MULTI-CAVITY HOT RUNNER INJECTION MOLD TOOL POLYMER

[1]Mr.P.Vinod, [2]Mr.K.Vijaykumar

Assistant Professor, Department of Mechanical Engineering, Vignan Institute of Technology & Science, Vignan Hills, Deshmukhi (V) , Pochampally (M), Nalgonda (Dist).

ABSTRACT: The aim of this paper is to design a multi-cavity injection mold. The theoretical part of this thesis scribes problematic of injection molding and injection mold design, namely runner systems, mold cooling and venting. Practical part of the thesis 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 submitted to injection molding process analysis and documented with assembly drawing. Injection mold designing was done in CAD application CATIA and evaluated by injection molding analysis by using ANSYS.

Keywords: ***Injection mold, Injection molding, Polymer***

I. INTRODUCTION

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 insulates. 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.). 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 design. 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.

II. THEORY SUMMARY

Theoretical part is divided into four parts. The first part describes division of polymer materials. In this part individual groups of polymer materials are shortly described and are accompanied with their basic properties. The second part deals with injection molding technology. This part describes injection molding process and injection molding cycle. Process of this technology is also mentioned. The third part is devoted to part design considerations and guidelines for molded plastic parts are described in this chapter. The fourth part of the theory is the biggest and focuses on injection mold design. It describes cold runner systems, hot runner system, mold ejection, mold cooling and venting of molds.

III. ANALYSIS

GOALS OF ANALYTICAL PART

The goal of this master thesis is to design a multi-cavity injection mold for the given part.

Individual goals of the master thesis:

- Elaborate theoretical part on the given topic Design 3D model of the given part
- Design model of hot runner injection mold
- Design 3D model of cold runner injection mold Evaluate individual designs
- Economical evaluation of the designed molds
- Run CAE analysis of selected injection mold type
- Draw 2D assembly drawing of selected injection mold type Evaluate the results of analytical part

IV. USED DESIGN SOFTWARE

CATIA is software developed by French company Dassault Systems. CATIA V5 is a system that is capable of covering the complete life cycle of a product.

Autodesk Simulation Mold flow Synergy 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 Mold flow 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.

4.1. INJECTED PART

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

Fig. 1 Render 3D model of the cup

4.2. Material

Chosen material for the injected part is polypropylene (PP). PP is a semi crystalline thermo-plastic material, which belongs to polyolefin group. Its crystalline 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. Process ability and dye ability of PP is very good. PP is used in a wide range of applications, for example in the food, textile and automotive industry. Chosen PP granulate is from manufacture Sabic Europe B.V, type PP RA12MN40. It is a random copolymer specially developed for injection molding. It is characterized by high melt flow rate (MFR) and good mechanical properties. Typical application for this type is in house wares and thin walled packaging. Data sheet of the material is enrolled as an appendix at the end of the thesis.

Table1. Material characterization

Trade name	RA12MN40
Density	905 [g/cm^3]
ITT (230 °C/ 2,16 kg)	40 [g/10 min]
Young modulus E	1340 [MPa]
Shear modulus G	481,3[MPa]
Parallel shrinkage	1,386 [%]
Perpendicular shrinkage	2,004 [%]
Maximum shear rate	100000 [1/s]
Maximum shear stress	0,25 [MPa]
Fillers	unfilled
Melt temperature	200 – 250 [°C]
Mold surface temperature	10 – 50 [°C]
Absolute maximum melt temperature	290 [°C]
Ejection temperature	107 [°C]

Table 2: Material characterization

Clamping force	3200 [kN]
Distance between tie bars	720 x 720 [mm]
Mold mounting plates	1040 x 1040 [mm]
Mold height	300 – 800 [mm]
Max. ejector stroke	250 [mm]
Max. ejector force	86 [kN]
Max. weight of moveable mold half	2900 [kg]
Injection pressure	2500 [bar]
Holding pressure	2500 [bar]
Max. shot weight	723 [g]
Effective screw length	23 [L/D]
Screw diameter	60 [mm]
Max. screw torque	2140 [Nm]
Nozzle contact force	90 [kN]

4.3. Mold multiplicity

Injection mold multiplicity is chosen according to various parameters, including mold complexity, productivity and required quality of injected products. In this case, four cavity injection molds was designed in order to accomplish thesis´s assignment.

4.3.1. Cavity

During injection cycles, cavity is exposed to high temperatures and pressures. Therefore cavity has to be made from hardened steel. Chosen material for the cavity is X210Cr12 which has old ČSN equivalent 19 436. Cavity was adjusted for the hot nozzles and for cooling purposes, series of cooling channels were drilled in to the cavity.

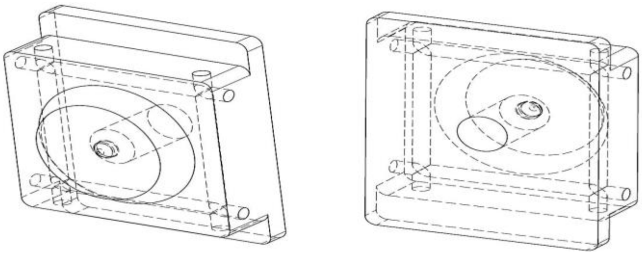

Fig. 2 Cavity 3D model

4.3.2. Core

Core is made from the same material as the cavity, as it has to withstand the same pressures and temperatures. From the manufacturing point of view, core had to be adjusted for proper insertion of stripper ring. For cooling purposes, a relatively big pocket had to be designed. The pocket was designed for an unconventional type of spiral core as conventional types from Hasco catalog are not suitable enough for this situation due to low heat removal coefficient. Due to leakage of coolant, cores were grooved to ensure proper insertion of sealing O-rings.

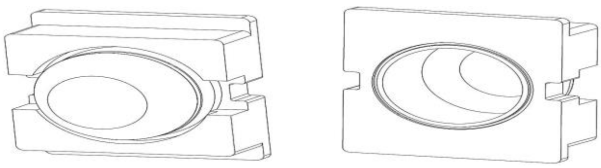

Fig. 3 Core 3D model

4.4. Mold frame

Mold plates and other standard parts of the mold frame were chosen from the Hasco catalog. Plate´s dimensions were chosen with regard to the way of part forming, cavity multiplicity and size of the injected part. The mold from catalog was inserted as an assembly of two plate mold. Basic layout of the assembly was changed and one additional plate was added to fix hot runner system in right place. Individual plates except thermal insulating plates are made from steel 1.1730 (C45U). Thermal insulating plates are made of thermal insulated material – synthetic resin filled with glass fiber. Mold size dimensions are 696 x 696 x 541 mm (L x W x H).

4.4.1. Hot sprue bushing

Hot sprue was selected from the Hasco catalog of standard parts. Chosen type is Z1055/3/30x85/12 with the sprue channel diameter of 12 mm.

4.4.2. Hot manifold

Hot manifold delivers melted material to four hot nozzles. Required hot runner system has to be custom made. This type of hot runner manifold of this size is not produced by Hasco and DME and cannot be found in their catalogs. It is designed for four hot nozzles with channel´s diameter 12 mm. Chosen material for production of the hot runner system is steel 1.2343 with ČSN equivalent 19 552. Dimensions of selected hot manifold are 280 x 406 x 37 mm (L x W x H).

4.4.3. Hot nozzles

Hot nozzles for designed injection mold were chosen from Hasco standard parts catalog. Selected type of four nozzles is Z3410/32x100. Each of the nozzles has channel’s diameter of 4.5 mm and a sufficient shot weight of 80 g, which is enough for this purpose. Nozzles are suitable for the selected type of material. Data sheet of the selected type is enrolled as an appendix at the end of the thesis.

4.4.4. Mold cooling

Because of multi cavitness of mold, mold cooling had to be divided into several cooling circuits. Cooling circuit accessories were selected from Hasco catalog and water with no additives was selected as a coolant.

4.4.5. Core cooling

In this case core spirals were used in core cooling circuits. Core spirals had to be specifically designed for this application. Standard part core spirals from Hasco catalog do not have required size and shape; therefore do not reach required heat removal efficiencies. Chosen material for designed core spirals is aluminum alloy 6082 (AlMgSi1). This alloy is medium strength alloy with excellent corrosion resistance. Core spirals from Hasco standard catalog are also produced of the same material.

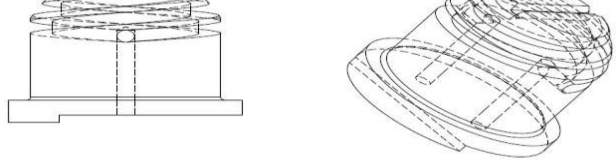

Fig. 4 Core spiral model

The mold contains two cooling circuits for the cores. Coolant is distributed in circuits through 14 mm channels and regulation of the flow is assured with plugs (Z940). Cooling circuits are ended with hose connectors and leakage prevention is secured with sealing O-ring (Z98).

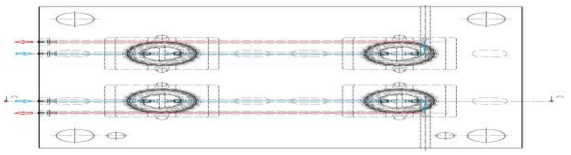

Fig. 5 Core spiral cooling circuits

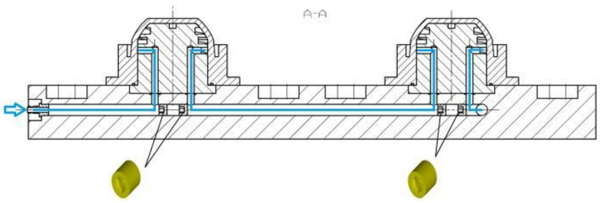

Fig. 6 Section A-A of core spiral cooling circuit

4.4.6. Sliders cooling

Sliders are cooled like cavities namely by cooling line network of drilled cooling channels.

Sliders are positioned vertically to each other and in order to bridge the space between them hoses were used. Furthermore, in order to cool the sliders in two layers of cooling channels and to maintain flow with only one input diverting coupling units were used. These standard parts also come from Hasco catalog.

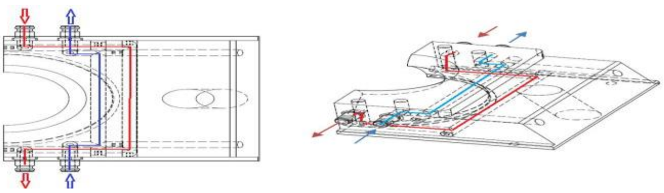

Fig. 7 Slider cooling

4.4.7. Ejection

For proper ejection of the parts from the mold, it is necessary that the parts remain on left (moveable) side of the ejection mold. In this case this is expected to happen due to shrink-age of the injected material on the core. Ejection of the individual parts from the core is realized by stripper rings. This type of ejection was designed to achieve ejection without any foot-print on the part and to eject the parts with uniform ejection force. The designed assembly consists of stripper rings and connection bars. Stripper rings are inserted into the core and assist to form individual parts. Because of the contact with melted material, they have to be made from the same material as core and cavity.

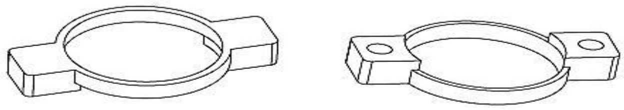

Fig. 8 Stripper rings

Stripper rings are connected with the ejection plates through steel bars. The bars are screwed in the rings and their connection with ejection plates is assured with screws. Selected material for the bars is steel 1.0060 (E335) which has the ČSN equivalent 11 600.

4.4.8. Manipulation system

To restrict the mold from opening in the main parting plane, two of standard parts locking devices from Hasco were installed.

4.5. Cold runner injection mold

Cold runner injection mold is the second variant of injection mold design for the given part. The main difference between the individual variants can be observed on the right stationary injection side. Left moveable side remained almost the same as in the first variant and the variant differ only slightly. Part forming, cooling, venting and manipulation system do not differ at all and therefore are not discussed in this chapter.

4.5.1. Mold frame

Concept of the mold frame is changed compared to hot runner injection mold. For this variant a three plate mold was chosen. This type of mold was chosen to assure proper ejection of cold runner system. Materials of individual plates are the same. Cold runner mold size dimensions are 696 x 696 x 498 mm (L x W x H).

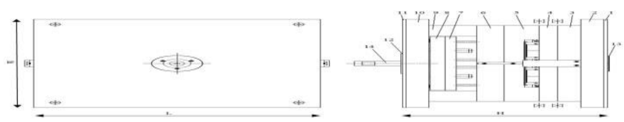

Fig. 9 Mold frame – cold runner mold

1 – Insulating p. right, 2 – Clamping p., 3 – Cavity support p., 4 – Cavity p., 5 – Core p., 6– Core support p., 7 – Ejector p. A, 8 – Ejector p. B, 9 – Riser bar, 10 – Setting p., 11 – Insulating p. left, 12 – Locating ring left, 13 – Locating ring right, 14 – Knockout

4.5.2 Cold runner system

The runner system in this cold runner mold consists of a cold sprue, runners and gates. The sprue delivers the melt to a runner, which in turn delivers the melt to the part-forming cavities. Frozen cold runner system is considered as a waste, which have to be grinded and sold or reused as a filler in different parts.

4.5.3. Cold sprue

Cold sprue delivers the melt into the cold runners and connects injection machine with the injection mold. The sprue was selected from Hasco catalog (Z51).Sprue channel is 6.5 mm wide in the diameter and is tapered by 1° angle. A hole was drilled to the sprue for proper insertion of a pin. The pin prevents the sprue from rotation during the injection

4.5.4. Cold runners and cold gate

Melt is delivered by cold runners in to the cold gate and then into the individual cavities. Cold runners were manufactured into the cavity support plate. Dimensions of the cold runner system were chosen with regard to the runners' length to avoid premature solidification of the melt. Cold runners has a parabolic cross-section with respectively graded diameters of 8 and 7 mm. Cold gate is manufactured in individual cavities with diameter of 1.8 mm. Type of selected gate is a point type gate, which requires three mold system.

Whole cold runner system represents waste material which is produced during every injection cycle. Due to long runner channels, size of the runner system is relatively and it has weight of 45 g.

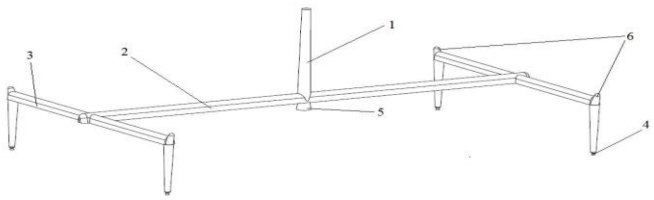

Fig. 10 Cold runner system

1 – Cold sprue (Z51), 2 – Cold runner – diameter 8 mm, 3 – Cold runner – diameter 9mm, 4 – Cold gate, 5 – Cold runner holder, 6 – Undercuts.

4.5.5. Ejection

Because of cold runner system, ejection is in this case more difficult. However, ejector assembly differs from hot runner mold only slightly. Basic concept of ejection with strip-per rings remained the same and an ejector pin for cold runner holder was added.

Due to three plate mold system, injection mold opens in two parallel parting planes with a time delay. Firstly mold opens in the secondary plane causing that cold runner system is divided from cavity support plate. Due to cold runner holder, cold runner system is held on the cavity plate. Then injection mold opens in the main parting plane, causing ejection of the injected parts and cold runner system. To assure a proper opening of the mold, a latch locking device was selected from Hasco catalog.

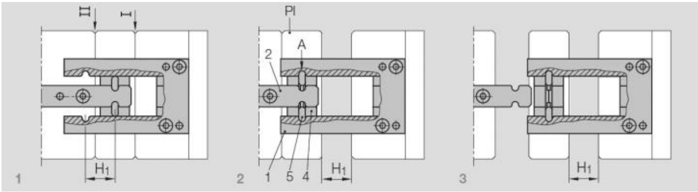

Fig. 11 Latch locking device

5. COMPARISON OF INDIVIDUAL VARIANTS

Individual variants differ in used runner system. First variant was designed with a hot runners system. The system distributes the melted material into the cavities through four nozzles, which were selected from Hasco catalog. Second variant was designed with a cold runner system and the melt is in this case distributed through cold runners. Variants have almost the same ejection side with runner holder designed in cold runner mold being the only different feature. Cooling system is the same in both designs.

First variant compared to the second one is designed with one additional plate. However, it has many advantages compared to the second variant. The main advantages are the faster injection cycle, no waste material being produced and sufficient usage in big series.

The biggest advantage of the injection mold with cold runner system is smaller initial costs. However, the size of the cold runner system is relatively big and the complexity of the injection mold is increased with the designed three plate mold.

5.1 Economical summary

Goal of economical summary was to compare hot runner system with cold runner system from economical point of view and approximately determine a number of working hours which are needed for profitable hot runner usage.

In this chapter Euro currency is transferred to CZK with rate given by Slovak National bank on 15. 4. 2015.

$$1 \text{ EUR} = 27{,}485 \text{ CZK} \quad (1)$$

Among production costs that are discussed in this chapter are include cost of hot runner system, material costs and energy costs.

5.1 Material costs

Chosen material is PP RA12MN40 from company Sabic Europe and it costs 1.40EUR per kilogram.

(2)

1.40 EUR = 38.381 CZK

5.1.1 Energy costs

Cost of electric energy on 16. 4. 2015 according to www.kurzy.cz is 31.6 EUR/MWh. Cost of electric energy is 0.866 CZK for 1 kW/h.

5.1.2 Hot runner system

Hot manifold

For cost estimation of hot runner system was Hasco catalog where prices of products are available used. Hasco offers a smaller hot runner system of the same shape for 2616 € and therefore the cost of produced hot runner system was set to 2900 €. Energy consumption was increased to 1800 W from Hasco´s 1470W.

2900 EUR = 79 503.5 CZK (3)

Nozzles

Hot nozzles were selected from Hasco catalog with cost of 811,88EUR for each.
4x 811.88 € = 3247.5 EUR = 89 030 CZK (4)

Table. 4. Expenses comparison

	Cost [EUR]	Cost [CZK]
Hot runner system	6347.5	174 016.5
Cold runner system	1 141.5	31 294.2
Total difference	**5 206**	**142 722.3**

Balance compares cold runner system with hot runner system. The result of the balance shows number of injection cycles required for hot runner system to be profitable. Tab.4. shows the cost difference of initial cost for individual runner systems. Changes in mold frame assembly and different cycle durations were neglected.

4.65 = 2.92x + 142 722 (5)

1.73 = 142 722 (6)

= 82 498 Cycles (7)

Duration in time:

= (x ∗ 15)/3600 (8)

= 343.7 = 344 hours (9)

Balance of the runner systems shows that 82 498 cycles are required to balance initial costs of hot runner system. The number of cycles equals to 344 working hours or 43 continuous shifts.

5.3 Final variant selection

Each of the variants has its advantages and disadvantages, but for an injection mold of this size and multiplicity, hot runner injection mold is the better option. Because of long cold runner channels, a lot of waste material is produced in every injection cycle. Injection cycle in hot runner mold is shorter and more suitable for automatized production. Further-more, this type of product is specifically produced in mass productions and therefore initial costs of hot runner system will pay off. Cold runner system mold would be more suitable for smaller number of cavities and for smaller production series.

In previous calculations price of injection molding cycle using hot runner system was compared to usage of cold runner. Costs of injection molding cycle with hot runner systemic 2.92 CZK and for the cold runner system it is 4.65 CZK. Balance of individual runner systems shows that 82030 injection cycles are required to pay the initial cost of hot runner system.

6. CAESIMULATION

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, figures in this chapter show results for only one of the cavities instead of four. In CAE analysis the symmetry was not contemn plated.

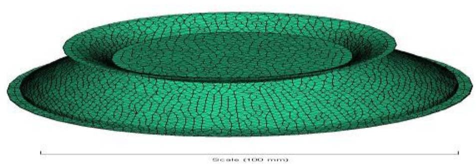

Fig. 12 Meshed part

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 and this solution was found compliant.

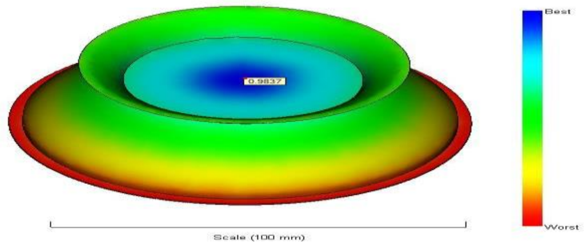

Fig. 13 Results of the gate location analysis

6.1. Analysis settings

After performing 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 material and injection molding machines are discussed in chapters 8.2 and 9. Selected molding machine is not included in Autodesk Mold flow 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 materials this one is labeled as Tool steel P20 with DIN steel equivalent 1.2311 with following characterizations.

Table5: Mold material

Density	7.8 [g/cm3]
Specific heat capacity	460 [J/kg.°C]
Thermal conductivity	29 [W/m.°C]
Young modulus	200000 [MPa]
Poisson number	0.33

6.2 Filling time

Filling is done simultaneously in all cavities proving that hot runner system is balanced. As seen from Fig. 53 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.

6.3 Clamp force

Size of the maximum closing force should be one of the estimation for selecting an appropriate 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.

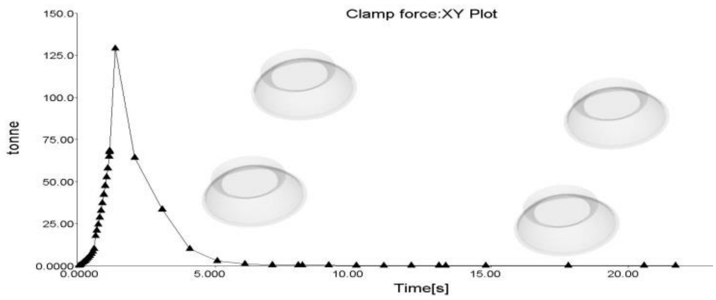

Fig.14 Clamp force

6.4 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.

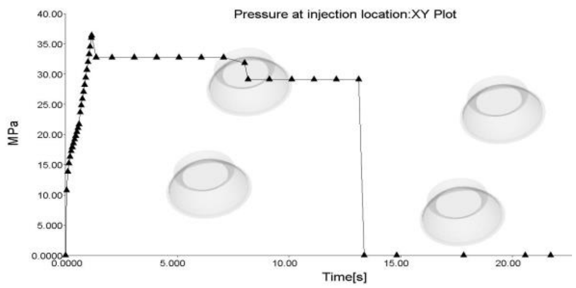

Fig.15 Pressure at time diagram

6.5 Shear rate

The highest shear rate allowed for the selected material according to data sheet is 100 000 s[1], 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 maximum shear rate is 47 627 s^{-1}.

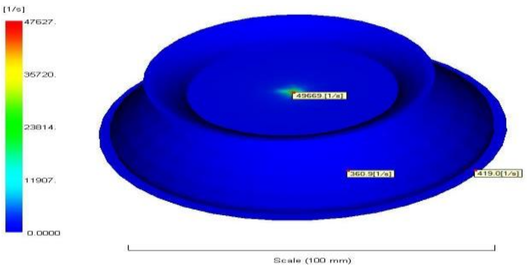

Fig.16 Shear rate analysis

6.5.1. 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. Problems are solved with the design that is discussed in previous sections of the thesis.

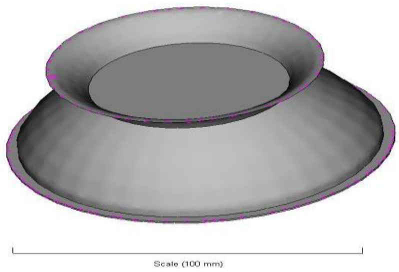

Fig. 17 Air traps – results

6.6. 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.

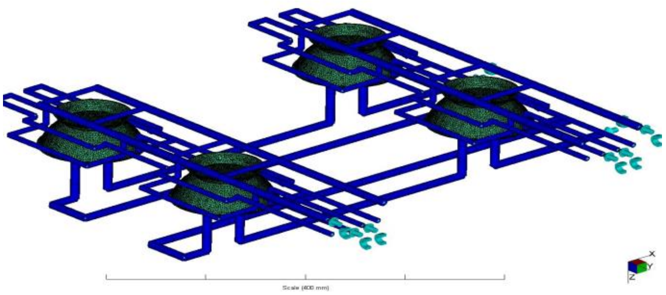

Fig. 18 cooling circuits

6.7 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.

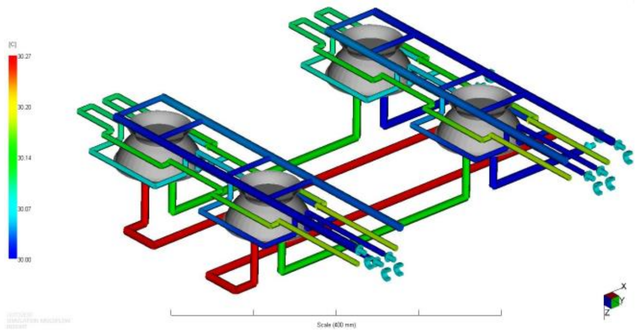

Fig. 19 Circuit coolant temperature – results

6.8 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 temperat

The result 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.

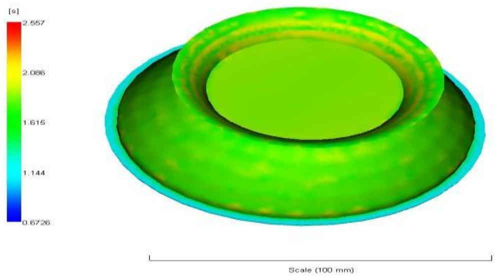

Fig. 20 Time to reach ejection temperature – results

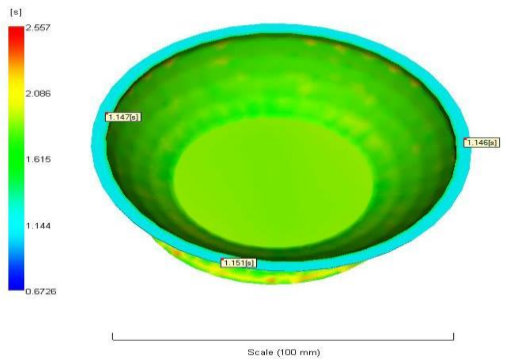

Fig. 21 Time to reach ejection temperature – results 2

6.9 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 top perimeter 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, despite 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.

7. RESULTS AND DISCUSSIONS

The aim of this master thesis was to design of multi-cavity injection mold in two variants for the same product. One variant with hot runner system and the second one with cold runner system. Next goal was to compare individual designs, do economical summary of individual variants and select final variant. After this, CAE analysis should have been run on chosen variant. The last point of assignment was to provide drawing documentation in form of assembly drawing with bill of material.

Injected part was a cup for desserts and yogurts. Selected material for this application had to be a material suitable for food packaging. Finally a polypropylene from Sabic Company was selected. As injection molding machine was selected Arburg Allrounder 720H.

Injection mold was designed with an effort to maximize the usage of standard parts in order 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.

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.

In next chapter individual variants were discussed and economical summary of used runner system was done. 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.

Injection molding process was simulated in program Autodesk Mold flow 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.

8. CONCLUSION

Analysis part of this master thesis is devoted to two multi-cavity injection mold designs, CAE analysis of injection process and to assembly drawing of selected variant.

Design itself was done in software CATIA V5R19 with support of standard catalog Hasco DakoModul. Autodesk Mold flow Insight 2014 was used for injection molding process simulation. Concept of the injection mold design is described in individual chapters.

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.

Individual designs with CAE analysis are burnt on attached CDs and attached in appendices.

9. BIBLIOGRAPHY

1. BEAUMONT, John P., R. NAGEL a R. SHERMAN. *Success fulinjection molding: process, design, and simulation; with CD-ROM.*Munich [u.a.]: Hanser, 2002. ISBN 34-461-9433-9.
2. REES, Herbert. *Mold engineering*. 2nd ed. Munich: Hanser Publishers, 2002, 688 s. ISBN 34-462-1659-6.
3. BEAUMONT, John P. *Runner and gating design handbook: tools for successful injection 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 make injection molds*. 3rd ed. Munich: Hanser Publishers, 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. *Injection molding: technology and fundamentals*. Cincinnati: Hanser, c2009, xxviii, 926 p. Progress in polymer processing. ISBN 15-699-0434-0.
8. MÉZL, M., *Základy technológie vstrekovaniaplastov*. Olomouc: Mapro, 2012. 301 s. ISBN 978-80-970749-7-5.
9. STRITZKE, Bernie. *Custom molding of thermo set elastomers a comprehensive approach 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