Gate Location and Cooling System Optimization

V. Senkerik, M. Stanek, M. Manas, D. Manas, A. Skrobak and J. Navratil

***Abstract*—**There are higher and higher quality and productivity requirements on plastic products. There increased requirements are mainly in automotive industry. This research paper deals with construction solution of an injection mold for specific product in automotive industry. It examines 4 designed gate position and material with different content of a filler and shape of the filler and its influence on deformation. Differences in deformations between individual versions are rather significant. There are also different layouts of drilled cooling channels and their influence on deformation compared. Analysis results shows that it influence mainly manufacturing production and also deformation. Eligible usage of these parameters can improve quality – lower product deformation.

***Keywords*—** Optimization, gate location, cooling system, injection molding, injection mold, PBT, polymer.

I. Introduction

INJECTION molding is the most commonly used manufacturing process for the fabrication of plastic parts. It is suitable for mass production of consumer articles, since raw material can be converted into inject by a single procedure.

The raw material, usually in the form of pellets or granules, is fed into the injection unit where it will be melted. The injection unit is generally a single-screw extruder in which the screw reciprocates coaxially against a hydraulically actuated cylinder. The continually rotating screw plasticates the granules to form a melt that is transported forward by the rotation. Because the injection nozzle is still closed during plastication, the melt is pushed to the front of the screw.

At the start of the cycle, the mold is closed by actuating the press, which on an injection molding machine is called the clamping unit. Before the melt, which is generated in and supplied by the plastification unit, is injected into the closed mold, the plasticating unit traverses against the mold, causing the injection nozzle of the plasticating unit to press against the sprue bushing of the mold. The pressure with which the nozzle is pressed against the sprue bushing must be adjusted in such a way that the joint remains sealed when the melt is injected afterwards. At the same time, the nozzle is opened and the melt can be pushed from the front of the barrel into the cavity of the mold.

As the cavity is filled, pressure builds up inside. This is counteracted by pressing the clamping unit against the mold under as much clamping force as possible to prevent melt from escaping out of the cavity through the mold parting lines.

The connection between the mold and plasticating unit is maintained until the filling process is complete. Generally, however, filling of the cavity does not mean the end of the process because the melt changes its volume on solidifying (freezing). In order that either more melt may be forced in to make up the difference in volume or to prevent the melt from running out of the mold, the connection must be maintained until the melt has frozen in the gate. The connection is broken by screwing back the plasticating unit, and closing the injection nozzle. Detaching of the nozzle causes thermal isolation between mold and plasticating unit because these are at totally different temperatures.

Since the plasticating process requires a certain amount of time, as soon as the nozzle is detached and closed, the plasticating unit usually starts rotating, drawing in - metering - more material, melting it and moving it to the front.

When the molding (molded part) has solidified to the extent that it can retain its shape without external support, the clamping unit opens the mold and the molding is pushed out of the cavity by ejectors. The cycle then repeats.

An important advantage of injection molding is that with it we can make complex geometries in one production step in an automated process. The injection molding technique has to meet the ever increasing demand for a high quality product (in terms of both consumption properties and geometry) that is still economically priced.

This is feasible only if the molder can adequately control the whole molding process, if the configuration of the molded part is adapted to the characteristics of the molding polymer material and the respective conversion technique.

There are major restrictions on wall thickness, which generally should not exceed a few millimeters, and on shape - it must be possible to demold the part.

Vojtech Senkerik is with the Tomas Bata University in Zlin, nam. T. G. Masaryka 5555, 76001 Zlin, Czech Republic (phone: +420576035062; e-mail: vsenkerik@ft.utb.cz).

Michal Stanek is with the Tomas Bata University in Zlin, nam. T. G. Masaryka 5555, 76001 Zlin, Czech Republic (e-mail: stanek@ft.utb.cz).

Miroslav Manas is with the Tomas Bata University in Zlin, nam. T. G. Masaryka 5555, 76001 Zlin, Czech Republic (e-mail: manas@ft.utb.cz).

David Manas is with the Tomas Bata University in Zlin, nam. T. G. Masaryka 5555, 76001 Zlin, Czech Republic (e-mail: dmanas@ft.utb.cz).

Adam Skrobak is with the Tomas Bata University in Zlin, nam. T. G. Masaryka 5555, 76001 Zlin, Czech Republic (e-mail: skrobak@ft.utb.cz).

Jan Navratil is with the Tomas Bata University in Zlin, nam. T. G. Masaryka 5555, 76001 Zlin, Czech Republic (e-mail: j1navratil@ft.utb.cz).

Fig. 1 Injection molding cycle

Typical injection moldings can be found everywhere in daily life; examples include toys, automotive parts, micro parts, household articles and consumer electronic goods.

For plastic injection molding, gate location has a significant effect on part quality and can determine if the part can be molded successfully. Some of the effects directly attributable to gate location are: mold venting, warpage, shrinkage, and overpacked and/or underpacked regions. In addition, as gate location has an effect on the thermo-mechanical processing history, it has a significant effect on the engineering properties of an injection-molded polymer, such as yield stress and impact strength.

Since, with all homogeneous plastic materials, solidification of the melt in the cavities of the mold is an effect influenced by the heat of the mold and since thermal conduction is critically influenced by the wall thickness, the gate must always be positioned at the thickest cross-section. If the gate is not at the thickest section, voids and sink marks will be caused. They result from too little holding pressure because of premature freezing of the gate area.

The position of the gate determines the direction of the material flow within the cavity. This causes so-called orientation, i.e. alignment of the molecules. Since the properties along and perpendicular to a molecule are very different, this also applies to many molded-part properties, e.g., the strength properties and shrinkage of moldings parallel to and perpendicular to the direction of flow. This effect, which is due to the orientation of the molecules, is all the more pronounced, the more the melt is sheared when it is freezing. The degree of orientation is therefore particularly high in thin-walled articles. The best values for tensile and impact strength are achieved in the direction of flow, while perpendicular to it, reduced toughness and increased tendency to stress cracking can be expected.

In addition, the runner system accommodates the molten plastic material coming from the barrel and guides it into the mold cavity. Its configuration, dimensions and connection with the molded part affect the mold filling process and, therefore, largely the quality of the product. A design which is primarily based on economic viewpoints (rapid solidification and short cycles) is mostly incompatible with quality demands especially for technical parts.

Injection molding process simulations (mold flow analysis) may be able to assist designer. These simulations have been developed over the past 30 years to provide critical information so that the best decisions get made. Through the early 1990s, these analyses had fairly limited capability and were quite expensive to use; however, significant progress has been made in process modeling, material characterization, and computational power. Today, simulations are extremely powerful and inexpensive, with results being available within hours or sometimes minutes.

The general reason to use a flow analysis is to improve our understanding of the molding process. Flow analyses are sometimes performed as standard operating procedure for marketing purposes or to verify design feasibility. To be more specific, however, analyses can be used to support critical decisions during the design, tooling, and processing stages of the molded part development process.

Computer-aided engineering (CAE) programs provide a flexible and economical means of recognizing potential errors early in the design and production process. The information gained from the simulation can assist in the optimization of the process, like cutting down cycle time, or part weight. It can also support the molder in fixing certain problems, which would otherwise have to be solved by trial-and-error- methods, which consume significant amounts of time, and waste material and energy. [1]–[6].

II. EXPERIMENT

This article deals with the design of injection mold for specific plastic product. Analyses are performed in program Autodesk Moldflow Insight 2010. It is plastic injection molding simulation computer-aided engineering software provides tools that help producers confirm and optimize the design of plastic parts and injection molds by accurately predicting the plastic injection molding process.

A. Product specification

Injected product is support frame for light module to front headlight of car, which allows side turn of lights in curves and horizontal adjustment of lights when the rear of car is loaded. It is located inside the headlight, which is not visible and looking at a car itself. This product will be manufactured in two versions, for left and right light, mirror the same.

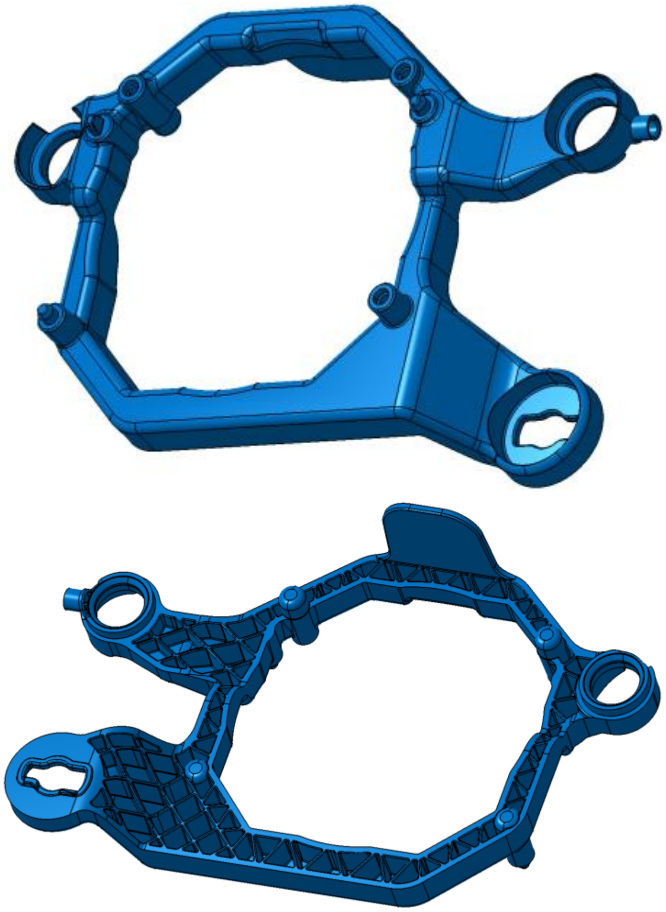

Fig. 2 Model

This part is required to have high stiffness, strength, shock resistance and shape stability at high temperatures, which are created by the light bulb. Injecting the product is complicated because it has a lot of ribs. The volume of one part is 35.21 cm^3. [3]

B. Materials

Several kinds of materials are examined for the product; from these it is chosen one that will be convenient the application. PBT is an examine material with glass fibers or beads at various filling concentrations. The materials are compared according to deformation – shrinkage of the product, which affects the filler itself and its content in polymer. [4]

Table I Materials

Family Name	PBT	PBT GF30	PBT GF50	PBT GB50
Filling	Pure material	30% glass fiber	50% glass fiber	30% glass beads
Ejection temperature	160°C	180°C	185°C	180°C
Mold surface temperature – set [°C]	80	80	80	80
Melt temperature – set [°C]	260	260	260	260
Absolute maximum melt temperature [°C]	280	280	290	280
Ejection temperature [°C]	160	180	185	180
Solid density [g/cm3]	1.2782	1.5360	1.7441	1.5360
Melt density [g/cm^3]	1.0598	1.3198	1.5355	1.3159

C. Gate location

Analyses of the cavity filling are performed to determinate how the gate location and number of gates in an injection molding piece affects its deformation. All examined versions of gate location are shown in Fig. 3-6.

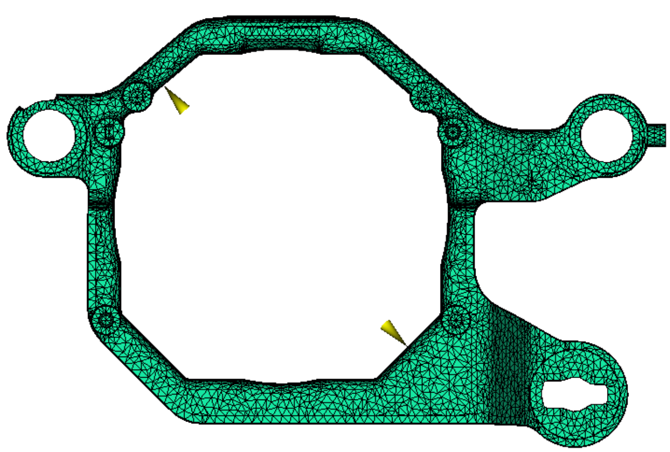

Fig. 3 Gate location version 1

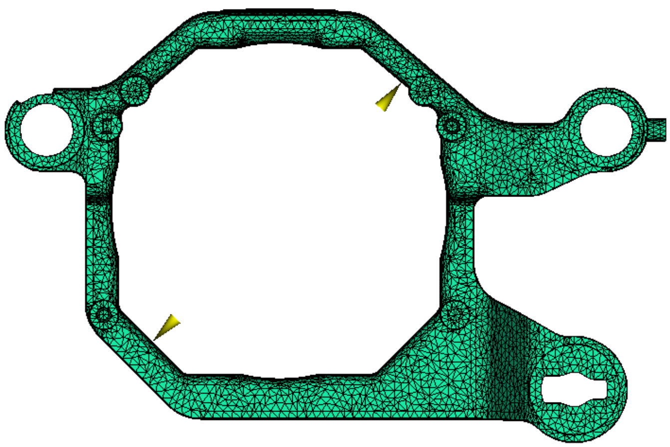

Fig. 4 Gate location version 2

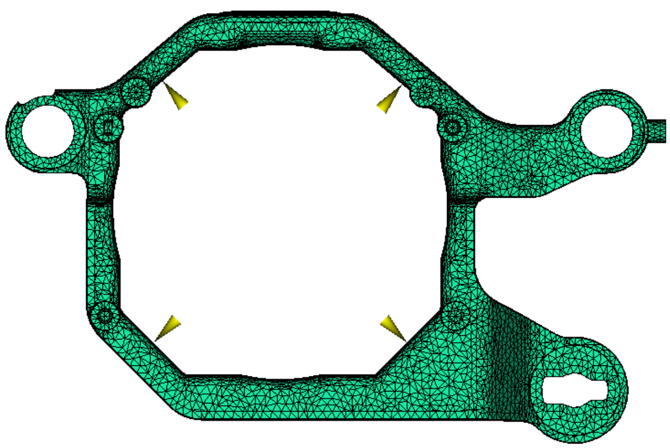

Fig. 5 Gate location version 3

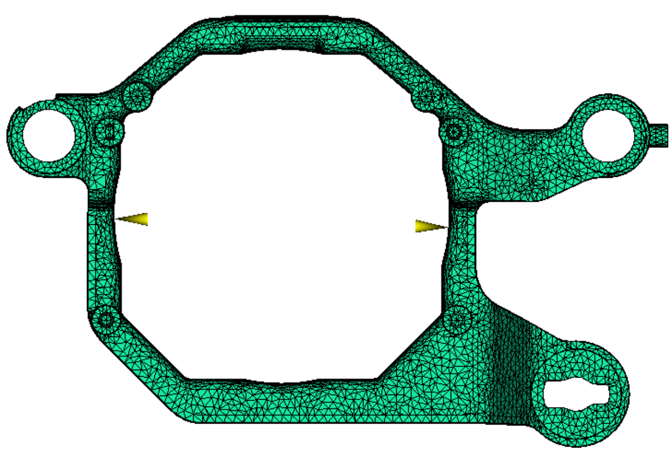

Fig. 6 Gate location version 4

Gate location versions 1 and 2 use only two gates which are placed diagonally in the corners of the product. Version 3 uses four gates and is combination of the first two versions. Version 4 uses only two gates which are directed horizontally. The gate locations are inside of the product for all versions. Process conditions for calculation of filling were chosen to minimize the influence on the deformations and they have been the same for all gates (Tab. II). Furthermore it is examined how filler type and its amount can affect deformation of the product.

Table II Process parameters

Injection time	Automatic
Injection + packing + cooling time	Automatic
Mold-open time	5 s
Velocity/ pressure switch-over	98 % volume filled
Pack/holding control	After 1.5 s switch to 80 %
Cooling liquid temperature	70° C
Cooling liquid flow rate	20 l/min

D. Runner system

The runner system accommodates the molten plastic material coming from the barrel and guides it into the mold cavity. Its configuration, dimensions and connection with the molded part affect the mold filling process and, therefore, largely the quality of the product. A design which is primarily based on economic viewpoints (rapid solidification and short cycles) is mostly incompatible with quality demands especially for technical parts. [5]

The simulation is done for the injection mold of a combination of hot and cold runner system (Fig. 7). Hot runner system provides reduction of waste, shortening the production cycle and maintains constant temperature during polymer flow from the machine. Cold runner system consists of a circular tunnel inlet channels ending in the mold cavity, which has the advantage that when form opens, the product separates from the cold runner system, thus eliminating one working operation and saving time.

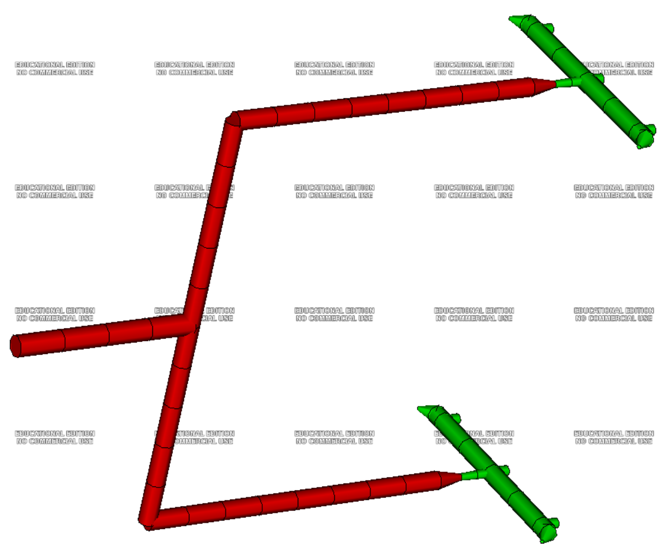

Fig. 7 Combination of hot and cold runner system

E. Cooling system

Attention is focused on the cooling system. Several injection molds were designed with different variants of the cooling system, which were subsequently analyzed. The aim was to design the simplest mold in terms of functionality and price and also maintaining acceptable quality.

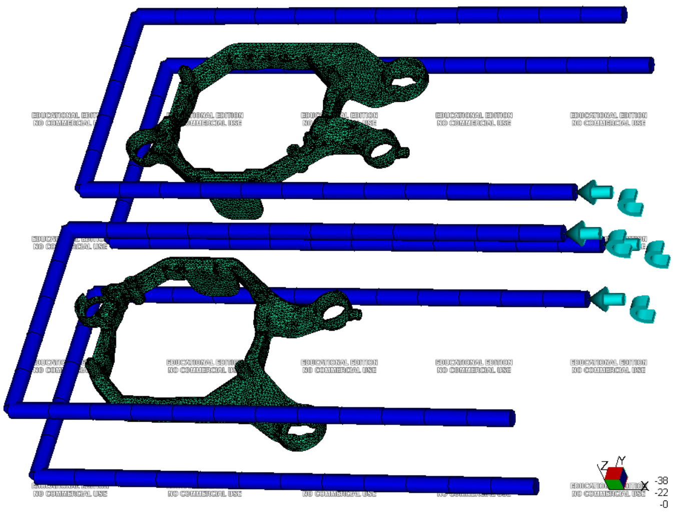

Fig. 8 Cooling system version 1

Cooling system version 1 (Fig. 8) has the simplest shape and a large distance between the cooling channels and sprue channels.

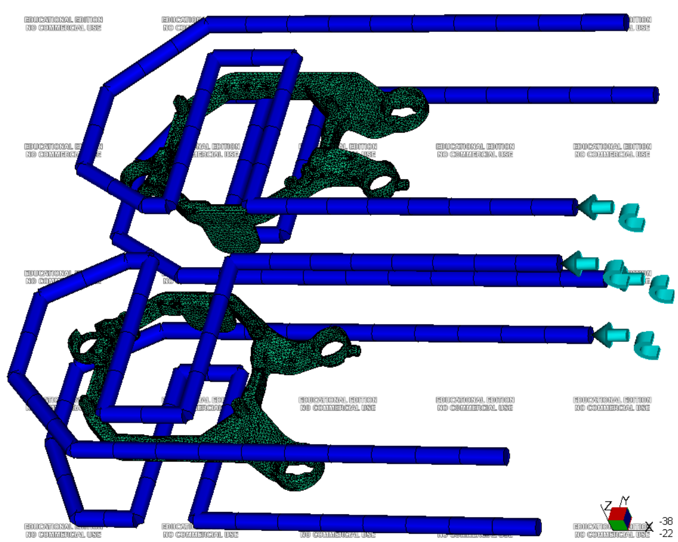

Fig. 9 Cooling system version 2

Cooling system version 2 (Fig. 9) has a more complex shape which partially copies the shape of the product and it cools the area around the sprue channels.

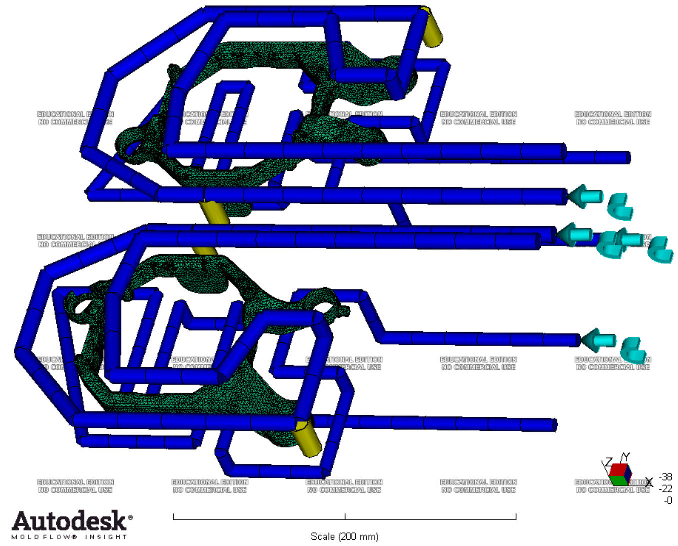

Fig. 10 Cooling system version 3

Cooling system version 3 (Fig. 10) has a very difficult storied shape that best copies the shape of the product and which contains two buffle elements for one cavity for cooling further parts of the inserts and cooling channels. Better heat distribution is achieved.

Water was chosen for cooling because of its high cooling effect. It is cheap and environmentally friendly, unlike oil. The channels have a diameter of 10mm, only cooling system version 3 uses diameter 8mm in the cavity because of better intertwine between the shape cavity, ejectors (Fig. 11) and replaceable inserts.

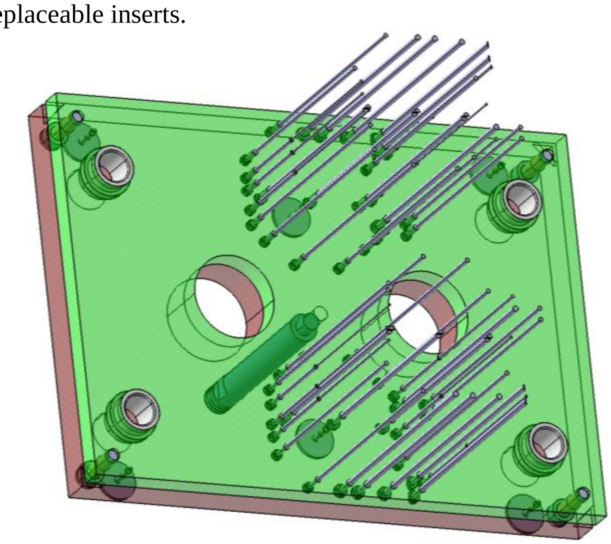

Fig. 11 Ejection system

The injection mold is designed as a real form for the production with all parts. Prismatic and cylindrical ejectors are used for ejection products in this injection mold. Cylindrical ejectors are the (3 ÷ 4) mm in diameter and prismatic (1.5 ÷ 2.5) mm.

Layout and number of ejectors varies depending on several versions of cooling systems - the cooling channels prevents placement of ejectors in some places. Their number depends on the complexity of cooling system; the more complex cooling system means lesser number of ejectors.

Ejector marks remain on the product, but that does not matter in this case, because they are kept in the inner part of product between the ribs. Ejectors are fixed in the ejection plate and ejector retaining plate. Some ejectors are fitted with shaped contact in the plates to prevent rotation and thus damage to itself or even injection mold. The functional ends of ejectors are tailored to shaped cavity. They must guarantee the safe ejection of the product.

III. Results and Discussion

A. Filling deformation results

Fig. 8 shows only one of the results of filling deformation, how the product is deformed under the influence of shrinkage and the effect of gate position.

The results of all materials and gate positions are shown in Tab. III.

Table III Deformations for individual materials and gate versions

Material	Gate version 1		Gate version 2		Gate version 3		Gate version 4	
	[mm]	[%]	[mm]	[%]	[mm]	[%]	[mm]	[%]
PBT	1.41	73.3	1.93	100	1.47	76.1	1.53	79.3
PBT GF30	1.41	73.1	1.74	90.2	1.65	85.4	1.22	63.3
PBT GF50	1.21	62.6	1.12	58	1.22	62.8	1.19	61.8
PBT GB50	1.22	63.2	1.67	86.4	1.27	65.8	1.31	68.1

The highest amount of deformation was chosen as 100%.

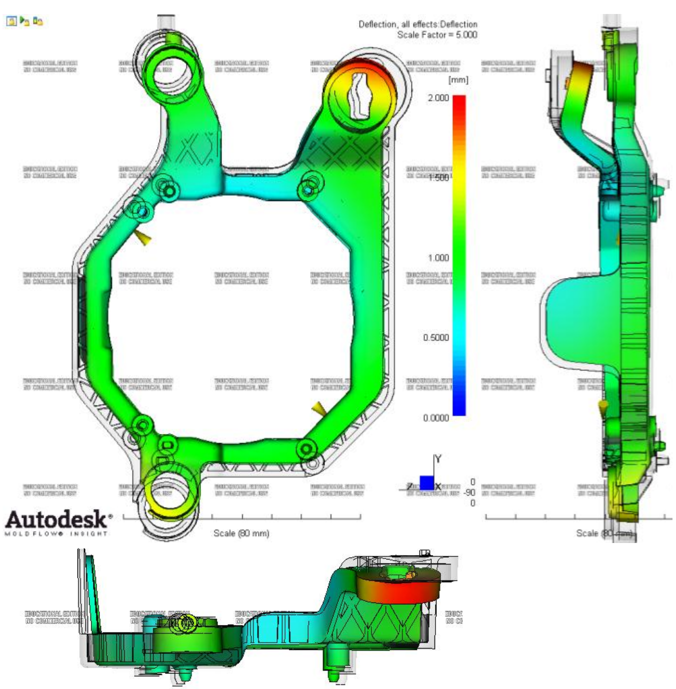

Fig. 12 Product deformation from PBT with gate location version 2 – the highest amount of deformation

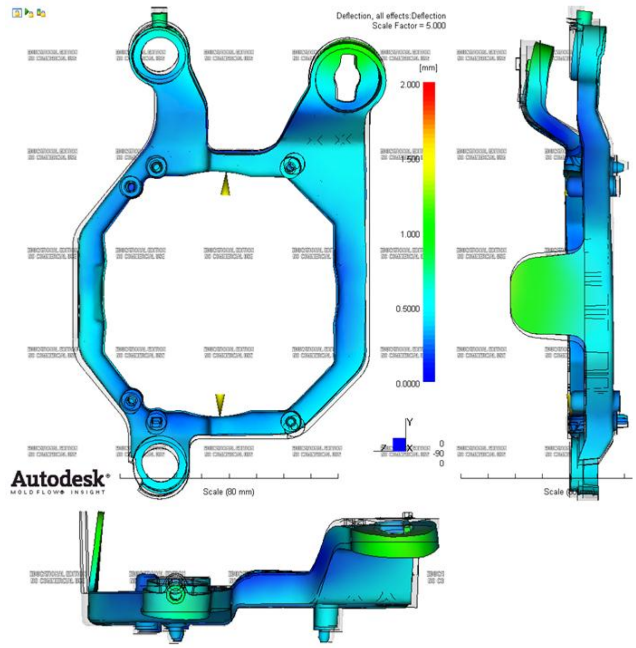

Fig. 13 Product deformation from PBT GF 30 with gate location version 4 - selected version

Finally, PBT GF30 material was chosen for design mold, which achieved satisfactory deformation amount. Increase in concentration of glass fibber causes wear of cavity mold. It works similarly as dispersed abrasive material in grinding paste. This effect increases demands on the cavity mold material, machining and heat treatment and injection machine wear. Filler also deteriorates the polymer flow and it must increase the values of process conditions during the injection processing. And materials with lower filler content have worse mechanical properties and thermal stability at thermal load. As the Tab. 2 indicates, the content of filler affects deformation, the higher the filler content, the lower the deformation.

Gate location of version 4 is chosen for the design of injection mold with the lowest deformation of the product for chosen material. This version is simple and has lower material consumption than version 3. The difference between maximum and minimum deformation of the selected material PBT GF30 is almost 30%, which is relatively high, because this deformation is caused only by the gate location.

B. Ejection time results

Analysis of time to reach the ejection temperature is very important. It displays the time needed for ejection of the product, measured since the beginning of injection molding process.

The longest time for cooling down to the ejection temperature is at protrusion because there is quite a thick wall. It would be appropriate to modify this part of the product so that there was less material and thus also reduce the ejection time and material consumption.

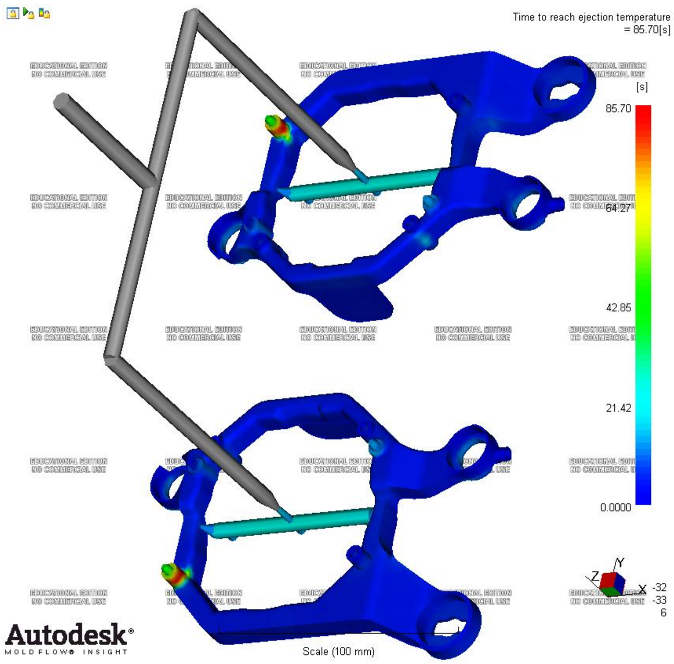

Fig. 14 Time to reach ejection temperature, cooling system version 1

One way to speed up cooling would be to insert another baffle element next to protrusion. The cooling system version 3 already has two baffles and adding another baffle would raise the pressure loss in cooling medium. Higher number of baffles means greater loss of pressure in cooling system.

Table IV Time to reach ejection temperature

Version of cooling system	Time to eject [s]	Difference [%]
Version1	85.70	100.00
Version 2	79.66	92.95
Version 3	77.98	90.99

The cooling system version 1 takes the longest time to reach the ejection temperature. The version 2 cools the examined product by nearly 7% faster than the version 1. The version 3 cools the product the fastest, by 2% faster than cooling 2 and up to 9% faster than cooling 1.

Fig. 9 shows that the product made using cooling system version 1 has the greatest deformation. Overall, the product is not too deformed in middle part, but side construction lug is twisted to main lengthwise plane of product, scale is 5:1. The total deformation is affected by shrinkage, orientation, cooling system, and also mentioned the gate position.

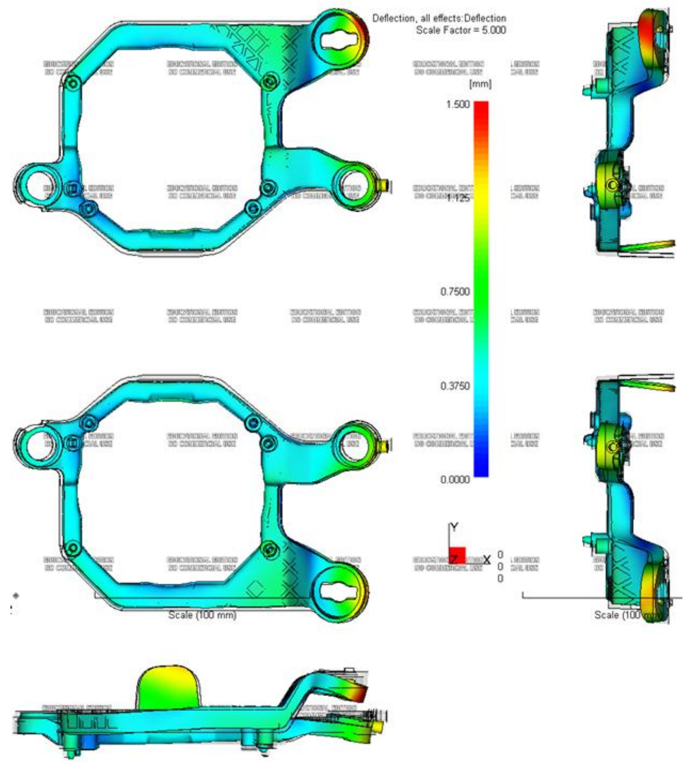

Fig. 15 Total deformation for the cooling system version 3

Table V Total deformation

Version of cooling system	Deformation [mm]	Difference [%]
Version1	1.477	98.66
Version 2	1.497	100.00
Version 3	1.441	96.26

In Tab. V is can be seen difference deformations. These deformations are very similar, because each deformation has the same design molds that differ only by cooling system. Thus, differences between the deformations are only a cooling effect on the product. The difference of the influence between cooling systems versions 1 and 2 is nearly 2%, which is relatively insignificant size and between 2 and 3 is nearly 4%. The cooling system version 3 has the most optimal cooling temperature field distribution and thus the product deforms the least.

IV. Conclusion

Filling and deformation analysis of the specific plastic product, which is support frame for light module to front headlight of car, was performed first. It was examined influence of gate location and number of gates on the deformation. Furthermore it was examined the influence of type and amount of a filler on this deformation. The selected version of gates was further used to design injection mold for next analyses.

In addition, several variants of cooling systems were designed with varied shape, complexity and usage baffle elements. They were analyzed and compared with each other. Complete analysis of filling, including runner and cooling system was performed. Analysis revealed differences between the designed cooling systems versions. By appropriate usage of these proposals a smaller deformation and increased productivity of the product can be achieved.

ACKNOWLEDGMENT

This paper is supported by the internal grant of TBU in Zlin No. IGA/FT/2012/041 funded from the resources of specific university research and by the European Regional Development Fund under the project CEBIA-Tech No. CZ.1.05/2.1.00/03.0089.

REFERENCES

[1] M. Stanek, M. Manas, D. Manas, V. Pata, S. Sanda, V. Senkerik, A. Skrobak, *"How the Filler Influence the Fluidity of Polymer",* Chemicke listy, Volume 105, 2011, pp.303-305

[2] Lam, Y. C., & Jin, S. *"Optimization of gate location for plastic injection molding."* Journal of Injection Molding Technology, 5(3), 180-180. (2001)

[3] S. Sanda, M. Manas, D. Manas, M. Stanek, V. Senkerik *"Gate Effect on Quality of Injected Part"*, Chemicke listy, Volume 105, 2011, pp.301-303

[4] M. Stanek, D. Manas, M. Manas, O. Suba, *"Optimization of Injection Molding Process",* International Journal of Mathematics and Computers in Simulation, Volume 5, Issue 5, 2011, p. 413-421

[5] G. Menges, W. Michaeli, & P. Mohren. "*How to make injection molds"*. 3rd ed. Munich: Hanser. 2001

[6] A. Calhoun, J. Golmanavich, & J. Ratzlaff. "Plastics technician's toolbox". Brookfield: Society of Plastics Engineers. 2002

[7] P. Adetunji. "The Role of Simulation and Compter Aided Analysis in Injection Molded *Products". RMIT University*

[8] J. Javorik, J., M. Stanek, *"The Shape Optimization of the Pneumatic Valve Diaphragms",* International Journal of Mathematics and Computers in Simulation, Volume 5, Issue 4, 2011, p. 361-369

[9] Stanek, M, Manas, M., Manas, D., Sanda, S., *"Influence of Surface Roughness on Fluidity of Thermoplastics Materials",* Chemicke listy, Volume 103, 2009, pp.91-95

[10] J. Javorik, M. Stanek, *"The Numerical Simulation of the Rubber Diaphragm Behavior,"* in Proc. 13th WSEAS International Conference on Automatic Control, Modelling & Simulation, Lanzarote, Spain, 2011, pp. 117-120.

[11] M. Stanek, D. Manas, M. Manas, O. Suba, *"Optimization of Injection Molding Process by MPX,"* in Proc. 13th WSEAS International Conference on Automatic Control, Modelling & Simulation, p.212-216.

[12] M. Stanek, M. Manas, T. Drga, D. Manas, *"Testing Injection Molds for Polymer Fluidity Evaluation",* 17th DAAAM International Symposium: Intelligent Manufacturing & Automation: Focus on Mechatronics and Robotics, Vienna, Austria, 2006, p.397-398.

[13] Stanek, M., Manas, M., Manas, D., *"Mold Cavity Roughness vs. Flow of Polymer"*, Novel Trends in Rheology III, AIP, 2009, pp.75-85

[14] J. Javorik, D. Manas, *"The Specimen Optimization for the Equibiaxial Test of Elastomers,"* in Proc. 13th WSEAS International Conference on Automatic Control, Modelling & Simulation, Lanzarote, Spain, 2011, pp. 121-124.

[15] H. Vaskova, V. Kresalek, *„Raman Spectroscopy of Epoxy Resin Crosslinking",* in Proc. 13th WSEAS International Conference on Automatic Control, Modelling &Simulation, Lanzarote, CanaryIslands 2011, p.357-360.

[16] L. Pekar, R. Matusu, P. Dostalek, J. Dolinay, *"The Nyquist criterion for LTI Time-Delay Systems"*, in Proc. 13th WSEAS International Conference on AutomaticControl, Modelling &Simulation, Lanzarote, CanaryIslands, 2011, p.80-83.

[17] Manas, D., Stanek, M., Manas, M., Pata V., Javorik, J., *"Influence of Mechanical Properties on Wear of Heavily Stressed Rubber Parts",* KGK – KautschukGummiKunststoffe, 62. Jahrgang, 2009, p.240-245.

[18] M. Stanek, M. Manas, T. Drga, D. Manas, *"Influence of Mold Cavity Surface on Fluidity of Plastics"*, Chapter 55 in DAAAM International ScientificBook 2007, DAAAM International, Vienna, Austria p.627-642

[19] M. Stanek et al., *"Simulation of Injection Molding Process by Cadmould Rubber"*, International Journal of Mathematics and Computers in Simulation, Vol.5, 2011, pp. 422-429.

[20] V. Pata et al., *"Visualization of the Wear Test of Rubber Materials"*, Chemicke Listy, Vol.105, 2011, pp. 290-292.

[21] M. Stanek et al., *"Plastics Parts Design Supported by Reverse Engineering and Rapid Prototyping"*, Chemicke Listy, Vol.103, 2009, pp. 88-91.

[22] S. Sanda et al., *"Injection Mold Cooling System by DMLS"*, Chemicke Listy, Vol.103, 2009, pp. 140-142.

[23] L. Pekar, R. Prokop, "Analysis of a Simple Quasipolynomial of Degree One*", in Proc. 13th WSEAS International Conference on Automatic Control*, Modelling & Simulation, Lanzarote, Canary Islands, 2011, p.86-89.

[24] M. Adamek, M. Matysek, P. Neumann, "Modeling of the Microflow Senzor*", in Proc. 13th WSEAS International Conference on Automatic Control*, Modelling & Simulation, Lanzarote, Canary Islands, 2011, p.137-140.

[25] J. Dolinay, P. Dostalek, V. Vasek, P. Vrba, "Teaching Platform for Lessons of Embedded Systems Programming*", in Proc. 13th WSEAS International Conference on Automatic Control*, Modelling & Simulation, Lanzarote, Canary Islands, 2011, p.158-161.

[26] R. Prokop, N. Volkova, Z. Prokopova, "Tracking and Disturbance Attenuation for Unstable Systems: Algebraic*", in Proc. 13th WSEAS International Conference on Automatic Control*, Modelling & Simulation, Lanzarote, Canary Islands, 2011, p.161-164.

[27] M. Stanek, M. Manas, D. Manas, S. Sanda, "Influence of Surface Roughness on Fluidity of Thermoplastics Materials, *Chemické listy*, Volume 103, 2009, p.91-95

[28] M. Manas, M. Stanek, D. Manas, M. Danek, Z. Holik, "Modification of polyamides properties by irradiation", *Chemické listy*, Volume 103, 2009, p.24-28

[29] M. Stanek, M. Manas, D. Manas, S. Sanda, "Plastics Parts Design Supported by Reverse Engineering and Rapid Prototyping", *Chemické listy*, Volume 103, 2009, p.88-91

[30] M. Stanek, M. Manas, T. Drga, D. Manas, "Polymer Fluidity Testing", *17th DAAAM International Symposium: Intelligent Manufacturing & Automation: Focus on Mechatronics and Robotics*, Vienna, Austria, 2006, p.395-396

[31] M. Stanek, M. Manas, T. Drga, D. Manas, "Testing Injection Molds for Polymer Fluidity Evaluation", *17th DAAAM International Symposium Intelligent Manufacturing & Automation: Focus on Mechatronics and Robotics*, Vienna, Austria, 2006, p.397-398

[32] M. Stanek, M. Manas, T. Drga, D. Manas, "Influence of Mold Cavity Surface on Fluidity of Plastics", Chapter 55 *in DAAAM International Scientific Book 2007*, DAAAM International, Vienna, Austria p.627-642

[33] H. Vaskova, V. Kresalek, „Raman Spectroscopy of Epoxy Resin Crosslinking", *in Proc. 13th WSEAS International Conference on Automatic Control*, Modelling & Simulation, Lanzarote, Canary Islands 2011, p.357-360.

[34] S. Sanda, M. Manas, M. Stanek, D. Manas, L. "Rozkosny, Injection Mold Cooling System by DMLS", *Chemicke listy*, Volume 103, 2009, p.140-142.

[35] M. Stanek, M. Manas, T. Drga, D. Manas, "Testing Injection Molds for Polymer Fluidity Evaluation", *17th DAAAM International Symposium: Intelligent Manufacturing & Automation: Focus on Mechatronics and Robotics*, Vienna, Austria, 2006, p.397-398.

[36] D. Manas, M. Manas, M. Stanek, M. Zaludek, S. Sanda, J. Javorik, V. Pata, "Wear of Multipurpose Tire Treads" *Chemické listy*, Volume 103, 2009, p.72-74.