02-13]. Available from: https://www.protolabs.com/en-gb/resources/design-tips/3d-cad-programs/

[8] IVORY, R. Advantages of CAE/CAM for plastic injection molding [online] [accessed 2023-02-06]. 2014. Available from: https://info.crescentind.com/blog/bid/69398/advantages-of-cae-cam-for-plastic-injection-molding

[9] SKYBA, R., HRCEK, S., SMETANKA, L., MAJCHRAK, M. Creation of calculation 3D model of slewing bearing. In: *Current methods of construction design. Lecture notes in mechanical engineering* [online].

[10] MEDVECKY, S., HRCEK, S., KOHAR, R., BRUMERCIK, F., KONSTANTOVA, V. (eds.). Cham: Springer, 2019. ISBN 978-3-030-33145-0, eISBN 978-3-030-33146-7, p. 149-157. Available from: https://doi.org/10.1007/978-3-030-33146-7_18

[11] STEININGER, J., HRCEK, S., SMETANKA, L., SKYBA, R. Optimisation procedure of inner geometry in spherical roller bearings with regard to their durability. *Scientific Journal of Silesian University of Technology - Series Transport* [online]. 2020, **106**, p. 173-181. ISSN 0209-3324, eISSN 2450-1549. Available from: https://doi.org/10.20858/sjsutst.2020.106.15

[12] HRCEK, S., BRUMERCIK, F., SMETANKA, L., LUKAC, M., PATIN, B., GLOWACZ, A. Global sensitivity analysis of chosen harmonic drive parameters affecting its lost motion. *Materials* [online]. 2021, **14**(17), 5057. eISSN 1996-1944. Available from: https://doi.org/10.3390/ma14175057

[13] HRCEK, S., SMETANKA, L., STEININGER, J., PATIN, B. Increase of durability in the area of line contact with curved body geometry. *Mechanism and Machine Theory* [online]. 2022, **176**, 105010. Available from: https://doi.org/10.1016/j.mechmachtheory.2022.105010

[14] SKYBA, R., HRCEK, S., SMETANKA, L., MAJCHRAK, M. Strength analysis of slewing bearings. *Transportation Research Procedia* [online]. 2019, **40**, p. 891-897. ISSN 2352-1457, eISSN 2352-1465. Available from: https://doi.org/10.1016/j.trpro.2019.07.125

[15] WEIS, P., KUCERA, L., PECHAC, P., MOCILAN, M. Modal analysis of gearbox housing with applied load. *Procedia Engineering* [online]. 2017, **192**, p. 953-958. ISSN 1877-7058. Available from: https://doi.org/10.1016/j.proeng.2017.06.164

[16] FATTORI, J. Tooling: how to properly size, gates, runners and sprues. Part 3. [online]. [accessed 2023-02-17] 2020. Available from: https://www.ptonline.com/blog/post/tooling-how-to-properly-size-gates-runners-and-sprues-part-3

[17] Confidence of fill result - Autodesk [online] [accessed 2023-02-08]. Available from: https://knowledge.autodesk.com/support/inventor/learn-explore/caas/CloudHelp/cloudhelp/2021/ENU/Inventor-Help/files/GUID-ABE48DE8-B01A-4A24-8AB4-E84A674AF3AA-htm.html

[18] Quality prediction result - Autodesk Moldflow Adviser [online] [accessed 2023-02-10]. Available from: https://help.autodesk.com/view/MFAA/2016/ENU/?guid=GUID-4F115A68-4469-4414-BB62-80B341FAC3AF