DESIGN AND ANALYSIS OF INJECTION MOULD WITH HOT RUNNER

PULAVENI SAGAR
M.Tech, Production Technology,
Sree Vaanmayi Institute Of Engineering & Technology, Bibi Nagar, Nalgonda.
E-mail: sagar.pulaveni@gmail.com

Ms. E. SANDEEP REDDY
Assistant Professor,
Dept. of Mechanical Engineering, Sree Vaanmayi Institute Of Engineering & Technology, Bibi Nagar, Nalgonda.

ABSTRACT

An understanding of the complex relationship between polymer structure and its process ability in hot runner injection moulding is essential for the correct selection and application of hot runner systems. In designing the hot runner system, there is often some flexibility with regard to the bore sizes selected within the manifolds, and the nozzles, which make up the system. If the pressure losses through the system are a concern, larger bores may be selected to reduce the pressure loss. The aim of this master project is to design a multi-cavity injection mold. The theoretical part of this thesis describes problematic of injection molding and injection mold design, namely runner systems, mold cooling and venting. Practical part of the project deals with two injection mold designs for the given part, which is a cup for yogurts and desserts. This is followed by comparison of the individual designs. The chosen injection mold design is sub-mitted to injection molding process analysis and documented with assembly drawing. Injection mold designing was done in CAD application CATIA V5R19 and evaluated by injection molding analysis in Autodesk Moldflow Synergy 2014 software.

Keywords: *Mold design, Hot runner gate design, Mold flow analysis.*

INTRODUCTION

Injection moulding is a manufacturing process for producing parts by injecting material into a mould. Injection moulding can be performed with a host of materials, including metals, glasses, elastomers, and most commonly thermoplastic and thermosetting polymers. Material for the part is fed into a heated barrel, mixed, and forced into a mould cavity where it cools and hardens to the configuration of the cavity. The manufacturing of thin-wall products is very important for the automotive industry because thinner components allow considerable overall weight savings, beneficial effects on the reduction of fuel consumption and improvement of environmental impact. In addition, the decrease in thickness allows significant cuts in production costs due to less material being used and shorter cycle times. All materials used for automotive applications such as metals, foams, plastics and composites are investigated in order to achiever eductions in product thickness. In particular, thin-wall fabrication of plastic products allows the realization of smaller and lighter parts which can withstand day-to-day use while maintaining their aesthetic appearance. Polymer materials have become inseparable part of the present. They excel in strength, low weight, and chemical resistance. Polymer materials are also good electric and heat insul-ants. Because of their price, properties, process and production technologies are used in an increasing number of sectors and gradually replacing the traditional materials (metals, glass, wood, etc.).

The polymers are processed by different technologies which for example include injection molding, extrusion, blow molding and compression molding. The most widespread tech-nology of polymer processing is injection molding. It is very accurate and efficient manu-facturing technology, which can produce products with complex shapes without additional adjustments. Due to high initial cost, this technology is used in large-scale production.

A tool that provides the final shape of the product is called an injection mold. The injection mold is very expensive and complex component. Due to the achievement of the required product quality during the long-term tool life, great emphasis is put on injection mold de-sign.

With the development of computer technology, the design of injection molds is realized with support of different CAD, CAM and CAE systems. They strongly speed up mold designing process. With the usage of these systems, injection molds can prevent from potential defects on the products and additional modification.

Because of the many inherent advantages in using plastic materials, there is an ongoing trend of replacing metal with injection-molded plastic parts in a wide variety of applications. More and more parts with critical end-use application requirements are becoming candidates for conversion to plastics. Plastics are lightweight, durable and corrosion-resistant; have a high strength-to-weight ratio; and, when used in transportation applications, for example, offer one of the easiest ways to increase fuel savings by making vehicles more lightweight. As plastics replace metals, the parts must be designed to take into account the properties of the specific plastic relative to the application requirements. One of the complicating factors for injection-molded plastic parts is that the properties of plastic materials effectively change during the manufacturing process. While this is not a problem in and of itself, problems can arise if the structural analyses are based on generic material data that does not accurately represent the actual properties of the molded part. These problems can include over engineering of components, which can lead to unnecessary costs and material usage, or under-engineering, which can result in part failure. Fiber-filled plastic materials are commonly used in metal replacement applications. When glass or carbon fibers are added to plastics, the elastic modulus can increase significantly with a negligible effect on part weight. This combination of low weight and high stiffness makes fiber-filled plastics ideal for high-performance applications. The key to unlocking the potential of these plastics lies in the orientation of the fibers. The orientation direction and the degree of orientation of the fibers determine the mechanical properties of the molded part. In areas where fibers are strongly aligned, the material will have higher strength characteristics in that direction, but will be relatively weak in the perpendicular direction (across the fibers). In areas where the fibers are more randomly oriented, the material will not achieve maximum strength, though the strength properties will not depend as much on the loading direction, creating a more isotropic like material.

Y.K. Shen, Y. J. Shie, and W. Y. Wu [7]: Microinjection is a branch of micro system technology. This paper employs analysis software to simulate three plastic materials (PP, PA, and POM) and four injection process parameters (injection time, mold temperature, injection temperature, and injection pressure) and applies them in simulation microinjection with the assistance of the Taguchi method, which is adopted in this paper. Further, the influences of these three plastic materials and four injection process parameters on microinjection moulding are analyzed. Through the simulation results, it is known that among the microinjection process parameters mold temperature is the most important moulding parameter. Also, during the microinjection process, mold temperature must be raised to be higher than the traditional mold temperature to prevent short shot that occurs when the melt 11 cools down too rapidly or when the melt's temperature is insufficient.

Y. K. Shen and W. Y. Wu [8]: Micro system technology enables product miniaturization, diversifies product functions, and improves quality, reliability and added value. This paper employs mold flow software to analyze three plastic materials (PP, PA, POM) and four injection moulding parameters (injection time, mold temperature, injection temperature, and injection pressure) and applies them in simulation microinjection moulding. During the process, the Taguchi method is used alternatively. All these are to obtain a better understanding of the relation between the three plastic materials as well as the four injection moulding parameters and microinjection moulding. Through the simulation results, it is known that mold temperature is the most important factor among the injection moulding parameters. Moreover, the mold temperature of microinjection moulding should be raised to be higher than the glass transformation temperature of the plastic material to avail the injection moulding of products. Y.

K. Shen, S. L. Yeh, and S. H. Chen [9]: Microinjection moulding is a branch of micro electro-mechanical system technology. This paper employs mold flow analysis software and draws a plan of simulation experiment items: three plastic materials (PS, PC, PMMA) and four moulding parameters (injection time, mold temperature, injection temperature, and injection pressure). The finite element method is used with the Taguchi method in microinjection moulding mainly to analyze the critical factors in the relation between the three plastic materials as well as the four injection moulding parameters and microinjection moulding. Two points about microinjection process are known through the simulation results: First, among the injection moulding parameters, mold temperature is the most important factor, and the next in importance is injection temperature. The third in line is injection pressure, and the least important among the four is injection time. Furthermore, the parameter values of mold temperature and injection temperature have mutual influences on each other. On the other hand, those of injection pressure and injection time affect each other's. Second, 12 mold temperature of microinjection moulding has to be higher than that of traditional injection, or otherwise short shot may occur. All of the above illustrate that temperature is the most critical factor of moulding parameters affecting precision products. Since products are small, high melting temperature is needed to increase the fluidity of the melt so that filling can be completed within a short period of time. However, it must be attended with meticulous care that temperature must not exceed the allowed temperature interval of the plastic material, or the plastic material may crack and cause potential problems after injecting the product such as bubbles formed in the plastic material, gas entrapping, and unsatisfactory mechanical properties.

Nelida Gracia, Esther Gonzalez, Juan Baselga, and Julio Bravo [10]: This paper uses ABS plastic materials of different levels and designs different values of thickness. C-MOLD is employed with the Taguchi method to simulate and find out the values of the injection temperature and injection pressure of the three plastic materials. The results of these three plastic materials differ according to their inherent characteristics. However, these plastic materials of different levels have one characteristic in common—that is, when thickness falls between 0.36~0.39mm, temperature variations are most in quantity. When fill-time decreases from 0.5s to 0.3s, temperature difference decreases to approximately half of the original value. ABS plastic material G-360 with the thickness of 0.39mm and fill-time of 0.3s has a temperature difference lower than 25℃. This means that when temperature is lower than 25℃, there is no possibility for residual stress to exist. In one single engineering plastic, different levels of engineering plastics are contained, and their constituents and the quantities of fiberglass all differ. Therefore, choosing suitable plastic materials is very important, and of course the environment and range in which the product is used and the structure of the product should also be taken into consideration so that the most suitable engineering plastic material can be chosen.

METHODOLOGY

INJECTION MOLDING

The injection molding process is one of the key production method for processing plastics. It is used to produce molded parts of almost any complexity that are to be made in medium to large numbers in the same design. There are major restrictions on wall thickness, which generally should not exceed a few millimeters.

Advantages:

- Direct route from raw material to finished part
- Very little finishing, or none at all, of molded parts
- Full automation
- High reproducibility
- Low piece costs for large volumes
- Possibility to make complex parts

Disadvantages:

- High investment costs
- Long period needed for injection mold making

- Injection molding machine is disproportionately big in comparison with injected part

MOLD DESIGN

An injection mold is a specialized piece of equipment used to form a plastic part. Nearly every mold is custom designed and built. There are modular molds that allow the exchange of inserts that can produce different parts and family molds that may produce different parts in a single molding cycle. However, it is most common that each mold is custom de-signed and built to produce a given part.

Injection molds must satisfy the following basic requirements

- Contain a core and cavity set(s) that defines the features of the part that it will form.
- Provide means for molten plastic to be delivered from the injection molding ma-chine to the part forming cavities.
- Act as a heat exchanger, which will
  - cool the part rapidly,
  - cool the part uniformly.
- Provide for the molded part to be ejected from the mold.
- Have a structure that will resist internal melt pressures which can potentially exceed 200 MPa and compressive forces from the molding machines clamp which can reach thousands of tons.
- In multi-cavity molds, provide uniformity to each cavity through steel dimensions, melt delivery, and cooling.

Runner systems

Injection mold can be classified in several ways and one of them is also by the type of run-ner system. According to the type of runner system injection molds are divided as:

- cold runner system,
- hot runner system,
- Combination of cold and hot runner systems.

DESIGN AND ANALYSIS

General guidelines for designing plastic parts have evolved over the years and are focused at issues related to manufacturability. These can include consideration of material shrink-age, part ejection, cooling, and mold filling. Probably the most troublesome issue related to successful development of a new plastic part is anticipating how it will shrink and warp after molding. Shrinkage of plastic parts varies from material to material and within a given material. This makes it difficult to design a mold and process that will produce a part to the desired size. In addition, variations in shrinkage within a given part develop residual stresses that act to warp the part. The stresses that do not warp the part will reside in the part and potentially cause delayed dimensional and structural problems.

USED SOFTWARE

CATIA V5R19

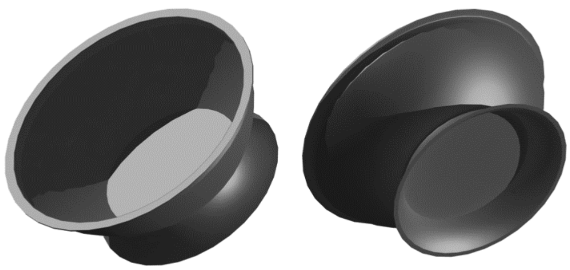
CATIA is software developed by French company Dassault Systemes. CATIA V5 is a sys-tem that is capable of covering the complete life cycle of a product. It offers part designing possibilities, various analysis, simulation and optimization to the creation of documentation and NC programs. The system is characterized by a significant level of industrial universality, which can be used in completely different areas of engineering. The wide range of modules that CATIA V5 features, allows users to create software solutions matched to the specific conditions and requirements. It can be automotive or aerospace, consumer goods as well as the production of machine tools or heavy machinery.

Autodesk Moldflow Synergy 2014

Autodesk Simulation MoldflowSynergy software, part of the Autodesk solution for Digital Prototyping, provides injection molding simulation tools for use on digital prototypes. Providing in-depth validation and optimization of plastic parts and associated injection molds, Autodesk Moldflow software helps study the injection molding processes in use today. Used by some of the top manufacturers in the automotive, consumer electronics, medical, and packaging industries. The software helps to reduce the need for costly mold rework and physical prototypes, minimize delays associated with removing molds from production, and get innovative products to market faster.

INJECTED PART

Injected part is a cup, which is used in food processing as cup for yoghurts and desserts.



Render 3D model of the cup

Material

Chosen material for the injected part is polypropylene (PP). PP is a semi crystaline thermo-plastic material, which belongs to polyolefin group. Its crystallinity is usually in range of 55-70 %. From the mechanical and chemical point of view PP is defined with good resistance. It is resistant to oils and chemical solvents. Processability and dyeability of PP is very good. PP is used in a wide range of applications, for example in the food, textile and automotive industry.

Material characterization

o Trade name RA12MN40

o Density 905 [g/cm3]

o ITT (230 °C/ 2,16 kg) 40 [g/10 min]

o Young modulus E 1340 [MPa]

o Shear modulus G 481,3[MPa]

o Parallel shrinkage 1,386 [%]

o Perpendicular shrinkage 2,004 [%]

o Maximum shear rate 100 000 [1/s]

o Maximum shear stress 0,25 [MPa]

o Fillers unfilled

Recommended processing :

o Melt temperature 200 – 250 [°C]

o Mold surface temperature 10 – 50 [°C]

o Absolute maximum melt temperature 290 [°C]

o Ejection temperature 107 [°C]

Injection molding machine technical characterization

o Clamping force 3200 [kN]

o Distance between tie bars 720 x 720 [mm]

o Mold mounting plates 1040 x 1040 [mm]

o Mold height 300 – 800 [mm]

o Max. ejector stroke 250 [mm]

o Max. ejector force 86 [kN]

o Max. weight of moveable mold half 2900 [kg]

o Injection pressure 2500 [bar]

o Holding pressure 2500 [bar]

o Max. shot weight 723 [g]

o Effective screw length 23 [L/D]

o Screw diameter 60 [mm]

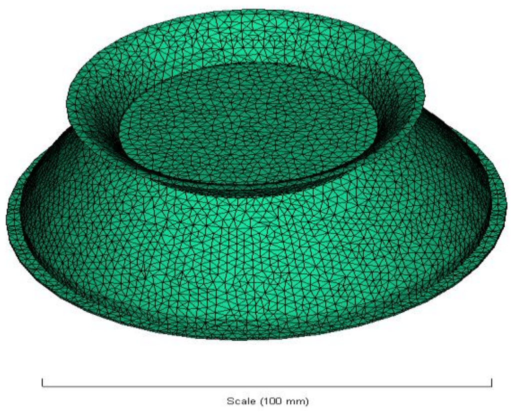
o Max. screw torque 2140 [Nm]

o Nozzle contact force 90 [kN]

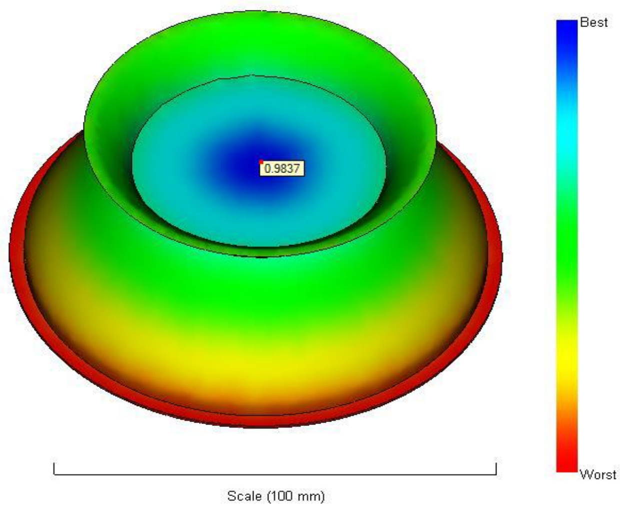
SIMULATION

Simulation of the injection process was done in Autodesk Mold flow Synergy 2014. Designed 3D model of the injected cup was increased for shrinkage. After importing to and before the analysis, 3D model of the part increased for shrinkage had to be meshed. The choice was from three types of meshes (mid plane, dual domain and 3D mesh).In this case Dual Domain was chosen which is sufficient for this analysis.

Due to the symmetry and better results clarification,



Before definition of all necessary parameters, gate analysis was performed to check for suitability of gate location. According to the gate analysis, chosen gate location is suitable from 98 % at the center of the part. Results of the analysis proved our assumption andthis solution was found compliant.



Results of the gate location analysis

ANALYSIS SETTINGS

After preforming the gate analysis all other necessary inputs were defined. These included selection of suitable material for injected parts, mold material, injection molding machine and other settings of process parameters. Selected molding machine is not included in Autodesk

Moldflow database and therefore an injection molding machine with similar parameters was chosen. Injection mold material was left default selected. According to the database of mold mate-rials this one is labelled as Tool steel P20 with DIN steel equivalent 1.2311 with following characterizations.

Mold material :
o Density 7,8 [g/cm3]

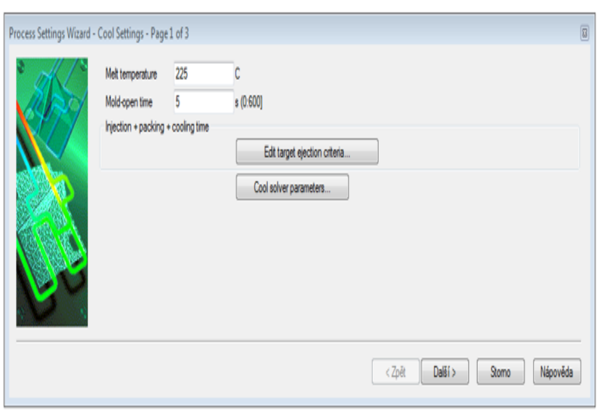
o Specific heat capacity 460 [J/kg.°C]

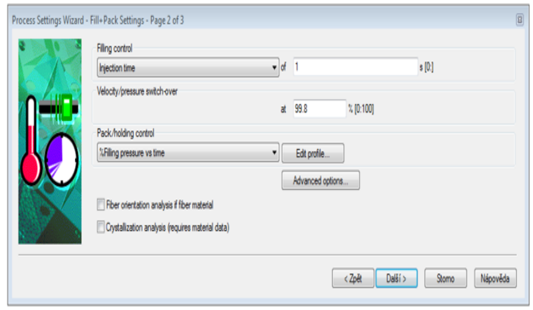
o Thermal conductivity 29 [W/m.°C]

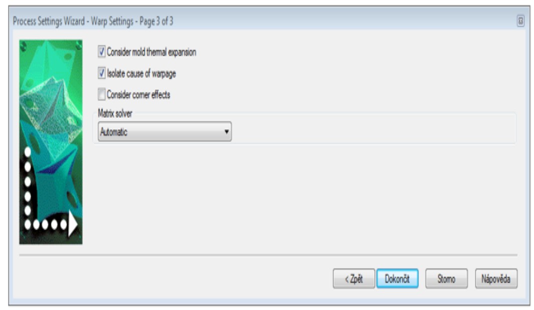
o Young modulus 200000 [MPa]

o Poisson number 0,33 [-]

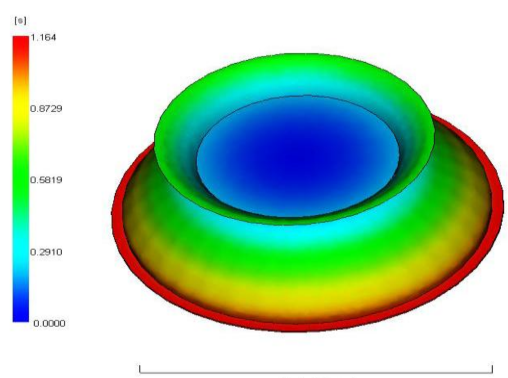
Process parameters
Adjusting the process parameters, first an analysis with automatically selected process parameters was performed. This was followed by examination of the results and new process parameters were set. This method was applied to achieve satisfactory results.







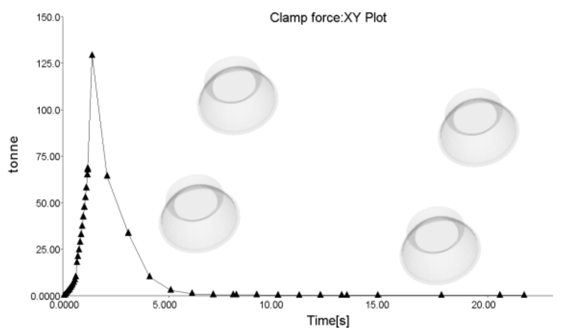
FILLING TIME
Filling is done simultaneously in all cavities proving that hot runner system is balanced. As seen from maximum value of fill time is 1,164 seconds and it can be observed in most distant place of the cavity where filling time reaches its maximum value. From this analysis we can set the filling time to 1,2 s. Cavities are filled in a relatively short time. Short injection cycles have positive effect on the orientation of polymer macromolecules. However, injection time cannot be disproportionately reduced as it would lead to material degradation and unfilled regions in the cavities.



Filling analysis – results

Clamp force

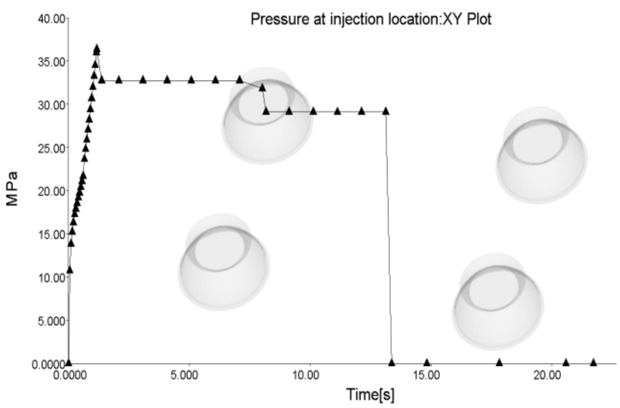
Size of the maximum closing force should be one of the estimation for selecting an appro-priate injection machine. Results of clamp force analysis determine course of clamping force. Maximum clamp force examined from the numerical results of this analysis is 1290 kN (129 tons). The biggest strength is required during application of filling pressure. Selected injection molding machine is able to produce the clamping force up to 3200 kN. Therefore meets the requirements regarding to clamping force with 20 % safety coefficient and it is selected correctly.



Clamp force

Pressure at injection location

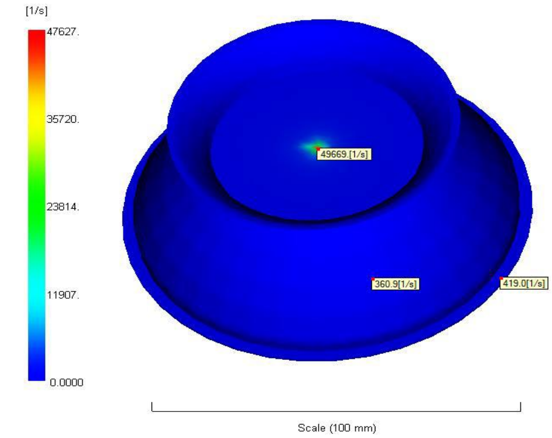
Maximum pressure during the filling is 36,4 MPa with pressure reaching its highest values at 1,16 seconds (end of cavity filling). Maximum injection pressure of the machine (250 MPa) was not exceeded.



Pressure at time diagram

SHEAR RATE

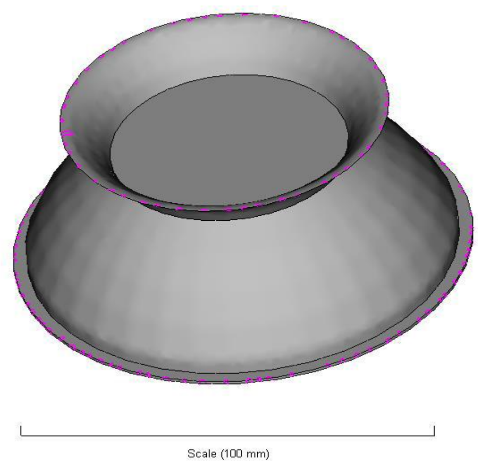
The highest shear rate allowed for the selected material according to data sheet is 100 000 s1, exceeding this value might lead to material degradation. The highest values can be observed in injection gate area. Maximum allowed value was not exceeded as the max-mum shear rate is 47 627 s-1.



Shear rate analysis

AIR TRAPS

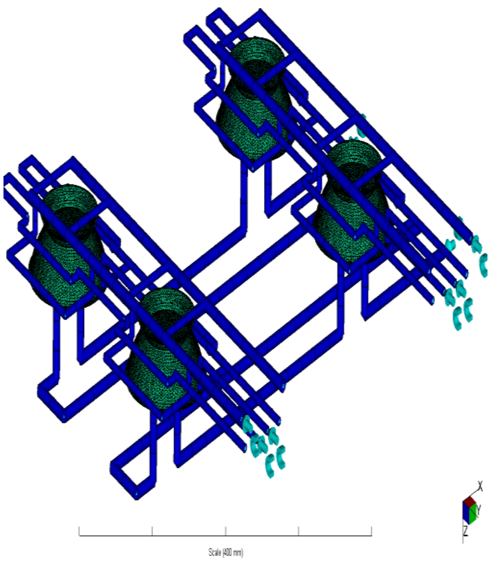
Results of this analysis are convenient for air traps predictions inside of the cavities. Trapped air is compressed and can result in a large temperature increase, which could damage the part. From examination of the results we can say that the air will be trapped in places that are filled last.



Air traps – results

Cooling analysis

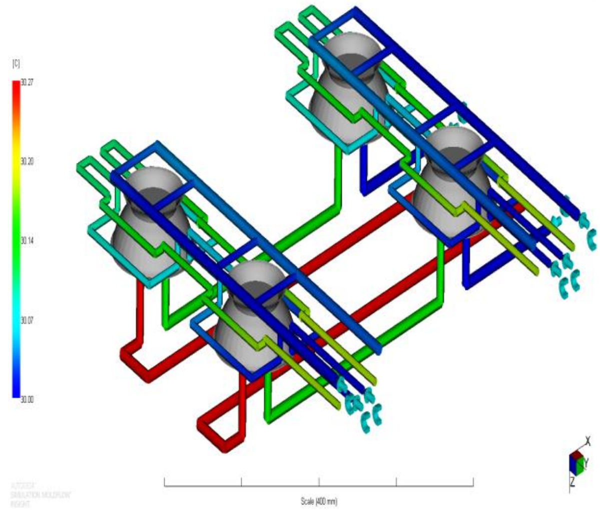
During injection process cavities are filled with melted material, which is cooled down to a suitable ejection temperature. Cooling system was designed as a network that consists of drilled channels. Water with no additives was selected as a coolant in this analysis. Coolant temperature was set to 35 °C.



Cooling circuits

CIRCUIT COOLANT TEMPERATURE

The results of this analysis show temperature of the coolant in the circuit. For the best heat removal efficiency from cavities, temperature difference between inlet and outlet should be less than 2 °C. This leads to uniform heat removal from the cavities. The requirement is accomplished as the temperature difference is only 0,37 °C. If the difference was more than 2 °C it would lead to uneven temperature field and deformations. This problem can be solved by division into several shorter cooling circuits or changing process parameters, namely by increasing the coolant pressure. The division into two shorter cooling circuits was done in the mold to assure the required temperature parameters.

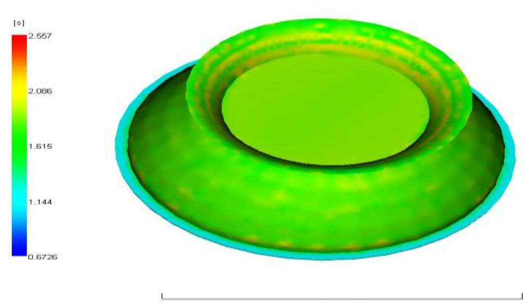


Circuit coolant temperature – results

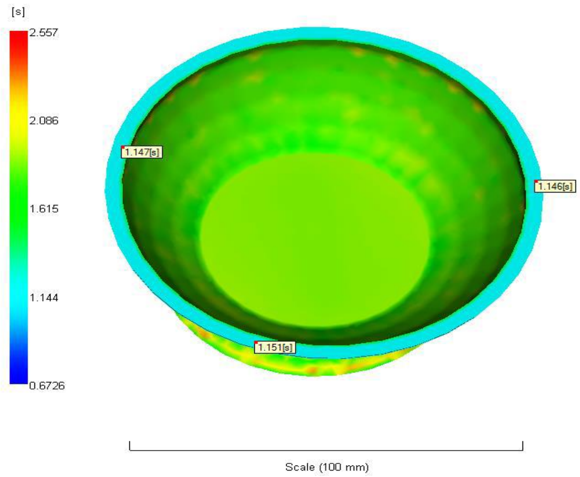
TIME TO REACH EJECTION TEMPERATURE

Time to reach ejection temperature is affected by the selected material as every material has its specific criteria for ejection. In this analysis ejection temperature was set to 85 °C for the whole part volume.

The results of this analysis show that time to reach ejection temperature is 2,56 s for the whole part volume. Short times are caused due to relatively small wall thickness. In places where stripper rings are situated, time to reach ejection temperature is only 1,15 s. There-fore time to reach ejection temperature can be set on 1,2 s. This setting will assure safe ejection of the parts with no deformation onto the parts´ surfaces that might be caused by the stripper rings.



Time to reach ejection temperature – results

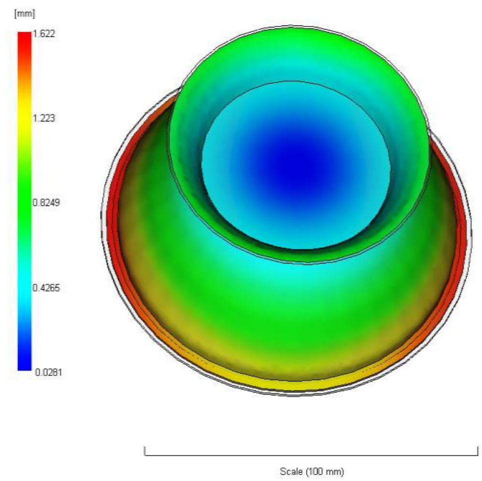


Time to reach ejection temperature – results 2

TOTAL DEFLECTION

The results of this analysis show that maximum total deformation is 1,622 mm. Results include all the influences on the deformation of injected parts. Size of shrinkage and total deformation can be adjusted by increasing holding pressure or more intensive cooling.

The maximum deflection values are situated at the topperimeter area of the part. These results can be justified by small wall thickness and insufficient cooling in this area. Due to occurrence of stripper rings no cooling circuits were designed in this area. However, de-spite of relatively big total deflection, deflection in the Z axis is only 0,17 mm, which means that the part will remain its flatness on its lid surface.



Total deflection

RESULTS AND DISCUSSIONS

Injection mold was designed with an effort to maximize the usage of standard parts in or-der to reduce the cost of mold and simplify designing in the 3D software. Mold multiplicity was assigned to four and firstly injection mold with hot runner system was designed. Mold frame, guiding and connecting elements were selected from Hasco catalog. Due to size of the hot runner system it has to be custom made, other hot runner accessories were selected from Hasco catalog. Cooling of the mold is done in 8 separate cooling circuits, with two of them for cavity cooling, another two for core cooling with remaining four cooling circuits related to slider cooling. Selected coolant was water with no additives. Uniform ejection of parts is provided with stripper rings. Injection molding process was simulated in program Autodesk Moldflow Insight 2014. From filling analysis can be observed that filling of the cavities is done simultaneously proving that hot runner system is balanced. Cavities are filled in a relatively short time and final filling value was set to 1,2 s. Results of the analysis confirmed correct selection of injection molding machine as no parameter is exceeded. From cooling analysis can be observed that time to reach ejection temperature on places where stripper rings are situated is 1,54 s. Therefore time to reach ejection temperature can be set on 1,2 s. Results provided from warp analysis show that the maximum total deformation is 1,62 mm. The maximum deflection values are situated at the top perimeter area of the part. Total deformation is relatively big, but it does not have any fatal consequences on usage of the part. These results can be justified by small wall thickness and insufficient cooling in this area. Due to occurrence of stripper rings no cooling circuits were designed in this area.

CONCLUSIONS

Concept of cold runner mold is similar to the first variant with hot runner system. However due to cold runner system a three plate mold concept had to be chosen. Things like mold cooling, sizes of plates and the way of part forming remained the same with ejector system differing only slightly in comparison to hot runner mold. Opening in individual parting planes is done with help of latch locking system selected from Hasco catalog. From calculation we can say that cost of one injection molding cycle with hot runner system is 2,92 CZK and for the cold runner system it´s 4,65 CZK. Balance of individual runner systems shows that 82 498 injection cycles are required to pay the initial cost of hot runner system.This number of cycles equals to 344 working hours or 43 continuous shifts. After considering pros and cons of individual variants, injection mold with hot runner was chosen. With help of software that is mentioned above 3D models of injection molds were de-signed. 3D models served like a keystone for production of 2D drawing documentation.

FUTURE SCOPE

Minimization of plastic injection molding defects using different types of hot and cold feeding systems by FEM simulation is recommended for further work .

BIBLIOGRAPHY

- [1] BEAUMONT, John P., R. NAGEL a R. SHERMAN. *Successfulinjectionmolding: process, design, and simulation; with CD-ROM.*Munich [u.a.]: Hanser, 2002. ISBN 34-461-9433-9.
- [2] REES, Herbert. *Moldengineering*. 2nd ed. Munich: HanserPublishers, 2002, 688 s. ISBN 34-462-1659-6.
- [3] BEAUMONT, John P. *Runner and gating design handbook: tools for successful in-jection molding*. 2nd ed. Cincinnati: Hanser, c2007, xvi, 308 p. ISBN 1569904219.
- [4] BOBČÍK, L. *Formy pro zpracování plastů: vstřikování termoplastů. Díl 1*. 2. upr.
- vyd. Brno: Uniplast, 1999. 133 s.
- [5] MENGES, Georg. *How to makeinjectionmolds*. 3rd ed. Munich: HanserPublishers, 2001, 612 s. ISBN 34-462-1256-6.
- [6] BOBČÍK, L. *Formy pro zpracování plastů: vstřikování termoplastů. Díl 2.* 1. vyd.
- Brno: Uniplast, 1999. 214 s.
- [7] KAMAL, Musa R, Avraam I ISAYEV a Shih-Jung LIU. *Injectionmolding: technol-ogy and fundamentals*. Cincinnati: Hanser,

c2009, xxviii, 926 p. Progress in polymerprocessing. ISBN 15-699-0434-0.

- [8] MÉZL, M., *Základytechnológievstrekovaniaplastov*. Olomouc: Mapro, 2012. 301 s. ISBN 978-80-970749-7-5.
- [9] STRITZKE, Bernie. *Custom molding of thermoset elastomers a comprehensive ap-proach to materials, mold design, and processing*. Munich: Hanser Publishers, 2009. ISBN 9781613443026.
- [10] SKOČOVSKÝ, P., BOKŮVKA, O., KONEČNÁ, R., TILLOVÁ, E., *Náuka o mate-riálipre odbory strojnícke.*, 2.vyd. Žilina: Žilinská univerzita v Žiline,2006. 349 s. ISBN 80-8070-593-3
- [11] MAŇAS, M., HELŠTÝN, J. *Výrobnístroje a zařízení: určenoproposl. fak. technolo-gické*. 1. vyd. Brno: VUT, 1990, 199 s. Učebnítextyvysokýchškol. ISBN 80-214-0213-X.
- [12] KAZMER, David. *Injectionmolddesignengineering*. Cincinnati: HanserGardner, c2007, xx, 423 p. ISBN 978-344-6412-668.
- 13] CAMPO, E. *Thecomplete part designhandbook: forinjectionmoldingofthermoplas-tics*. Cincinnati: HanserGardnerPublications, c2006, xxi, 870 p. ISBN 978-156-9903-759.
- [14] ZEMAN, Lubomír. *Vstřikováníplastů: úvod do vstřikovánítermoplastů*. 1. vyd. Pra-ha: BEN - technickáliteratura, 2009, 247 s. ISBN 978-80-7300-250-3.
- [15] OSSWALD, Tim A, Lih-Sheng TURNG a Paul J GRAMANN. *Injection molding handbook*. 2nd ed., Updated 2nd ed. Cincinnati: Hanser Gardner Publications, c2008, xvii, 764 p. ISBN 9783446407817.
- [16] ČAMAJ, M. *Návrhvstrekovacejformy*. Zlín, 2013. 70 s. Bakalárskápráca. UTB Zlín.