

PREVENTING FLASH OCCURRENCE IN THE INJECTION PLASTIC MOULD

MARTIN HABRMAN ¹, ZDENEK CHVAL ¹, KAREL RAZ ¹, PHILIPP STREIT ², DOMINIK STUEMPFL ²

¹ University of West Bohemia, Faculty of Mechanical Engineering, Regional Technological Institute, Pilsen, Czech Republic

² Ostbayerische Technische Hochschule Amberg-Weiden, Fakultät Maschinenbau, Umwelttechnik, Amberg, Germany

DOI: 10.17973/MMSJ.2022_10_2022096

mhabrman@rti.zcu.cz

This article deals with the injection moulding into the plastic mould. This is an unconventional approach to injection moulding, especially because the 3D printing is used to produce the mould. The mould with the same shape was made of a conventional material (aluminium) for comparison. The properties of the plastic mould were then compared to the aluminium mould. The flash was seen as the main shortcoming. The modifications of the mould were suggested in two variants, one of them was implemented.

The advantage of the flash solution was described. Finally, other solution options were summarized based on the performed testing.

KEYWORDS

Injection moulding, aluminium mould, 3D printed plastic mould, flash

INTRODUCTION

Plastic injection moulded products are produced by the means of heat and pressure. This production process takes place on an injection moulding machine. The granulate is heated and homogenized in the injection unit and melted there. The melted material is injected into the mould by the injection pressure. The mould is placed between two plates and consists of a movable and a fixed part. There is a dividing plane between these two parts of the mould. Due to the pressure that tends to open the mould, it is necessary for the clamping unit to exert a closing force. Pressure resistance is required from the mould material. The closing force has to be stronger than the opening force. After injection and cooling of the moulded part to the ejection temperature, it is ejected from the mould. In the meantime, the melted plastic material is prepared in the injection unit for the next moulded part. Replacing of the mould is both time consuming and expensive. For this reason, materials with a longer service life are chosen for the injection moulds. The most often chosen materials are steel, aluminium etc. Aluminium usually has a shorter service life. However, it is easier to mill than steel. [Wang 2018], [Shoemaker 2006]

This article is focused on a small series production by injection moulding. For a small, planned number of moulded parts, plastic material was chosen for the mould. Plastic is not expected to have its service life as long as the commonly used conventional materials. However, it is sufficient for the testing and production of moulded parts up to a hundred pieces. Another advantage is the fact that it for 3D printing is easier to use plastic than metal. 3D printing is very close to the production method known as Rapid Prototyping. The essence of this method is to produce a product from a 3D model as fast

as possible. The production process is also characterized by the shortest possible path to the product. Only a 3D CAD model was needed for the 3D printed mould. It was necessary to make a drawing for the aluminium mould, create a program for machining, etc. The 3D printed mould was many times easier to produce. 3D printing was also chosen to produce the mould.

The plastic material is HP PA12GB filled with glass fibres and the 3D printer used is HP JET FUSION 4200 series. All made by the same manufacturer, Hewlett-Packard Company. The raw material used in HP PA12 is powder. The 3D printed part is produced using the patented Multi Jet Fusion method from Hewlett-Packard Company. This method is part of the Powder Bed fusion method. The accuracy of the final part is up to 0.1 mm. Also, their heat resistance (according to the datasheet of the material) is sufficient for the planned number of moulded parts. [Wang 2018], [Shoemaker 2006]

INJECTION MOULDING

The injection moulding was performed into both the plastic mould from HP PA 12 and the mould made of a conventional material. Aluminium was chosen to facilitate the milling of the mould.

The 3D model of the mould was the same for both technologies. The milling of the aluminium mould was performed according to commonly used tolerances. The accuracy of a 3D printed mould depends only on the accuracy of the 3D printer. However, the flatness of the 3D printed mould is worse than the one of the milled moulds. The flatness depends on the clamping and movement of the cutter in the case of a milled mould. The flatness of the 3D printed mould depends on the printing process used. The flatness of a 3D printout created using the Multi Jet Fusion, Powder Bed fusion 3D printing process depends on the orientation of the parts in the powder container. The recommended orientation with an inclination of 45 ° was used for the production of the mould. The deviation was greater than 0.1 mm, both in the surface in the dividing plane and on the opposite side. Unfortunately, the parallelism has not been analysed. It would be useful to investigate the effect of the container orientation on the accuracy of the 3D printout on the future. Under current conditions, the aluminium mould was more accurate. [Markovicova 2019], [Kim 2016], [Shoemaker 2006]

A specially modified mould was designed for injection moulding. Its advantage is the possibility of producing cavity as a plastic insert. Its use saves the amount of material needed. This specially modified mould is in section in Figure 1.

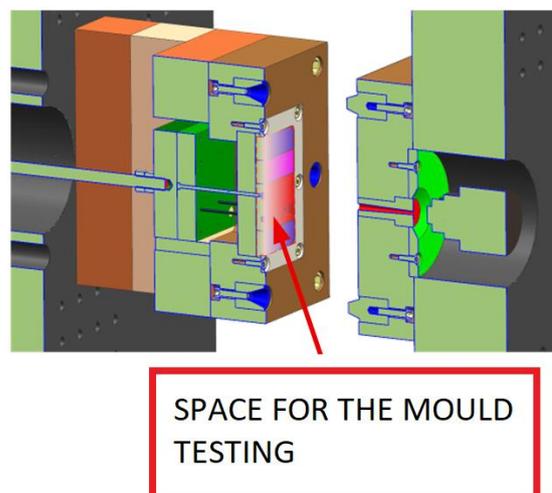


Figure 1. Specially modified mould

The fixed mould plate was modified for this testing. There is the space for the cavity parts similar like on movable mould plate.

There is marked space for testing mould part in the Figure 1. This space is used for inserting part of mould. These parts are shown in Figure 2.

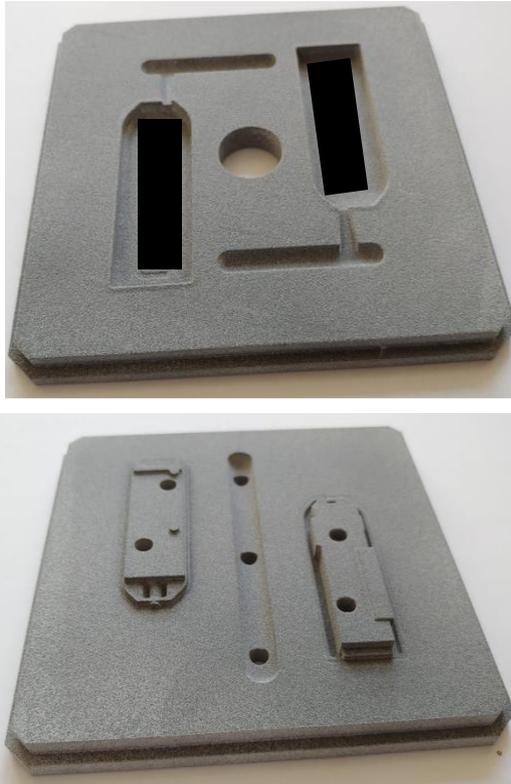


Figure 2. Fixed (bottom) and moveable (upper) mould parts

The logo of customer is hidden in the Figure 2 by black colour. Fixed mould is on the left side and moveable mould is on the right side in the Figure 2. The mould occupies the entire space for mould testing. The mould size is 121x121mm. The height of the mould is 10 mm. Due to the shape of the cover, split injection channels were designed. The inlet channel is purchased and made of steel because the fix mould has the hole in the middle (steel insert is used). The semi-circular canal has a radius of 3 mm. The cross-section of the gate (inlet) is 6x2.5 mm for both types of cover.

The injected part is the cover of the electronics (flash drive). The size of the part is 60 x 20 x 10 mm.

The modified mould with the plastic insert was mounted on the electrical moulding machine Arburg 470 E1000 – 29. The values of basic moulding parameters were set for all moulds. The moulding values are shown in Table 1.

Parameter	Value	Unit
Injection temperature	200	°C
Injection volume	13	cm ³
Injection pressure	350	bar
Filling time	0,3	sec
Packing volume	5	cm ³
Packing time	10	sec
Colling time	60	sec

Table 1. Basic injection parameters

The testing was performed under the conditions shown in Table 1. The volume of the injected part with the runner system is 12.46 cm³. There is a volume cushion 0.54 cm³. Parameters in Table 1 were set with respect to the chosen injected material. The most frequently used injection material is Polypropylene (PP) and was therefore also chosen as a suitable material for

the testing. The base values are shown in Table 2. The producer is INEOS Group Ltd. [INEOS 2008]

Parameter, Test conditions	Value	Unit
Test Methods		
Melt Flow Rate, 30°C/2.16kg ISO 1133	25	g/10min
Flexural Modulus, 23°C ISO 178	1 200	MPa
Tensile Strength ISO 527-1, 2	32	MPa
Notch tensile strength, 23°C ISO 180/1A	3.2	kJ/m ²
Notch Charpy Impact Strength, 23°C ISO 179/1eA	2.4	kJ/m ²

Table 2 Basic parameters of PP [INEOS 2008]

The resulting moulded part can be seen at Figure 3. The test data and the injection volume were written on moulded parts. The part produced by using the aluminium mould is shown in Figure 3 left and the part produced using the plastic mould on the right side. The flash on the part produced using the plastic mould is marked with a description.

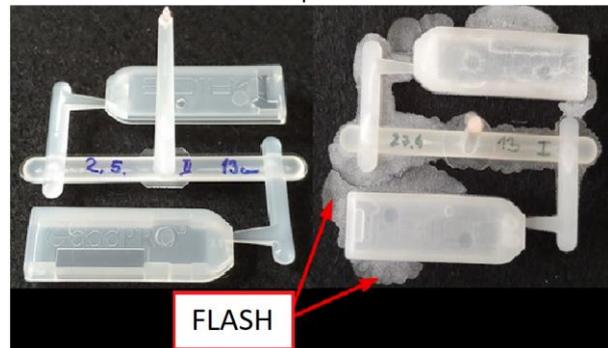


Figure 3. Injection moulding into left – Aluminium mould, right – HP PA 12 mould

Figure 3 clearly shows the difference between the injection moulding into aluminium and plastic moulds. The part produced using the aluminium mould is transparent, clear, and has a good surface quality.

The part produced using the aluminium mould has better properties. Another visible disadvantage of 3D printed plastic moulds is the flash. This article focuses on this problem. [Soskic 2021]

One of the possible causes of poorer surface quality of injected part is roughness of mould. It can also cause the flash. Too much roughness is causing the leakage of the melt. It will also verify the accuracy of the 3D print surface quality. The roughness of the aluminium mould was produced by the final value Ra 3.2. A HOMMEL-ETAMIC T8000 device was used to measure and verify the roughness. The sensor used was TKU300 with measuring range 300 µm. The measured length was 4.8 mm. The evaluated value is Ra 3.12 micrometres. The roughness of the aluminium mould is made within tolerance. The roughness of the 3D printed parts is not stated by the printer manufacturer. This value depends on the capabilities of the 3D printer and cannot be changed so much. There is also the effect of positioning of the model in the printing container. This effect has to be analysed in future. An Olympus LEXT confocal microscope was used to measure the roughness of the 3D printed mould. The measurement made in two mutually perpendicular directions. The number of fields of view in each direction was 10. The lens shot five times during the measurement. The analysis was performed by aligning the samples using three points. This suppressed the noise reduction. λc coefficient was automatically assigned 0.8 mm = 800 µm. The roughness of the evaluation according to the

standard ISO 4287 was $Ra = 10.7$. The roughness of the 3D printed plastic mould is much greater than the roughness of the aluminium mould. The shape of the flash is not around all injected part uniform. The more roughness certainly causes the flash but not so much in this case.

FLASHING

The overflow of plastic into the parting plane is called the flash. Flash occurs due to the gap existing at the parting plane via which the melted plastic flows outside the mould cavity. The flash has to be cut out of the moulded part. However, it is better to avoid it. Removing of it costs time and money. It is much better to focus on the reason of the flash creation.

The flash can be caused either by the design of the mould or by the improperly chosen injection moulding process. Of the process settings, the most frequent causes of the flash are the following: A small closing force on the injection moulding machine, which is caused by an improperly selected injection moulding machine. The force resulting from the expansion of the melt then easily overcomes the closing force and the mould opens. Therefore, the closing force of the injection moulding machine must be considered when selecting the injection pressure. Furthermore, an improperly selected process parameter may be too high melt temperature. A higher melt temperature improves the fluidity of the melt. This also leads to melt leakage more into the parting plane. The design of the injection mould can affect the flash as follows: First influence is the improper venting system, allowing the melt to flow into the venting system. However, both moulds do not have a venting system and therefore this issue can be ruled out. Second influence is the gap between the moving and the fixed part of the mould. Due to the manufacturing accuracy of the 3D printed mould, the gap can cause this flash. [Godec 2021] [Shoemaker 2006]

One of the solutions is to increase the thickness of the mould in order to have both plastic inserts in perfect contact (there will be contact only between these two plastic parts). The increased thickness is by 0.2mm. The result will be a contact of two moulds. There are two options how to increase the thickness of the mould. Full-area or only local offset from the edge of the cavity. Design of the full-area offset is shown in the Figure 4.

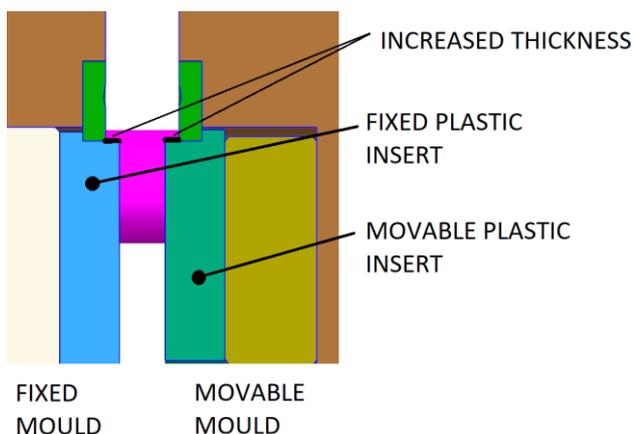


Figure 4. Design with the increased thickness of both plastic inserts

It should be noted that these are the plastic moulds. The PA12 plastic has a lower pressure resistance than aluminium. There is a risk that it will be deformed by the clamping force. Therefore, it was decided to deform the local offset from the edge of the cavity. There are many possibilities to solve it. The aim of this article is not to analyse their different shapes. The designed shape can be seen in Figure 5 on the right side. The local offset

from the edge of the cavity is shown by the hatched area (in cross-section). The dimensions of this modification are also in Figure 5. The goal of this modification is to stop the flash creating. The name of this geometry modification is the flash stopper.

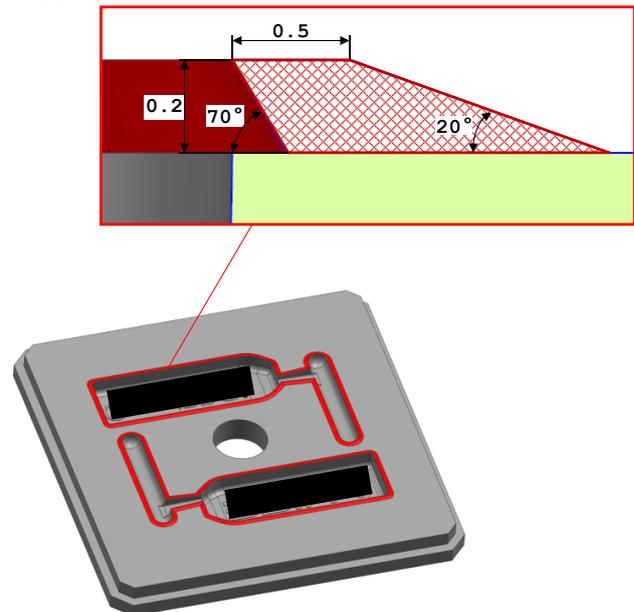


Figure 5. Flash stopper model

The flash stopper is shown in red in the Figure 5. This was done both in the cavity and in the canals. If it were not done at the canals, there would be a risk of the flash forming in the parting plane. The height of the flash stopper is the same as the flash height. This height of the flash was measured on the injection moulded part. Angles are designed with respect to the assumed deformation. The deformation must not occur on the 20° side. The deformation of the flash stopper is formed by the 70° chamfer. These parameters are shown in the detail in the Figure 5. The deformed part of the flash stopper must not interfere with the cavity. [Spirk 2017] [Kim 2016]

SOLUTION TESTING

The same settings used on the machine during the testing of the previous moulds were used in this case as well.

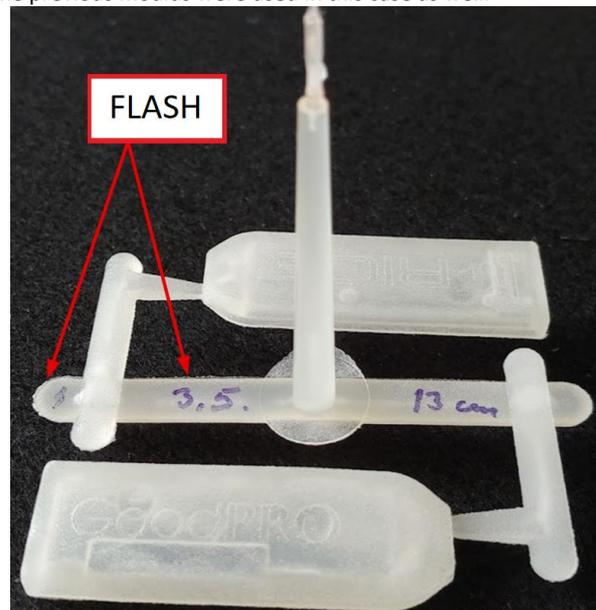


Figure 6. Injected part into plastic mould with the flash stopper

The blocking was successful, and the flash was not formed, as you can see in the Figure 6.

Figure 6. shows also the injection moulded parts still with the runner system. This runner system will be separated from the injection moulded parts. Fortunately, the flash stopper successfully prevents the flash formation and the injection moulded part was created without flash. This solution is therefore successful. A small part of the flash can be seen along the channel but not around the inlet channel. There is a gap intentionally left between the inlet purchased steel channel and the moveable mould. The volume of the flash on the channels is minimal, as seen in the picture Figure 6.

CONCLUSION

The successful solution of the flash was introduced and tested in this article. The variant with the flash stopper saved printing time and plastic material (it means the waste in the flash). This variant was chosen because of this reason. The lifetime of the mould was planned to be 100 pieces. This variant was more time consuming for preparing than the variant with an increased thickness of the plastic mould. It was necessary to do the profile of the flash stopper around the edge of the mould cavity and channels. The option with increased thickness of the entire surface was also tested. The result was the same, it means the injection moulded part without a flash. However, this second option is already more demanding to maintain the accuracy of the entire surface during 3D printing (the effect of the heat deformation can occur more often within thicker part. The effect of mould deformation on the accuracy of the moulded part was not investigated. The disadvantage of this variant is the bigger loading of the plastic insert with higher thickness. The clamping force is transferred to the frame of the mould. Especially the plate under the mould part is more stressed by bending. Both variants solved the problem with the flash. It is up to the designer to decide whether to save material or time. As we can see, it is possible to solve the problem of moulds by design.

Another option is to solve the problem with the flash by changing the processing parameters. Reducing the injection temperature or pressure also helped significantly.

Based on the implementation of this experiment, the problem with the flash in plastic mould can be summarized as follows: If the flash occurs, first adjust the process parameters (it doesn't work all the time). This is an easy adjustment that can help. If the flash remains on the injection moulded part, adjust the mould design. This research will continue with the testing of various designs of the flash stopper with various dimensions.

ACKNOWLEDGMENTS

This paper has been prepared under project 'Research of additive technologies for future usage in mechanical engineering - RTI plus', CZ.02.1.01/0.0/0.0/18_069/0010040 of the Pre-application research for ITI of the Ministry of Education of the Czech Republic aimed at supporting research, development and education.

REFERENCES

[Wang 2018] Wang, M.L., Chang, R.Y. and Hsu, Ch.H. Molding Simulation: Theory and Practice. Munich: Hanser Publishers, 2018, ISBN: 978-1-156990-619-4

[Shoemaker 2006] Shoemaker, J. Moldflow Design Guide, Munich: Hanser Publishers, 2006, ISBN: 978-1-56990-403-9

[Spirk 2017] Spirk, S., Krizek, M. and Jenicek, S. Polyurethane foam behaviour during impact. Machine Modelling and Simulations, Skelne Teplice, Slovakia, 2017. Zilina: Editing Centre of University of Zilina DOI: 10.1051/mateconf/201815702050

[Kim 2016] Kim, H., Zhao, Y., and Zhao LH. Process-Level Modelling and Simulation for HP's Multi Jet Fusion 3D Printing Technology. 1st International Workshop on Cyber-Physical Production Systems (CPPS), Vienna, Austria, 2016. Vol. 1

[Markovicova 2019] Markovicova L., Zatkalikova, V. and Hanusova, P. Carbon Fiber Polymer Composites. Conference Quality Production Improvement – CQPI. Poland, 2019. Poland: Sciendo, pp 276-280

[Soskic 2021] Soskic, Z. et al. Production cost model of the multi-jet-fusion technology. Proceedings of the Institution of Mechanical Engineers, Part C, 2021. Vol. 10, No. 10, pp 1917-1929

[Godec 2021] Godec, D. Bmadic, V. and Breski, Z. Optimisation of Mould Design for Injection Moulding - Numerical Approach, Tehnicki Glasnik-Technical Journal, 2021, Vol. 15, No. 2, pp 258-266

[INEOS 2008] Datasheet of 100-GB25. INEOS. 2008: INEOS Olefins & Polymers Europe

CONTACTS:

Martin Habrman
University of West Bohemia, Faculty of Mechanical Engineering,
Univerzitni 8, Pilsen 306 14, Czech Republic
Email: mhabrman@rti.zcu.cz
T: +420 377 638 745, M: +420 720 642 984

Zdenek Chval
Email: zdchval@rti.zcu.cz, T: +420 377 638 741

Karel Raz
Email: kraz@rti.zcu.cz, T: +420 377 638 751

Philipp Streit
Ostbayerische Technische Hochschule Amberg-Weiden,
Fakultaet Maschinenbau, Umwelttechnik, Amberg, Germany
Email: ph.streit@oth-aw.de

Dominik Stuempfl
Email: d.stuempfl@oth-aw.de