Műszaki Tudományos Közlemények vol. 19. (2023) 32–36.
DOI: Hungarian: https://doi.org/10.33895/mtk-2023.19.05
English: https://doi.org/10.33894/mtk-2023.19.05

DESIGN OF AN INJECTION MOLDING TOOL WITH EXCHANGABLE INSERTS

Kristóf KERESZTES,[1] Attila Levente GERGELY[2]

Sapientia University of Transylvania, Faculty of Mechanical Engineering, Târgu Mures, Romania

[1] keresztes.kristof@student.ms.sapientia.ro

[2] agergely@ms.sapientia.ro

Abstract

The aim of this work was to design an injection molding tool with replaceable inserts. The tool design is made for a BOY 15S injection molding machine. Based on the interpretation and comparison of the simulation results of the filling process, we optimized the plastic part design, the runner system and the location of the gates and cavities in order to maximize productivity and efficiency while minimizing cost. In the process of the tool design, we also considered the potential problems regarding the manufacturing process of the tool.

Keywords*: injection molding, tool design, optimization, simulation.*

1. Introduction

1.1. The basics of injection molding

The basic principle of injection molding is the "injection" of a low viscosity plastic melt into a closed cavity. This is done through a narrow passage. Low viscosity is essential for a fast and complete filling of the mold. Complete filling (for thermoplastics) is also facilitated by high injection pressure. Due to the fundamental nature of the injection molding process, the cooling process of the melt only lasts a few seconds at most and results in a final product of high dimensional accuracy. In addition, the advantage of injection molding is that it allows the production of complex shaped parts that require no further machining other than the removal of plastic that has solidified in the runner system.

1.2. The injection molding machine

The injection molding machine is responsible for melting the plastic (usually in the form of granules) and injecting it into the mold. It is also responsible for opening the mold along the parting plane and operating the ejection system. Its mechanism is optimized for fast and efficient injection molding and ejection of the product after solidification.

The mold is designed for the BOY 15S injection molding machine located at the University of Sapientia EMTE Târgu Mures, Faculty of Mechanical Engineering.

1.3. The injection molding tool

The injection molding tool is the component that allows the shaping of the part. It contains the runner system and the mold cavities that are filled to create the part. The type of tool used depends on the part. The production of more complex parts requires more complex tooling, so it is important to think cost-effectively from the very first steps of part design. The mold designed in this article has a simple, two-plated design) (**Figure 1**).

It is also the tool's task to convey the melt from the nozzle of the injection molding machine to the mold cavity, this is done by means of the inlet / runner system.

The opening through which the plastic melt enters the mold cavity from the runner system is called the gate. The size and position of the gates have a profound influence on the quality of the final product. It can sometimes make the difference between a perfect and a defective part, so correct sizing and positioning is of particular importance.

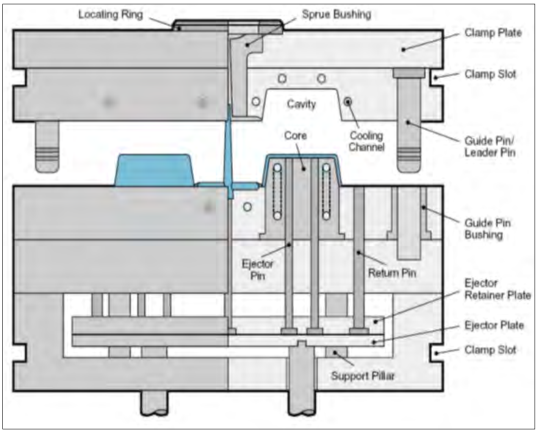

Figure 1. *Construction of a two-plated injection molding tool.* [1]

2. Methodology

Autodesk Inventor was used to create the assembly model of the plastic part, inserts and tool housing, and to perform the stress analysis. The plastic flow of the melt was simulated using SolidWorks Plastics. The parameters required to run the simulations were chosen from the table of the parameters of the BOY 15S injection molding machine.

In the calculations, at each step, we chose the less favorable parameter of either polyethylene or polypropylene, therefore the results provide a significant safety factor.

2.1. Tool housing

In designing the tool housing, the selected components were based on the simple two-plated tool structure. The correct movement of the two tool halves during tool opening and closing process is ensured by pins and corresponding bushings, the 3D models of which were downloaded from the online parts catalogue of Meusburger GmbH & Co KG.

2.2. Inserts

The design process of the inserts consists of several steps. The first step is to design the shape of the mold cavity, which is determined by the plastic part. When designing the plastic part (and therefore the mold cavity as well), it is important to avoid shapes that are problematic for injection molding. Examples include right-angled corners, thin recesses, sections with staggered wall thicknesses, etc.

3.Results

3.1. Cavity filling simulations

To perform the simulations, a plastic component model is needed. This model forms the final product.

3.1.1. Determining the optimal gate position

The first result of the evaluation of the simulations is the determination of the optimal gate position. This was chosen based on the number and size of air pockets and weld lines that appear in the simulations, since these are manifested as aesthetic defects in the final product [2]. The overall quality of the filling process was also taken into consideration.

In determining the optimal gate position, injection pressure, and a gate dimension were kept constant so that the quality of the filling process could be studied as a function of gate position.

In addition to the results of simulations with different barrier positions, cost-effective design also played an important role in this decision.

Figure 2 shows the final gate position. This will ensure that the melt front fills the model's protrusions longitudinally, which helps to eliminate weld lines. [3]

3.1.2. Determining the optimal plastic part model

The plastic part model represents a keychain with the logo of the Sapientia University of Transylvania. The prominent parts of the 3D model create a number of geometric corners, which present difficulties for the injection molding process. In the optimization process of the model, the priority was to change these problematic shapes

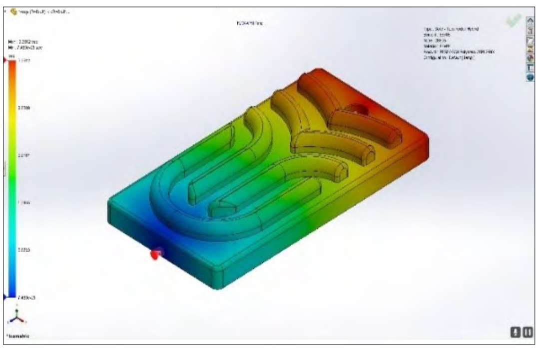

Figure 2. *Optimal gate position for symmetrical filling.*

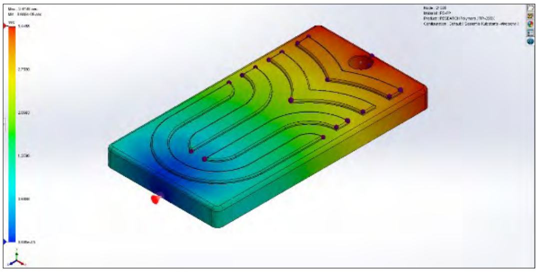

Figure 3. *Optimized model with reduced and chamfered protrusions.*

in such a way that the deviation from the original part was minimized (**Figure 3**).

The simulations helped in the optimization of the plastic part model to ensure that the cavity is fully filled. This was done by modifying the geometric features that result in air pockets and weld lines. [3]

3.2. Design of the injection molding tool

3.2.1. Design of core and cavity retainer plates

Due to the interchangeable inserts of the mold, the space required for the insert must be provided in the two mold halves, which measure 80 × 100 [mm × mm]. The inserts have a thickness of 15 mm, and they are positioned 13 mm deep in the two retainer plates to ensure full contact of the inserts - and only the inserts - during the closing of the tool.

The ejector rods must penetrate the moving side of the core retainer plate to allow ejection of the plastic part. This can be achieved by drilling holes, but this would mean that different holes would have to be drilled to accommodate different inserts (which have varying rod sizes and placements). As one of the main concerns is to facilitate the interchangeability of the inserts, this solution is not ideal.

In order to make the moving side of the tool suitable for different inserts, a through-slot must be designed in the core retainer plate. The dimensions of this slot are slightly smaller than the size of the insert, as shown in **figure 4**. This is done to achieve a shoulder that provides a support surface for the insert..

To verify the functionality of the mold base, stress analysis was performed. The maximum load is known due to the injection molding machine parameters (50 tons). The analysis was performed using AutoDesk Inventor Stress Analysis (**Figure 5**). The material used in the simulations is AISI 1045 steel. The mechanical properties of this

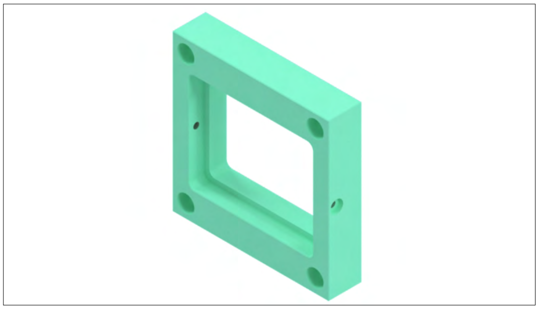

Figure 4. *More retainer plate machined for insert accommodation and ejection system operation.*

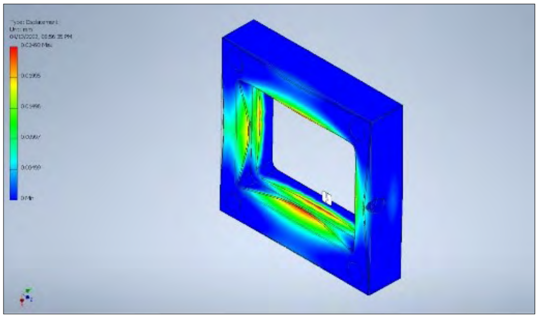

Figure 5. *Deformation of the moving side mold base under 50 tonnes of load force (AutoDesk Inventor Stress Analysis).*

Figure 6. *Side view of the parting plane (marked with a blue line at the bottom of the part).*

material can be found in the AutoDesk Inventor material catalogue. The Young's Modulus of AISI 1045 is around 200 GPa.

The maximum deformation at 50 tonnes of closing force did not exceed 25 µm, as it is shown in **figure 5**. This value is low enough to be considered negligible for injection molding.

3.2.2. Determining the parting plane

When designing an injection molding tool, it is important to consider the position of the parting plane. Since cost-effective design is a major objective, the part's base plane has been chosen as the parting plane (**Figure 6**). This offers several advantages.

Primarily, it ensures that the part remains in the moving side of the mold during mold opening, thus making part ejection possible. Next, the position of the parting plane means that only one of the inserts needs to be machined, reducing production costs and time. There is one drawback to the choice of this particular parting plane: the edges of the part that are in the parting plane cannot be rounded, nor chamfered.

3.2.3. Runner system and cavity design

The inlet channel diameter was chosen to be 2 mm, as this diameter is suitable for injection molding all the listed plastics for the designed mold (**Table 1**), since the maximum flow path is 70 mm (from the nozzle to the gates).

The relationship between the diameter of the sprue and the diameter of the runner system channels is given by the following formula:

$$D_{previous} = D_{next} \cdot \sqrt{n} \tag{1}$$

where n is the number of channels in the junctions, $D_{previous}$ is the diameter before the junction and D_{next} is the diameter after the junction.

The runner diameter calculated from the size of the sprue is 2,82 mm. Since the sprue channel is tapered (0,5° taper angle), the value obtained is the average diameter. Among the sprues distributed by Meusburger GmbH, one can be found with an average diameter of 2,91 mm. This is sufficiently close to the calculated value.

The positioning of the cavities is determined by the factors previously mentioned: the chosen gate position and the dimensions of the base plates. Since the dimensions of the insert are 80 x 100 [mm x mm], four cavities can be placed leaving a sufficiently thick (6 mm) wall between the tool cavities and the outside of the insert (**Figure 7**).

The volume calculated from the 3D model of the designed plastic part is 2.35697 cm^3, so the volume of the four cavities is 9.42788 cm^3. The combined volume of the inlet manifold and runner system is 3.80619 cm^3, so the volume of polymer melt required for one injection molding cycle is 13.23407 cm^3. The maximum injection volume per cycle is 96 cm^3.

3.2.4. Determining the dimensions of the gates

When designing the gates, the aim is to minimize the size of the gate. The reason for this is to make the removal of plastic solidified in the gates easier from the product of the injection molding cycle.

Table 1. *Maximum flow path of plastics in a 2 mm diameter channel* **[4]**

Plastic	Max. flow path in a 2 mm diameter channel [mm]
Polyethylene (LDPE)	720
Polyethylene (HDPE)	400
Polypropylene	870
Polystyrene	670
Polyamide 66	560
Polyamide 6	510
ABS	370

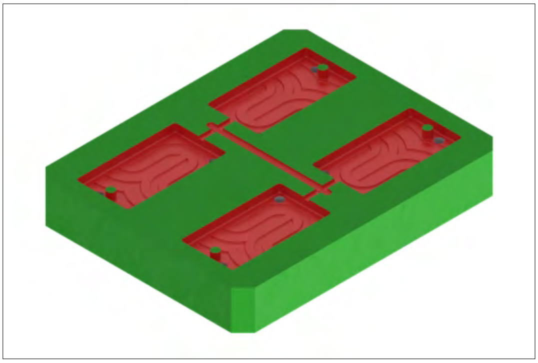

Figure 7. *Model of the moving side insert.*

The limiting factors are the maximum shear rate and the calculated pressure drop at the gates. These values depend on the used plastic **[5]**.

In the first step, the maximum shear rate was considered: **[5]**

$$\dot{\gamma} = 6 \cdot \dot{V}/(w \cdot h^2) \tag{2}$$

where $\dot{\gamma}$ – shear rate [1/s], $\dot{V}$ – volume flow [cm^3/s], w – gate width [m] and h – gate height [m].

The final dimensions of the barriers are: w = 2 mm, h = 0,75 mm. The resulting shear rate: $\dot{\gamma}$=17645.426 1/s. Among the different polyethylene and polypropylene types, the lowest maximum shear rate value was found to be that of HDPE and LDPE: 40 000 1/s **[6]**, which is not exceeded by the calculated result.

Next, the calculated pressure drop across the entire runner system was investigated. From the results of the simulations, the pressure drop is found to be less than 75 MPa. The used injection molding machine is able to produce more than twice the calculated pressure (P_{max} = 168 MPa).

3.2.5. Ejection system design

The ejector rods used in this design are cylindrical and have a flat face. In order to calculate the minimum diameter, we had to determine the force required for ejection using formula (3) **[5]**. The value of this force is 1970 N (rounded up).

$$F = \mu \cdot cos(\Phi) \cdot E \cdot CTE \cdot (T_2 - T_1) \cdot A_{eff} \qquad (3)$$

where μ – coefficient of friction, E – Young's Modulus [Pa], CTE – coefficient of thermal expansion [1/°C], A_{eff} – shrinkage area of the plastic piece [m^2], ε – material specific elongation, Φ – taper angle of the wall of the tool cavity that is stressed during the shrinkage of the plastic [°].

Knowing the force required to eject the parts, the minimum diameter required to resist the shearing force by the plastic parts can be calculated. **[5]**

$$\Omega_{ejectors} > 2 \cdot \frac{F_{ejection}}{\sigma_{max} \cdot h} \qquad (4)$$

where $\Omega_{ejectors}$ – circumference of the face of the ejector rods [m] (shear line), σ_{max} – yield strength [Pa] and h – wall thickness of the plastic part at the point of contact with the ejector rod [m].

The resulting minimum total circumference of the ejector rods is 41.042 mm. The designed inserts have 8 ejector rods, so the minimum diameter of one ejector rod is 1.63301 mm. Including a safety factor, the rounded value is 3 mm; such ejector rods are available from Meusburger GmbH, which simplifies sourcing and obtaining the part.

The ejector rods were also checked for deflection: their stress was 34.83725 MPa, while the maximum allowable value was 2.86467 GPa.

The shape of the plastic part limits the positioning of the ejector rods, which can only come into contact with the plastic part where its surface is flat (**Figure 8.**).

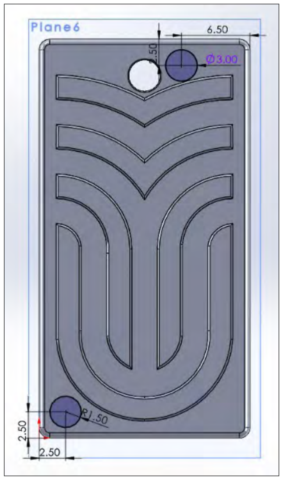

Figure 8. *Positioning of ejector rods on the plastic part.*

4. Conclusions

We have successfully completed the design of an injection molding tool with exchangeable inserts. Thanks to the mold's design, it will be possible to injection mold different plastic parts without the need for the lengthy and costly process of changing the tool. Thanks to the implementation of several safety factors, the tool will be suitable for analyzing the injection molding process of different types of plastics.

This has been the first step towards enabling students at our University to study the injection molding process with an industrial injection molding machine.

References

[1] Mikó B.: *Műanyagfröccsöntő szerszám tervezése és gyártása*. Budapest, 2006. 1–6.

[2] Seow L. W., Lam Y. C.: *Optimizing Flow in Plastic Injection Molding*. Journal of Materials Processing Technology, 78/3. (1997) 333–341. https://doi.org/10.1016/S0924-0136(97)00188-X

[3] Mehdi M.: *Intelligent Optimization of Mold Design and Process Parameters in Injection Molding*. Springer. Springer International Publishing, Cham, 2019. https://doi.ord/10.1007/978-3-030-03356-9

[4] Dunai A., Macskási L.: *Műanyagok fröccsöntése*. 2003. 357–358.

[5] Kazmer O. D.: *Injection Molding Design Engineering*. Hanser Gardner Publications Inc, München, 2016. 176–178, 265–273.

[6] Maximum of Shear Stress and Shear Rate. 2021.