# Multi-cavity Injection Mold Design

Bc. Michal Čamaj

Master thesis
2015

Tomas Bata University in Zlín
Faculty of Technology

Univerzita Tomáše Bati ve Zlíně
Fakulta technologická
Ústav výrobního inženýrství
akademický rok: 2014/2015

# ZADÁNÍ DIPLOMOVÉ PRÁCE
(PROJEKTU, UMĚLECKÉHO DÍLA, UMĚLECKÉHO VÝKONU)

| | |
|---|---|
| Jméno a příjmení: | **Bc. Michal Čamaj** |
| Osobní číslo: | **T13471** |
| Studijní program: | **N3909 Procesní inženýrství** |
| Studijní obor: | **Konstrukce technologických zařízení** |
| Forma studia: | **prezenční** |
| Téma práce: | **Návrh více-dutinové vstřikovací formy** |

Zásady pro vypracování:

1. **Vypracujte literární rešerši na dané téma**
2. **Vytvořte 3D model zadaného výrobku**
3. **Navrhněte vstřikovací formu včetně výkresové dokumentace**
4. **Vypracujte tokovou analýzu navrženého řešení**
5. **Vytvořený návrh zhodnoťte**

Rozsah diplomové práce:

Rozsah příloh:

Forma zpracování diplomové práce: **tištěná/elektronická**

Seznam odborné literatury:

1. **REES, H. Mold Engineering**
2. **BEAUMONT, J.P. et al. Successful injection molding process, design, and simulation**
3. **BOBČÍK, L. Formy pro zpracování plastů: vstřikování termoplastů, Díl 1 a 2**

Vedoucí diplomové práce: **Ing. Jan Navrátil**
Ústav výrobního inženýrství

Datum zadání diplomové práce: **30. ledna 2015**

Termín odevzdání diplomové práce: **13. května 2015**

Ve Zlíně dne 30. ledna 2015

UNIVERZITA TOMÁŠE BATI VE ZLÍNĚ Fakulta technologická L.S.

doc. Ing. Roman Čermák, Ph.D.
*děkan*

prof. Ing. Berenika Hausnerová, Ph.D.
*ředitel ústavu*

Příjmení a jméno: Michal Čamaj  Obor: KTZ

# PROHLÁŠENÍ

Prohlašuji, že

- beru na vědomí, že odevzdáním diplomové/bakalářské práce souhlasím se zveřejněním své práce podle zákona č. 111/1998 Sb. o vysokých školách a o změně a doplnění dalších zákonů (zákon o vysokých školách), ve znění pozdějších právních předpisů, bez ohledu na výsledek obhajoby [1)];
- beru na vědomí, že diplomová/bakalářská práce bude uložena v elektronické podobě v univerzitním informačním systému dostupná k nahlédnutí, že jeden výtisk diplomové/bakalářské práce bude uložen na příslušném ústavu Fakulty technologické UTB ve Zlíně a jeden výtisk bude uložen u vedoucího práce;
- byl/a jsem seznámen/a s tím, že na moji diplomovou/bakalářskou práci se plně vztahuje zákon č. 121/2000 Sb. o právu autorském, o právech souvisejících s právem autorským a o změně některých zákonů (autorský zákon) ve znění pozdějších právních předpisů, zejm. § 35 odst. 3 [2)];
- beru na vědomí, že podle § 60 [3)] odst. 1 autorského zákona má UTB ve Zlíně právo na uzavření licenční smlouvy o užití školního díla v rozsahu § 12 odst. 4 autorského zákona;
- beru na vědomí, že podle § 60 [3)] odst. 2 a 3 mohu užít své dílo – diplomovou/bakalářskou práci nebo poskytnout licenci k jejímu využití jen s předchozím písemným souhlasem Univerzity Tomáše Bati ve Zlíně, která je oprávněna v takovém případě ode mne požadovat přiměřený příspěvek na úhradu nákladů, které byly Univerzitou Tomáše Bati ve Zlíně na vytvoření díla vynaloženy (až do jejich skutečné výše);
- beru na vědomí, že pokud bylo k vypracování diplomové/bakalářské práce využito softwaru poskytnutého Univerzitou Tomáše Bati ve Zlíně nebo jinými subjekty pouze ke studijním a výzkumným účelům (tedy pouze k nekomerčnímu využití), nelze výsledky diplomové/bakalářské práce využít ke komerčním účelům;
- beru na vědomí, že pokud je výstupem diplomové/bakalářské práce jakýkoliv softwarový produkt, považují se za součást práce rovněž i zdrojové kódy, popř. soubory, ze kterých se projekt skládá. Neodevzdání této součásti může být důvodem k neobhájení práce.

Ve Zlíně 7. 5. 2015

Čamaj

---

[1)] *zákon č. 111/1998 Sb. o vysokých školách a o změně a doplnění dalších zákonů (zákon o vysokých školách), ve znění pozdějších právních předpisů, § 47 Zveřejňování závěrečných prací:*

*(1) Vysoká škola nevýdělečně zveřejňuje disertační, diplomové, bakalářské a rigorózní práce, u kterých proběhla obhajoba, včetně posudků oponentů a výsledku obhajoby prostřednictvím databáze kvalifikačních prací, kterou spravuje. Způsob zveřejnění stanoví vnitřní předpis vysoké školy.*

*(2) Disertační, diplomové, bakalářské a rigorózní práce odevzdané uchazečem k obhajobě musí být též nejméně pět pracovních dnů před konáním obhajoby zveřejněny k nahlížení veřejnosti v místě určeném vnitřním předpisem vysoké školy nebo není-li tak určeno, v místě pracoviště vysoké školy, kde se má konat obhajoba práce. Každý si může ze zveřejněné práce pořizovat na své náklady výpisy, opisy nebo rozmnoženiny.*

*(3) Platí, že odevzdáním práce autor souhlasí se zveřejněním své práce podle tohoto zákona, bez ohledu na výsledek obhajoby.*

[2)] *zákon č. 121/2000 Sb. o právu autorském, o právech souvisejících s právem autorským a o změně některých zákonů (autorský zákon) ve znění pozdějších právních předpisů, § 35 odst. 3:*

*(3) Do práva autorského také nezasahuje škola nebo školské či vzdělávací zařízení, užije-li nikoli za účelem přímého nebo nepřímého hospodářského nebo obchodního prospěchu k výuce nebo k vlastní potřebě dílo vytvořené žákem nebo studentem ke splnění školních nebo studijních povinností vyplývajících z jeho právního vztahu ke škole nebo školskému či vzdělávacího zařízení (školní dílo).*

*[1] zákon č. 121/2000 Sb. o právu autorském, o právech souvisejících s právem autorským a o změně některých zákonů (autorský zákon) ve znění pozdějších právních předpisů, § 60 Školní dílo:*

*(1) Škola nebo školské či vzdělávací zařízení mají za obvyklých podmínek právo na uzavření licenční smlouvy o užití školního díla (§ 35 odst. 3). Odpírá-li autor takového díla udělit svolení bez vážného důvodu, mohou se tyto osoby domáhat nahrazení chybějícího projevu jeho vůle u soudu. Ustanovení § 35 odst. 3 zůstává nedotčeno.*

*(2) Není-li sjednáno jinak, může autor školního díla své dílo užít či poskytnout jinému licenci, není-li to v rozporu s oprávněnými zájmy školy nebo školského či vzdělávacího zařízení.*

*(3) Škola nebo školské či vzdělávací zařízení jsou oprávněny požadovat, aby jim autor školního díla z výdělku jím dosaženého v souvislosti s užitím díla či poskytnutím licence podle odstavce 2 přiměřeně přispěl na úhradu nákladů, které na vytvoření díla vynaložily, a to podle okolností až do jejich skutečné výše; přitom se přihlédne k výši výdělku dosaženého školou nebo školským či vzdělávacím zařízením z užití školního díla podle odstavce 1.*

## ABSTRAKT

Cieľom tejto diplomovej práce je navrhnúť mnohonásobnú vstrekovaciu formu. Teoretická časť popisuje problematiku vstrekovania, konštrukciu foriem, menovite vtokové systémy, chladenie a odvzdušnenie vstrekovacích foriem. Praktická časť sa zaoberá dvoma konštrukčnými návrhmi vstrekovacej formy pre daný diel, ktorý je kelímek na dezerty. Nasleduje vyhodnotenie a výber konštrukčného riešenia. Vybraný návrh je podrobený tokovej analýze a doložený zostavným výkresom. Návrh vstrekovacej formy bol realizovaný pomocou 3D modelovacieho programu CATIA V5R19 a tokové analýzy boli spravcované v softeware Autodesk Moldflow Synergy 2014.



Kľúčovéslová: Vstrekovacia forma, Vstrekovanie, Polymér



## ABSTRACT

The aim of this master thesis 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 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 V5R19 and evaluated by injection molding analysis in Autodesk Moldflow Synergy 2014 software.



Keywords: Injection mold, Injection molding, Polymer

## ACKNOWLEDGEMENTS

I would like to express my sincere gratitude to all people who supported me during the work on the thesis.

I am especially grateful to my supervisor, Ing. Jan Navrátil, for profesional guidance, given advice and time that he has given me throughout writing the thesis.

I declare I worked on this Master Thesis by myself and I have mentioned all the used literature.

I hereby declare that the print version of my Bachelor's/Master's thesis and the electronic version of my thesis deposited in the IS/STAG system are identical.

In Zlín 22.4. 2015

......................................

Signature

# CONTENTS


**INTRODUCTION ........ 11**
**I THEORY ........ 12**
**1 DIVISION OF POLYMER MATERIALS ........ 13**
1.1 THERMOPLASTIC MATERIALS ........ 13
1.1.1 Amorphous thermoplastic polymers ........ 14
1.1.2 Semicrystalline thermoplastic polymers ........ 15
1.2 THERMOSETS ........ 16
1.3 ELASTOMERS ........ 16
**2 INJECTION MOLDING ........ 18**
2.1 INJECTION MOLDING ........ 18
2.1.1 Injection molding cycle ........ 18
**3 PART DESIGN ........ 20**
3.1 GUIDELINES FOR MOLDED PLASTIC PARTS ........ 20
3.1.1 Designing the primary wall ........ 20
3.1.2 Corners, fillets and radii ........ 21
3.1.3 Draft angles and undercuts ........ 21
3.1.4 Ribs, gussets, and bosses ........ 21
3.1.5 Marks and signs ........ 22
3.1.6 Threads ........ 23
**4 MOLD DESIGN ........ 24**
4.1 RUNNER SYSTEMS ........ 24
4.2 COLD RUNNER MOLDS ........ 24
4.3 HOT RUNNER MOLDS ........ 28
4.3.1 Hot runner manifoldand drops ........ 29
4.3.2 Hot drops (nozzles) ........ 30
4.4 MOLD EJECTION SYSTEMS ........ 31
4.4.1 Means of ejection ........ 32
4.5 MOLD COOLING ........ 33
4.5.1 Cooling line networks ........ 33
4.5.2 Baffles and bubblers ........ 34
4.5.3 Coolants ........ 35
4.6 VENTING ........ 36
**5 THEORY SUMMARY ........ 37**
**II ANALYSIS ........ 38**
**6 GOALS OF ANALYTICAL PART ........ 39**
**7 USED SOFTWARE ........ 40**
7.1 CATIA V5R19 ........ 40
7.2 AUTODESK MOLDFLOW SYNERGY 2014 ........ 40
7.3 HASCO DAKO MODUL ........ 40
**8 INJECTED PART ........ 41**

8.1 Injected part specification ..... 41
8.2 Material ..... 41
**9 INJECTION MOLDING MACHINE ..... 43**
**10 INJECTION MOLD DESIGN ..... 44**
10.1 Mold multiplicity ..... 44
10.1.1 Part forming ..... 44
10.1.2 Cavity ..... 46
10.1.3 Core ..... 47
10.1.4 Sliders ..... 48
10.2 Hot runner injection mold ..... 48
10.2.1 Mold frame ..... 48
10.2.2 Hot runner system ..... 50
10.2.3 Mold cooling ..... 51
10.2.4 Ejection ..... 55
10.2.5 Venting ..... 56
10.2.6 Manipulation system ..... 57
10.3 Cold runner injection mold ..... 57
10.3.1 Mold frame ..... 57
10.3.2 Cold runner system ..... 59
10.3.3 Ejection ..... 61
**11 COMPARISON OF INDIVIDUAL VARIANTS ..... 63**
11.1 Economical summary ..... 63
11.1.1 Material costs ..... 63
11.1.2 Energy costs ..... 63
11.1.3 Hot runner system ..... 64
11.1.4 Production costs – cold runner ..... 64
11.1.5 Production costs – hot runner ..... 65
11.1.6 Balance ..... 65
11.2 Final variant selection ..... 66
**12 CAESIMULATION ..... 68**
12.1 Analysis settings ..... 69
12.1.1 Process parameters ..... 70
12.2 Filling time ..... 71
12.3 Clamp force ..... 71
12.4 Pressure at injection location ..... 72
12.5 Shear rate ..... 73
12.6 Air traps ..... 73
12.7 Cooling analysis ..... 74
12.8 Circuit coolant temperature ..... 75
12.9 Time to reach ejection temperature ..... 75
12.10 Total deflection ..... 77
**RESULTS AND DISCUSSIONS ..... 78**
**BIBLIOGRAPHY ..... 81**
**LIST OF ABBREVIATIONS ..... 84**

**LIST OF FIGURES** ........ **86**
**LIST OF TABLES** ........ **88**
**APPENDICES** ........ **89**

# 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 insulants. 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 technology of polymer processing is injection molding. It is very accurate and efficient manufacturing 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 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.

# I. THEORY

# 1 DIVISION OF POLYMER MATERIALS

The term polymer refers to many “mers” or “monomers.” These monomers are combined by various means to produce a long molecular chain, or polymer. [1]

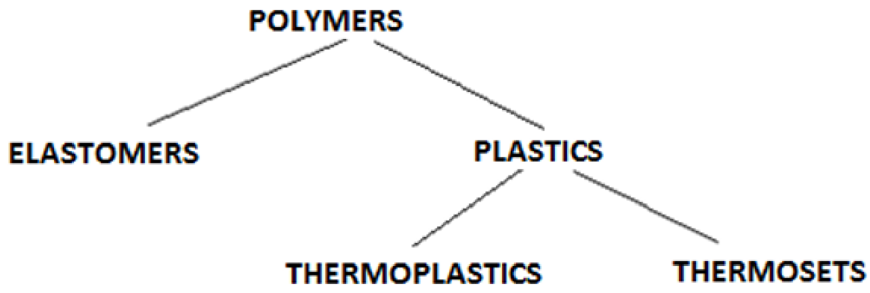



*Fig. 1. Division of polymer materials [4]*

## 1.1 Thermoplastic materials

Thermoplastic materials are made of linear or branched polymer units comprised of repeating monomers. Thermoplastics comprise about 94 % of the volume of material used in the plastics industry. Thermoplastics can be repeatedly heated, melted, and formed into a shape or product. The individual atoms within a molecular chain are held together by very strong primary bonds. These are commonly called covalent bonds. The bonds holding individual polymer chains are much stronger when compared to the secondary covalent bonds. These secondary bonds are commonly referred to as van der Waals forces and have generally less than 5 % strength of the primary bonds. Thermoplastic materials are subdivided into amorphous and semicrystalline thermoplastics. [1, 4]

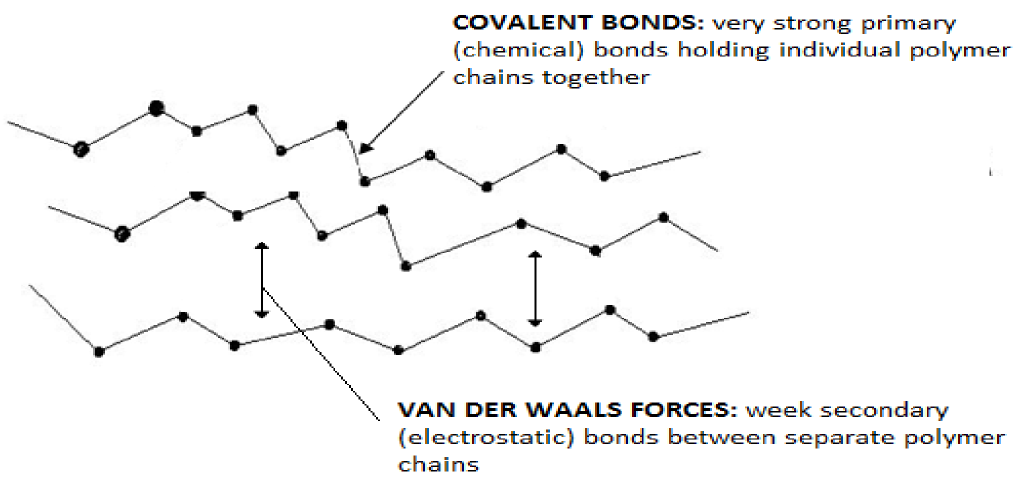



*Fig. 2. Primary and secondary bonds between separate polymer chains [1]*

### 1.1.1 Amorphous thermoplastic polymers

Area of application for amorphous thermoplastics polymers is under the glass transition temperature (Tg). Amorphous polymers are solid in this area. With temperature increasing over Tgamorphouspolymers go over to plastic state in which they are processed. [4]

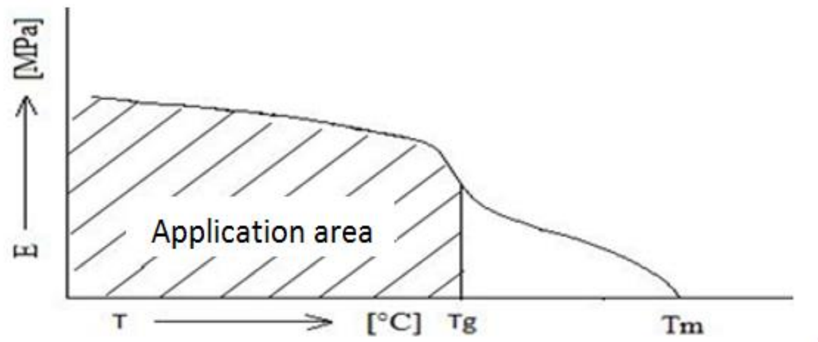



*Fig. 3. Application area – Amorphous thermoplastic polymers [4]*

Some characteristics of amorphous materials include the following [1,8]:

- Random arrangement of the polymer chains.
- Commonly transparent to translucent in their natural form.
- Less chemical resistant than semicrystalline thermoplastics.
- No distinct melt temperature.
- More prone to stress cracking with sustained stress or strain.

Some common amorphous materials are Polystyrene (PS), Polycarbonate (PC), Polyvinyl chloride (PVC) or Polymethyl methacrylate (PMMA). [8]

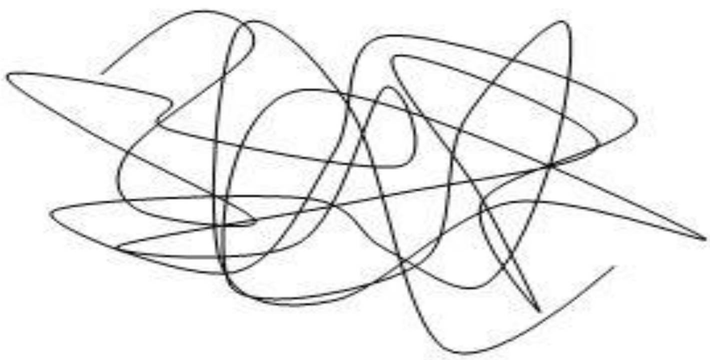


*Fig. 4. Amorphous structure [1]*

### 1.1.2 Semicrystalline thermoplastic polymers

Within semicrystalline plastic materials some percentage of the polymer chains is formed into densely packed crystalline regions. Application area for semicrystalline polymers is above the glass transition temperature. In this area semicrystalline polymers have good combination of strength and toughness. [1, 4]

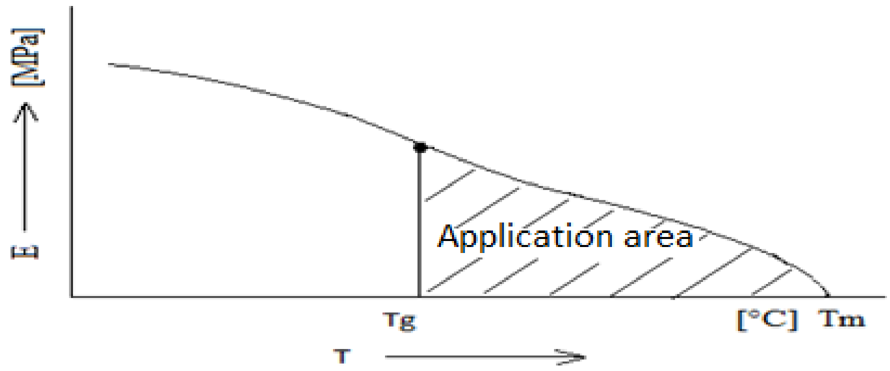



*Fig. 5. Application area – Semicrystalline thermoplastic polymers [4]*

The characteristics of semicrystalline materials include [1, 10]:

- Opaque to translucent. Some clarity may be gained though thinning of the cross section, very quick cooling and the use of nucleating agents to control crystal growth.
- Highly organize crystalline structure withinamorphous regions.
- Density is affected by cooling rate. Fast cooling will inhibit crystal growth and thereby reduce density.
- A more distinct melt temperature.
- High shrinkage during cooling due to compact nature of the crystal structure.

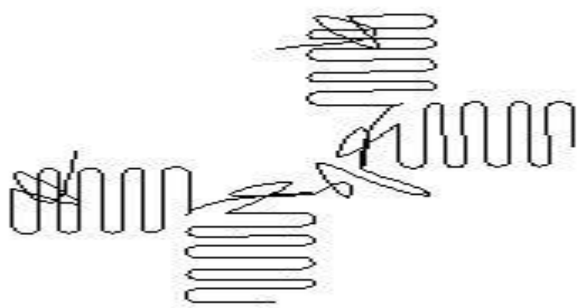


*Fig. 6. Semicrystalline structure [1]*

Some common semicrystallinethermoplastic polymers are Polyethylene (PE), Polypropylene (PP), Polyamide (PA6, PA66) or Polyetheretherketon (PEEK). [8]

## 1.2 Thermosets

Thermosets are crosslinked or network type polymers. A thermosetting material is characterized by the fact that it can be heated and formed into a product only once. During this process molecules crosslink and can be formed into a single molecule. Once "cured," the material cannot be remelted like a thermoplastic material. This can often give thermosetting materials an advantage in performance at elevated temperatures. The crosslinking process is regarded as a slow process. When considering a molding process, as thermoset polymers are forced into a mold cavity, they continue to flow, or creep, into small fissures until crosslinking is complete. Thermoplastics, on the other hand can freeze almost instantaneously (at least on the outer surface). In comparison, solidifying thermoplastic can take seconds, while crosslinking thermoset can take minutes. Thermoset polymers are difficult to recycle. Some common thermoset polymers are polyurethanes, synthetic or epoxies resins. [1, 8, 9, 10]

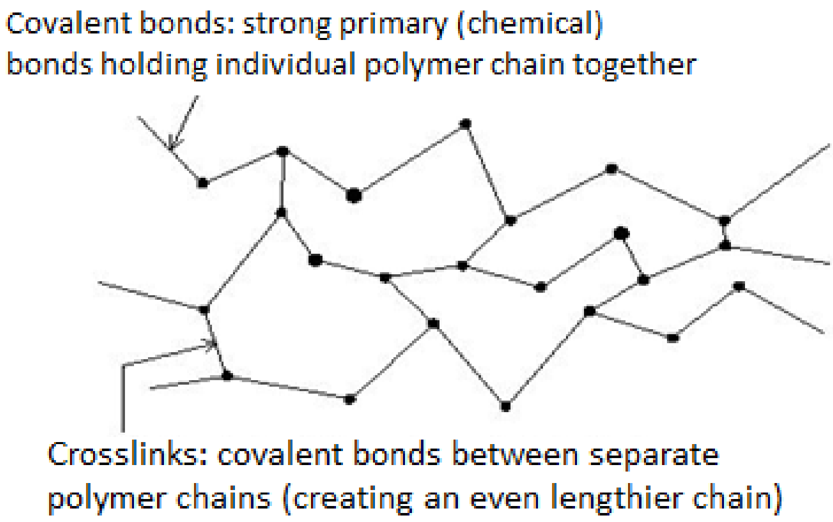



*Fig. 7. Common structure of a thermosetting material [8]*

## 1.3 Elastomers

Elastomers are composed of long, flexible, and more or less mixed-up macro-molecular chains that present only relatively low physical interactions. This low level of interactions between the chains governs the elasticity of the material. Elastomers are categorized into natural or synthetic classifications. Natural rubber is produced from the latex of the Heveabrasiliensis plant (rubber tree). Natural rubber is chemically known as cis-1,4 polyisoprene. Elastomers are elastic materials, they have the ability to deform substantially under the application of a force and then snap back to almost their original shape when the force is removed. [9, 10]

Compared to thermoplastics, elastomers present various specific characteristics [9]:

- At ambient temperature, their rigidity (or module) is low.
- They are deformable, which means they can sustain large reversible deformation without breaking.
- They are resilient, which means they are able to recover their initial geometry after repeateddeformations, by releasing quantitatively, on the outside, the energy that was previously supplied to deform it. This last characteristic highly depends on the elastomer nature, temperature, and the number of repeated deformations.

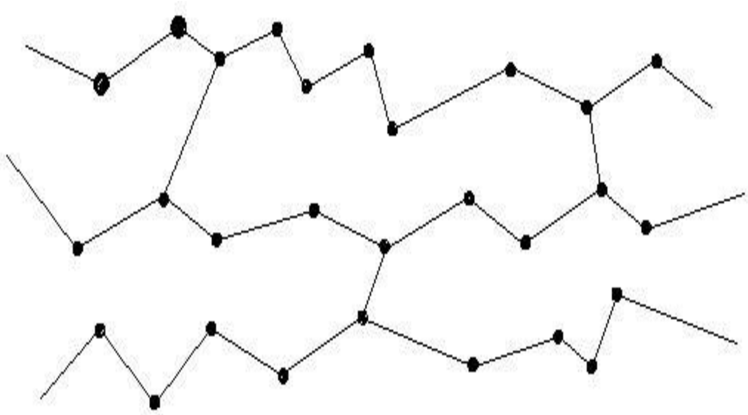


*Fig. 8. Common structure of elastomer materials [8]*

# 2 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. [3, 5]

Advantages [5, 17]:

- 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 [17]:

- High investment costs
- Long period needed for injection mold making
- Injection molding machine is disproportionately big in comparison with injected part

## 2.1 Injection molding

The molded parts are produced discontinuously in cycles. The overall injection molding cycle may be described as the total time required to produce one complete shot of one or more parts, depending on the number of cavities in a given mold. The injection molding cycle is not merely the time that the polymer melt remains in the mold. It includes the time necessary for the mold to close and clamp, any safety or delay time required at the start of the cycle, the injection time (time required to fill the cavity), the hold time, the time required to cool the molten material, mold opening, and the ejection time. The sum of these elements is known as the total overall cycle. [3, 5, 14]

### 2.1.1 Injection molding cycle

The hopper is fed with pellets which are delivered into the barrel. Contact between the pellets and the barrel transfers heat to the plastic. The mechanical action of crushing the pellets between the screw flights, the barrel, and each other adds more heat to the compound.

Up to 70 % of the needed heat is provided by shear heating during this crushing. The screw rotates and moves simultaneously backwards with the plastic part accumulating in the front of it. A nonreturn valve at the front of the screw prevents flow back along the flights. Then the screw pushes forward the plastic material, which flows through the nozzle, the sprue, and the runner system. After the gates are filled, the plastic flows into the cavity. The plastic material starts to cool as soon as it touches the cold surface of the mold. It cools down and at the ejection temperature the part is ejected from the mold. [1]

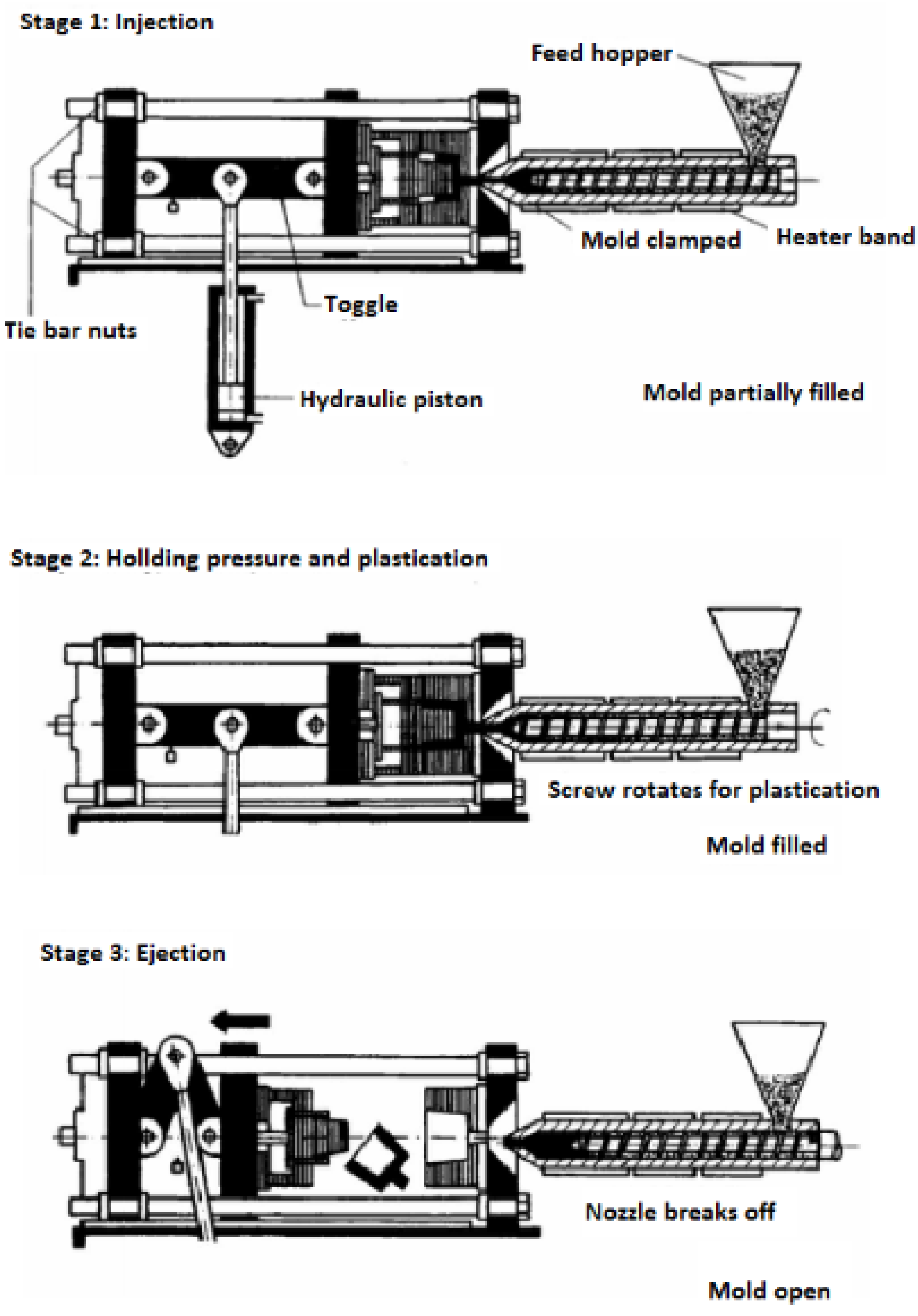



*Fig. 9. Injection molding cycle [5]*

# 3 PART DESIGN

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 shrinkage, part ejection, cooling, and mold filling. Probably the most troublesome issue related to successfuldevelopment 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. [1, 2, 3]

## 3.1 Guidelines for molded plastic parts

One of the primary objectives for the designer of injection molded plastic parts is to maintain a uniform wall thickness in the part. A uniform wall will minimize injection molding problems, particularly those related to shrinkage. [1, 7, 13]

### 3.1.1 Designing the primary wall

Injection molded plastic parts are generally designed around the use of relatively thin walls. Unless the injection molded plastic part is foamed or is produced with gas assist injection molding, its walls are normally less than 5 mm. When determining the thickness of the primary wall one must consider not only the structural, functional, and aesthetic issues related to the wall but also their impact on manufacturability. Manufacturability issues include consideration of the injection pressure required to fill the mold cavity, cooling time, and the influence on ejection from the mold. These will require consideration of available injection pressure and injection rates, mold rigidity, ejection techniques, and melt delivery means. [1]

Constant wall thickness

Maintaining a constant wall thickness should be the primary objective of the product designer. Each region in a part that has a different thickness will tend to shrink differently. These variations in shrinkage will not only complicate achieving the desired size of the molded part but are also major contributors to warpage and residual stresses. Variation in wall thickness also affect the mold filling and packing, or the compensation, phases of the molding cycle. Irregular, and difficult to predict, filling patterns can result. This can further complicate orientation-induced shrinkage as well as cause problems with venting, gas traps, and weld lines. In addition, a thick region fed through a relatively thin region can result in hesitation effects during mold filling which potentially can cause no fills. If variations in wall thickness cannot be avoided then try to keep the variation to a minimum and provide a gradual transition rather than a sudden change. [1, 12]


*Fig. 10. General guidelines in maintaining the constant thickness [1]*

### 3.1.2 Corners, fillets and radii

As most plastics are notch sensitive and therefore sharp corners should be avoided. This is particularly true with an inside corner which will act as a stress concentrator under load. Stress concentration will increase with the ratio of the corners fillet radius to the wall thickness. It is preferred that the inside radius be as large as possible. A larger radius not only can improve the part structurally but it will also reduce expected warpage developed from unbalanced cooling in corners. [1]

### 3.1.3 Draft angles and undercuts

Draft angles provide easier ejection from the mold. The range of the draft angles is to several degrees, depending on the material shrinkage, surface of the walls and other product design requirements. Undercuts in a part can significantly increase the difficulty of ejecting molded part from a mold and therefore should be avoided. [1, 3]

### 3.1.4 Ribs, gussets, and bosses

Ribs, gussets, and bosses are features that are added to the primary wall for structural, assembly, or other functional reasons. When adding these features to the primary wall, the plastics designer must again consider manufacturability. This includes consideration of part ejection, venting of air during mold filling, and effects on mold filling and packing. [1]

<u>Ribs</u>

Ribs are primarily used to increase the rigidity of a part or a specific region of the part. To achieve the desired rigidity of a given part an excessive thickness might be needed in the primary wall. This would negatively affect part cost. The addition of ribs can often attain the required rigidity while maintaining the more desirable thinner wall. The base thickness of the rib should be from 50 % to 75 % of the primary wall thickness depending on the material shrinkage characteristics. Generally a rib thickness of 50 % of the primary wall is recommended for high-shrink materials, and 75 % for low shrink materials. [1, 4]

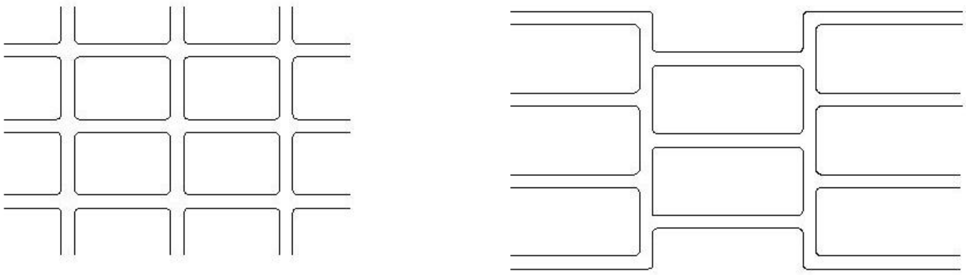


*Fig. 11. Different layouts of ribs [4]*

<u>Gussets</u>

Gussets are thin features, much like a rib, and are normally used to reinforce a local feature in a part. The reinforced feature could be a side wall, boss, or some other projection. Gussets are normally triangular in shape and should be designed using the same guidelines for thickness as for a rib. In addition, they should be no more than 4 times the primary wall thickness in height and 2 times the primary wall thickness in width. [1]

<u>Bosses</u>

Bosses are normally either solid or hollow round features that project off the primary wall. They can be used for assembly with self-tapping screws, expansion inserts, force-fit plugs, and so forth. Bosses are often supported with ribs or gussets. A boss should not be attached directly to a side wall, as the intersection with the wall will result in a thick region, causing sink or void. [1]

### 3.1.5 Marks and signs

Preferred design of marks and signs on the injected part is gained by combination of countersunk and extruded area of the sign. Countersunk marks are difficult to produce and extruded marks do not fit the purpose. [1, 4]

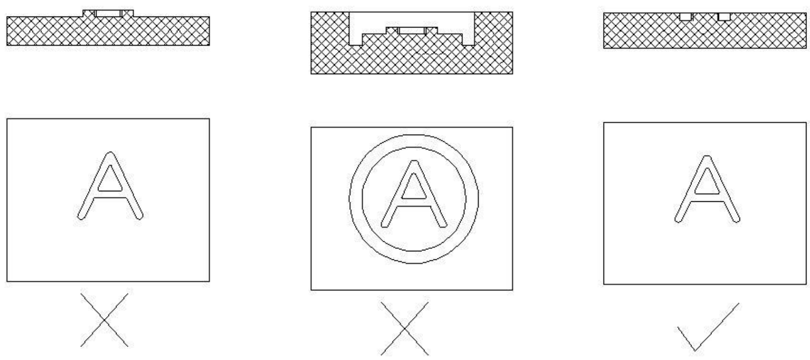


*Fig. 12. Different ways of marks designing [4]*

### 3.1.6 Threads

Threads on plastic parts are characterized with lower strength and with soft threads causing problems with forming the part. Threads should be rounded or trapezoidal with higher pitch. These types of threads are firmer and are easy to manufacture. Especially low costs can be gained by designing dashed threads. [4]

# 4 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 designed and built to produce a given part. [1, 12]

Injection molds must satisfy the following basic requirements [1, 15]:

- 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 machine 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.

## 4.1 Runner systems

Injection mold can be classified in several ways and one of them is also by the type of runner system. According to the type of runner system injection molds are divided as [2, 3]:

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

## 4.2 Cold Runner Molds

For thermoplastic materials, a cold runner mold refers to a mold in which the runner is cooled, solidified, and ejected with the molded part(s) during each molding cycle. Approximately 70 % of molds today are cold runner molds. [3]

The runner system in a cold runner mold normally consists of a sprue, runner and at least one gate. In a single-cavity mold, the cavity is generally placed in the center of the mold, and the sprue delivers the melt directly to the center of the cavity. In a multi-cavity mold, sprue delivers the melt to a runner, which in turn delivers the melt to the part-forming cavities. [3]

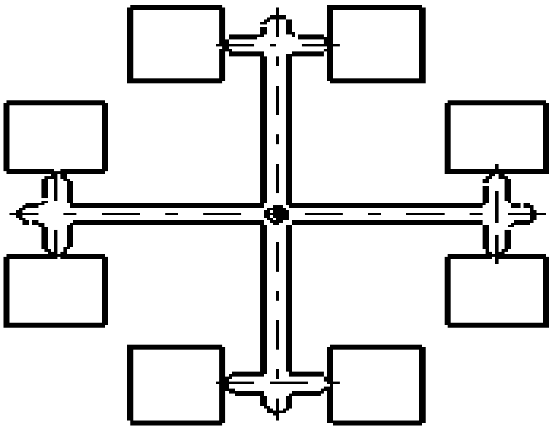


*Fig. 13. Example of cold runner system [4]*

The cold runner system mold is by far the most basic and most common type of mold. It is simpler, less expensive to construct, and easier to operate and maintain than a hot runner mold. [3]

Advantages of cold runner systems

Cold runner systems have several advantages over hot runner systems. Because of their simplicity, they are cheaper to build. In addition, the cost to maintain a cold runner system is less because there are no heaters, heater controllers, thermocouples, and other hot runner components to maintain. Operation is also much simpler as there is no need to tend to the various heater controllers or deal with the many potential problems such as gate drool, clogging, leaking, material degradation or problems in the runner manifold, and so forth. [3]

Disadvantages of cold runner systems

A major disadvantage of the cold runner system is the fact that the unwanted frozen runner must be dealt with. This requires the need to separate the runner from the molded parts and then sell or grind the runner for reuse. This step of regrinding introduces additional potential for material contamination and the need for granulators and their maintenance. The reground material can often be fed back into the process, at a controlled ratio, with the virgin material. As the reprocessed material will differ somewhat from the virgin material, it can be expected that the method of reintroducing regrind can alter the molding process and the properties of the molded part. [3, 1]

Cold sprue

A cold sprue is normally formed inside of a sprue bushing. It is designed to be easily replaced and is normally purchased from a supplier of standardized mold components. The replaceable sprue bushing provides a number of different attributes. Structurally it must withstand repeated impacts from the injection nozzle whenever it engages the sprue bushing. The interface between the sprue bushing and the nozzle must not be deformed as leaking or ejection problems might occur. [3]

The sprue also experiences high melt flow rates and melt pressures. To withstand these structural challenges, the sprue bushing is commonly made from hardened steel. The sprue must also provide for its ejection from the mold.  For this purpose, the flow channel of the sprue is normally tapered and polished in the direction of draw. [3]

The cold runner

The cold runner is one of the most influential components of a successful molding, but is often grossly misunderstood and its impact underestimated. The ideal runner has a full round cross sectional shape which provides the optimum ratio of the perimeter of the runner geometry to cross-sectional area of the runner. However, the full round has the disadvantage of requiring the two halves of the runner to be machined. These two half runners must then closely match up to form the full round runner when the mold is closed. Due to the added cost of the full round runner and the potential for misalignment, alternatives to full round runners are often used.  Alternative runner shapes include: trapezoidal, parabolic and half round. These alternatives are often much easier to machine because there is no concern of matching up runner halves as with the full round runner. [3]

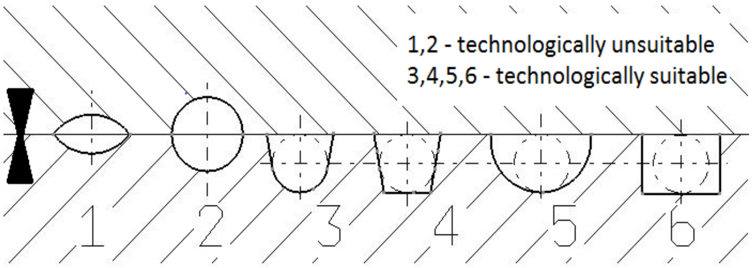



*Fig. 14. Cross sections of a runner [4]*

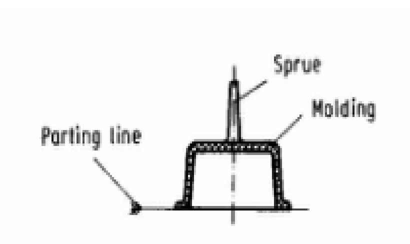
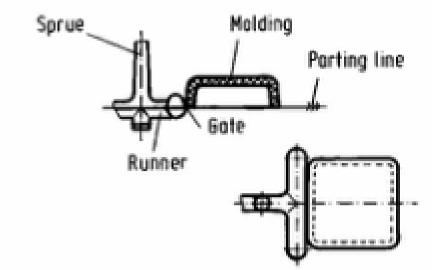
Gate designs

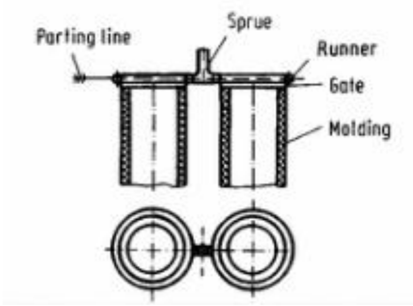
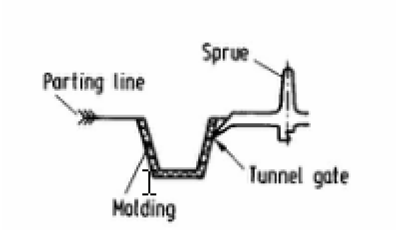
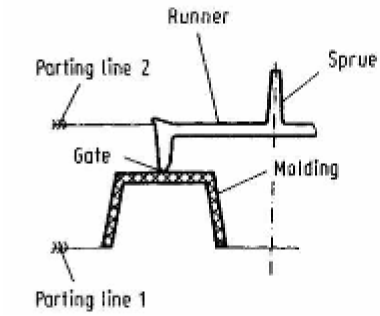
The gate is the link between the part and the runner system. It is normally a restricted area that facilitates separation of the runner from the part. The size, shape, and placement of the gate can significantly affect the ability to successfully mold a product. The key figure of the gate is to allow easy, potentially automatic, separation of the part from the runner system, while allowing filling and packing of the part. [3]

Gate types

Some of the common gate types are listed in table 1 below.

*Tab. 1. Gate types [3, 5]*

| Type of gate | | Characteristics |
|---|---|---|
| Sprue (gate) |  | Applications: It is used for temperature-sensitive and high-viscous materials, high-quality parts and those with heavy sections.<br>Pros: Results in high qualityand exact dimensions.<br>Cons: Post operations for sprue removal, visible gate mark. |
| Edge gate |  | Application: For parts with large areas such as plates and strips<br>Pros: No weld lines, high quality, exact dimensions.<br>Cons: Post operations for gate removal. |

| | | |
|---|---|---|
| Ring gate |  | Application: For sleeve shape<br><br>Pros: Uniform wall thickness around circumference<br><br>Cons: Slight weld line, post operations for gate removal. |
| Tunnel gate |  | Application: Primarily for smaller parts in multi-cavity molds and for elastic materials<br><br>Pros: Automatic gate removal.<br><br>Cons: For simple parts only, because of high pressure loss. |
| Pinpoint gate (three-plate mold) |  | Application: For multi-cavity molds and center gating.<br><br>Pros: Automatic gate removal.<br><br>Cons: Large volume of scrap, higher mold costs. |

## 4.3 Hot runner molds

During the past several decades, the use of hot runners has increased to approximately 30 % of all new molds. Most hot runners are purchased from a company that specializes in their design and manufacture (DME, Hasco, Mold Masters and others). The hot runner is then assembled into a mold. [3]

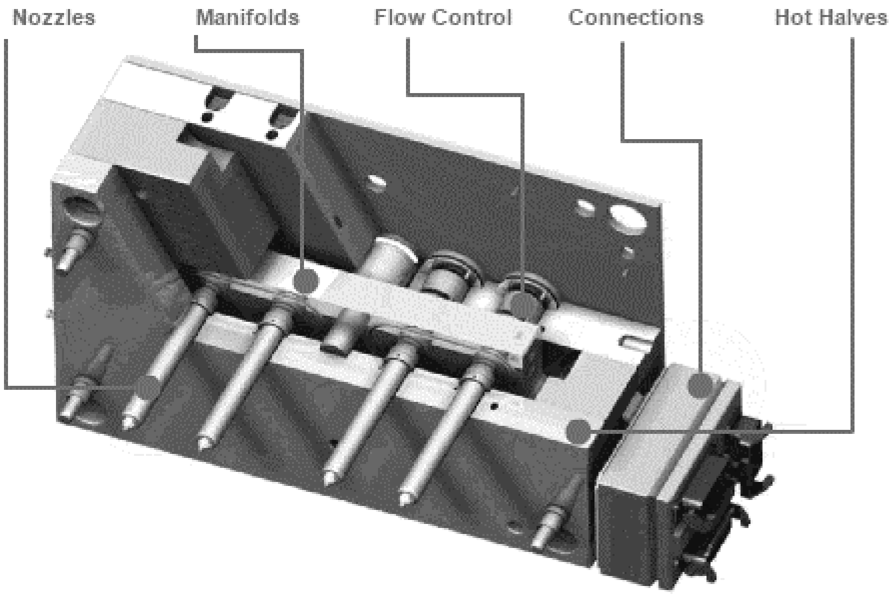



*Fig. 15. Hot runner system [31]*

Advantages and disadvantages of hot runners [3]:

- Eliminates the need to deal with the left over cold runner
- Faster cycle time
- Reduced clamp stroke vs. three-plate
- Reduced energy for plastification, filling pressure and granulators
- Improved automation
- Cleaner work environment
- Use of stack molds

Disadvantages of hot runners [3, 4]:

- Complex injection mold design
- More energy-expensive than cold runner
- The need of temperature sensors and regulators

### 4.3.1 Hot runner manifoldand drops

There are several combinations of manifolds and drops used to create the various hot runner systems [3]:

- Externally heated manifold and drops
- Externally heated manifold with internally heated drops
- Internally heated manifold and internally heated drops

- Insulated manifold and drops

Hot manifolds are set between the clamping plate and the cavity plate in the stationary half of the mold. Manifolds must be thermally insulated from the other parts of the mold, this is usually done with an air gap. They are made from steel in various different shapes e.g. I, H, X, Y act. [3]

*Fig. 16. Commonly used manifold types [28]*

Externally Heated Manifold and Drops

Externally heated systems have the ability to provide the lowest pressure drop of all molds, except for insulated hot runners. The flaw channels are cylindrical in cross section and generally have larger diameter than a cold runner mold. Both the larger diameter and the fact that there is no growing frozen layer in the runner system contribute to the relatively low pressure drop of these types of molds. Disadvantages of an externally heated runner system include the potential for leaking of molten plastic and the amount and location of the required heat from the heaters. Improper design or operation can result in plastic leaking between the drop and the manifold. [1, 3]

### 4.3.2 Hot drops (nozzles)

Hot drops are the part of the hot runner system, which delivers the melt from the hot manifold to the part-forming cavity. There are a number of basic requirements they must meet [3]:

- Conduct heat to the gate (prevent gate from freezing).
- Provide thermal separation between the hot drop and cold part-forming cavity.
- Provide clean separation of the melt and the frozen part.
- Minimize flow restrictions or areas of material hang-up.
- Provide good temperature control of the melt.

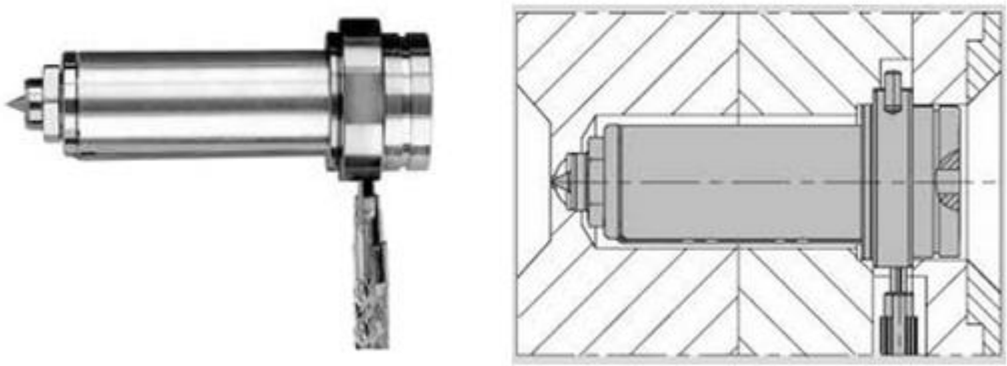


*Fig. 17. Hot nozzle Techni Shot from Hasco [28]*

## 4.4 Mold ejection systems

Ejection of plastic parts from a mold can present a significant challenge. It is critical when designing a plastic part, which is to be injection molded, that the part is designed considering how it is to be ejected from the mold. In other words, the part´s design is strongly influenced by this requirement. Basic considerations include location of the parting line of the mold relative to the part during mold opening, tapering vertical surfaces that must be drawn from a core or cavity, and undercuts that will prevent the part from being ejected. [1, 2]

After plastic is injected into the cavity, it begins to cool and shrink. This shrinking plastic develops a significant pressure on the mold´s core. The part must be pushed off the core by some means. If the pressure is excessive, the force required to eject the part may result in damaging it. Therefore, it is important to minimize the negative effect of the pressure created by the shrinking plastic. This is done by tapering the inside walls of the part. [1, 12]

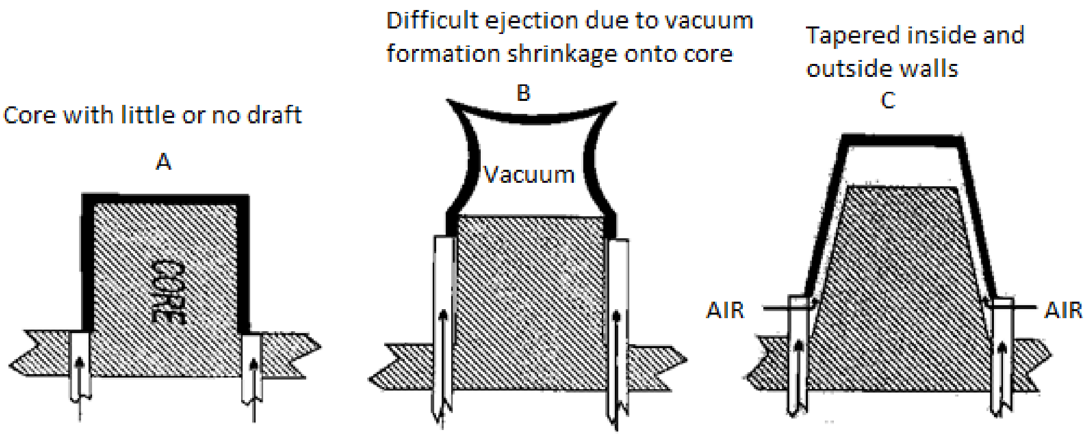



*Fig. 18. Basic ejection problem – Five sided box [1, 12]*

### 4.4.1 Means of ejection

Pins

Pin ejection is the most common and least expensive mean of ejection. Use of pins sometimes results in excessive stress because of their small contact area with the part. This mean of ejection is simple and functionally guaranteed. However, after ejection pins remain footprint on the product. Therefore it is not suitable to use pins on the appearance surfaces of the injected part. [1, 4]

Ejector sleeves

Ejector sleeves are used for small parts with cylindrical cores or for round, internal features on a part such as a hollow boss. Although this type of ejection is more expensive, it provides an excellent positive force on the part. Most common ejector sleeves are limited to small cylindrical parts with less than 50 mm diameter. This mean of ejection doesn´t leave footprint on the surface. [1, 4]

Stripper plates

Stripper plates provide the same positive features of ejector sleeves but can easily accommodate larger parts. Stripper plates are used in a three-plate cold runner mold. Unlike an ejector sleeve, stripper plates are not limited to cylindrical shapes. The plate normally includes stripper inserts that contact the core and the plastic part. Unlike the ejector sleeve, the stripper plate cannot be used to eject internal features on a part like a boss. [1, 4]

Stripper rings

Stripper ring ejection is similar to stripper plate ejection expect for the shape of the stripper. Instead of a bulky plate, the ring borders the edges of the part only. This feature is usually used with a large cylindrical part in a single-cavity mold. [1, 4]

Blade ejection

Blade ejectors are often applied to straight edges of parts where an increase in contact area is necessary and cannot be achieved by pins. The blade can be a flat section machined from a round pin. [1, 4]

Air ejection

Air ejection is yet another mean of ejection that can be used alone or in combination with any of the above mechanical methods. This mean of ejection finds use in ejection process of thin-wall products of larger sizes. [1, 4]

## 4.5 Mold cooling

Injection mold acts as a heat exchanger. The aim of cooling system is to achieve optimal short injection cycle while maintaining all technological production requirements. Coolant efficiency is significantly affected by the coolant´s flow rate through the cooling channels. Turbulent flow is much more efficient in extracting heat as compared to laminar flow. A Reynolds number of at least 10 000 is preferred to ensure efficient cooling. [1, 6]

### 4.5.1 Cooling line networks

Cooling circuits within a mold can be laid out as series or parallel, and as combination of these two. Each type has advantages and disadvantages.[1]

In a parallel circuit, water is fed to multiple parallel branches from a single source or manifold. The primary advantage of a parallel circuit is that because its layout results in shorter flow lengths, it has less chance of exceeding the pumps pressure limits. Other benefit of parallel circuit in comparison with series circuit is that the channels are easier to clean. [1, 24]

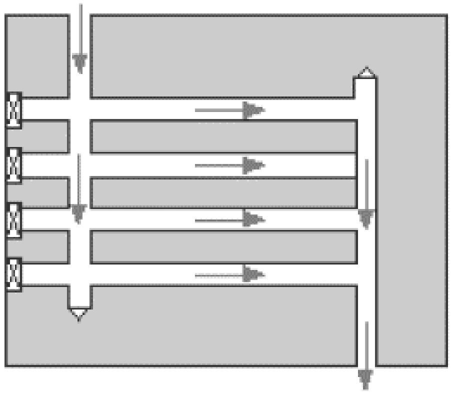


*Fig. 19. Parallel circuit [24]*

In a series circuit there is a single inlet and a single outlet with no branching. This results in a relatively long flow path for the water. Advantages of this type are that the flow rate is more assured to be constant along the entire length of the circuit, as it is not divided into various branches that could have flow imbalances. If a circuit were clog, the clog would be obvious as there would be no water flow. The two major concerns with regard to the series circuit are rise of temperature and pressure drop. [1, 24]

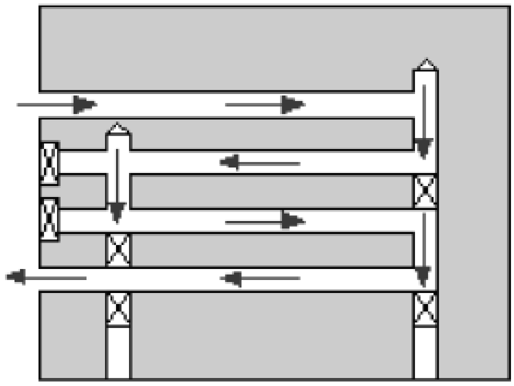


*Fig. 20. Series circuit [24]*

### 4.5.2 Baffles and bubblers

Baffles and bubblers are generally used to cool inside restrictive cores. Often multiple baffles or bubbles are used to cool the inside of larger cores or to get closer to corners in box-shaped parts. [1, 24]

A baffle is actually a cooling channel drilled perpendicular to a main cooling line, with a blade that separates one cooling passage into two semi-circular channels. The coolant flows in one side of the blade from the main cooling line, turns around the tip to the other side of the baffle, and then flows back to the main cooling line. A bubbler requires a feed line and a separate return line. The lower feed line feeds water up through a tube to the top of the bubbler hole. The water emerges from the tube and returns down the outside of the tube to the return line. [1, 24]

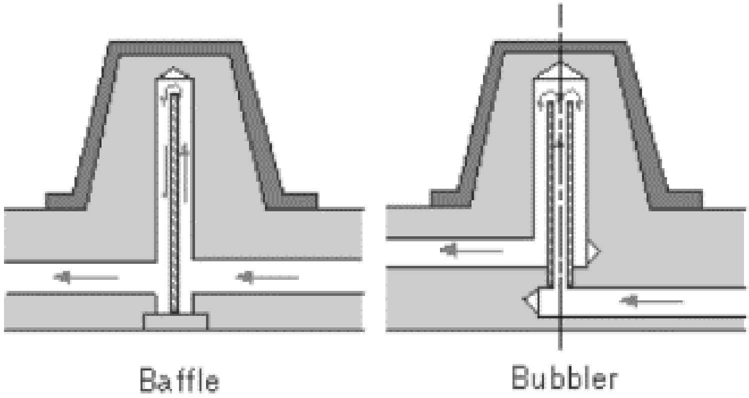



*Fig. 21. Baffle and bubbler [24]*

#### 4.5.3 Coolants

Coolants may include water, water glycol solutions of various ratios, and oil. The coolant with the best thermal conductivity should be matched to the required temperature. Water is the best medium for heat extraction but is limited to temperatures between freezing and boiling. For personal safety reasons and to avoid damaging equipment, water is not normally used temperatures below 5 °C and above 90 °C. Below 5 °C, glycol solutions are normally used. Oil is most commonly used for temperatures above 90 °C. [1, 2, 4]

*Tab. 2. Coolant types [4]*

| TYPE | ADVANTAGES | DISADVANTAGES | NOTE |
|---|---|---|---|
| water | High heat transfer, low viscosity, low price, ecologically suitable | Usage under 90 °C *, corrosion development **, scale settling | *) Withstands higher temperature in pressure circuits **) problem can be solved with additives |
| oil | Usage possible over 100 °C | Worse heat transfer | |
| glycols | Restriction of corrosion and system plugging | Environment pollution | |

## 4.6 Venting

The theory of venting is simple: The air inside the cavity space must be allowed to escape so that the inrushing plastic can fill the whole space. Without venting the air inside of the mold have no space to escape. As the air is compressed, its heat content is now concentrated in a small volume, resulting in a large temperature increase. The temperature can reach several hundred degrees Celsius and cause the leading edge of the inrushing plastic to burn. There are several ways of venting air or gas out of the mold and it can be done by [2]:

- parting line venting
- venting of channels and grooves
- vent pins
- insert venting
- venting of ribs
- venting the bottom of a cavity

In old mold building methods, these poorly filled areas and burned edges of the product were used to indicate where the cavity needed venting, after the mold was tried out. In a well-engineered mold, however, this need should be recognized before the mold is built, and adequate venting must be designed into it from the start. Even though the experienced designer anticipates most areas where air could be trapped, some venting may still have to be added after testing. [2]

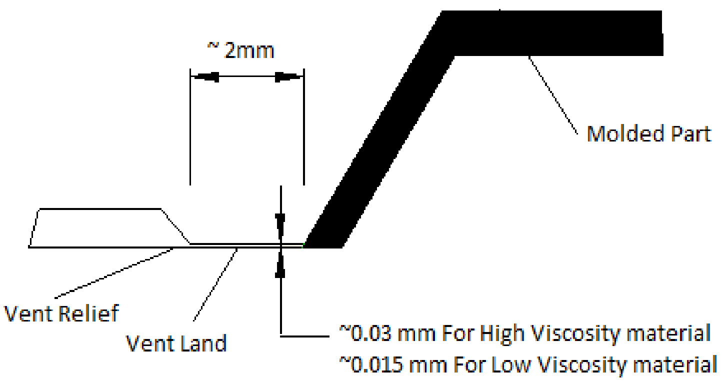



*Fig. 22. Common vent design [1]*

# 5 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. Pros and cons of this technology are 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.

# II. ANALYSIS

# 6 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 3D 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

# 7 USED SOFTWARE

## 7.1 CATIA V5R19

CATIA is software developed by French company DassaultSystemes. CATIA V5 is a system 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. [27]

## 7.2 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. [25, 30]

## 7.3 Hasco Dako Modul

Hasco Dako Modul is software from German standard part producer company Hasco GmbH in cooperation with company Dako. The software provides an electronic catalog of products that are offeredby Hasco. The application supports the selection of the product, followed by export of its geometry. Format of the geometry is compatible with the leading design engineering programs like CATIA, SolidWorks, Autodesk Inventor, etc. [28]

# 8 INJECTED PART

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

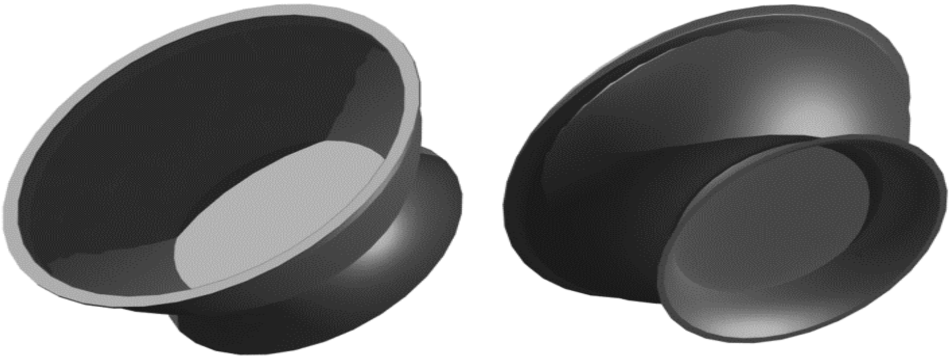


*Fig. 23. Render 3D model of the cup*

## 8.1 Injected part specification

The cup consists of two areas. With upper area for the dessert and bottom area for cup stability, cup has the total height of 47 mm and a lid diameter of 112 mm. Constant wall thickness is 1 mm and the lid thickness is 0,8 mm. The design of the bottom area makes the cup different in comparison with other cups. However, because of that the cup must be processed by injection molding technology instead of thermoforming technology. In order to eject parts properly the mold has to be designed with slide mechanisms.

## 8.2 Material

Chosen material for the injected part is polypropylene (PP). PP is a semicrystaline thermoplastic 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. [2, 4]

Chosen PP granulate is from manufactureSabic 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 are in housewares and thin walled packaging. Data sheet of the material is enrolled as an appendix at the end of the thesis. [29]

<u>Material characterization [29, 32]:</u>

- 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 100 000 [1/s]
- Maximum shear stress 0,25 [MPa]
- Fillers unfilled

Recommended processing [29, 32]:

- Melt temperature 200 – 250 [°C]
- Mold surface temperature 10 – 50 [°C]
- Absolute maximum melt temperature 290 [°C]
- Ejection tempertature 107 [°C]

# 9 INJECTION MOLDING MACHINE

Injection molding machine was chosen in compliance with the mold size, clamping force and calculated stroke volume. Selected injection machine is ArburgAllrounder 720H from Arburg. [19]

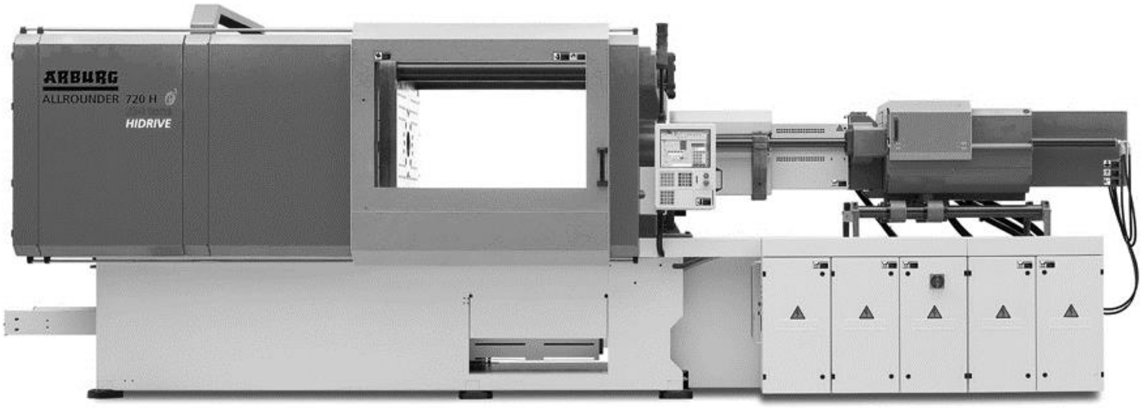



*Fig. 24.ArburgAllrounder 720 H [19]*

Technical characterization [19]:

| | |
|---|---|
| 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] |

# 10 INJECTION MOLD DESIGN

Injection mold design is supported with two designs for the given part. First variant of injection mold is designed with hot runner system. The second variant is designed with cold runner system. Both molds were designed in software CATIA V5with an effort to maximize the usage of standard parts from Hasco DakoModul.

## 10.1 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 mold was designed in order to accomplish thesis´s assignment.

### 10.1.1 Part forming

Because of the bottom area of the designed cup, it has to be formed in two parting planes perpendicular to each other. Main parting plane opens to Z direction and individual parts are formed with core and cavity in this direction. Secondary parting plane is formed by a pair of sliders. Because of material´s shrinkage, individual forming components were enlarged for provided shrinkage values from material´s data sheet.

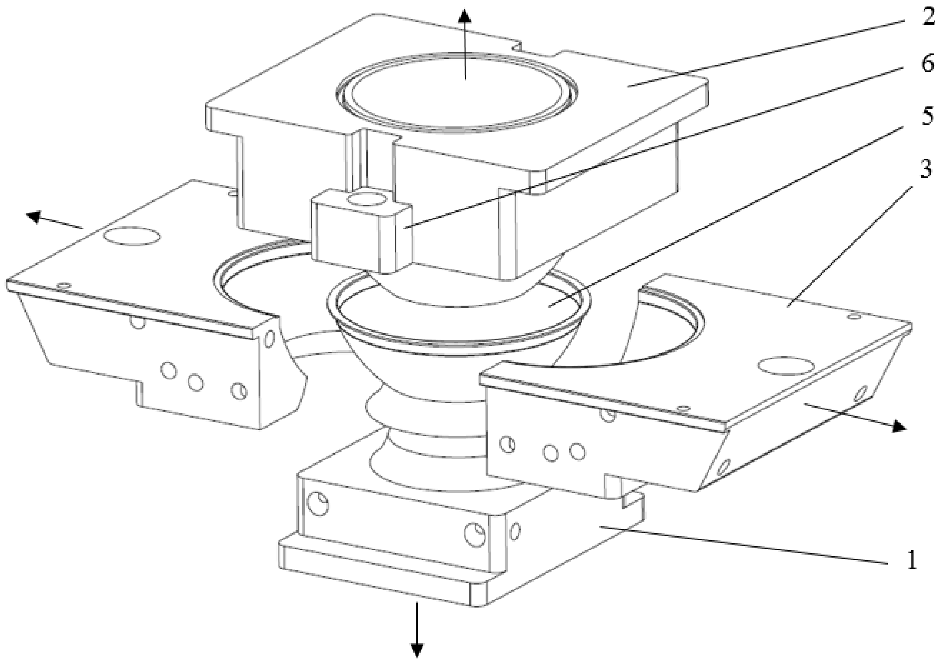



*Fig. 25. Part-forming elements*

1 – Cavity, 2 – Core, 3 – Slider, 4 – Stripper ring, 5 – Cup

When mold opens sliders are forced to spread, because of the connection with the guide angle pillars. They move on the guide rails, which are screwed to the core plate. Precision movement of the sliders is assured by the guide strips.

Movement of the sliders stops when they reach the end of the guide angle pillars. At this point location of sliders is secured against further movement with couple of spring plungers for the each slider. When mold closes, holding force of spring plunders is overcome and the connection with guide angle pillars is reunited.

During the injection process a big pressures are generated and therefore sliders must be locked in the right position. They lean on the wear plate, which is mounted to locking heel.

Length of the guide angle pillars and positions of the spring plungers were calculated to gain sufficient space for a proper ejection of the injected parts. The guide angle pillars are clamped in the cavity plate with 24° angle.

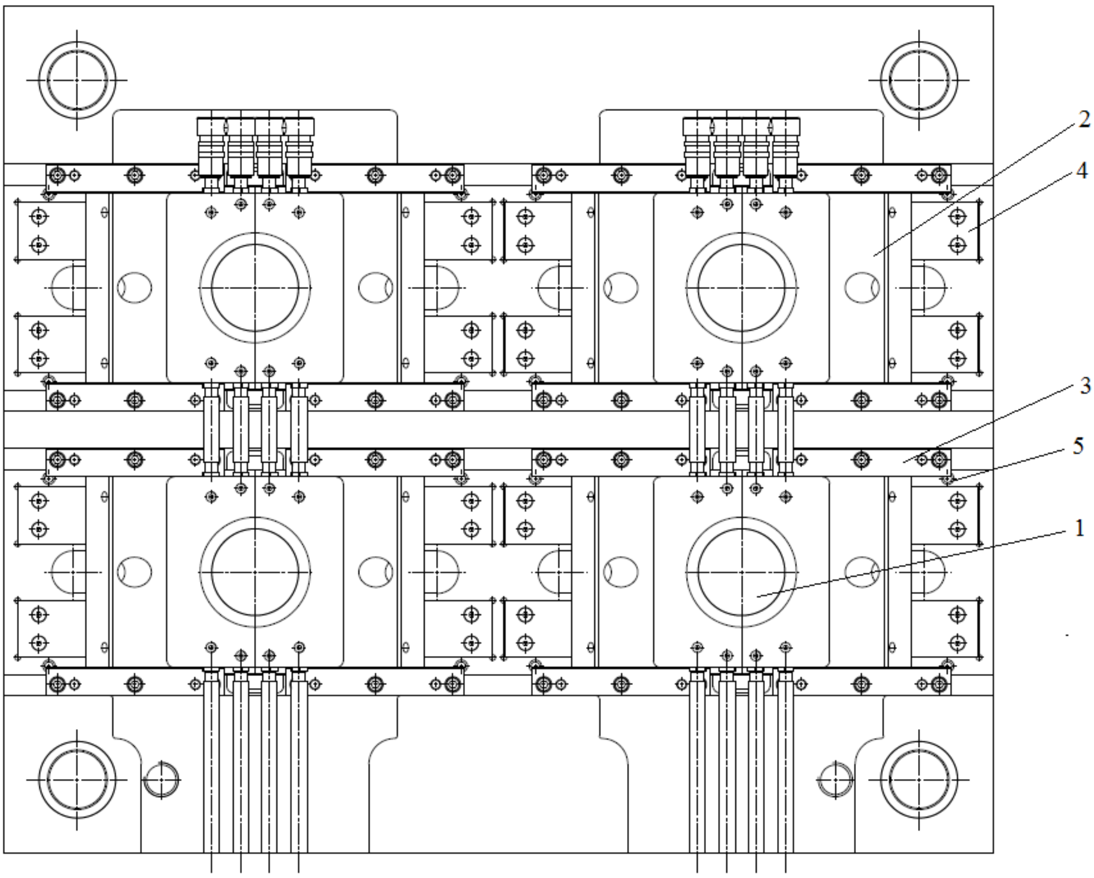



*Fig. 26. Core plate*

1 – Core, 2 – Slider, 3 – Guide strip (Z185W), 4 – Guide rail (Z186W), 5 – Spring plunger (Z374)

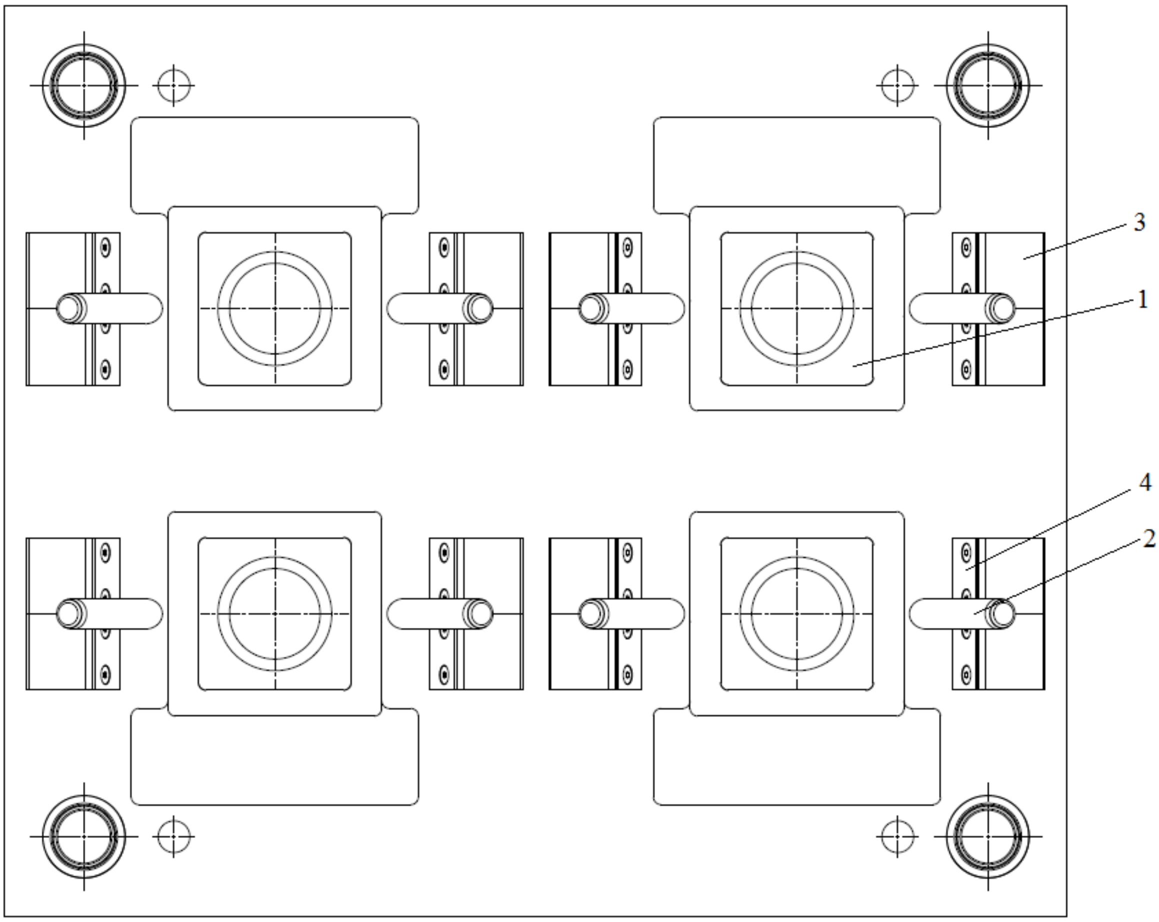



*Fig. 27. Cavity plate*

1 – Cavity, 2 – Guide angle pillar, 3 – locking heel, 4 – wear plate

### 10.1.2 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.

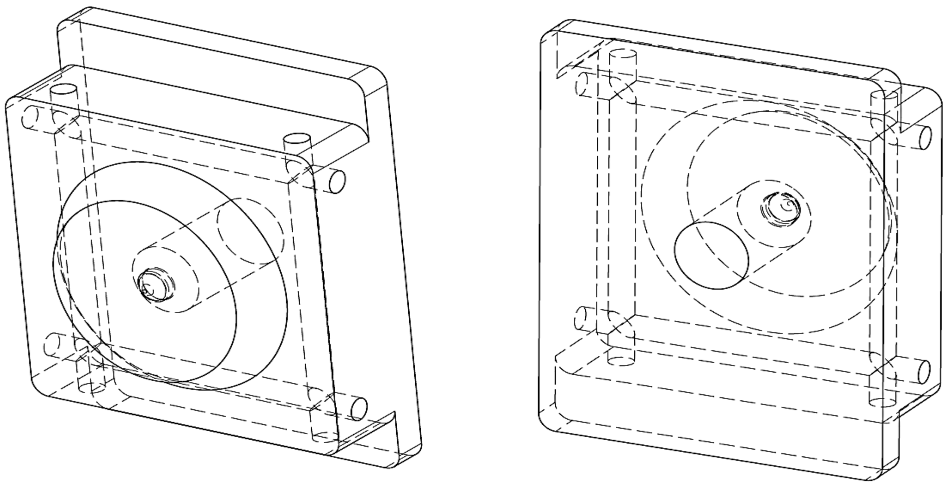


*Fig. 28. Cavity 3D model*

### 10.1.3 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 a 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.

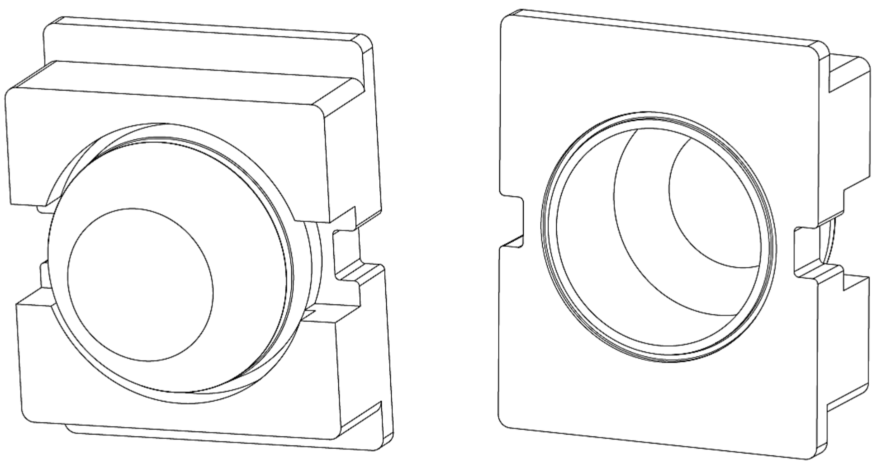


*Fig. 29. Core 3D model*

### 10.1.4 Sliders

Sliders constitute the secondary parting plane and form the perimeter of the part.Material for thesliders is the same as for the cores and cavities. For cooling purposes, series of cooling channels were designed.

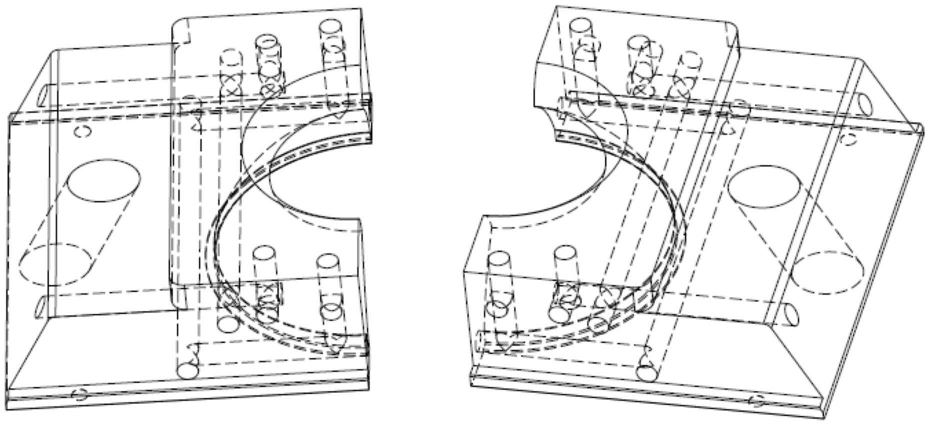


*Fig. 30. Sliders*

## 10.2 Hot runner injection mold

Hot runner injection mold for four cavities was designed with effort to maximize the usage of standard parts in order to reduce the cost of mold and simplify designing in the 3D software.

### 10.2.1 Mold frame

Mold plates and other standard parts of the mold frame were chosen from the Hascocatalog. 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).

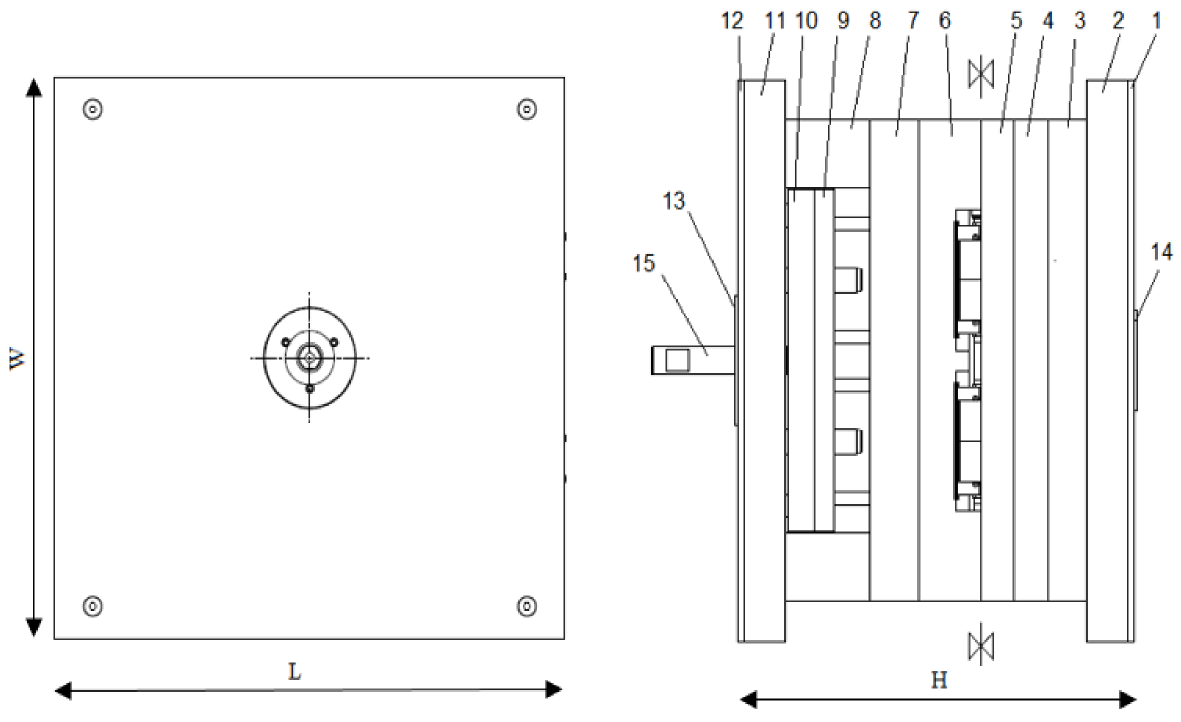



*Fig. 31. Injection mold frame*

1 – Insulating p. right, 2 –Clamping p., 3 – Additional p., 4 – Cavity support p., 5 – Cavity p., 6 – Core p., 7 – Core support p., 8 – Riser bar, 9 – Ejector p. A, 10 – Ejector p. B, 11 – Setting p., 12 – Insulating p. left, 13 – Locating ring left, 14 – Locating ring right, 15 – Knockout

<u>Guiding and connecting elements</u>

Guiding and connecting elements for this injection mold were taken from Hasco standard part catalog. Into this group of elements belong leader pins, bushings, screws, sleeves and center rings. Injection mold is clamped to the injection machine for clamping and setting plates.

Centering of both sides of the injection mold on injection molding machine is done by locating rings. Both locating rings were selected from Hasco catalog, K100 on the right side and K500 on the left side. Mutual location of individual plates and their centering is assured with leader pins, bushings and sleeves.

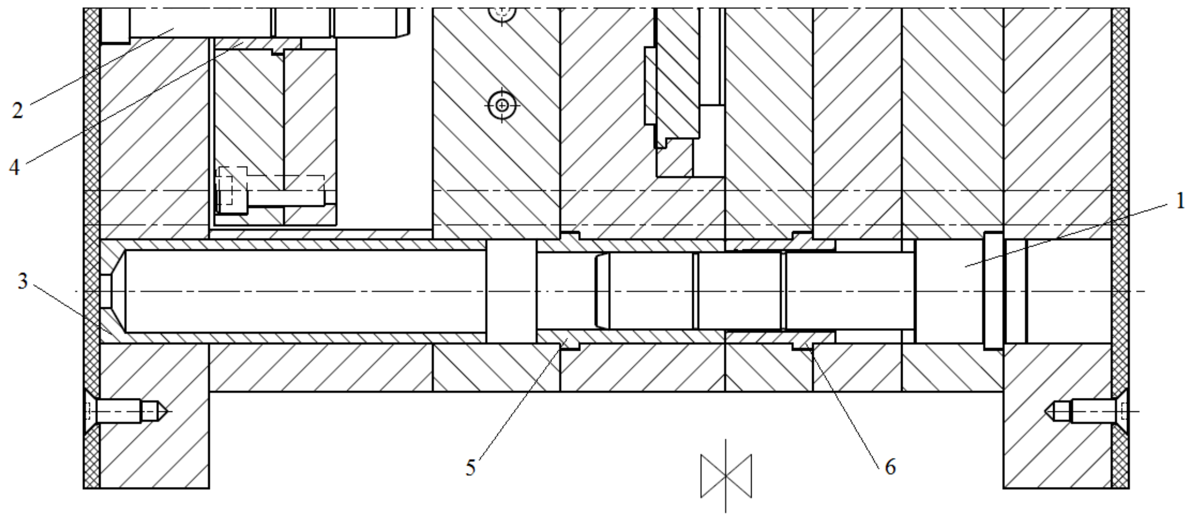



*Fig. 32. Injection mold guiding elements*

1,2 – Leader pins; 3 – Sleeve; 4,5,6 – Bushings

### 10.2.2 Hot runner system

Hot runner system keeps the injected material in melted phase during the whole injection cycle, which reflects into shorter injection times. Another advantage of hot runner system is that no vast material is being produced. This kind of material has to be crushed and grinded. This is often considered as it leads to big money savings, especially in large series.

<u>Hot sprue bushing</u>

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

<u>Hot manifold</u>

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 steele 1.2343 with ČSN equivalent 19 552. Dimensions of selected hot manifold are 280 x 406 x 37 mm (L x W x H).

<u>Hot nozzles</u>

Hot nozzles for designed injection mold were chosen from Hascostandard 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.

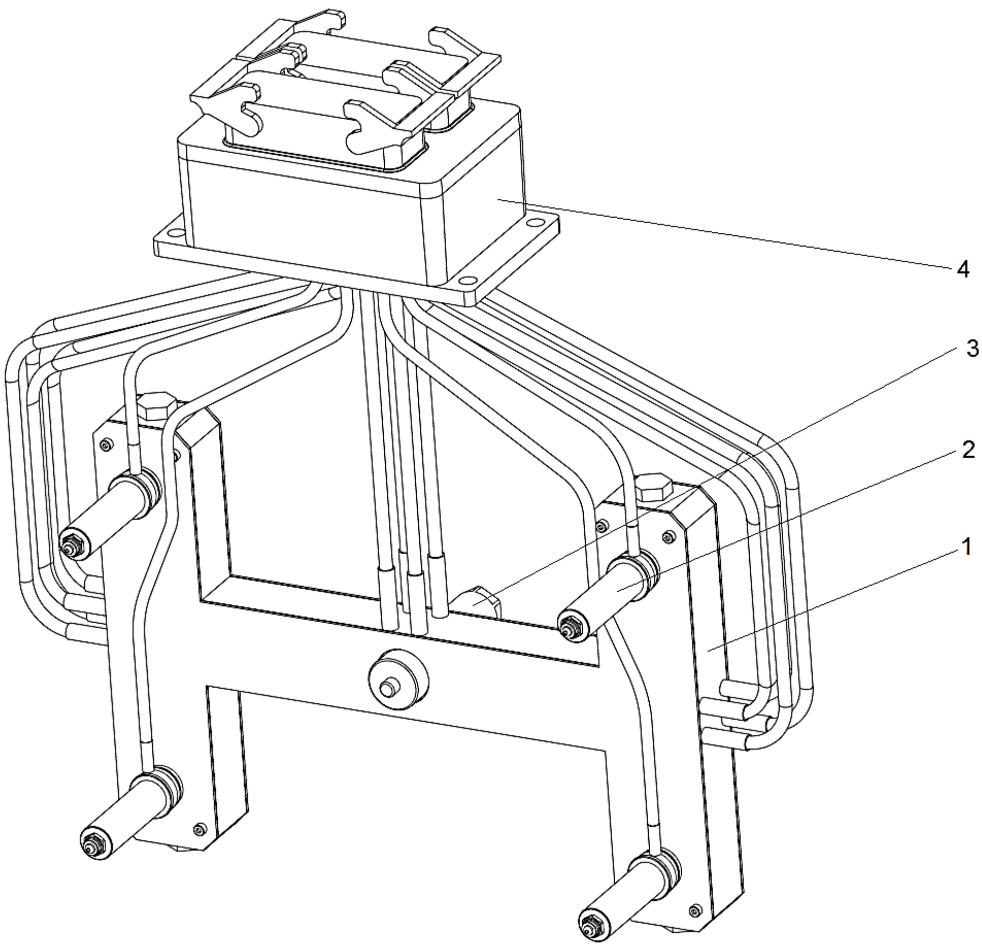



*Fig. 33. Hot runner system*

1 – Hot manifold, 2 – hot nozzle, 3 – hot sprue bushing, 4 – junction box

### 10.2.3 Mold cooling

Because of mold 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.

<u>Cavity cooling</u>

Cavities are cooled by cooling line network of drilled cooling channels. The network of cooling channels forms a parallel circuit. There are two parallel circuits for cavity cooling purposes, with each of them providing cooling for two cavities. The diameters of the distributive channels are 14 mm and branched channels have diameter 10 mm. Coolant is regulated in the line network with plugs (Z940). Hose connectors (Z87) are located in the bottom side of the mold. To prevent leakage of coolant into the space between surfaces of individual plates, cooling system is equipped with sealing O-rings.

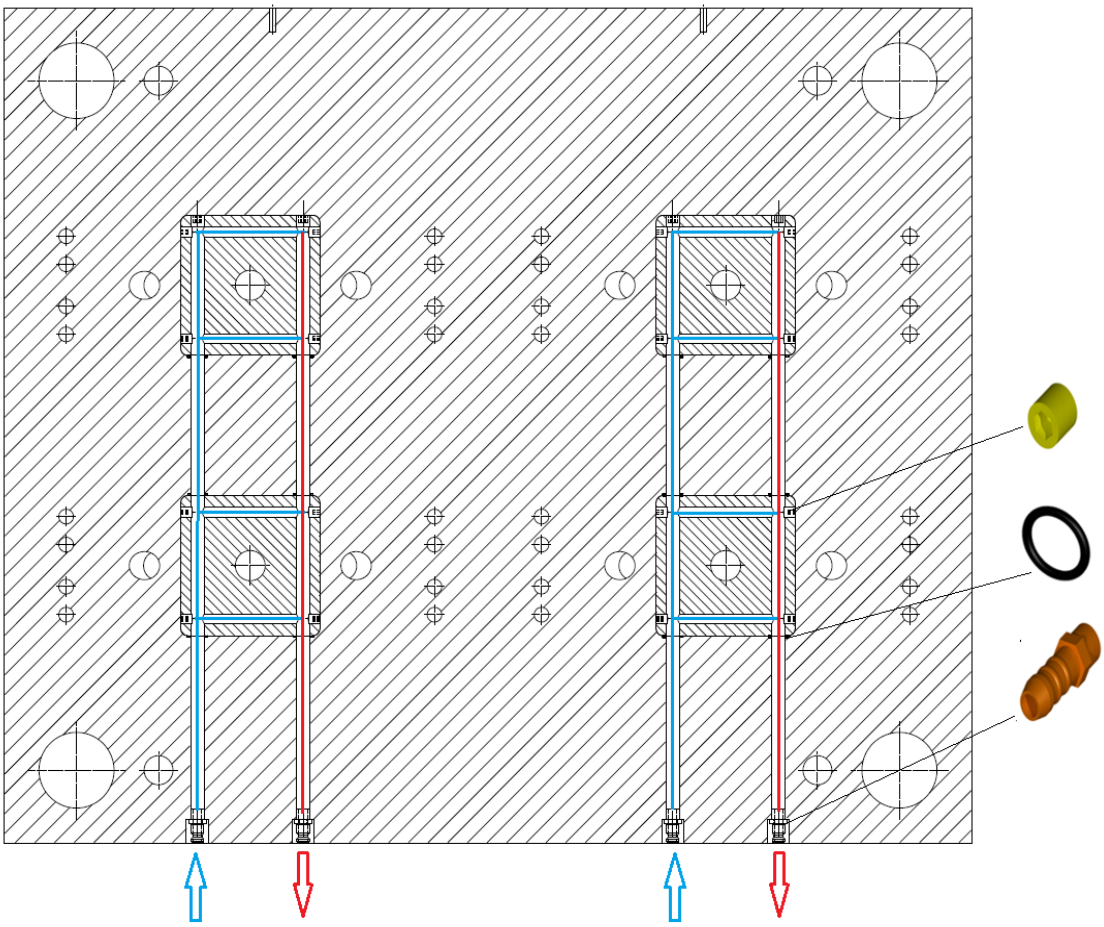


*Fig. 34. Cavity cooling circuits*

<u>Core cooling</u>

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. [24]

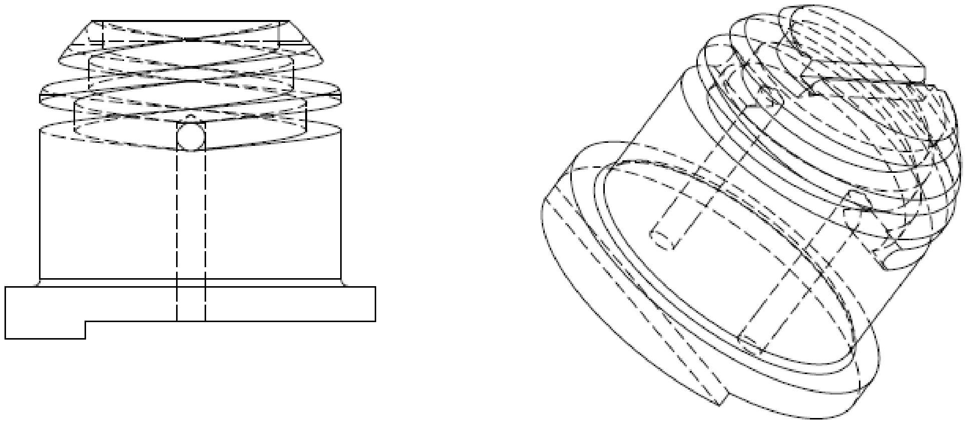


*Fig. 35. Core spiral model*

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).

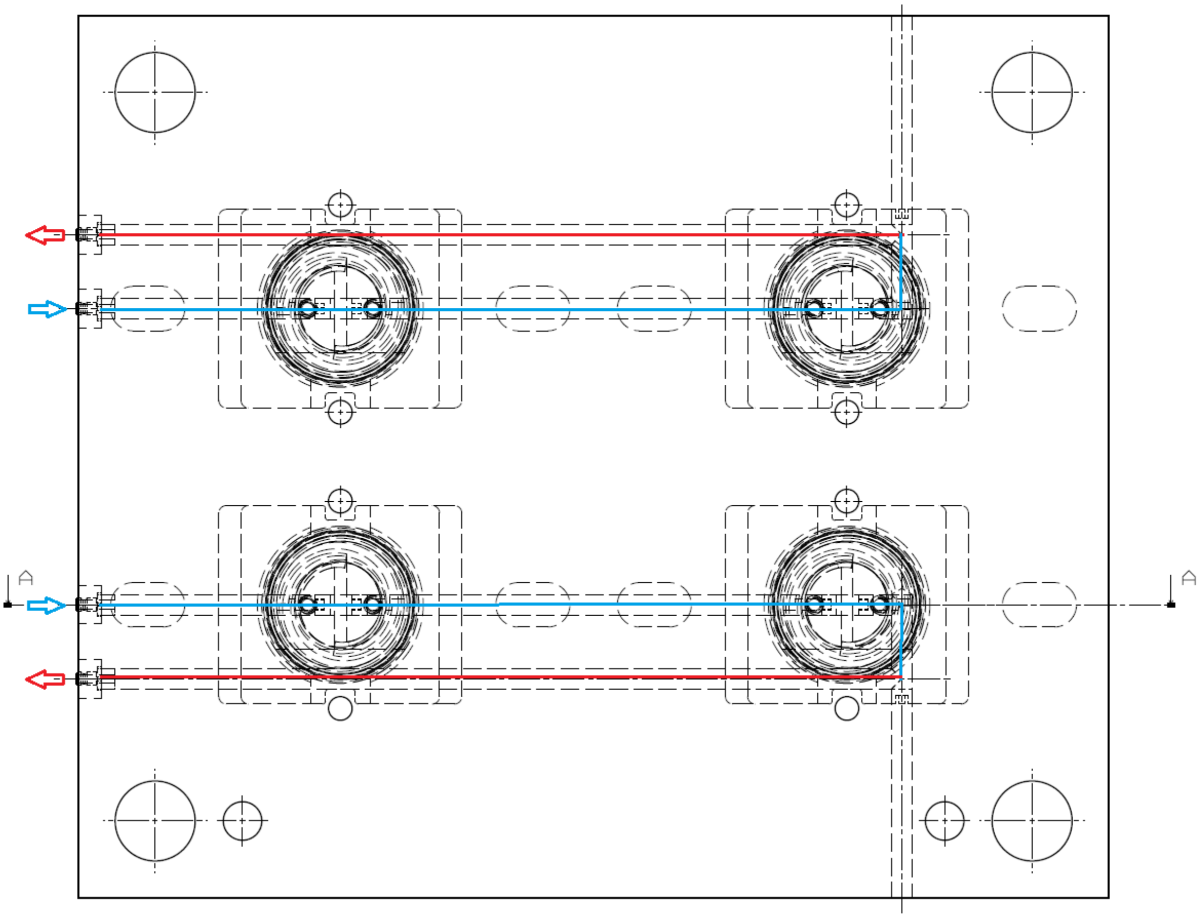


*Fig. 36. Core spiral cooling circuits*

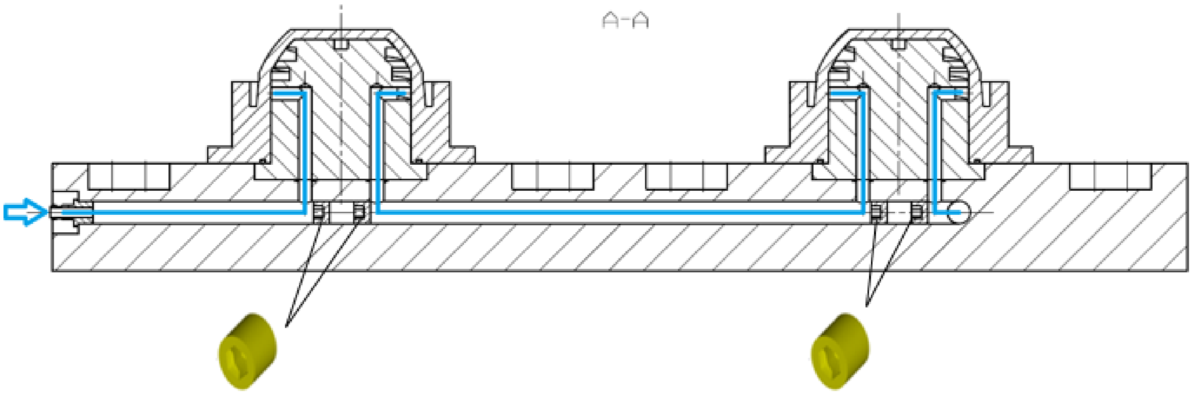



*Fig. 37. Section A-A of core spiral cooling circuit*

Sliders cooling

Sliders are cooled like cavities namely by cooling line network of drilled cooling channels. There are four cooling circuits all together for sliders´ cooling with the each of them for two sliders.

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.

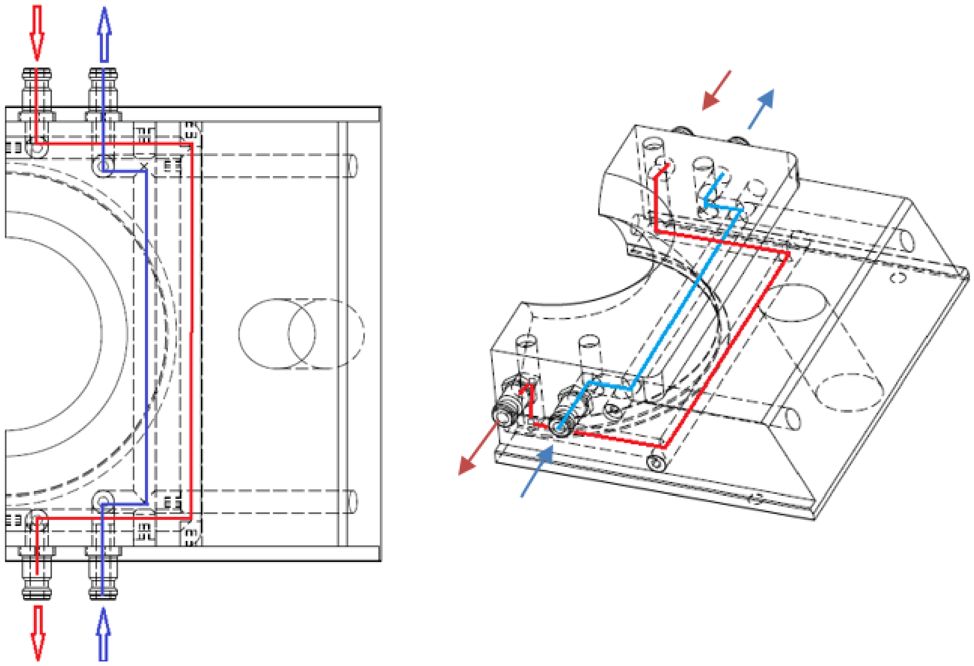


*Fig. 38. Slider cooling*

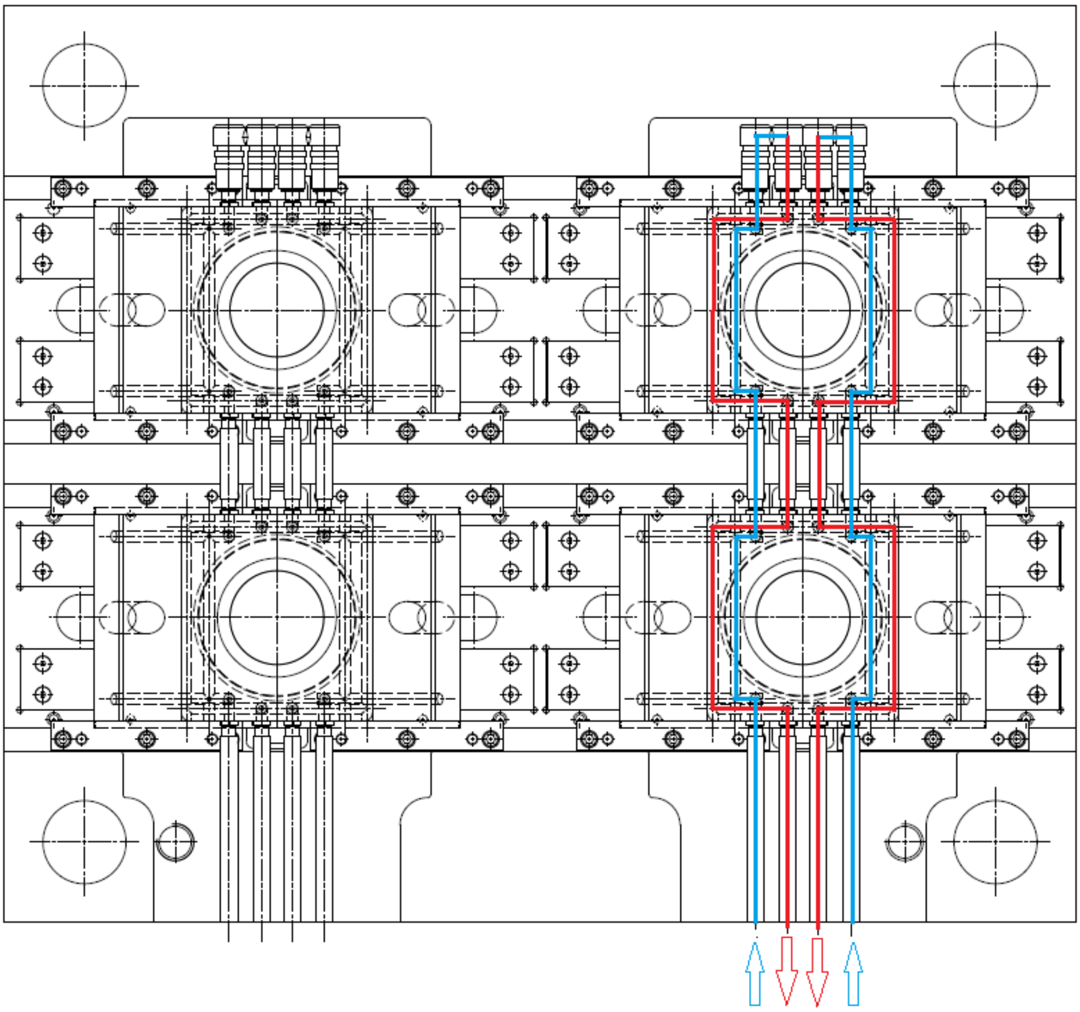


*Fig. 39. Slider cooling circuits*

### 10.2.4 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 shrinkage 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.

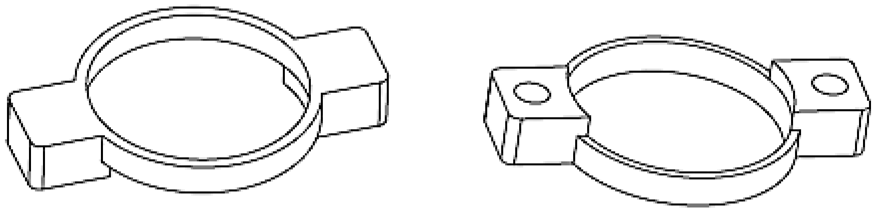


*Fig. 40. Stripper rings*

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 steel1.0060 (E335) which has the ČSN equivalent 11 600.

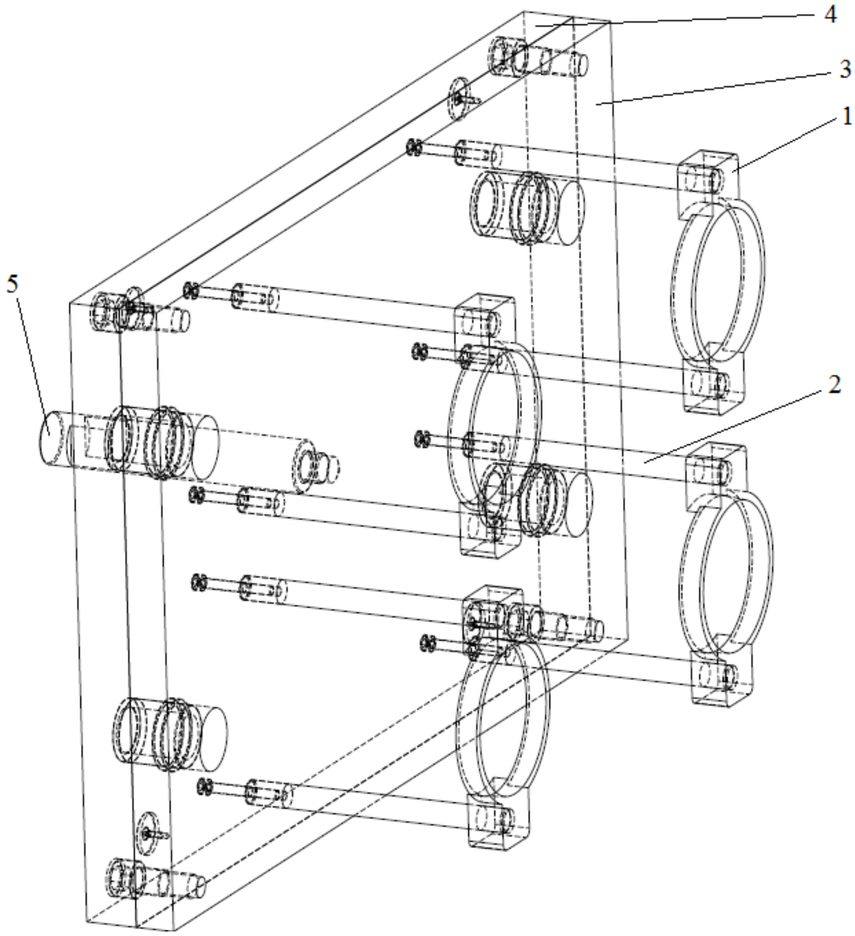



*Fig. 41. Ejector assembly*

1 – Stripper ring, 2 – Connection bar, 3 – Ejector plate A, 4 – Ejector plate B, 5 – Knock-out.

### 10.2.5 Venting

Venting plays an important part in injection mold designing. Without venting the air inside of the mold have no space to escape. As the air is compressed, its heat content is now concentrated in a small volume, resulting in a large temperature increase. In the designed injection mold, it is supposed that location of possible air traps will be in the edges of the part, places that are filled last. In this case we count on leakage of the air into parting planes, namely in the space between sliders, cores and cavities.

### 10.2.6 Manipulation system

For easier handling and manipulation with injection mold, mold was equipped with lifting eyes from Hasco.

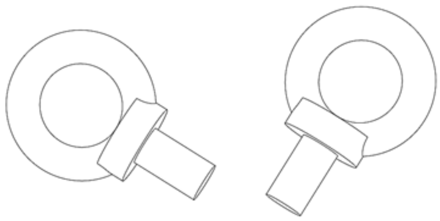


*Fig. 42. Lifting eyes from Hasco*

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

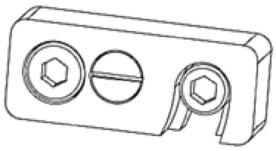


*Fig. 43. Lock device*

## 10.3 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.

### 10.3.1 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 runnermold size dimensions are 696 x 696 x 498 mm (L x W x H).

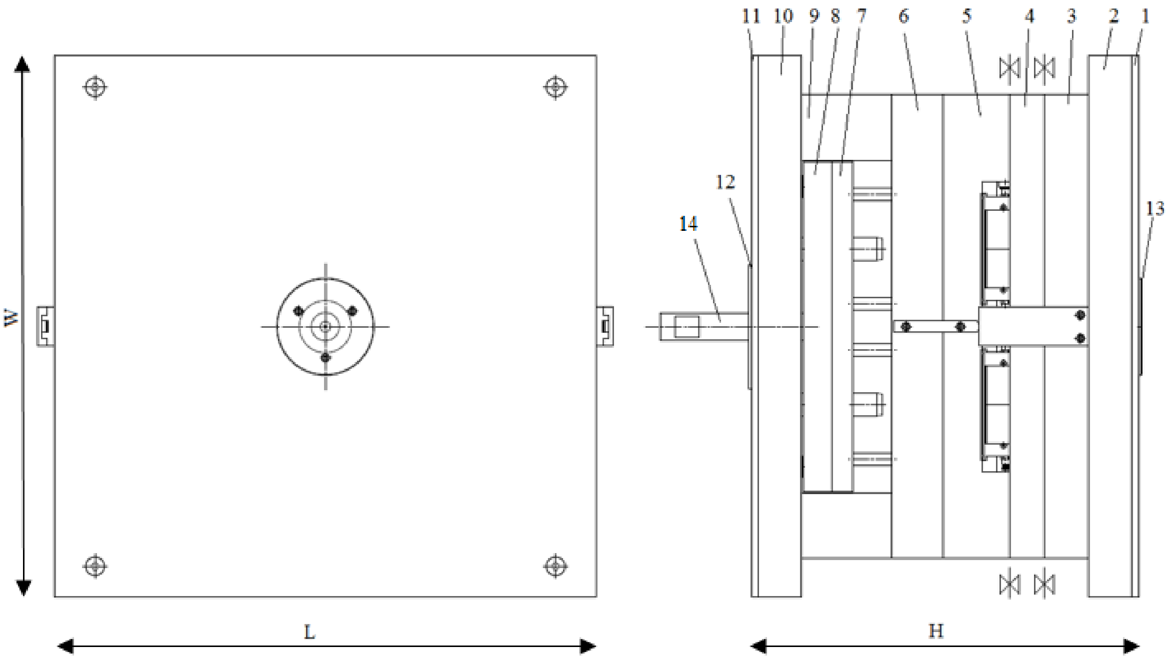



*Fig. 44. Mold frame – cold runner mold*

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

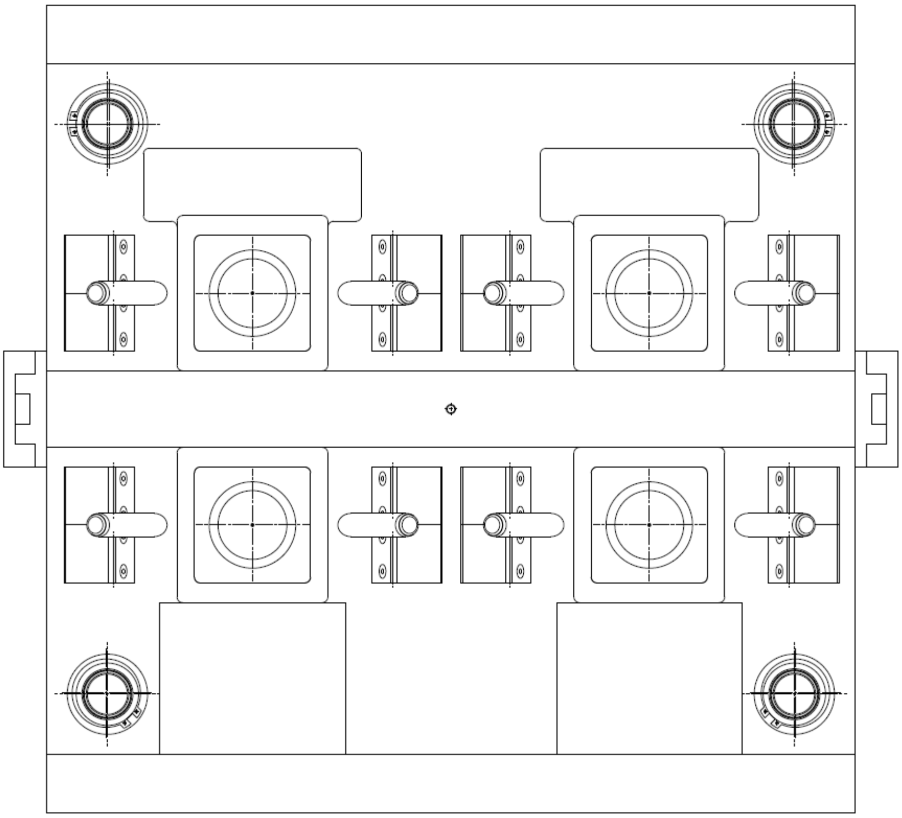


*Fig. 45. View to main parting plane – injection side*

### 10.3.2 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.

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.

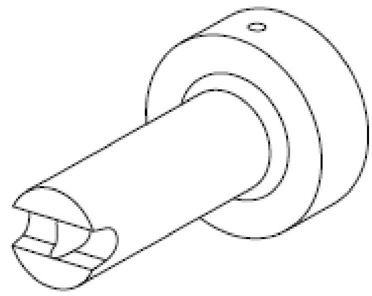


*Fig. 46. Cold sprue*

<u>Cold runners and cold gate</u>

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 withrespectivelygraded 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.

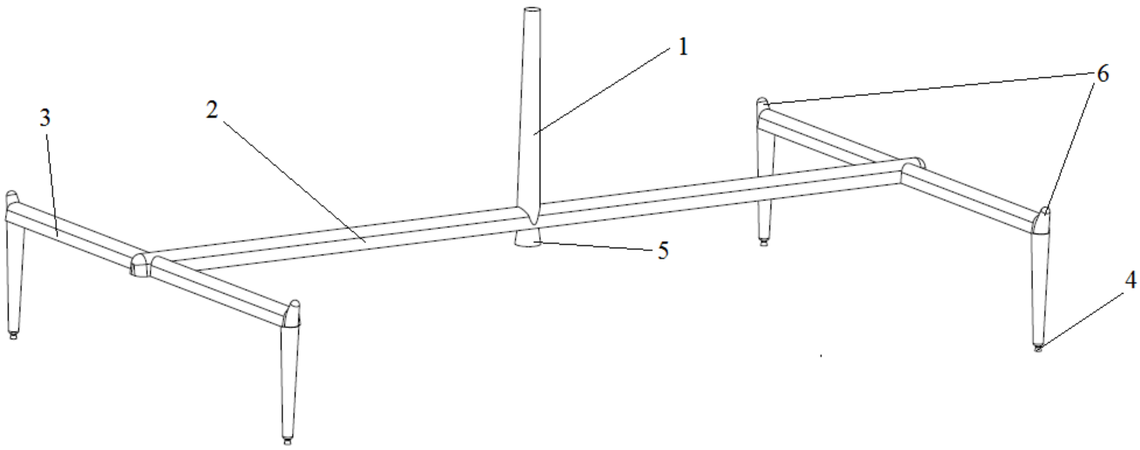



*Fig. 47. Cold runner system*

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

### 10.3.3 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 stripper rings remained the same and an ejector pin for cold runner holder was added.

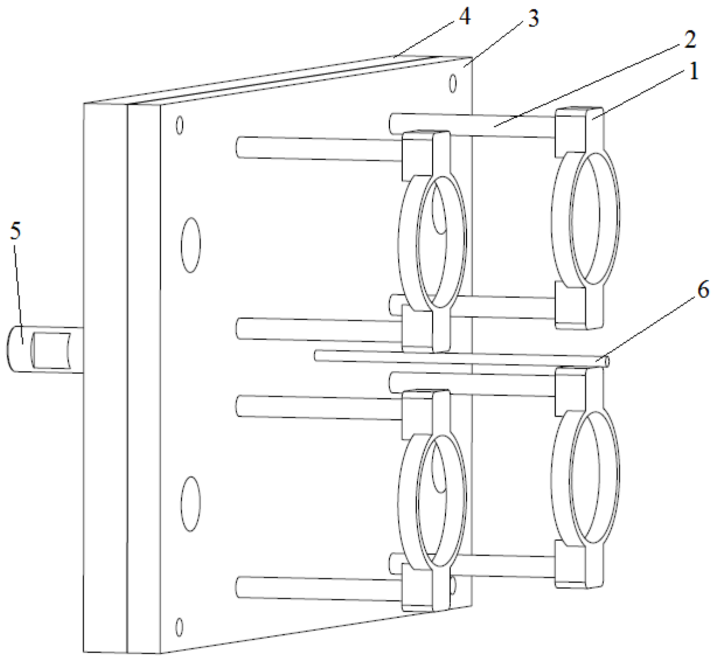



*Fig. 48. Ejector assembly – cold runner mold*

1 – Stripper ring, 2 – Connection bar, 3 – Ejector plate A, 4 – Ejector plate B, 5 – Knock-out, 6 – Ejector pin.

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.

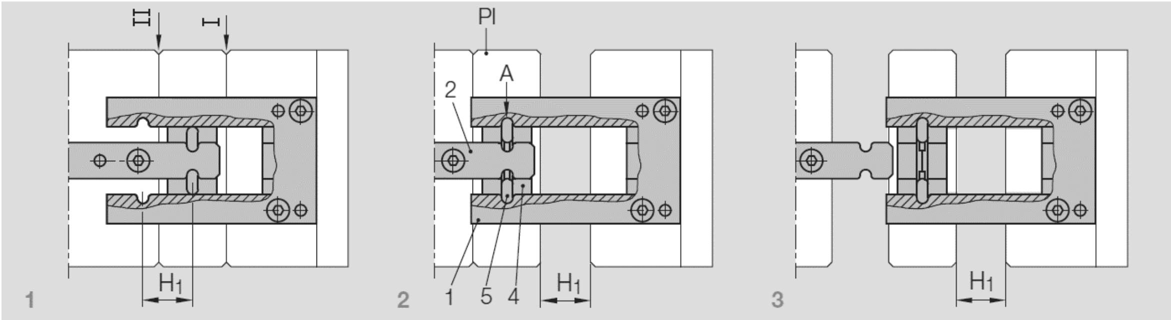



*Fig. 49. Latch locking device [28]*

# 11 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.

## 11.1 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. [21]

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

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

### 11.1.1 Material costs

Chosen material is PP RA12MN40 from company Sabic Europe and it costs 1,40EUR per kilogram. [22]

$$1{,}4\ \text{EUR} = 38{,}381\ \text{CZK} \qquad (2)$$

### 11.1.2 Energy costs

Cost of electric energy on 16. 4. 2015 according to www.kurzy.cz is 31,6 EUR/MWh.[23]

31,6 € = 866,31 CZK (3)

Cost of electric energy is 0,866 CZK for 1 kW/h.

#### 11.1.3 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 (4)

Nozzles

Hot nozzles were selected from Hasco catalog with cost of 811,88EUR for each.

4x 811.88 € = 3247,5 EUR = 89 030 CZK (5)

Total cost of hot runner system is 168 533,5 CZK.

Energy consumption

Total electric energy consumption is sum of manifold and nozzles electric consumptions.

$$P_{HR} = P_M + P_N \tag{6}$$

$$P_{HR} = 1\,800 + (4 * 400) \tag{7}$$

$$P_{HR} = 3\,400\,W = 3{,}4\,kW \tag{8}$$

$$C_{EHR} = 3{,}4 * 0{,}866 = 2{,}94\,CZK/hour \tag{9}$$

#### 11.1.4 Production costs – cold runner

Total weight of used material in one injection cycle.

$$m_T = m_P + m_{CR} \tag{10}$$

$$m_T = 76 + 45 = 121\,g = 0{,}121\,kg \tag{11}$$

Material costs:

$$C_{MCR} = 0{,}121 * 1{,}4 = 0{,}169\,EUR = 4{,}65\,CZK \tag{12}$$

Cost of used material with cold runner system is 4,65 CZK for one injection cycle.

### 11.1.5 Production costs – hot runner

Material costs:

Weight of one shot (4 injected parts) with hot runner system is 0,076 kg.

$$C_{MHR} = m_P * 1{,}4\ EUR \quad (13)$$

$$C_{MHR} = 0{,}076 * 1{,}4 = 1{,}1048\ EUR = 2{,}91\ CZK \quad (14)$$

Cost of used material with hot runner system is 2,91 CZK for one cycle.

Energy costs:

Energy costs for one hour are 2,94 CZK/hour. Number of injection shots per hour (3 600 s) with injection cycle being 15 s.

Number of injection cycles per hour:

$$s = \frac{3600}{15} = 240\ cycles \quad (15)$$

Calculation of energy consumption for one injection cycle:

$$C_{ES} = \frac{C_{EHR}}{240} = 0{,}012\ CZK \quad (16)$$

Total cost of energy and material for one injection cycle.

$$C_{HR} = C_{MHR} + C_{ES} \quad (17)$$

$$C_{HR} = 2{,}91 + 0{,}012 = 2{,}92\ CZK \quad (18)$$

Total cost (material and enegy) of hot runner system is 2,92 CZK for one cycle.

### 11.1.6 Balance

*Tab. 3.Expenses of runner systems*

| | Cost [EUR] | Cost [CZK] |
|---|---|---|
| **Hot runner system** | | |
| Hot manifold | 2 900 | 79 503,5 |
| Hot nozzle (4x) | 3 247,5 | 89 030 |
| Junction box + cables | 200 | 5 483 |
| **Total** | **6 347,5** | **174 016,5** |
| | | |
| **Cold runner system** | | |
| Cold sprue | 59 | 1617,5 |
| Latch locking system (2x) | 1 082,5 | 29 676,7 |
| **Total** | **1 141,5** | **31 294,2** |

*Tab. 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.

$$4{,}65x = 2{,}92\text{x} + 142\,722 \tag{19}$$

$$1{,}73x = 142\,722 \tag{20}$$

$$x = 82\,498 \text{ Cycles} \tag{21}$$

<u>Duration in time:</u>

$$t = (\text{x} * 15)/3600 \tag{22}$$

$$t = 343{,}7 = 344 \text{ hours} \tag{23}$$

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.

## 11.2 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. Furthermore, 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 systemis 2,92 CZK and for the cold runner system it is 4,65 CZK. Balance of individual runner systems shows that 82 030 injection cycles are required to pay the initial cost of hot runner system.

# 12 CAESIMULATION

Simulation of the injection process was done in Autodesk MoldflowSynergy 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 (midplane, dual domain and 3D mesh).In this case Dual Domain was chosen which is sufficient for this analysis.

Due to the symmetry and better resultsclarification, figures in this chaptershow results for only one of the cavities instead of four. In CAE analysis the symmetry was not contemplated.

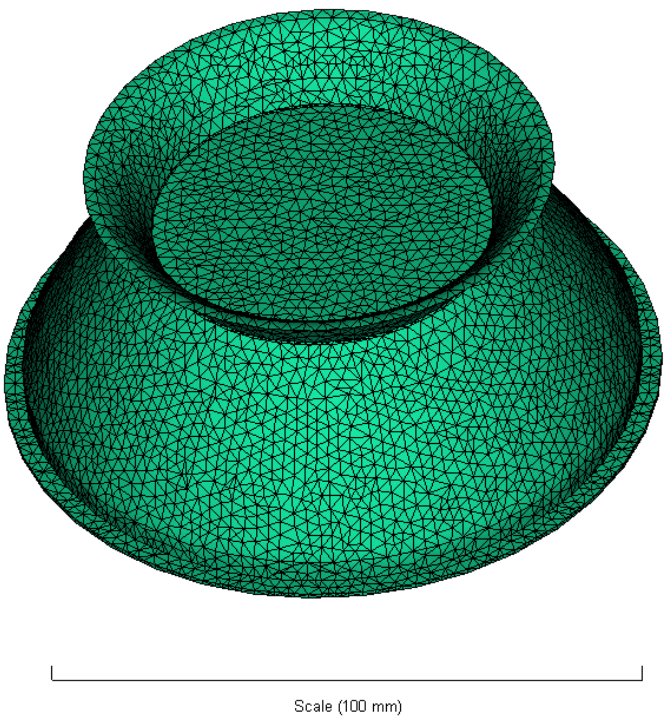



*Fig. 50. Meshed part*

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 andthis solution was found compliant.

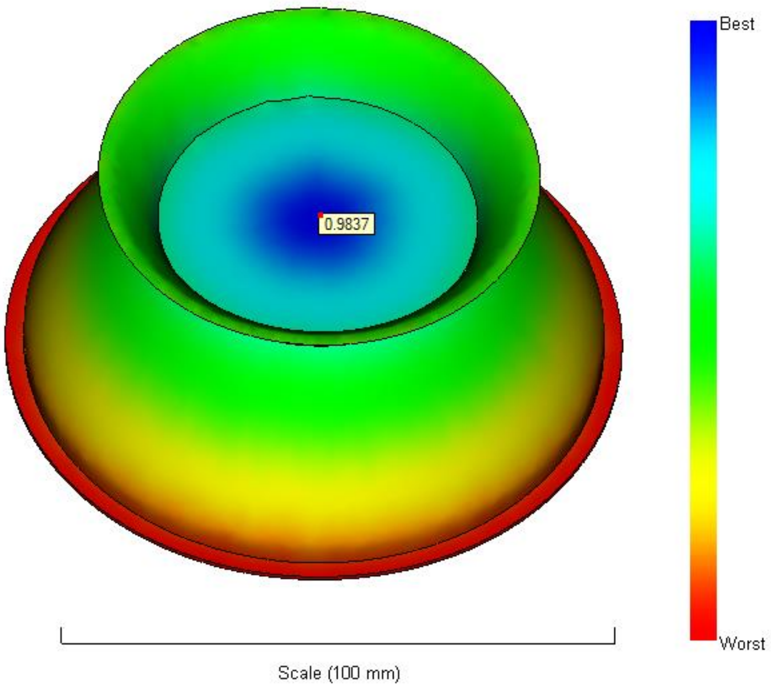



*Fig. 51. Results of the gate location analysis*

## 12.1 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 material and injection molding machines are discussed in chapters 8.2 and 9. 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 materials this one is labelled as Tool steel P20 with DIN steel equivalent 1.2311 with following characterizations.

Mold material [32]:

- 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 [-]

### 12.1.1 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.

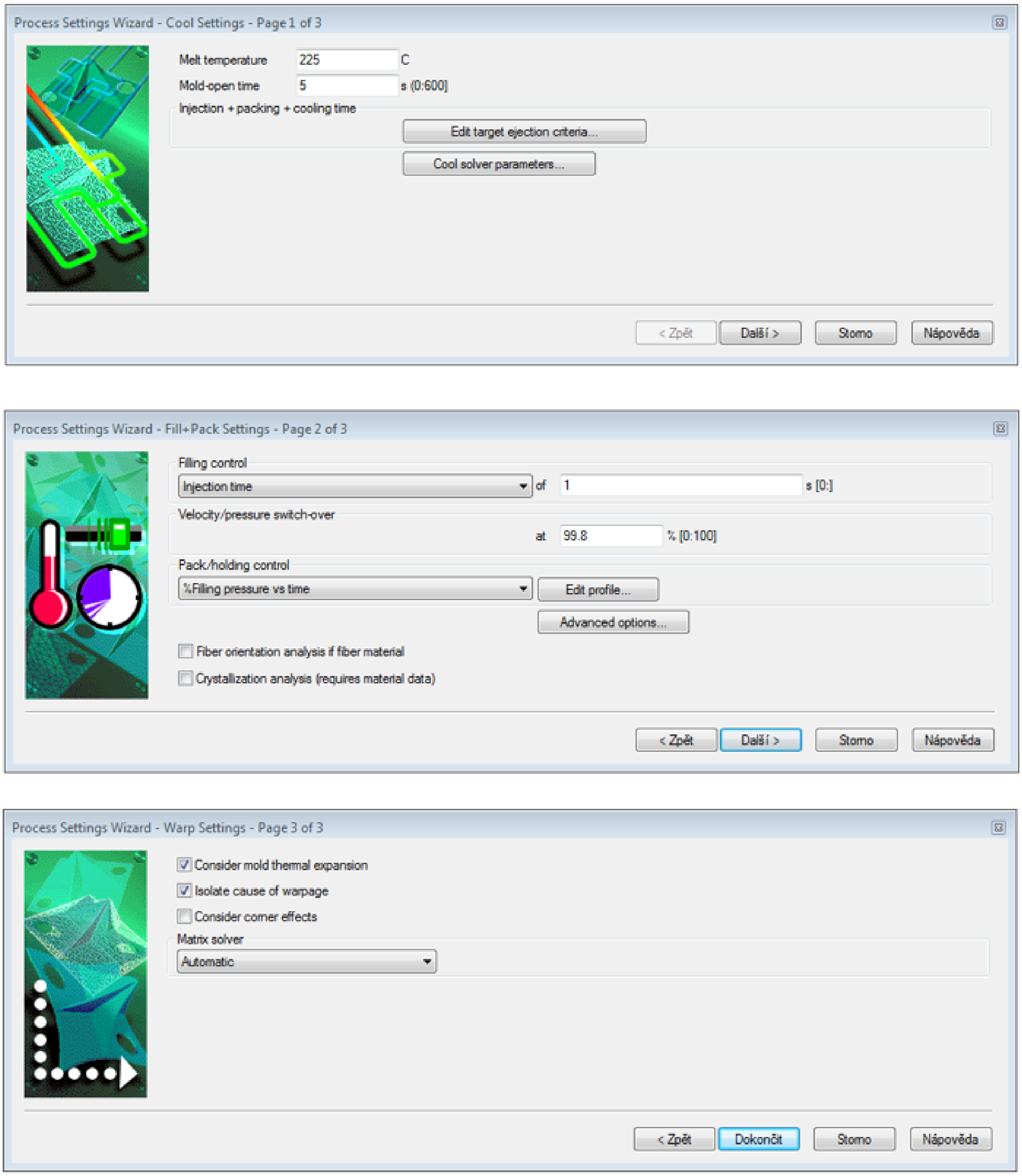



*Fig. 52. Process parameters*

## 12.2 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.

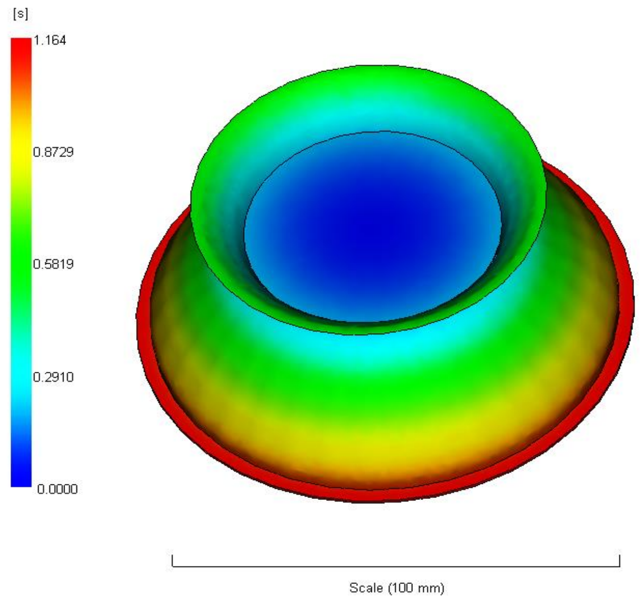



*Fig. 53. Filling analysis - results*

## 12.3 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.

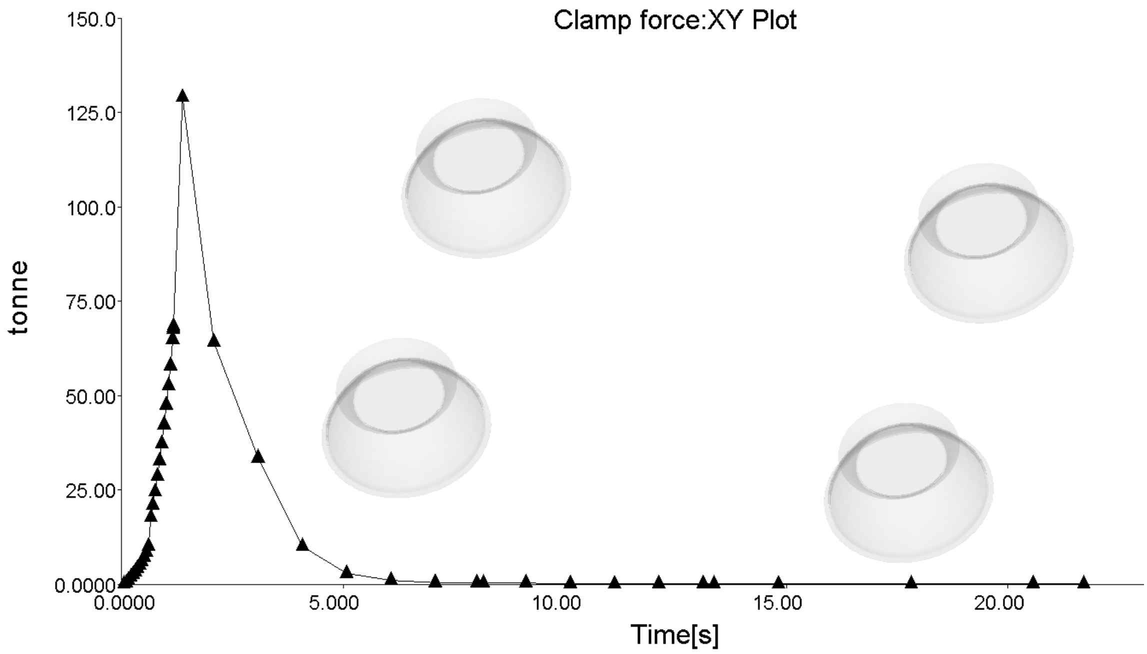



*Fig. 54. Clamp force*

## 12.4 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.

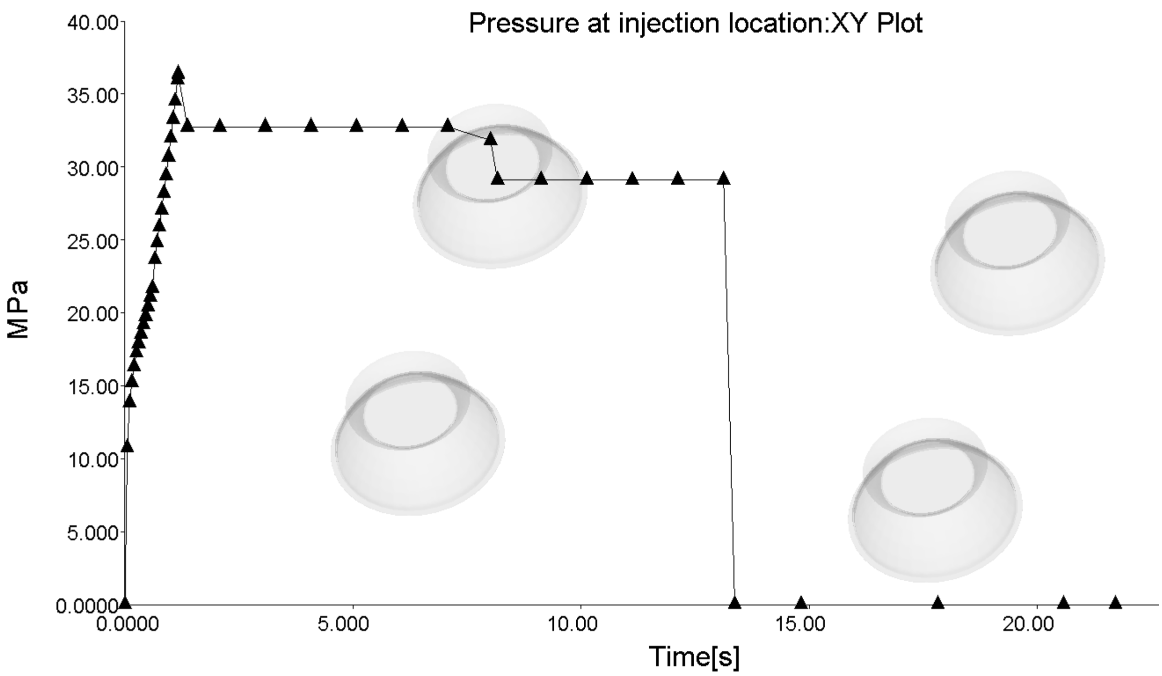



*Fig. 55. Pressure at time diagram*

## 12.5 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 $s^{-1}$.

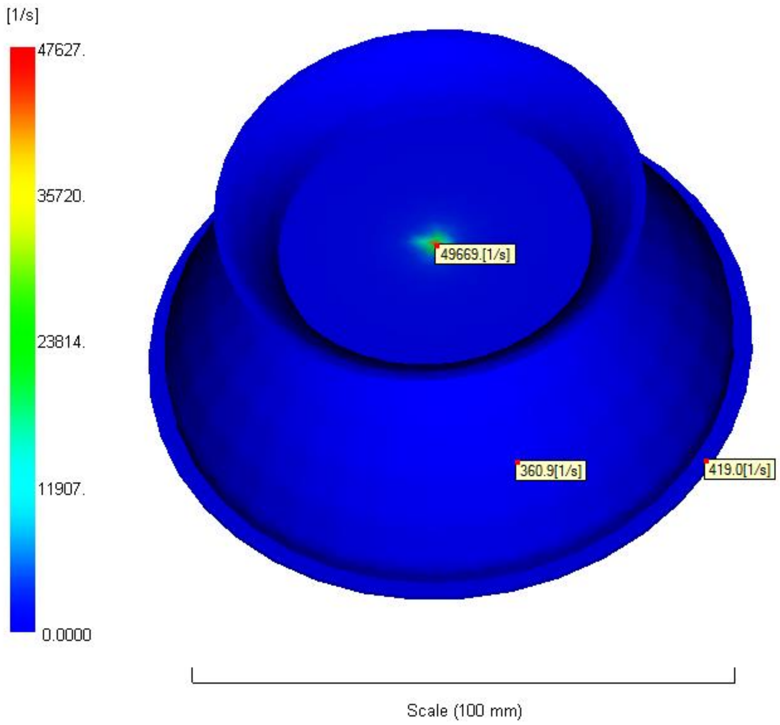



*Fig. 56. Shear rate analysis*

## 12.6 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 designthat is discussed in previous sections of the thesis.

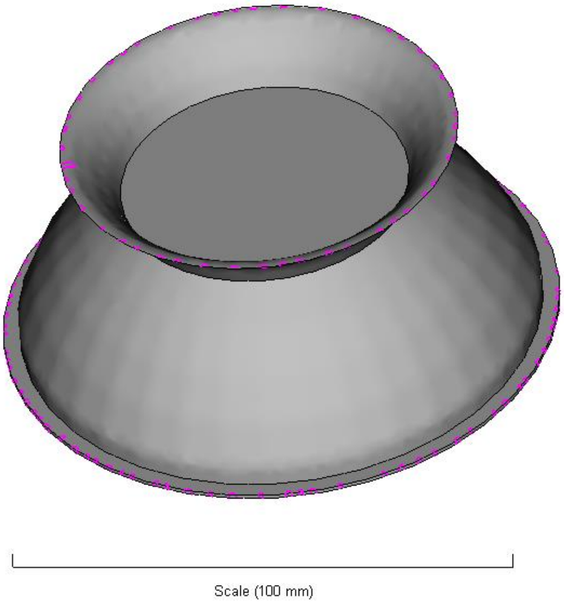



*Fig. 57. Air traps - results*

## 12.7 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.

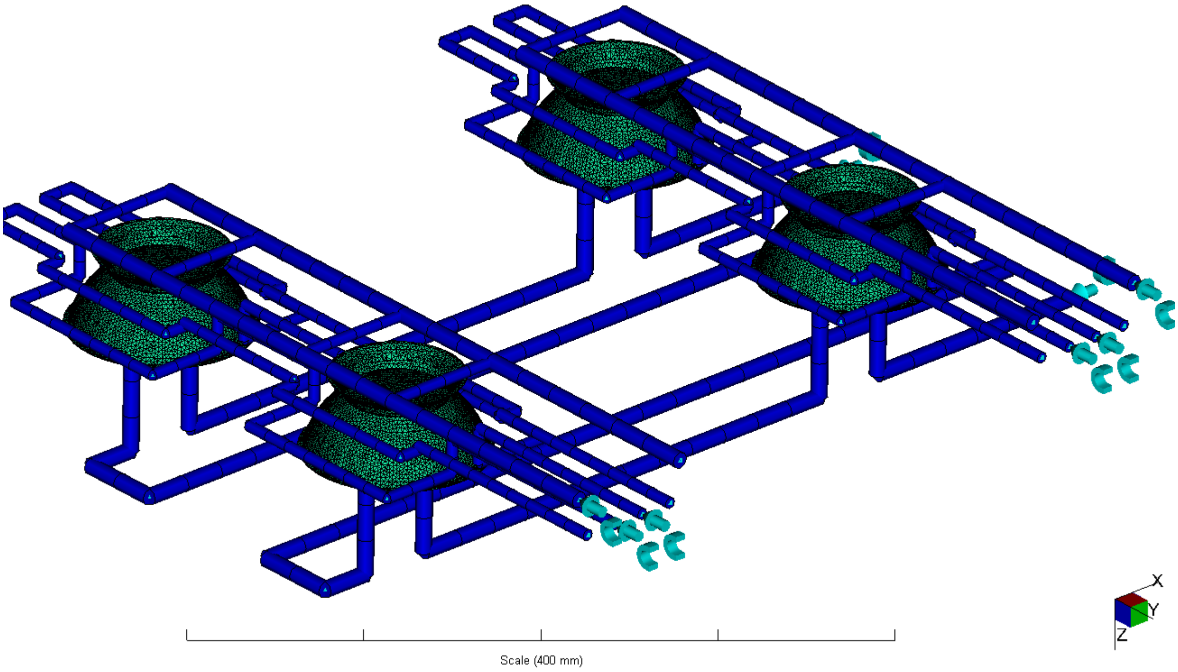



*Fig. 58. Cooling circuits*

## 12.8 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.

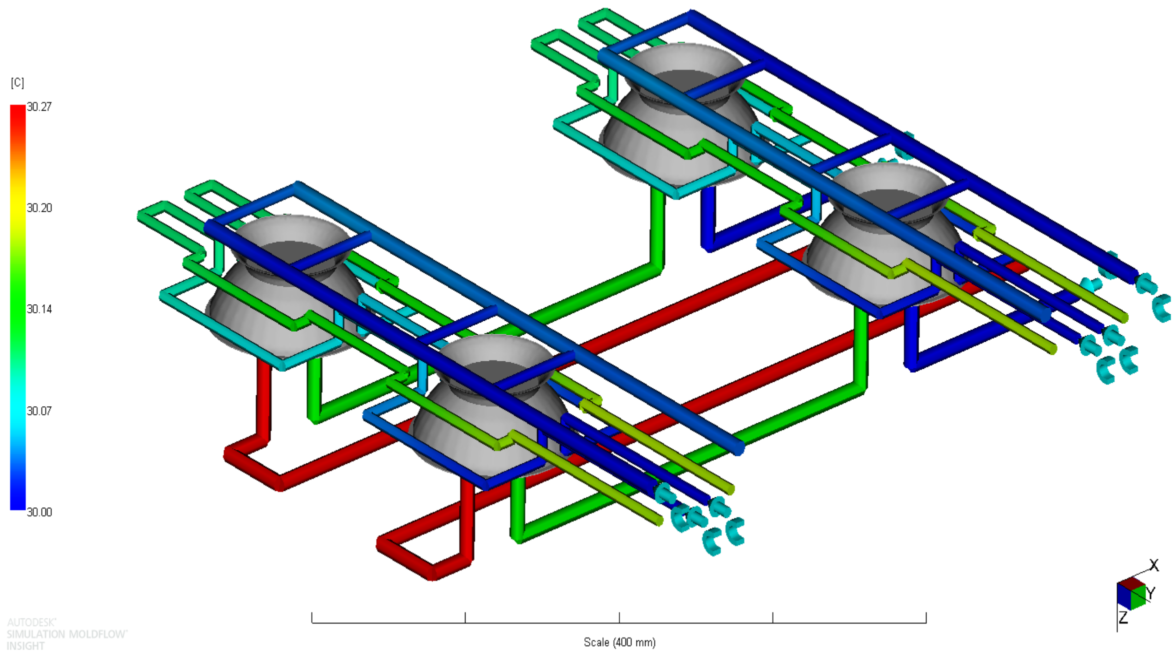



*Fig. 59. Circuit coolant temperature - results*

## 12.9 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. Therefore 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.

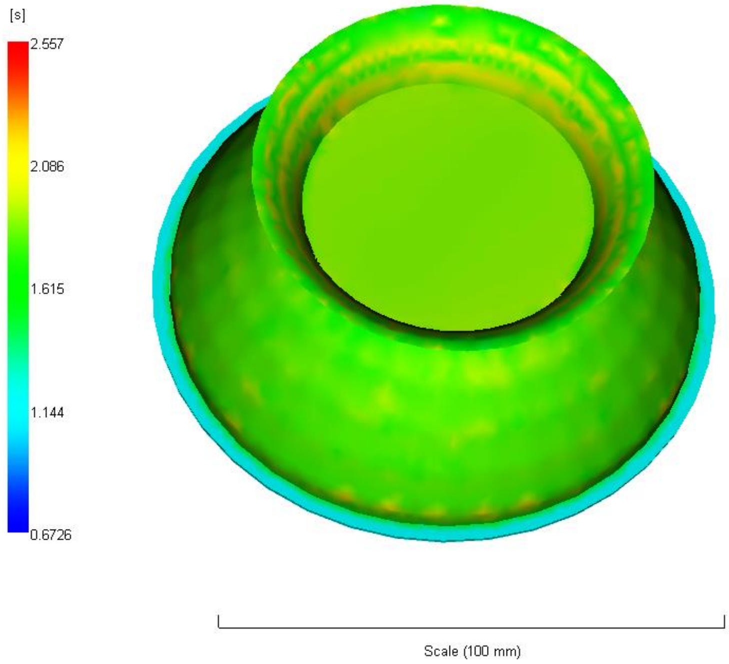



*Fig. 60. Time to reach ejection temperature - results*

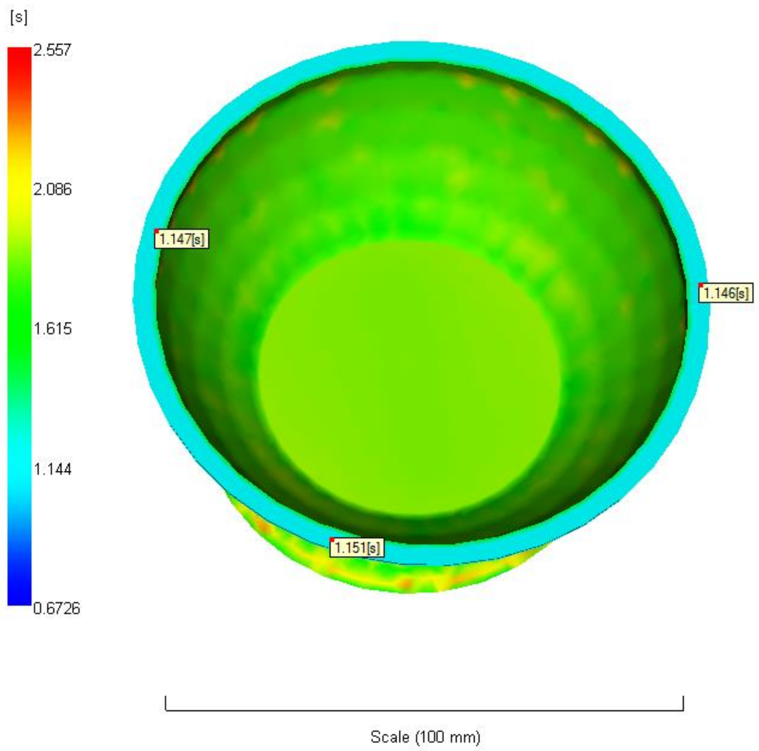



*Fig. 61. Time to reach ejection temperature – results 2*

## 12.10 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 andinsufficient 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.

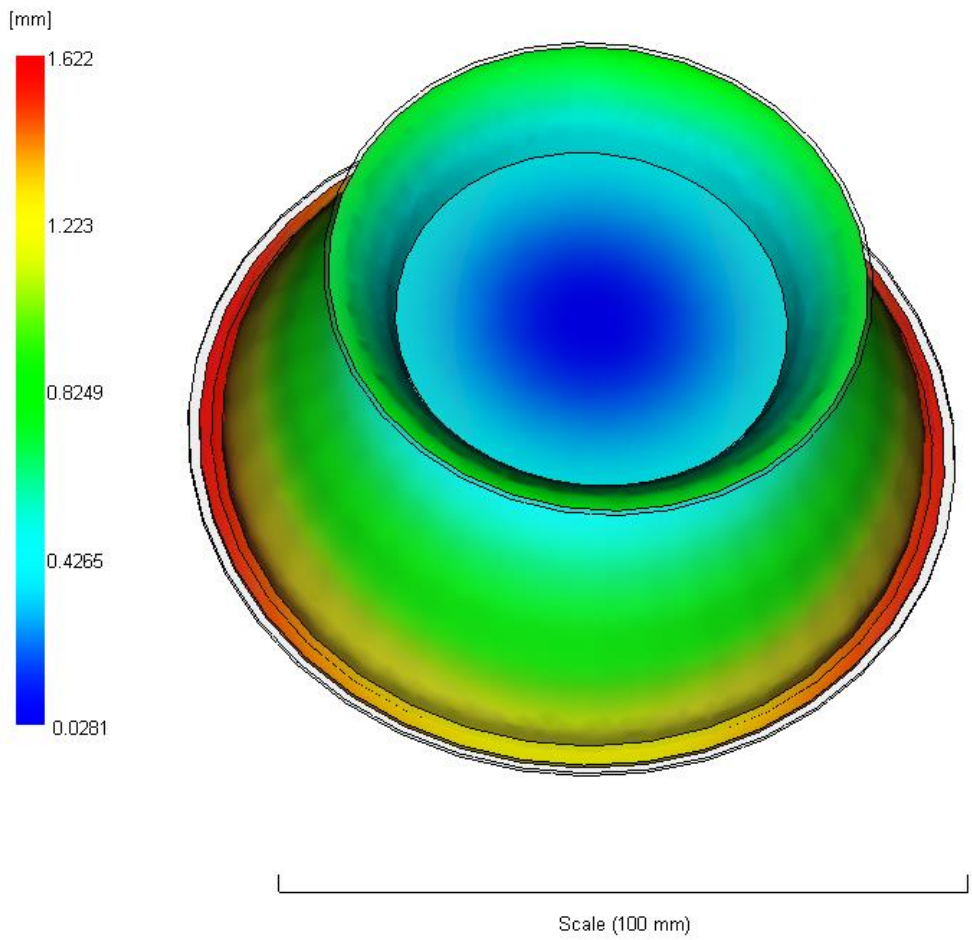



*Fig. 62. Total deflection*

# 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 ArburgAllrounder 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 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.

# ANALISIS 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 Moldflow 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 designed. 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.

# 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 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 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: technology 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 approach to materials, mold design, and processing*. Munich: Hanser Publishers, 2009. ISBN 9781613443026.

[10] SKOČOVSKÝ, P., BOKŮVKA, O., KONEČNÁ, R., TILLOVÁ, E., *Náuka o materiálipre 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. technologické*. 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: forinjectionmoldingofthermoplastics*. Cincinnati: HanserGardnerPublications, c2006, xxi, 870 p. ISBN 978-156-9903-759.

[14] ZEMAN, Lubomír. *Vstřikováníplastů: úvod do vstřikovánítermoplastů*. 1. vyd. Praha: 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.

Internet sources:

[17] *Vstřikování Plastů* [online]. 2005 [cit. 2014-11-10]. Dostupný z WWW: <http://www.ksp.tul.cz/cz/kpt/obsah/vyuka/skripta_tkp/sekce_plasty/04.htm>.

[18] *Plasty* [online]. [2002] [cit. 2014-11-12]. Dostupný z WWW: <http://www.ateam.zcu.cz/plasty.pdf>.

[19] *Arburg*[online]. c2001-2007 [cit. 2014-15-3]. Dostupný z WWW: <http://www.arburg.com >.

[20] *Formplast* [online]. 2014 [cit. 2013-15-3]. Dostupný zWWW: <http://www.formplastgmbh.de/en/injection_moulding_machine.php?maschinen_id=4872>.

[21] *Slovenskánárodnábanka* [online]. 2015 [cit. 2015-16-4]. Dostupné z WWW: <http://www.nbs.sk>.

[22] *Inetrplastics* [online]. 2015 [cit. 2015-16-4]. Dostupné z WWW: <http://www.interplastics.sk>.

[23] *Kurzyenergií* [online]. 2015 [cit. 2015-16-4]. Dostupné z WWW: <http://www.kurzy.cz/komodity>.

[24] *Aircraft materials* [online]. 2015 [cit. 2015-10-4]. Dostupné z WWW:<http://www.aircraftmaterials.com/data/aluminium/6082.html>.

[25] *Smartplast* [online]. 2015 [cit. 2015-10-3]. Dostupné z WWW:<http://www.smartplast.cz/ami.php>.

[26] *Plastics* [online]. 2015 [cit. 2015-10-1]. Dostupné z WWW: <http://plastic-knowledge-kanyakumari.blogspot.cz/2011/10/tool-design-mould-cooling-system-and.html>.

[27] *Technodat* [online]. 2015 [cit. 2015-10-3]. Dostupné z WWW: <http://www.technodat.cz/catia-v5>.

[28] *Hasco* [online]. 2015 [cit. 2015-10-3]. Dostupné z WWW:<http://www.hasco.com>.

[29] *Sabic* [online]. 2015 [cit. 2015-10-3]. Dostupné z WWW: <http://www.atlantic-polymers.pl/plastics/pp/RA12MN40.pdf>.

[30] *Moldflow* [online]. 2015 [cit. 2015-10-3]. Moldflow Insight. Dostupné z WWW: <http://images.autodesk.com/adsk/files/moldflow_insight_detail_brochure_us.pdf>.

[31] *Synventive* [online]. 2015 [cit. 2015-10-3]. Dostupné z WWW: <http://www.synventive.com/synventive-hot-runner-systems.aspx>.

Electronicprograms:

[32] *AutodeskMoldflowInsight 2014* [computer program]. Ver. EducationalEdition-ServicePack 1. AutodeskInc., 2009 [cit. 2015-04-14].>.

# LIST OF ABBREVIATIONS

| | |
|---|---|
| 2D | Two-dimensional space |
| 3D | Three-dimensional space |
| CAD | Computer-aided design |
| CAE | Computer-aided engineering |
| $C_E$ | Cost of electric energy |
| $C_{EHR}$ | Eneryconsumtion - hot runner system |
| $C_{ES}$ | Energy consumption for one injection cycle |
| $C_{EHS}$ | Cenaenergiíhorkéhovtokovéhosystému |
| $C_{HR}$ | Cost of electric energy and material – hot runner |
| $C_{MCR}$ | Cost of material – cold runner |
| $C_{MHR}$ | Cost of material – hot runner |
| E | Young's modulus [MPa] |
| G | Shear modulus [MPa] |
| H | Height [mm] |
| HRC | Hardness according to Rockwell |
| L | Length [mm] |
| MFR | Melt flow rate index [g/10 min] |
| $m_{CR}$ | Cold runner weight |
| $m_p$ | Weight of parts |
| $m_T$ | Total weight |
| $P_{HR}$ | Input (power) hot runner |
| $P_M$ | Input (power) manifold |
| $P_N$ | Input (power) nozzle |
| PA6 | Polyamid 6 |
| PC | Polycarbonate |

| | |
|---|---|
| PE | Polyethylene |
| PEEK | Polyetheretherketon |
| PMMA | Polymethylmethacrylate |
| POM | Polyoxymethylene |
| PP | Polypropylene |
| PS | Polystyrene |
| PVC | Polyvinyl chloride |
| Ra | Arithmetic average of the roughness profile [μm] |
| s | Number of injection cycles |
| t | Time [s] |
| Tg | Glass–liquid transition temperature [°C] |
| Tm | Melting temperature [°C] |
| W | Width [mm] |

# LIST OF FIGURES

*Fig. 1. Division of polymer materials [4]* ........ 13
*Fig. 2. Primary and secondary bonds between separate polymer chains [1]* ........ 13
*Fig. 3. Application area – Amorphous thermoplastic polymers [4]* ........ 14
*Fig. 4. Amorphous structure [1]* ........ 14
*Fig. 5. Application area – Semicrystalline thermoplastic polymers [4]* ........ 15
*Fig. 6. Semicrystalline structure [1]* ........ 15
*Fig. 7. Common structure of a thermosetting material [8]* ........ 16
*Fig. 8. Common structure of elastomer materials [8]* ........ 17
*Fig. 9. Injection molding cycle [5]* ........ 19
*Fig. 10. General guidelines in maintaining the constant thickness [1]* ........ 21
*Fig. 11. Different layouts of ribs [4]* ........ 22
*Fig. 12. Different ways of marks designing [4]* ........ 23
*Fig. 13. Example of cold runner system [4]* ........ 25
*Fig. 14. Cross sections of a runner [4]* ........ 27
*Fig. 15. Hot runner system [31]* ........ 29
*Fig. 16. Commonly used manifold types [28]* ........ 30
*Fig. 17. Hot nozzle Techni Shot from Hasco [28]* ........ 31
*Fig. 18. Basic ejection problem – Five sided box [1, 12]* ........ 31
*Fig. 19. Parallel circuit [24]* ........ 34
*Fig. 20. Series circuit [24]* ........ 34
*Fig. 21. Baffle and bubbler [24]* ........ 35
*Fig. 22. Common vent design [1]* ........ 36
*Fig. 23. Render 3D model of the cup* ........ 41
*Fig. 24.ArburgAllrounder 720 H [19]* ........ 43
*Fig. 25. Part-forming elements* ........ 44
*Fig. 26. Core plate* ........ 45
*Fig. 27. Cavity plate* ........ 46
*Fig. 28. Cavity 3D model* ........ 47
*Fig. 29. Core 3D model* ........ 47
*Fig. 30. Sliders* ........ 48
*Fig. 31. Injection mold frame* ........ 49
*Fig. 32. Injection mold guiding elements* ........ 50

*Fig. 33. Hot runner system* ........ 51
*Fig. 34. Cavity cooling circuits* ........ 52
*Fig. 35. Core spiral model* ........ 53
*Fig. 36. Core spiral cooling circuits* ........ 53
*Fig. 37. Section A-A of core spiral cooling circuit* ........ 54
*Fig. 38. Slider cooling* ........ 54
*Fig. 39. Slider cooling circuits* ........ 55
*Fig. 40. Stripper rings* ........ 56
*Fig. 41. Ejector assembly* ........ 56
*Fig. 42. Lifting eyes from Hasco* ........ 57
*Fig. 43. Lock device* ........ 57
*Fig. 44. Mold frame – cold runner mold* ........ 58
*Fig. 45. View to main parting plane – injection side* ........ 59
*Fig. 46. Cold sprue* ........ 60
*Fig. 47. Cold runner system* ........ 60
*Fig. 48. Ejector assembly – cold runner mold* ........ 61
*Fig. 49. Latch locking device [28]* ........ 62
*Fig. 50. Meshed part* ........ 68
*Fig. 51. Results of the gate location analysis* ........ 69
*Fig. 52. Process parameters* ........ 70
*Fig. 53. Filling analysis - results* ........ 71
*Fig. 54. Clamp force* ........ 72
*Fig. 55. Pressure at time diagram* ........ 72
*Fig. 56. Shear rate analysis* ........ 73
*Fig. 57. Air traps - results* ........ 74
*Fig. 58. Cooling circuits* ........ 74
*Fig. 59. Circuit coolant temperature - results* ........ 75
*Fig. 60. Time to reach ejection temperature - results* ........ 76
*Fig. 61. Time to reach ejection temperature – results 2* ........ 76
*Fig. 62. Total deflection* ........ 77

# LIST OF TABLES

*Tab. 1. Gate types [3, 5]* ........................................ 27

*Tab. 2. Coolant types [4]* ........................................ 35

*Tab. 3.Expenses of runner systems* ........................................ 65

*Tab. 4. Expenses comparison* ........................................ 66

# APPENDICES

| | |
|---|---|
| AI | Data sheet of injected material |
| AII | Injection molding machine – data sheet |
| AIII | Data sheet of hot nozzle |
| AIV | Injection mold – right side |
| AV | Injection mold – left side |
| PVI | Drawing documentation |
| AVII | CD – Hot runner mold, injection molding analysis |
| AVIII | CD – Cold runner mold |

# AI DATA SHEET OF INJECTED MATERIAL

## SABIC® PP RA12MN40

### PP random copolymer for Injection moulding

**Description:**
This random copolymer is medium ethylene modified, clarified and anti-static. It is specially developed for injection moulding applications. Special characteristics are high stiffness, good transparancy and high gloss. Typical applications are in housewares and thin walled packaging.

**Health, Safety and Food Contact regulations:**
Material Safety Data Sheets (MSDS) and Product Safety declarations are available on our Internet site http://www.SABIC-europe.com

The product mentioned herein is in particular not tested and therefore not validated for use in pharmaceutical/ medical applications.

This grade material is UL registered under File E111275 (www.ul.com)

**Typical values** Revision 20110418

| Properties | Unit (Si) | Values | Test methods |
|---|---|---|---|
| **Polymer properties** | | | |
| **Melt flow rate (MFR)** | | | ISO 1133 |
| at 230 °C and 2.16 kg | g/10 min | **40** | |
| at 230 °C and 5 kg | g/10 min | - | |
| **Density** | kg/m³ | **905** | ISO 1183 |
| **Mechanical properties** | | | |
| **Tensile test** | | | ISO 527 |
| stress at yield | MPa | **29** | |
| stress at break | MPa | **32** | |
| strain at break | % | **>50** | |
| **Flexural test** | | | ASTM D 790 |
| Flexural modulus | MPa | **1200** | |
| **Izod impact notched** | | | ISO 180/4A |
| at 23 °C | kJ/m² | **4.5** | |
| at 0 °C | kJ/m² | **2.2** | |
| at -20 °C | kJ/m² | - | |
| **Charpy impact notched** | | | ISO 179 |
| at 23 °C | kJ/m² | **3.3** | |
| at 0 °C | kJ/m² | - | |
| at -20 °C | kJ/m² | - | |
| **Hardness Shore D** | - | **68** | ISO 868 |
| **Thermal properties** | | | |
| **Heat deflection temperature** | | | |
| at 1.80 MPa (HDT/A) | °C | **49** | ISO 75/A |
| at 0.45 MPa (HDT/B) | °C | **76** | ISO 75/B |
| **Vicat softening temperature** | | | |
| at 10 N (VST/A) | °C | **128** | ISO 306/A |
| at 50 N (VST/B) | °C | **66** | ISO 306/B |

# AII INJECTION MOLDING MACHINE– DATA SHEET

## Technical data | 720 H

| Clamping unit with clamping force | max. kN | 720 H 3200 |
|---|---|---|
| Opening force \| stroke | max. kN \| mm | — \| 600 |
| Mould height, fixed \| variable | min.-max. mm | — \| 300-800 |
| Platen daylight fixed \| variable | max. mm | — \| 900-1400 |
| Distance between tie bars (w x h) | mm | 720 x 720 |
| Mould mounting platens (w x h) | max. mm | 1040 x 1040 |
| Weight of movable mould half | max. kg | 2900 |
| Ejector force \| stroke | max. kN \| mm | 86 \| 250 |
| Dry cycle time EUROMAP ² | min. s - mm | 1,5 - 504 |

| Injection unit | | 1300 | | | 2100 | | | 3200 | | |
|---|---|---|---|---|---|---|---|---|---|---|
| with screw diameter | mm | 55 | 60 | 70 | 60 | 70 | 80 | 70 | 80 | 90 |
| Effective screw length | L/D | 22 | 20 | 17 | 23 | 20 | 17,5 | 23 | 20 | 18 |
| Screw stroke | max. mm | | 240 | | | 280 | | | 320 | |
| Calculated stroke volume | max. cm³ | 570 | 678 | 923 | 792 | 1078 | 1407 | 1232 | 1608 | 2036 |
| Shot weight | max. g PS | 521 | 620 | 844 | 723 | 984 | 1286 | 1125 | 1469 | 1860 |
| Material throughput | max. kg/h PS | 86 | 96 | 115 | 125 | 145 | 175 | 185 | 215 | 250 |
| | max. kg/h PA6.6 | 43 | 48 | 58 | 62 | 74 | 88 | 93 | 110 | 125 |
| Injection pressure | max. bar | 2380 | 2000 | 1470 | 2500 | 2000 | 1530 | 2500 | 2000 | 1580 |
| Holding pressure | max. bar | 2380 | 2000 | 1470 | 2500 | 2000 | 1530 | 2500 | 2000 | 1580 |
| Injection flow ² | max. cm³/s | 952 | 1134 | 1542 | 1132 | 1540 | 2012 | 1188 | 1552 | 1964 |
| Screw circumferential speed ² | max. m/min | 55 | 60 | 70 | 51 | 60 | 69 | 41 | 47 | 53 |
| Screw torque | max. Nm | 1510 | 1640 | 1920 | 2140 | 2500 | 2550 | 3140 | 3510 | 3510 |
| Nozzle contact force \| retraction stroke | max. kN \| mm | | 90 \| 550 | | | 110 \| 600 | | | 110 \| 600 | |
| Heating capacity \| zones | kW | | 22,9 \| 8 | | | 31,4 \| 8 | | | 38,4 \| 8 | |
| Feed hopper | l | | — | | | — | | | — | |

| Drive and connection with injection unit | | 1300 | 2100 | 3200 |
|---|---|---|---|---|
| Net weight of machine | kg | 15000 | 16500 | 18000 |
| Emiss. sound press. level DIN EN 201:1997 | dB(A) | | 65 +3 | |
| Oil filling | l | 260 | 360 | 360 |
| Drive power ² | max. kW | | — | |
| Electrical connection ³ | kW | 93 | 122 | 153 |
| Total | A | | — | |
| Machine | A | 160 | 200 | 250 |
| Heating | A | 35 | 50 | 63 |
| Cooling water connection | max. °C | | 30 | |
| | min. Δp bar | | 1,5 \| DN 25 | |

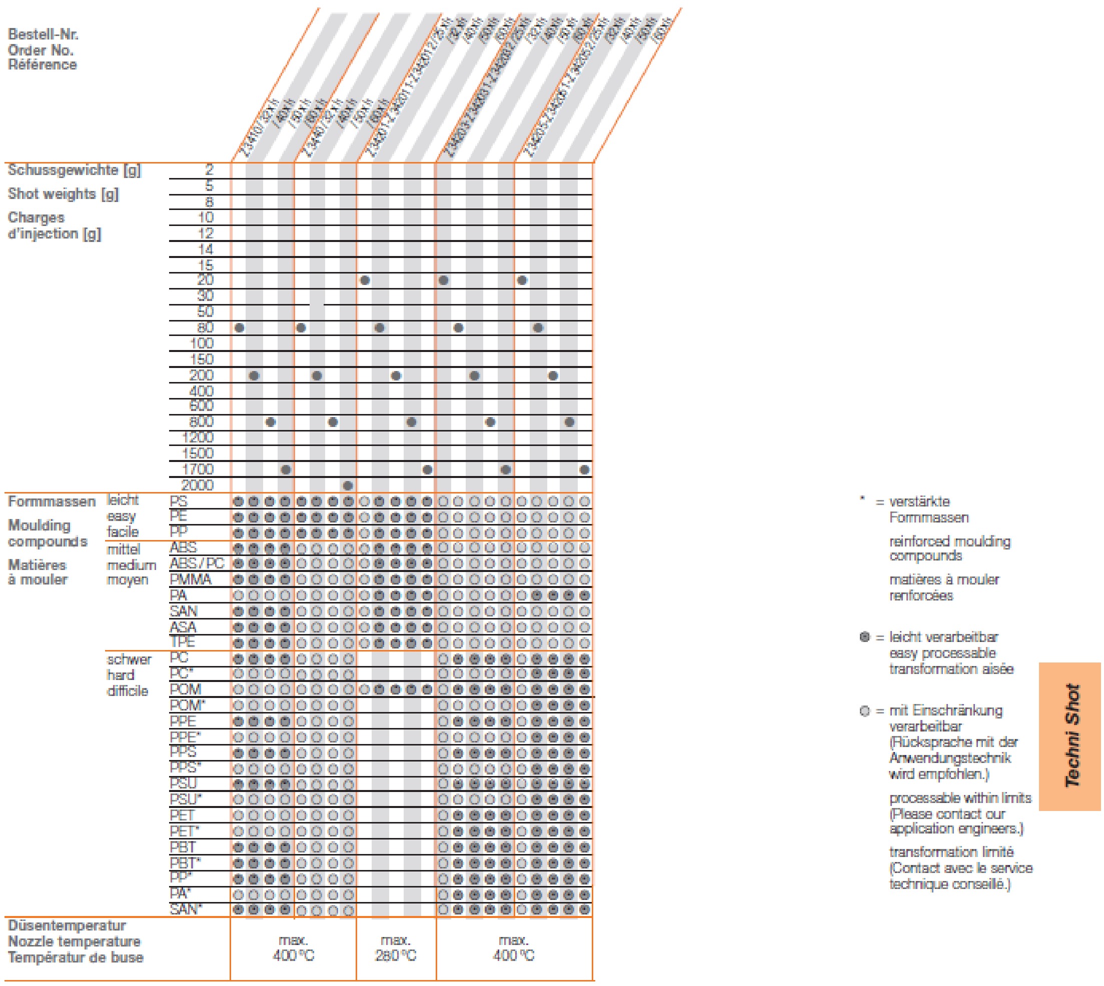
| Machine type with EUROMAP size designation ¹ |
|---|
| 720 H 3200-1300 \| 2100 \| 3200 |

**Upon request: other machine types and mould installation heights, screws, drive powers etc.**
All specifications relate to the basic machine version. Deviations are possible depending on variants, process settings and material type. Depending on the drive, certain combinations, e.g. max. injection pressure and max. injection flow may be mutually exclusive.

1) Clamping force (kN) - large injection unit = max. stroke volume (cm³) x max. injection pressure (kbar)
2) Specifications depend on the drive variant / drive configuration.
3) Specifications relate to 400 V/50 Hz.
[ ] Specifications apply to alternative equipment.

# AIII DATA SHEET OF HOT NOZZLE

**Z 3410/... - Z 342052/...**




Die Schussgewichts-Angaben sind Richtwerte, sie sind abhängig von den zu verarbeitenden Massen, dem Fließweg/Wanddickenverhältnis sowie weiteren Verarbeitungsparametern.

In Verbindung mit Heißkanal-Verteilerblöcken verringern sich die Gewichtsangaben. Bei relativ hohen Spritzdrücken und ungünstigen Verarbeitungsbedingungen ist der nächst größere Düsentyp zu verwenden.

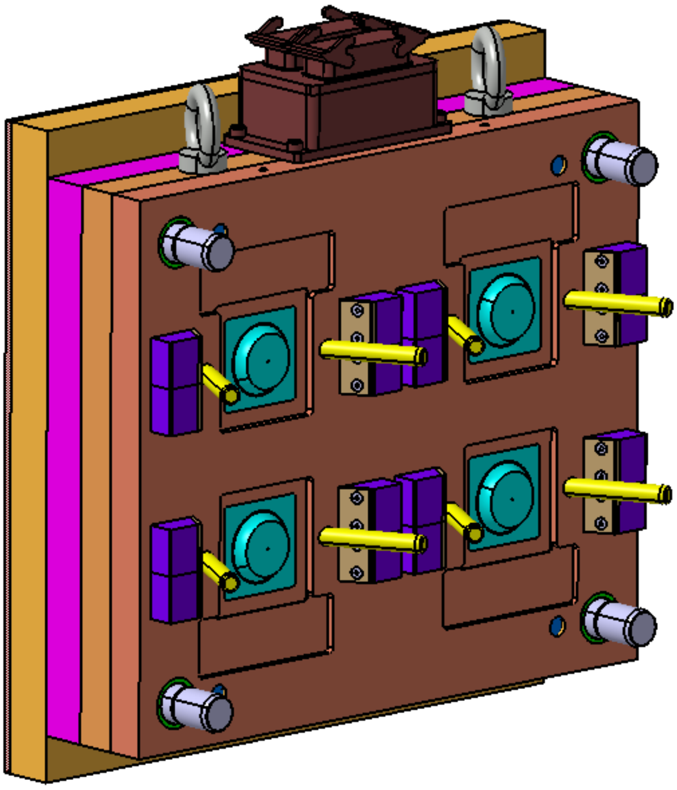
The shot weights given are approximate values only. They are depending on kind of resin to be processed, the flow path/wall thickness ratio as well as other processing parameter.

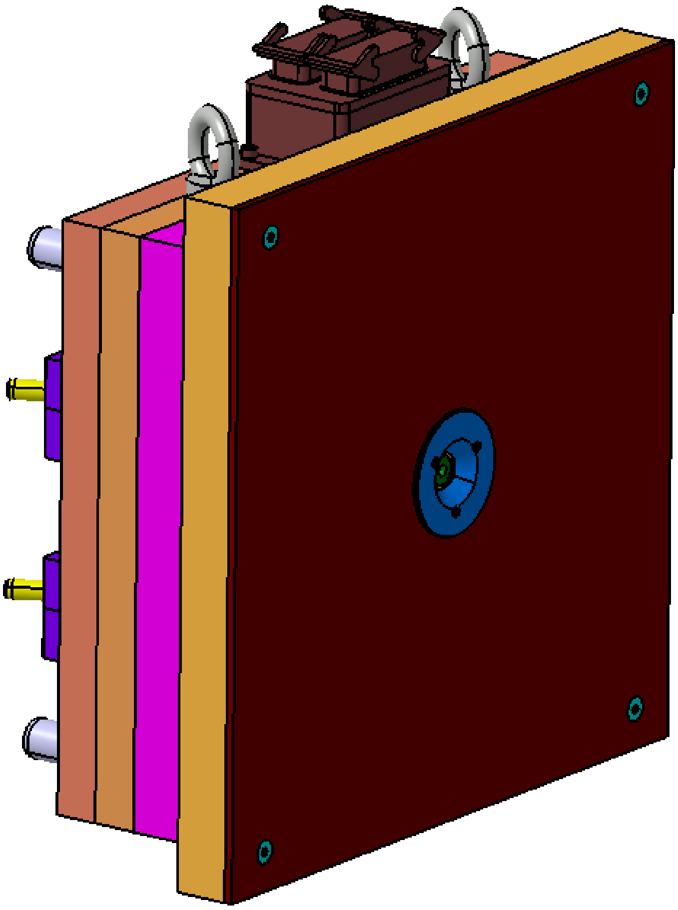
When used with hot-runner manifolds the shot weights have to be reduced. With relatively high injection pressure and difficult processing conditions the next larger nozzle size has to be used.

Les données concernant les charges d'injection ne sont que des valeurs indicatives et dépendant des masses à traiter, du rapport écoulement/épaisseur des parois, de même que d'autres paramètres de traitement.

Les indications concernant les charges doivent être diminuées en cas d'utilisation avec bloc à canal chaud. Pour des pressions relativement élevées, veuillez choisir le type de buse inmédiatement supérieur.

**AIV   INJECTION MOLD – RIGHT SIDE**





## AV INJECTION MOLD – LEFT SIDE