Injection Mold Design

Bc. Radomír Navrátil

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. Radomír Navrátil**
Osobní číslo:	**T12443**
Studijní program:	**N3909 Procesní inženýrství**
Studijní obor:	**Konstrukce technologických zařízení**
Forma studia:	**prezenční**
Téma práce:	**Konstrukce vstřikovací formy**

Zásady pro vypracování:

1. **Vypracujte literární rešerši na dané téma.**
2. **Proveďte konstrukci 3D modelu plastového dílu.**
3. **Navrhněte vstřikovací formu pro výrobu zadaného dílu.**
4. **Proveďte zhodnocení návrhu pomocí analýz.**

Rozsah diplomové práce:
Rozsah příloh:
Forma zpracování diplomové práce: tištěná/elektronická

Seznam odborné literatury:

dle zadání vedoucího DP

Vedoucí diplomové práce:	Ing. Michal Staněk, Ph.D. 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: RADOMÍR NAVRÁTIL

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 13.5.2015

....

---

[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).

ii 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

Tato diplomová práce se zabývá konstrukcí vstřikovací formy pro plastový díl.

V teoretické části je popsáno rozdělení plastů, jejich reologie, problematika vstřikování, pravidla volby vstřikovacího stroje, konstrukce vstřikovací formy a vstřikovaného dílce. Úkolem praktické části je nakreslit 3D model plastového výrobku, provést příslušné analýzy, zhotovit konstrukci vstřikovací formy. Při konstrukci je využíván program Catia V5R19, Autodesk Moldflow 2014 a normálie firmy HASCO.

Klíčová slova: plasty, forma, technologie vstřikování, catia

ABSTRACT

This master thesis is focused on the construction of injection mold proceess for plastic product.

In the theoretical part are presented type of plastics, their rheology, problems of injection molding, correct option of injecting machine, the design of injection mold and produced part.

The aim of the practical part is to design 3D model of plastic part, create correspond analysis and then to make the design of injection mold. The Catia V5R19 software, Autodesk Moldflow 2014 and the standard parts from company HASCO are used for whole design.

Keywords: plastic, mold, injection molding technology, catia

Poděkování, motto a čestné prohlášení, že odevzdaná verze bakalářské/diplomové práce a verze elektronická, nahraná do IS/STAG jsou totožné ve znění:

Prohlašuji, že odevzdaná verze bakalářské/diplomové práce a verze elektronická nahraná do IS/STAG jsou totožné.

Contents

INTRODUCTION

When the Hyatt brothers, John and Isaiah, built and patented the first injection moulding machine in 1872, it is doubtful if they could possibly have imagined the impact this invention would have on the world. At first, plastics were applied for consumer products, because they had not big function and requirements. The economical benefits were obvious and moreover products had esthetic design, low weight and does not corrode. Plastics were step by step used in wider field by discovering new types and modifications with higher strength, heat resistance and other special characteristics due to using improving additives and fillers.

Nowadays, the most popular plastic producing method is thermoplastic injection molding. This technology enable to produce parts without shape limitation for application in automotive and aircraft industry, even food packaging etc.

The injected parts are made by injection molding machine, which contains injection mold. The mold giving to part final shape, appearance and surface quality, then it should to ensure entire stability and keep desired characterization of product. The mold has to be resistant against high pressure, to have shape, to be dimensionally accurate and enable to smooth releasing from the cavity surface.

Design and mold production alone is very budget and time demand. One of the possibilities how to safe especially time of designing and producing is using typified components from specialized companies. Also CAD/CAM/CAE software is necessary, these systems enable to eliminate mistakes as early as possible. The result is less modifications in process and safe large amount from the budget. [2], [3], [4]

I. THEORETICAL PART

1 INJECTION MOLDING

The material used for molding, which is generally available as granulate or powder, is plasticized in an injection unit and injected into a clamped and closed mold under pressure, reached high values. The main advantage of injection molding is that it is a very effective and economical method of mass production. Injected parts with fine tolerances can be produced in one step, usually completely automatically. In general after-processing is not necessary. It is also possible to integrate different functions into one part to avoid the formation of different components that would be more expensive. [2], [4]

To guarantee a high quality in the injection molded parts the following points have to be considered: [2]

• The material has to be plasticized and injected carefully to avoid negative effects on the material properties.

• The process settings (such as pressures and temperatures) concerning the machine and mold have to remain constant with regard to time and space.

• Basic parts of an injection molding machine.

1.1 Injection molding process

In injection molding the mold and the plasticizing area are separated from each other. The mold on the other hand, is kept cold enough for demolding of the thermoplastic injection molded part or warm enough for crosslinking of the thermosets. The plasticized material is injected into the clamped closed mold. Completely automated production is possible if the mold is installed with a vertical parting line. This enables the parts to fall down out of the mold after demolding. Injection molding machines are typically used for the processing of thermoplastics. There are two types of injection unit available: a piston injection unit and a screw piston injection unit (reciprocating). Nowadays the reciprocating screw method is the most common. For the processing of thermosets only screw piston machines can be used. This is because without the screw, the dwell time would be too long and the risk of early crosslinking would be too high. [2]

Molding Process Windows

Process windows are the ranges of processing conditions, such as melt temperature, pressure, and shear rate, within which a specific plastic can be fabricated with acceptable or optimum properties by a particular fabricating process. The window for a specific plastic part can vary significantly if changes are made in its design and the fabricating equipment used. To mold parts at the lowest cycle time, the molding machine would be set at the lowest temperature. If due to machine and plastic variables rejects develop, then one moves the machine controls to achieve higher temperatures and/or lower pressures and thus restore quality. This is a simplified approach to producing quality parts, since only two variables are being controlled. [1]

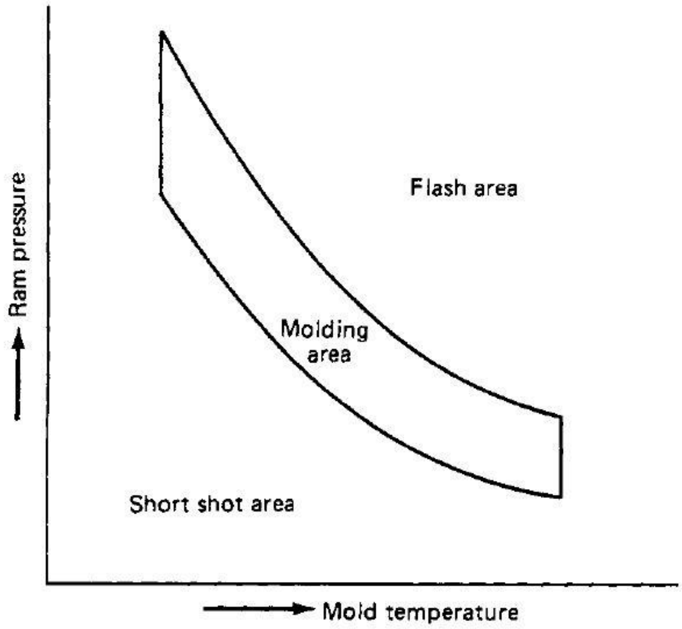

Fig. 1 Molding area diagram [1]

1.2 Molding cycle

Injection molding cycle has certain order: [2]

- Mold closing,
- Injection,
- Packing,
- Cooling and plastication,
- Mold opening

Thermoplastics are injected into relatively cold mold. The temperature of the mold must be sufficiently below the melting temperature of the material for it to solidify. This is because solidification is a physical process.

Thermosets and classical elastomers are injected into a hot mold to make the crosslinking of the material possible. Crosslinking is a chemical process.

In the screw piston injection unit, the material is dosed and plasticized simultaneously. The material is kneaded thoroughly by a rotating, axially movable screw. It is heated up to the processing temperature by the heat transfer of the hot cylinder wall and by friction between material and wall of the cylinder. The material is transported by the screw to the screw tip. As the nozzle opening (cylinder opening) is still closed, the screw moves backwards. As soon as enough material is in the area in front of the screw tip, the screw is stopped. This is controlled by a limit switch or by a stroke measure device. This is the end of the plasticizing and dosage stage. In screw piston injection machines, the material is plasticized more homogeneously and has to stand less thermal stress than in piston injection units, as the plasticizing itself happens just shortly after the injection.
A single injection molding cycle can be broken down into three distinct stages: plasticization, mold filling and cooling with solidification. [2]

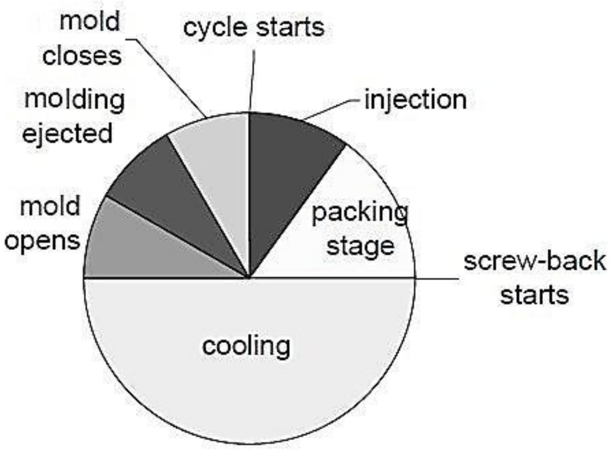

Fig. 2 Breakdown of an injection molding cycle [2]

Plasticization

The polymer flow rate is governed by the material processing conditions of the plasticization stage: a combination of material rheology, barrel temperature and shear, back pressure and screw speed. The main aim is to produce a homogeneous melt for the next stage where the material enters the mold. Molding parameters which control the plasticization stage are cylinder temperature, screw back temperature and back pressure. [2]

Filling

Here the injection unit delivers a pre-set certain amount of molten polymer to the mold tool. Mold filling parameters are of great importance to the end result especially when considering factors such as warpage (orientation effects) and surface finish (skin formation).

Filling dynamics are also thought to be the major factor in affecting the levels of residual stress. It is important that injection speeds are reproducible as slight changes can cause variations in the end product. Injection speeds which are too high can cause jetting and degradation and thus affect mechanical properties. A low speed may cause an increase in pressure requirements due to a thicker frozen layer and short shots which lead to incomplete filling of the mold.

The important thing is that the speeds are reproducible from one shot to the next. Important molding parameters for filling are the injection speed and injection pressure.[2]

Packing and Solidification

Certain amount of the material is in the tool, filling must be completed by tool packing, the part is cooled and finally ejected. The purpose of the packing stage is to add extra material to compensate shrinkage caused by the increasing density of the solidifying polymer. If the additional polymer were not injected the component would shrink and warp due to non uniform cooling. [2]

2 MATERIALS

We can divided polymers into many groups according their chemical, physical characteristics or their usage.

2.1 Structure of Polymers

The word polymer derives from the Greek word *poli*, which means many and the word *meros*, which means parts. This is because polymers are made up of a number of smaller repeated units called monomers. The simplest and most commonly used monomer is ethylene. Chemically it consists of two carbon atoms (C) and four hydrogen atoms (H). It can be represented in the two ways. The lines in this diagram represent bonds that exist between the atoms to form a molecule. [2]

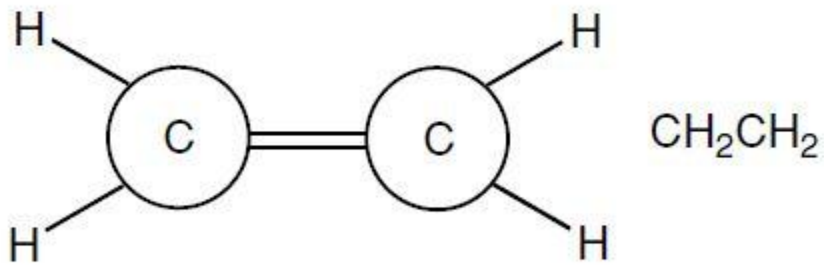

Fig. 3 An ethylene molecule [2]

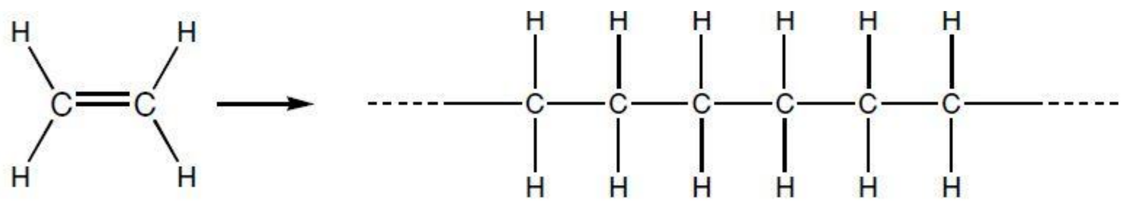

Fig. 4 Polyethylene molecule [2]

It is the existence of the double bond between the carbon atoms in ethylene, which allows the creation of polyethylene. This happens when the monomers are combined by a process called polymerization to form. A chain of useful polymer may consist of 200-2000 monomers joined together. This particular type of polymerization is called addition polymerization. [2]

2.2 Plastics

Plastics are made up of polymers and other materials that are added to increase the functionality. The actual polymer content within a plastic can vary widely from less than 20% to nearly 100%. There is a great range of materials availabl. Plastics can be subdivided into three main categories, thermoplastics, thermosets and elastomers. This distinction is based on both the molecular structure and the processing routes that can be applied. These three classes of materials will now be introduced. [2]

2.2.1 Thermoplastic

These materials melt and flow when heated and solidify as they cool. On subsequent reheating they regain the ability to flow. This means they can be reprocessed and hence recycled by re-melting them.

Thermoplastics are used to make consumer items such as drinks containers, carrier bags and buckets. When thermoplastics solidify they can take one of two molecular structures: an amorphous structure or a semi-crystalline structure.

When semi-crystalline materials are cooled the molecular structure tends to become highly ordered and crystals are formed. The size of these crystalline regions varies according to both the structures of the chains themselves and the cooling rate. These materials displays sharp melting points unlike amorphous materials that soften. Semi-crystalline materials also tend to shrink more due to this molecular rearrangement. This shrinkage will be more in the direction of flow due to the molecular realignment caused by the process of injection molding. [2]

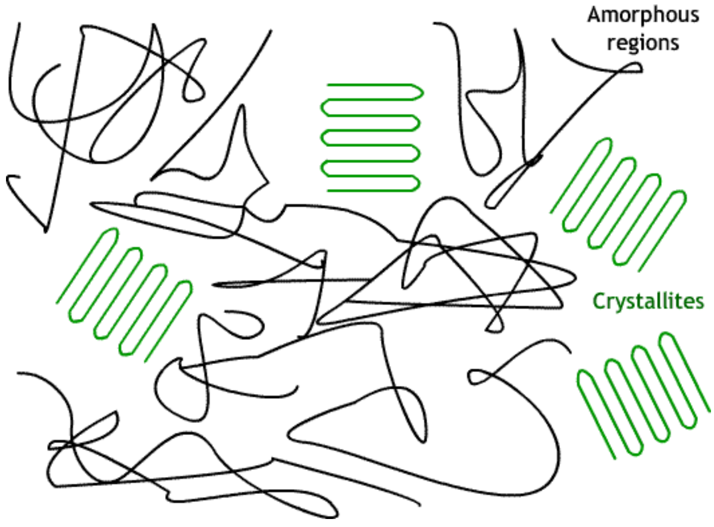

Fig. 5 Semicrystalic polymer structure [17]

2.2.2 Thermosets

Thermoset injection molding compounds change their structure when injected. Before injection molding, they still consist of thread-shaped molecules similar to thermoplastics. However, during a process termed curing the molecules crosslink forming a highly dense network of bonds. This makes the material stiff and brittle and the thermoset molded parts can then no longer be melted. Thermoset materials decompose before they can melt, therefore, they cannot be reprocessed in the same way as thermoplastics. The differences in the arrangement of molecules between thermoplastics and thermosets. Thermosets are often used where their strength and durability can be utilized. [2]

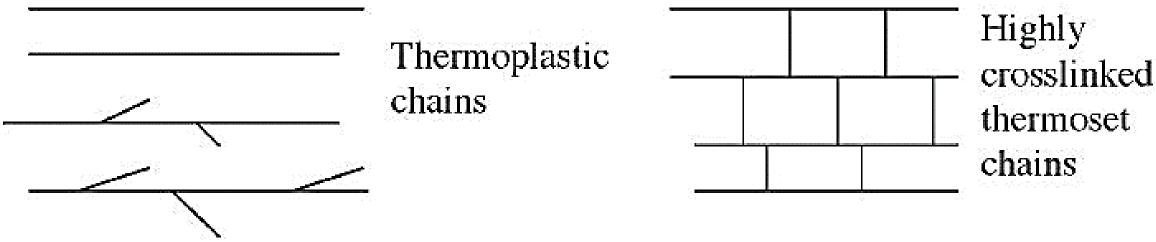

Fig. 6 Arrangements of thermoplastic and thermoset molecular chains [2]

Once thermoset plastics have crosslinked they can no longer be melted. Thermoset materials can thus be deformed only elastically and, in contrast, cannot be deformed plastically. The rigidity of the thermoset materials depends on how narrow or wide the spaces between the network of crosslinking of the molecules are. A material with a wide gap can be deformed elastically to a large degree. Such thermosets are also known as elastomers. [2]

2.2.3 Elastomers

Elastomers are highly elastic polymers, which can be significantly deformed for common conditions by small force without violation. This deformation is mostly refundable. In the molecular chains have reactive places allowing chemical curing reaction. Curing runs just in presence of curing agents, which is mostly sulfur. During vulcanization is plastic rubber changed into the cured rubber. [4]

2.3 Rheology of polymers

Rheology deals with deformation and flow and examines the relationship between stress, strain and viscosity. Most rheological measurements measure quantities related to simple shear such as shear viscosity and normal stress differences. Material melt flows can be split into three categories, each behaving differently under the influence of shear as shown in Figure 7 dilatent (shear thickening), Newtonian and Non-Newtonian pseudoplastic (shear thinning) behave. [2]

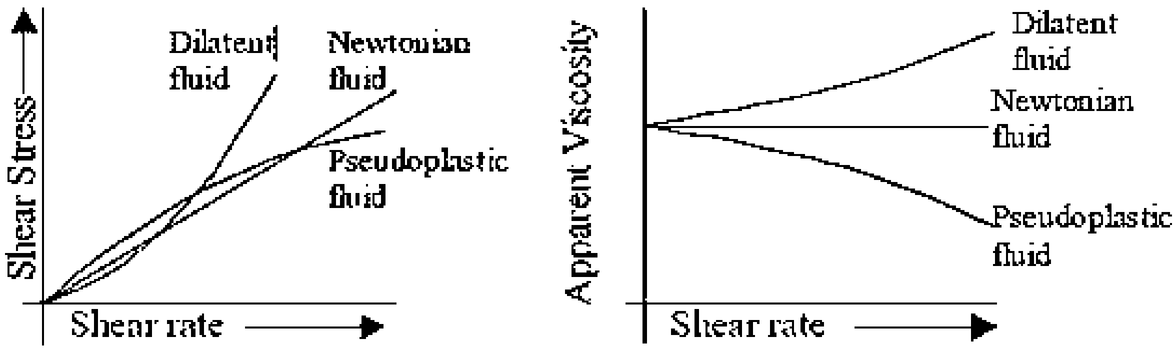

Fig. 7 Typical stress/shear relationships (left) and apparent viscosity/shear curves (right) for dilatent, Newtonian and pseudoplastic fluids [2]

High viscosity melts flowing into low viscosity melts will force the low viscosity material in front. Low viscosity melts flowing into high viscosity melts will jet through areas of least resistance, giving an effect termed ‘melt fingering’. This type of effect can also be seen in gas assisted injection molding where the core component, gas, has an effective viscosity of zero. The injected gas follows the path of least resistance, making channels in the hottest, thickest and least viscous parts of the melt stream. Breakthrough of the core component in co-injection molding can cause unwanted surface defects, whereby the core material can be visible in the corners of the moldings. Therefore, a high percentage of co-injection studies have investigated the relationship between the relative viscosities of the materials used within the process and the resultant skin/core distribution to reinforce these findings. [1],[2]

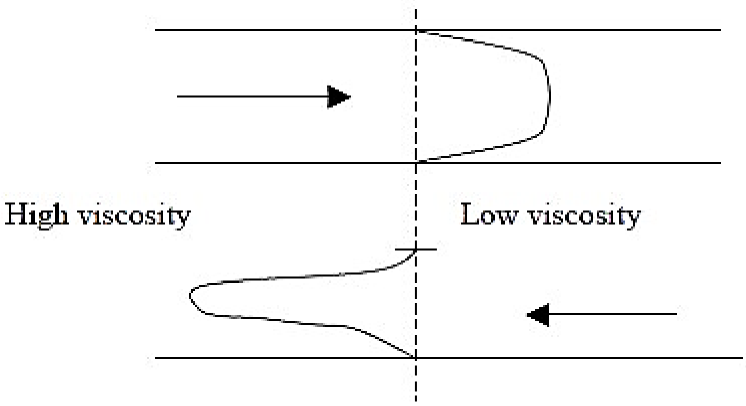

Fig. 8 Interface behave of melts of different viscosity [2]

Linearized shrinkage

The linearized shrinkage is developed as the melt flows in the cavity. These flow-induced effects can be attributed to shear forces and extensional forces induced to the polymer during filling and packing.

The filling stage is characterized by a so-called fountain flow. A shear field is developed due to the variations in the velocity field and the shear field makes the molecules oriented in the direction of the main strain direction. If there is time, and if the polymer would be kept in its molten state, this orientation is recovered. When the melt closest to the mold solidifies, the molecules keep their orientation in the flow direction as well as their molecular elongation. They will therefore have less frozen in orientation and will also have less tendency to shrink relative to flow direction. This gradient of oriented molecules causes the skin to be in compression while the core is in tension and consequently a warpage effect arises. The variations of the cooling rate, the direction and the velocity of flow, part thicknesses, etc., over the part geometry make this shrinkage effect complex. The balance of the strains and their directions results in so called “residual stress” which in its turn can contribute to warpage. [10]

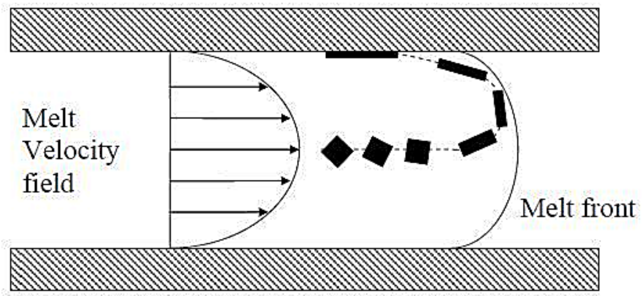

Fig. 9 The fountain flow near the melt front. The deformation and orientation of a material element is depicted as it gets closer to the melt front [10]

Volumetric shrinkage

All polymers exhibit shrinkage when cooled from the melting temperature to the solid state. The two classes of thermoplastics, amorphous and semi-crystalline, show a linear dependency of the specific volume on the temperature in the melted state. In the solid state, the specific volume of the semi-crystalline polymers decreases exponentially whereas the amorphous polymer keeps a linear dependency, although with a different slope. This can be visualized by the pvT diagram, where the specific volume is plotted as a function of the temperature for different pressures. [10]

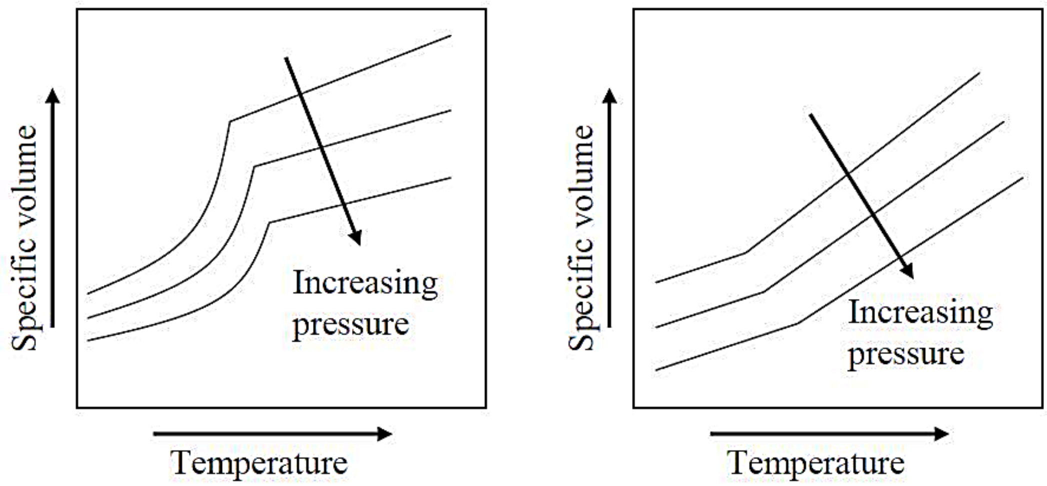

Fig. 10 Example of pvT-diagram for semi-crystalline polymers (left) and amophous polymers (right). The specific volume is plotted as a function of the temperature and for different pressures. [10]

Shrinkage and injection molding process

Different area shrinkage refers to differences in thicknesses, in packing pressure or in mold wall temperatures. Thinner regions solidify more rapidly than thicker regions, if the cooling rate is constant. The frozen layer will represent a greater percentage of the thickness in the thin section than in the thick section. Consequently the thinner regions will solidify first at a higher pressure which results in lower shrinkage. Different mold wall temperatures will give the same effect: cooler areas will extract heat from the hotter areas, and they will solidify first at a higher pressure causing a lower shrinkage than for the hotter area.

Moreover, warpage is affected by variations in the process settings, that is, other variations than changing the material and the mold design. The main relationship between the different molding parameters and the shrinkage. [10]

3 INJECTION MOLDING MACHINE

Injection molding machine is assembled from:

- Injection unit
- Machine base with hydraulics
- Control unit and control cabinet
- Clamping unit with the mold

3.1 Injection Units

The first aim of the plastication stage is to produce a homogeneous melt for the next stage where the material enters the mold. A second important function of the injection unit is the actual injection into the mold. Here, it is important that injection speeds are reproducible as slight changes can cause variations in the end product.

Once the material has passed through the hopper, it enters the injection barrel. The barrel will consist of a number of separately controlled heating zones. The heat is generated from conduction of heat from the cylinder and also the heat generated by the shearing action of the screw on the material feedstock. Polymers are not particularly good conductors of heat; therefore the polymer thickness in any section of the screw tends to be kept low. The amount of shear is material dependent, mainly viscosity related and controlled by the machine screw back and back pressure. [2]

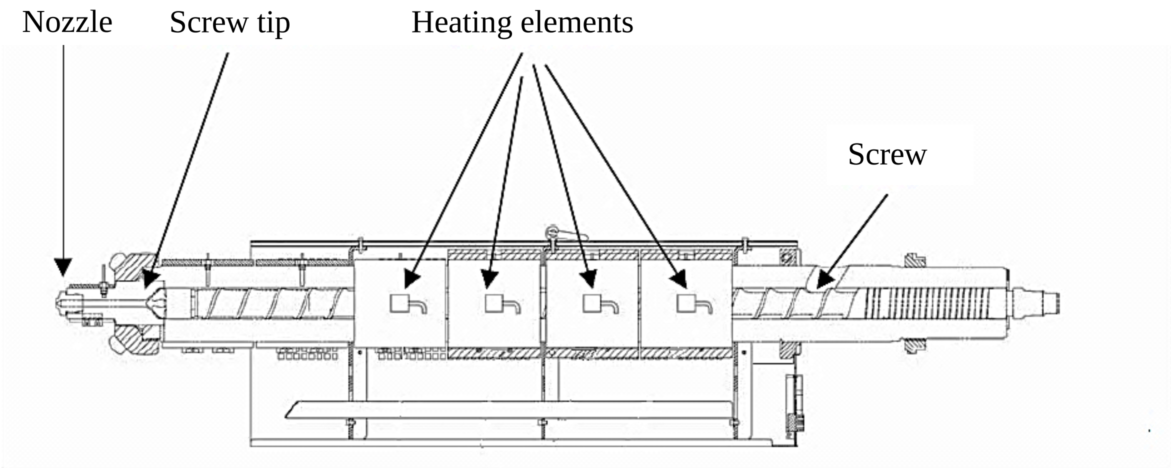

Fig. 11 Injection unit [2]

Shot Capacity

The shot capacity is the full amount as a weight or volume of material injected during molding from the screw. This is usually given as a shot capacity for polystyrene, and will vary with material. The shot size is the amount of material required to fully fill a molding tool. [2]

Plasticizing Capacity

This is the maximum rate at which the injection unit can deliver polymer melt. In extrusion this is a continuous process. However, it should be remembered that injection is an intermittent process, therefore the plasticizing rate will be lower. To calculate the melting rate consideration should be given to the overall cycle time.

The effectiveness of plastication depends on the shot size, cylinder capacity, screw design, screw speed and heater band power. It will also vary from material to material. [2]

3.2 Clamping Unit

The clamping units of injection machines are described and rated separately to the injection unit. The clamping units are required to enable mounting and holding of the two mold halves. They must also provide sufficient clamping force during injection and cooling to enable effective molding. The mold halves must also open and close accurately and smoothly to enable part injection and begin the next process cycle. Injection machines can be run by hydraulics, a hydraulic and toggle combination or by electrical power. The clamping units on injection molding machines use hydraulic force.

Figure 13 shows a clamping unit. The stationary platen is attached to the machine with four tie rods connecting it to the movable platen. Direct hydraulic clamping system The clamp ram moves the moving platen until it reaches the stationary platen and the pressure begins to build up. The ejectors are fitted onto the moving platen and can be activated once the tool is opened and the moving platen retracted. [2]

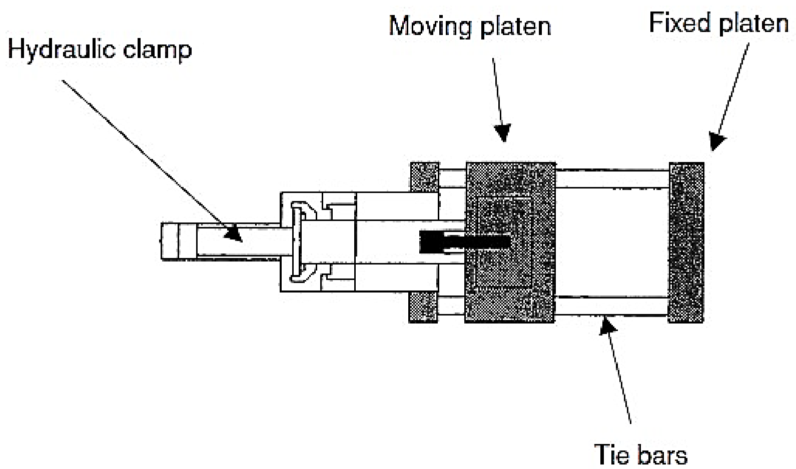

Fig. 12 Clamping unit [2]

3.2.1 Selection of the Clamping Unit

To select a clamping unit consideration must be given to the following factors: [1]

• Injection mold size

- dimensions
- centering
- ejector coupling
- mold weight

• Projected surface area (amorphous and semi-crystalline materials require different clamp forces)

- table
- amorphous cm^2 x 4 = kN
- semi-crystalline cm^2 x 6-7 = kN

• Select higher clamping force with split fallower molds

- with thin walled parts and thermoset materials.

The type of clamp and the clamping force are the main specifications of a clamping unit. However, there are other design features which also need consideration. These are: [1]

- Maximum daylight
- Space between tie bars
- Clamp stroke
- Clamp speed
- Knockout stroke.

4 INJECTION MOLD

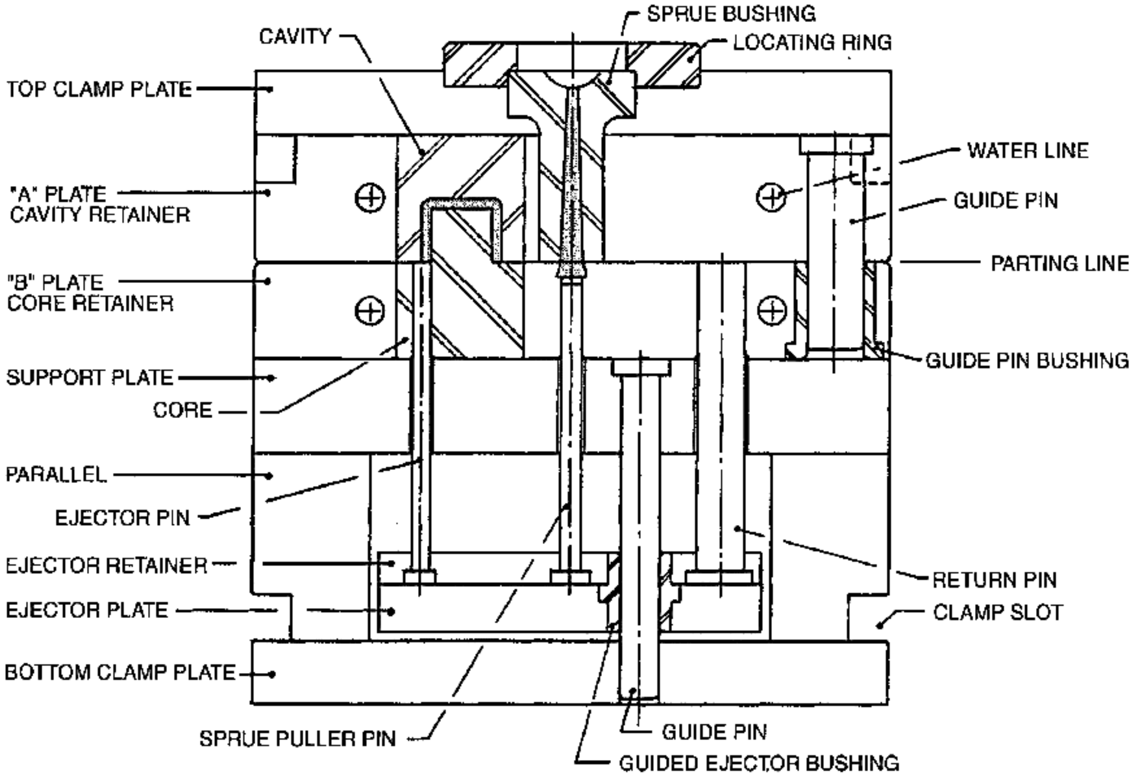

Fig. 13 Injection mold design terminology [1]

4.1 Types of Molds

Injection mold designs differ depending on the type of material and component being molded. Mold tool design and component design are equally important considerations for success. Component design is beyond the scope of this book but the various tooling, gating, temperature control and ejection systems that make up the mold tool will be considered here. After parts are injection molded they must be ejected. A variety of mechanisms can be employed such as ejector pins, sleeves, plates or rings. The design standard for injection mold tools is the two-plate design. [2]

4.2 Feed System

The feed system accommodates the molten polymer coming from the barrel and guides it into the mold cavity. Its configuration, dimensions and connection with the molding greatly affect the mold filling process and subsequently, the quality of the product. A design that is based primarily on economic viewpoints, (rapid solidification and short cycles) is mostly incompatible with quality demands. The two main areas that need to be considered are the runner system and the gate. [2]

When designing runner systems the three primary considerations are as follows: [2]

1. The shape of the runner
2. The runner layout
3. The runner dimensions.

More specific demands of the runner design could include the following points: [2]

1. The cavity should fill with a minimum of weld lines
2. The cavities fill at the same time
3. Restrictions to flow should be as low as possible
4. Share of the total shot weight should be as low as possible
5. Should be easily demolded
6. Appearance of the product should be unaffected
7. Length as short as is technically feasible to reduce losses in temperature and pressure and keep scrap to a minimum.
8. Cross-section as large as required to allow a longer or equal freezing time to that of the component (to allow effective packing of the part).

Cold Runner

Runner systems are machined straight into the mold plates, their temperature therefore being that of the mold temperature, (i.e., usually 20 °C to 120 °C). The material passes through the runner to the cavities which are filled and packed by holding pressure and then the molten material in the runner freezes with the rest of the component during cooling. [2]

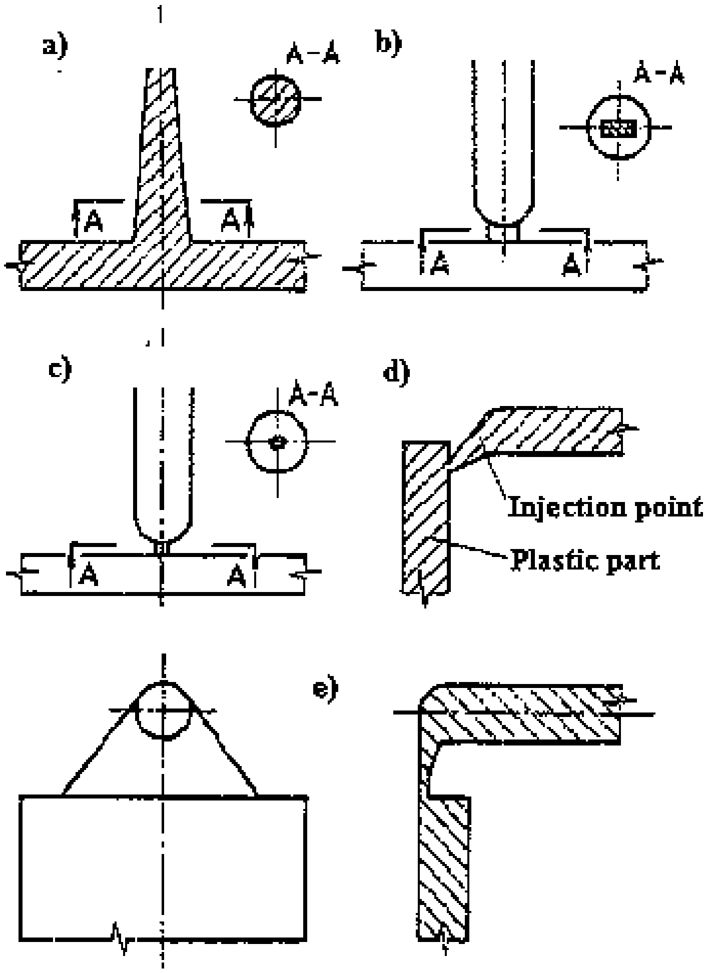

Fig. 14 Examples of injection points [11]

a) Sprue gate, b) Restricted gate c) Circular restricted gate d) Submarine gate e) Fan gate

Hot Runners

Hot-runner systems are employed for so-called ‘‘runnerless’’ injection molding of parts from thermoplastic resins. It is also advantageous to use partial hot-runner systems, i.e. those with secondary runners. With proper design, lower pressure losses can be achieved in hot-runner systems than in comparable molds with solidifying runner systems. Thus, it is possible to produce extremely large parts such as automobile bumpers with suitable hotrunner systems. Economical production of parts in stack molds has become possible only through the use of hot-runner technology. By completely eliminating the solidifying secondary runners, the injection capacity of an injection molding machine can be better utilized. This may also result in a reduction in the filling time, which can lead to a reduction in cycle time. In principle, however, hot-runner systems do not reduce the cycle time. The design principles employed for various hotrunner systems can differ considerably. This applies to both the hot- runner manifold and the hot-runner nozzles, the type and design of which can have a considerable influence on the properties of a molded part. [16]

The use of the hot runner technique for feeding multi-impression and large area moldings is now firmly established. The advantages of hot runner moldings are as follows: [2],[6]

• Melt enters the cavities in a more controlled condition than with a sprue and runner system, as the temperature control in the hot runner is adjustable to finer limits

• A possible reduction in post-molding finishing operations to remove large sprue gate witness marks

• The elimination of cold sprues and runners in multi-impression molds which would normally be scrapped or reworked

• Hot runners enable single impression, large area moldings to be edge-gated, whilst keeping the molding in the center of the machine platen

• Effective increase in the shot capacity of the machine as, once the hot runner is filled, the injection capacity can be fully concentrated into the cavities

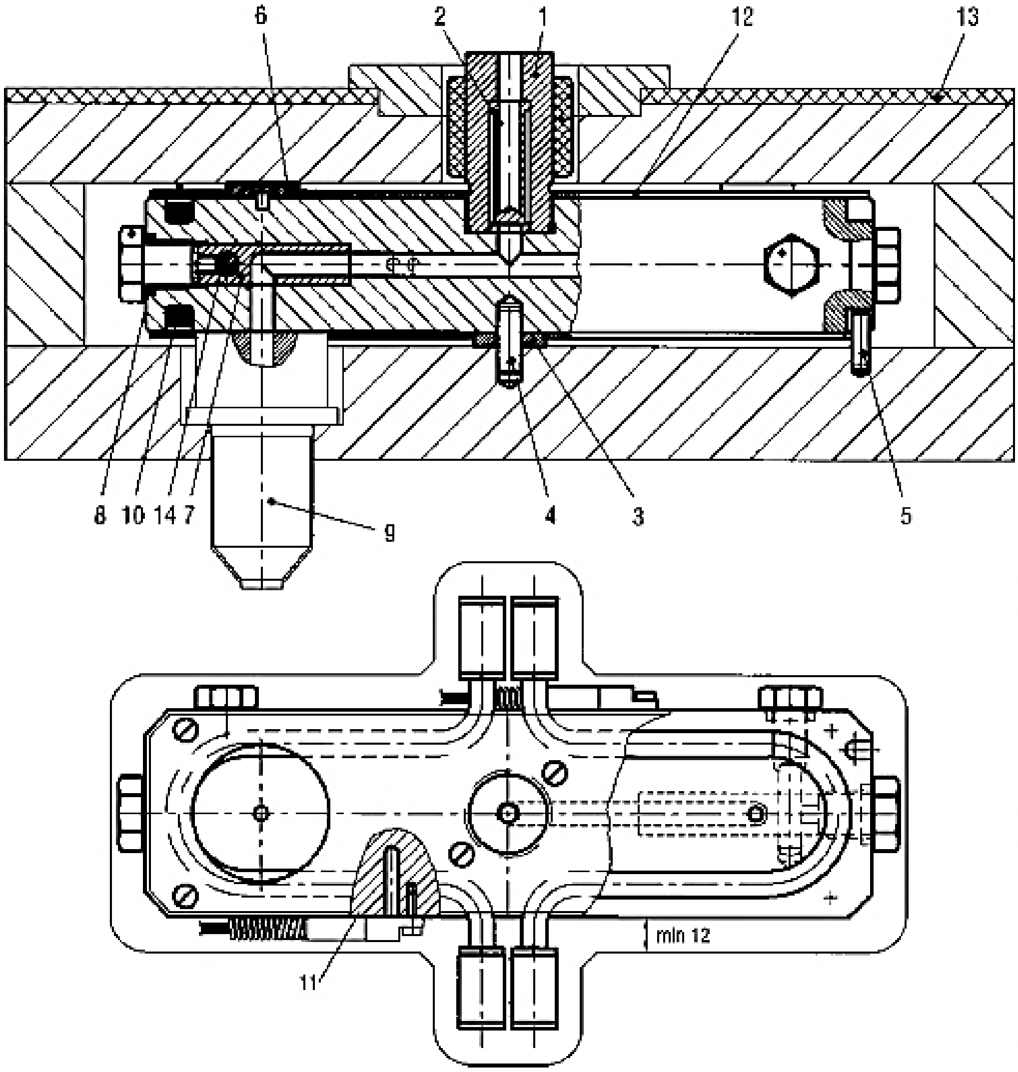

Fig. 15 Hot runner system design [18]

1 – Melt inlet, 2 – Filter insert, 3 –Spacing underlay, 4,5 – Locating pin, 6 – Spacing underlay, 7 – Plug, 8 – Sealent ring, 9 – Hot runner nozzle, 10 – Heating, 11 – Thermocouple, 12 – Plate, 13 – Insulating sheet, 14 - Pin

4.3 Release of undercuts

Release of undercuts generally requires additional design features in the mold such as opening of the mold along several planes, for instance. Additional release surfaces can be provided by slides and split cavities. Molds with slides release external undercuts with the aid of :

- angle pins,
- cams,
- hydraulically or pneumatically actuated mechanisms.

Angle Pins

Angle pins are also used to pull out core pins. Angle pins use the normal movement of the molding machine to remove the core pin as the mold opens at the parting line. Motorized racks and pinions are used to unscrew threaded cores. Figure 16a shows the use of an angle pin with the mold in the closed position. The lock angle is greater than the angle of the angle pin butts against the back surface of the slide. This positive locking device is used to ensure that the core pin is in the proper position and keeps the pin from retracting as the material is injected into the mold. As the mold opens, the piece part stays with the movable portion of the mold. [1]

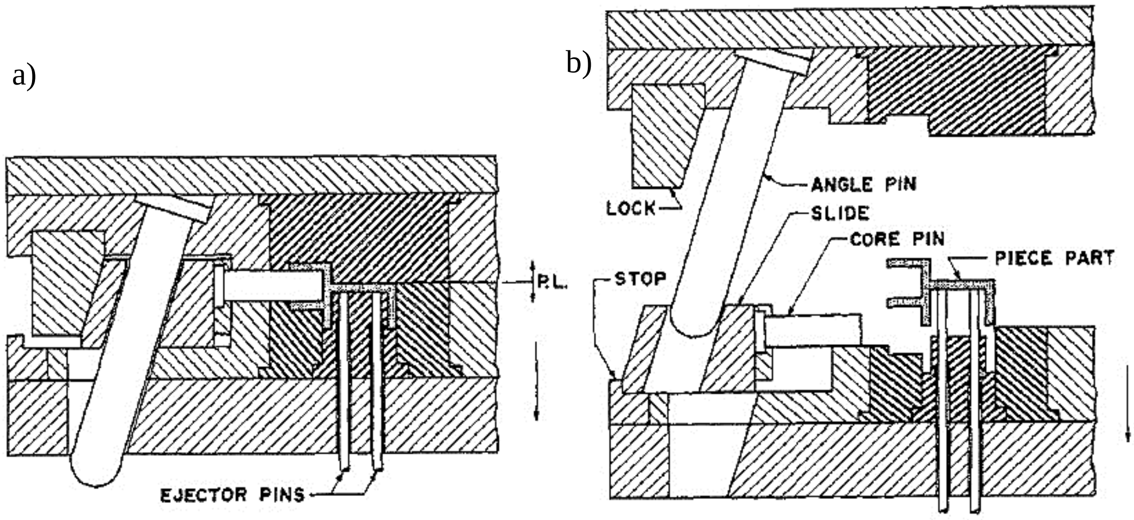

Fig. 16 Release of part by angle pin [1]

4.4 Ejection system

The conventional mold ejector system moves between the clamp plate and support plate in a space provided. The ejector plate and pin plate are guided by return pins that ride on bearing surfaces in the support plate and core plate. The ejector plate carrying the ejector pins must move freely in the mold. The number and location of the ejector pins are determined by the size and shape of the piece part. [1]

There are various types of ejectors used to release molded parts: [16]

- ejector pins,
- ejector sleeves,
- stripper plates, stripper bars, stripper rings,
- slides and lifters,
- air ejectors,

- disc or valve ejectors, etc.

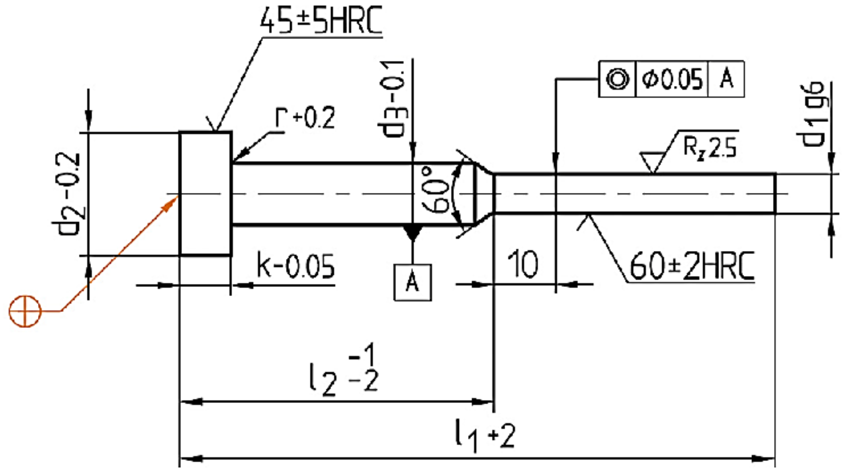

Fig. 17 Prismatic ejector pin [15]

The type of ejector depends on the shape of the molded part. The pressure on the surface of the section of the molded part to be ejected should be as low as possible in order to avoid deformation. Profiled ejector pins should be prevented from turning. [16]

5 COOLING SYSTEM

With thermoplastics the main purpose of the mold system is to minimize both the cycle time and thermal differences in mold part cooling. Mold cooling is therefore essential for both cost saving and quality control. Uniform cooling improves product quality by preventing differential shrinkage, high residual stress and mold release problems.

Therefore in designing an injection mold tool, the size and layout of the cooling channel is an important part. [2]

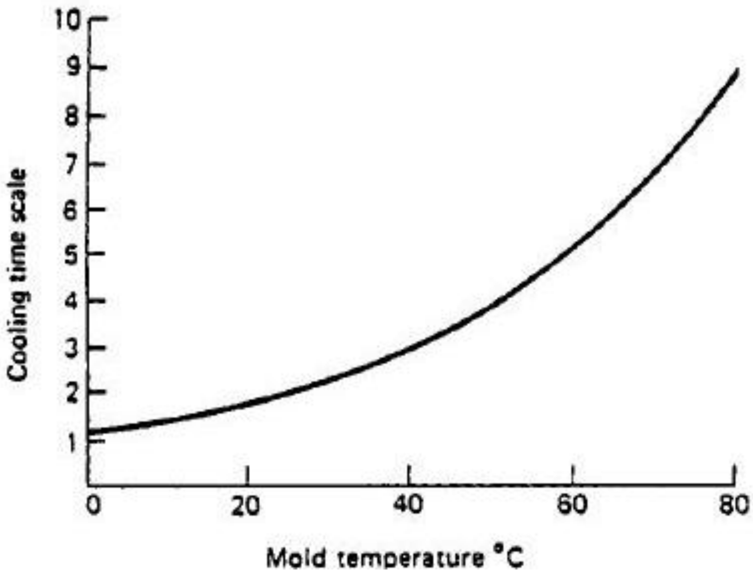

Fig. 18 Effect of mold temperature on cooling time [1]

Controlled cooling channels are essential in a mold for proper process and design. The cooling medium must be in turbulent flow, rather than laminar flow, in order to transfer heat out of the molded part at an adequate rate. The coolant is usually water, but can be any liquid or gas (such as air) that can absorb heat and transfer it efficiently away from its source. Coolants are used in molds, chillers, etc. Water is one of the most effective and low-cost coolants. It may be mixed with an antifreeze such as ethylene glycol for operation below the freezing point. [1]

In principle, the molding may be released from the mold as soon as its outer layer is sufficiently rigid, at a temperature called the mold release temperature. The inside of the molding will often still be considerably hotter than the outer part. Minimum cooling time required to reach mold release temperature is governed by: [1]

- Wall thickness of the molding
- Difference between polymer and mold temperatures

- Difference between mold release temperature of the article and mold temperature

The minimum cooling time may be estimated from the following equation: [1]

$$S = \frac{-t^2}{2\pi\alpha} \log_e \left[\frac{\pi}{4} \frac{(T_r - T_m)}{(T_c - T_m)} \right] \quad (1)$$

Where: *S* = minimum cooling time (sec)

t = thickness of molding (cm)

(Y = thermal diffusivity of material

T, = ejection temperature of molding

(sq cm/sec) (often the heat distortion temperature is used)

T_m = mold temperature (ºC)
T_c = cylinder temperature (ºC)

Table 1 Heat transfer coefficients [11]

Material	λ(W/mK)
Silver	410
Aluminium	204
Copper	395
$CuBe_2$	113
Steel soft	44
Steel chrome	40
Steel nickel	26
Plastic	0,2-1,2
Air	0,04
Water	0,19
Oil	0,16

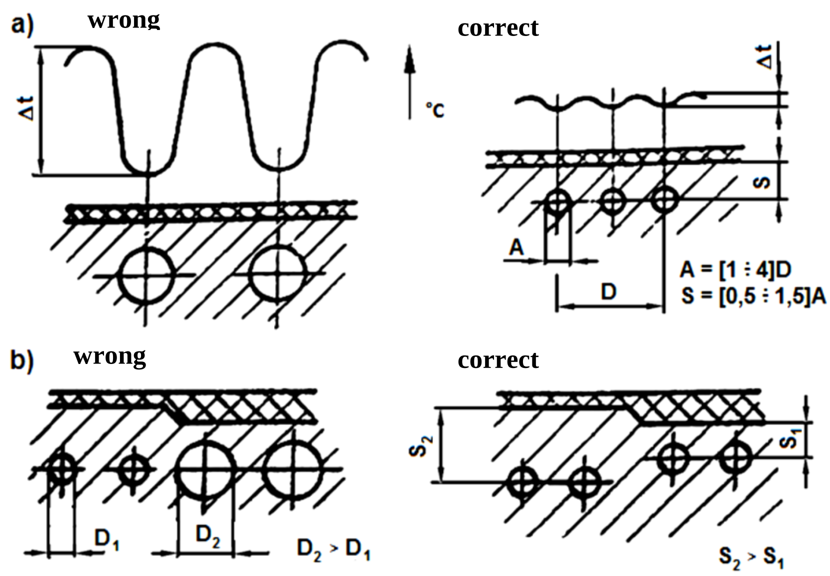

Fig. 19 Cooling influence of position cooling channels [5]

a) *Channel position influence on a cooling of cavity*
b) *Product cooling with different thickness*

5.1 Cooling media

As a cooling media we call agents, which are managed influence keeping mold on optimal working temperature. Medium choice depends mainly on mold design, cooling requirements and production technology. [4]

There are divided on two elementary types: [4]

- Active - directly are resourced of temperature in the mold – according to mold design keep mold cavity surface in the require temperature.
- Passive – influencing mold heating regime by physics characteristics. It can be conductive or insulating materials, heat pipe etc.

The most using active medium is water, which flows in channels created in mold. Heat transfer efficiency is given by size and quality of channel surface, then by flow type and temperature difference of medium. The flow should be turbulent (Re >> 2300) and medium temperature difference at inlet and outlet from 3 till 5ºC.

Tempering by electrical parts is used especially when in needed to heat mold for higher temperature level, when losses to surroundings are higher than heat delivered by injected material. [4]

Table 2 Active temperating media [4]

TYPE	PROS	CONS
water	good heat transfer, low viscosity, cheap price, ecological harmlessness	usable up to 90ºC *) corrosion creations **)
oil	tempering over 100ºC	worse heat transfer
glycol	lower incidence of corrosion and clogging	environmentally unfriendly, aging

Explanatory notes

*) in high pressure circuit possibly to use water even in higher temperatures

**) possibly to reduce by water additives

6 MULTI-COMPONENT INJECTION MOLDING

Multi-shot moulding has been around for over thirty years and is used as a method of placing materials either side by side, one on top of the other within an overlap, or superimposition of one shot onto another. To do this special tooling and machinery is required. Common examples of such mouldings are keypads with the numbers made of one colour and the letters molded-in using another colour. The advantage of this technique being the elimination of the printing processes which would otherwise be required to mark the keypads. Multi-colour automotive taillights are also made by this technology. Another common application of this technology is to combine hard and soft materials to produce a 'soft feel' component. Handles such as on doors or toothbrushes are common products benefiting from this technology. [2]

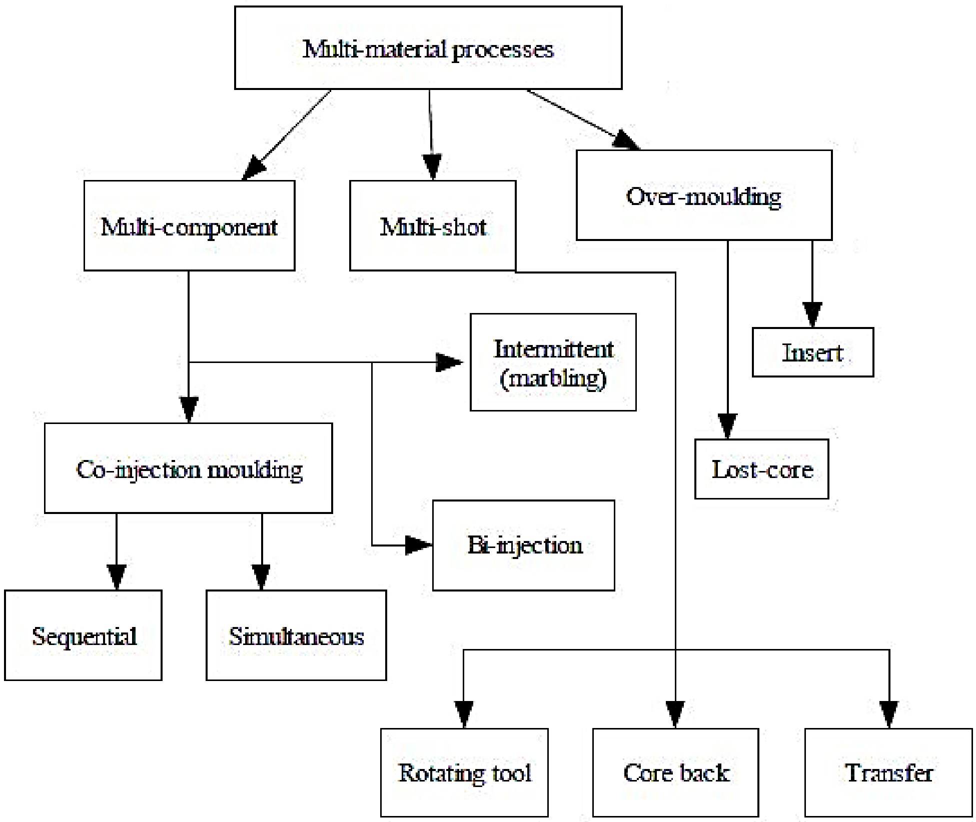

Fig. 20 Multi material injection molding technologies [2]

6.1 Machine Technology

Multiple injection units can be arranged around the clamping units as combinations of horizontal and vertical units in piggy back or right angle configurations. [2]

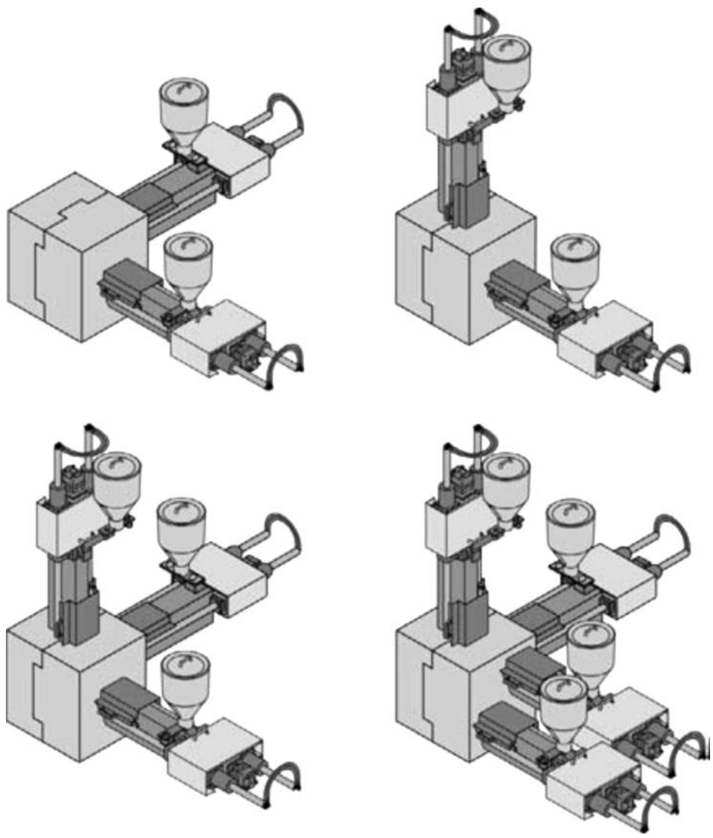

Fig. 21 Possible arrangements of injection units [7]

Using a vertical injection unit can save space and hence is the most used position for multi-shot moulding at the split-line. For mold changing, the units can be slid towards the nozzle. Where the vertical position cannot be used, perhaps because of lack of factory height, the second unit can be positioned at right angles. The position of the unit can be adjusted both horizontally and vertically, although the former is available as an option rather than as standard, by some manufacturers. Again as in the vertical unit, it can be moved to the nozzle-side to change the mold. [2]

6.1.1 Core Back Molding

Core back is a tooling controlled process. Core back molding, thought of simplistically, is one tool taking multiple shots within a single machine cycle. It allows different areas of the tooling to be opened or closed to specific material feeds. This is achieved through the use of moving slides or inserts. [2]

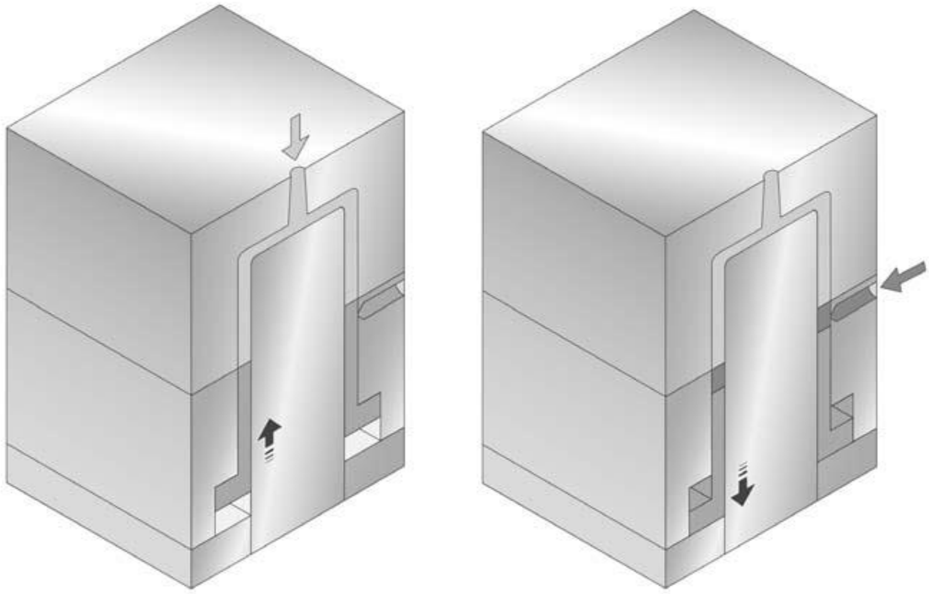

Fig. 22 Core back molding [7]

two-component process may consist of the following stages: [2]

1. The first material is injected into the cavity
2. Using a core puller to activate a slide seal, a further area of cavity is revealed
3. The second material is injected into the cavity
4. The completed multi-shot component is ejected.

6.1.2 Rotating Tool

In this method, the mould rotates through 180° for a two-shot part or 120° for a three-shot part. Rotational capability can be machine or tool based. There can be an integral rotary capability designed into the tool or the machine can be equipped with a rotary attachment to the moving platen. The choice usually comes down to economics. If rotational capability is to be used regularly it is cheaper to have it on the machine, than to continually buy more expensive tooling. A rotary platen must have an accurate indexing device to control the rotation and the stroke needs to be both fast and cushioned to prevent damage. The platen must also have the facility to mount ejector pins. [2]

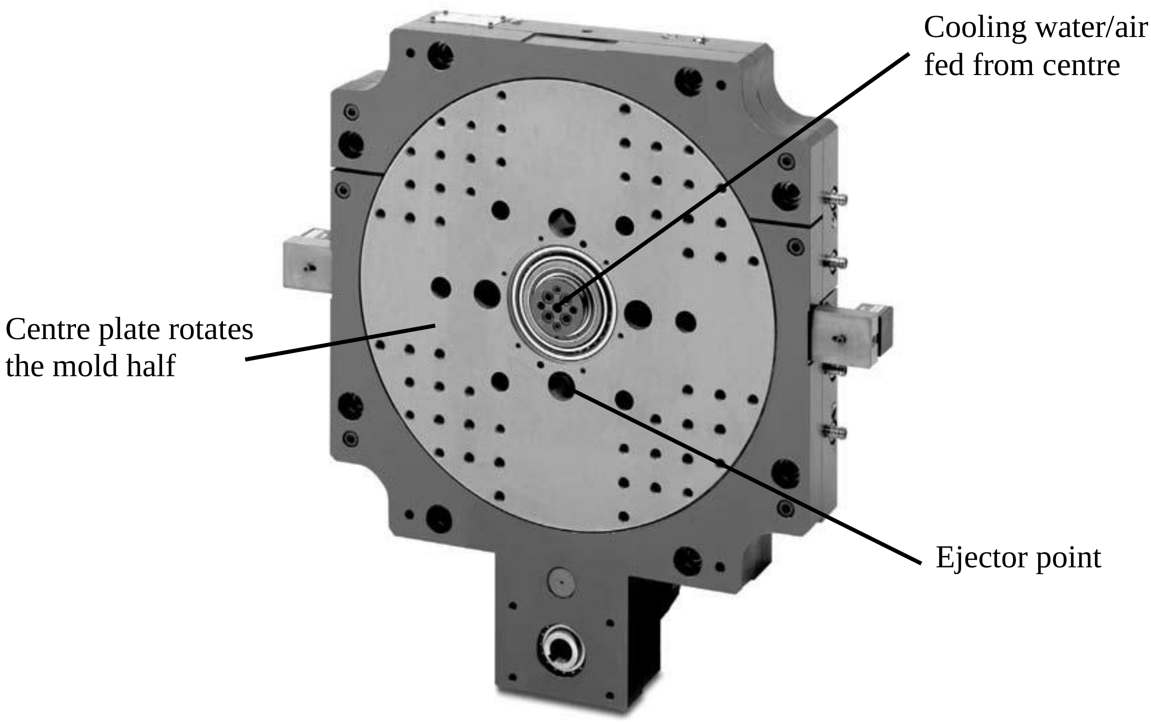

Fig. 23 Indexing unit [7]

The process proceeds in parallel so at any stage there is a shot being produced by each cavity. This makes the overall cycle time per moulding shorter than the core back technique described earlier. Generally the moulding produced in the first cycle should be expected to melt only on the very surface. This gives the good material separation required but still forms an adequate bond. This does however require good control of the process. [2]

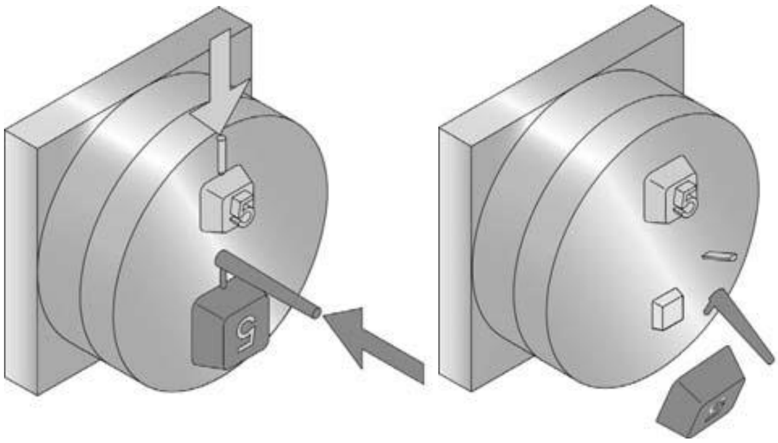

Fig. 24 Multi-component rotating mold [7]

6.1.3 Transfer Molding

In this method, instead of rotating the mold, a robot is used to transfer the molding to the next cavity where it can then be over-molded. In this example the robot will move the upper moulding to the lower larger cavity as the tool opens after each cycle. Like rotary methods, moulding proceeds in parallel with a molding produced in each of the cavities during any cycle. Therefore the cycle time will be dependent on the moulding requiring the longest moulding time. As in rotating methods, a good bond is required whilst maintaining distinct separation of materials. High accuracy is required in placing the insert to get good definition and registration on the final component. A means must therefore exist to hold the preform accurately in place before the second material is injected onto it. Transfer moulding is not restricted to one machine. Robotics can be used to move the preform to a second machine. [2]

6.1.4 Over-Molding

In this process, a component termed a preform is placed into the tool of an injection moulding machine. A second material is then moulded onto or around the preform. Two methods fall under this category: insert moulding and lost core moulding. [2]

6.1.5 Insert Molding

Insert moulding with plastics is a two-step process whereby a first preform component is placed into the open mould cavity. Injection then proceeds as with traditional moulding methods with injection of a molten plastic onto the preform. This process is not limited to two material components and the resultant mouldings can be transferred in this way until the required number of layers is achieved. Inserts can be loaded by hand or by the use of robots. Inserts must be accurate in both their dimension and their placement into the overmoulding tool to prevent tool damage and provide accurate registration of one material on another. A means must also exist to hold them in place within the tool. In this way it has similar requirements to that of in mould lamination techniques commonly used to decorate plastics with films or foils. [7]

6.2 Material Selection for multi-component injection molding

In selecting material combinations for multi-material moulding applications, consideration must be given to the combination of properties required in the final product. In many cases, there must be a certain level of adhesion between skin and core in order to maintain mechanical integrity. This can be achieved in two ways:

1. The materials are compatible and offer some degree of bonding at the interface.
2. A method must be found to mechanically interlock them. In the case of over-moulding and multishot this can be done with clever usage of material properties and tool design.

Not all parts require adhesion. In fact in some cases the requirement may be the exact opposite. If joints are to be produced, it is necessary that the mouldings can move freely at the interface. Examples include over-molding to produce what will be the moving arms and legs on dolls and other similar toys or to produce ball and socket joint mouldings. In cases like these, materials must be selected for their immisciblity to ensure smooth regions of movement. Where adhesion is required, good interfacial bond strength is a pre-requisite, otherwise the properties may come from a significantly reduced section thickness. For good adhesion, a certain amount of interdiffusion is required between the melts. This can be achieved when there is a high compatibility, or solubility between the melts. Tables of compatible and incompatible material combinations are shown in Table 3. However caution is required when using such tables, since it has been shown that changing from one particular grade of material to another can affect the bond strength. Occasionally manufacturers may also seemingly disagree on the adhesion properties of materials. Since processing conditions also affect adhesion, experimentation may be required to as certain optimum conditions for any given material combination. Additives are available to aid the compatibility of materials called compatibilisers. Through the addition of these materials it is possible to chemically bond some non-adherent materials. These substances usually contain a third polymer that bonds to, or is soluble in, the two materials. But number of chemical sites available at the interface and the molten contact time are limiting factors in the final bond strength as there is generally little interfacial mixing. [2], [7]

Table 3 Material adhesion compatibility [7]

	ABS	ASA	EVA	PA 6	PA 66	PBT	PC	HDPE	LDPE	PET	PMMA	POM	PP	PPO mod	PS-GP	PS-HI	SAN	TPU
ABS	+	+	+			+	+	-	-	+	+	-	-	-	*	*	+	+
ASA	+	+	+			+	+	-	-	+	+	-	-	-	*	-	+	+
EVA	+	+	+					+	+				+		+	+	+	
PA 6				+	+	*	*	*	*			-	*	-	-	-	+	+
PA 66				+	+	*	*	*	*			-	-	-	-	-	+	+
PBT	+	+		*	*	+	+	-	-	+	-	-	-	-	-	-	+	+
PC	+	+		*	*	+	+	-	-	+		-	-	-	-	-	+	+
HDPE	-	-	+	*	*	-	-	+	+	-	*	*	-	-	-	-	-	-
LDPE	-	-	+	*	*	-	-	+	+	-	*	*	+	-	*	-	-	-
PET	+	+				+	+	-	-	+	-	-		-	-	-		+
PMMA	+	+				-		*	*	-	+		*	-	-	-	+	
POM	-	-		-	-	-	-	*	*	-		+	-	-	-	-	-	
PP	-	-	+	*	-	-	-	-	+		*	-	+	-	-	-	-	-
PPO mod	-	-		-	-	-	-	-	-	-	-	-	-	+	+	+	*	-
PS-GP	*	*	+	-	-	-	-	-	*	-	-	-	-	+	+	+	-	-
PS-HI	*	-	+	-	-	-	-	-	-	-	-	-	-	+	+	+	-	-
SAN	+	+	+	+	+	+	+	-	-		+	-	-	*	-	-	+	+
TPU	+	+		+	+	+	+	-	-	+			-	-	-	-	+	+

Key (-) : No adhesion, (*) : Poor adhesion, (+) : Good adhesion
PS-GP = general purpose polystyrene
PS-HI = high impact polystyrene

As well as adhesion, there are other material characteristics that also need to be considered when molding with materials of different generic families. Examples are the levels of relative shrinkage and thermal expansion values, these may need to be matched or careful consideration given to the requirements before final material selection takes place.

Certainly in the case of many co-injection techniques, differences in mould shrinkage and thermal expansion can lead to problems such as sink marks, warpage and residual stresses. With over-moulding techniques, differences in shrinkage or the coefficient of linear thermal expansion can produce high stresses between restrained materials.[2]

II. PRACTICAL PART

7 MASTER THESIS AIM

At the master thesis was established an objectives:

- To create literary research for following theme,
- To design 3D model of injected plastic part,
- To create injection mold assembly design,
- To draw 2D documentation of the injection mold.

Theoretical part contains regarding to injection molding, rheology, injection mold machine characteristics, injection mold desing and their function parts. The injection mold assembly elements are briefly described in previous chapters.

The practical part objective was to design 3D model of the plastic parts. Design is released from real part. To this plastic product was designed 3D injection mold model and 2D documentation with bill of material as well. The design was provided by CATIA V5R19 and typified elements HASCO.

8 USED SOFTWARE

8.1 Catia V5R19

CATIA (***C**omputer-**A**ided **T**hree-Dimensional **I**nteractive **A**pplication*) is integrated computer designing system (CAD/CAM/CAE), developed by French company Dassault Systèmes and used mainly in airplane and automotive industry. This is software, by which is possible to create injection mold design from 3D model, assembly up to 2D draw documentation, as well as machining or loading simulation.

8.2 Hasco R1/2015

The Hasco module was developed as library of 3D standardized parts needed to mold design. The application contains wide amount documentations and guidelines, how standardized parts to use the best. The parts from library it is possible save to formats compatible with the wide range of designing software (Catia, SolidWorks, Unigraphics, AutoCad, Inventor, atd.).

8.3 Autodesk Moldflow Insight 2014

The Autodesk Moldflow Insight (later just moldflow) as a constituent part of Autodesk solution for digital prototyping is tool for injection molding prototyping. Moldflow offers to provide deep solutions, to evaluate plastic part and injection mold and thereby helps to study injection molding processes used in present practice. Software Moldflow utilize front world producers in automotive industry, consumer electronics, medical materials and also packaging in order for expenses saving. [14]

9 PLASTIC PART

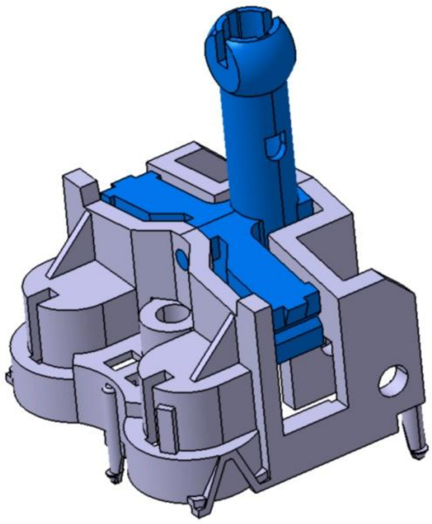

Fig. 25 Given two component plastic part

Injected in-mold assembly consists two part used in electro technical industry as a switch. First shot part is shell with platform dimensions 20x18 mm, which has two symmetric halves. The shell design contains three flexible connections due to join into electro technical assembly. Also other design necessities are occurred like ribs, holes and varying wall thickness from 0.5 mm up to 1.2 mm to ensure proper properties. Part volume is 0.87 cm^3 and weight 0.96 g.

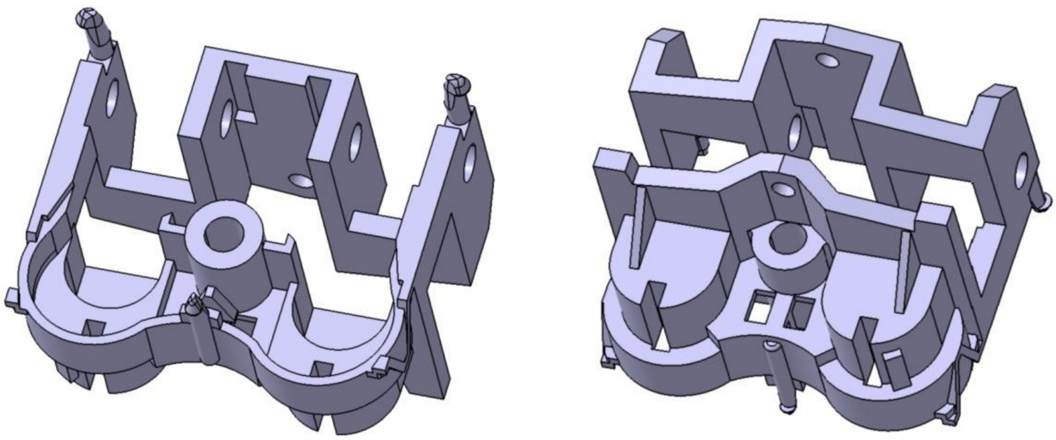

Fig. 26 First shot part

The second shot part has T shape, hollow tube and small shaft moving in holes of first shot part in 60 degrees range. Volume part is 0.21 cm^3 and weight 0,26 g.

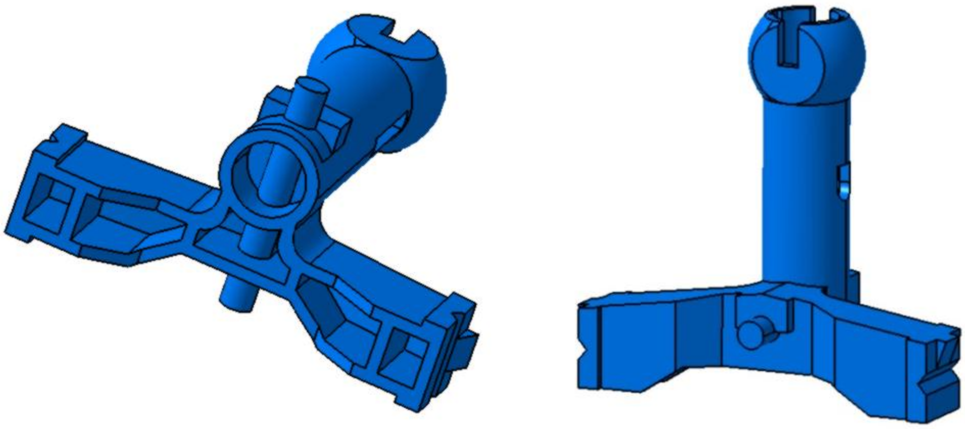

Fig. 27 Second shot part

9.1 Material

Requirements were for the parts material poor adhesion to each other polymer, appropriate mechanical strenght and electrical properties. Then for the first shot material is needed to use polymer with higher melting point temperature in compare with the second shot material used. The bonding uncompatibility is of both material is important as well. Therefore were picked polyamide 66 as a first shot material and for second shot material polyoxymethylen.

9.1.1 Polyamide 66

Polyamide 66 is one of the most versatile engineering thermoplastics. It is popular in every major market using thermoplastic materials. Because of its balance of strength, ductility and heat resistance, Polyamide 66 is an outstanding candidate for metal replacement applications. Polyamide 66 is very easy to process with a very wide process window. This allows it to be used for everything from complex, thin walled components to large thick walled housings. As a first shot material was chosen Staramide R1000MR.

Table 4 Staramide R1000MR properties

Elastic modulus	2690 MPa
Shear modulus	962 MPa
Melt temperature	300 °C

Mold surface temperature	80 ºC
Ejection temperature	205 ºC
Maximum shear stress	0,5 MPa
Maximum shear rate	6000 1/s

9.1.2 Polyoxymethylen

POM is a semi-crystalline polymer (75–85% crystalline) with a melting point of 175 °C. POM is a tough material with a very low coefficient of friction. However, it is susceptible to polymer degradation catalyzed by acids, which is why it is stabilized. POM is typically very difficult to bond. As a second shot material was chosen Hostaform C1301.

Table 5 Hostaform C1301 properties

Elastic modulus	3100 MPa
Shear modulus	3100 MPa
Melt temperature	210 ºC
Mold surface temperature	90 ºC
Ejection temperature	134 ºC
Maximum shear stress	0,45 MPa
Maximum shear rate	40000 1/s

10 MOLDFLOW ANALYSIS

10.1 Analysis results (first shot)

The process settings of the first part injection molding were: melt temperature 300ºC and 80ºC mold surface temperature. Due to cavity dimensions and melt flow were achieved fill time only 0.42 s. Thin walls ensured quick part cooling, therefore all surface where ejector pins are affect is ejection temperature reached within 3 seconds.

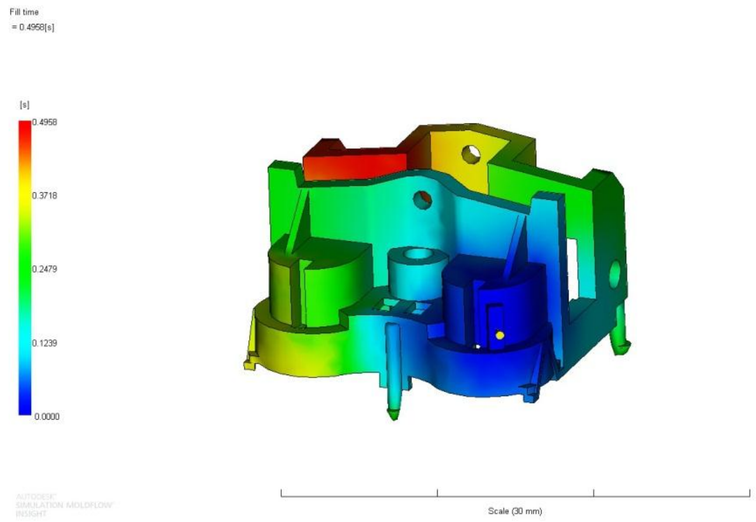

Fig. 28 Fill time of the first shot part (cavity filled during 0.49s)

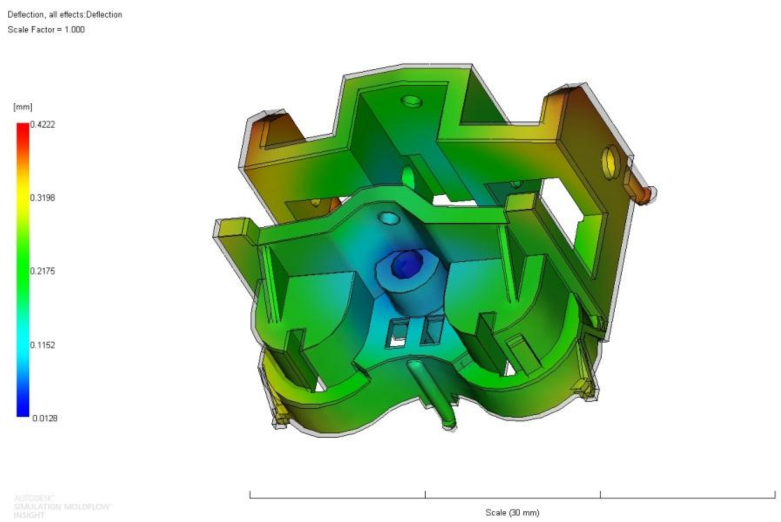

Fig. 29 Deflection – all effects (maximal deflection – 0,42 mm)

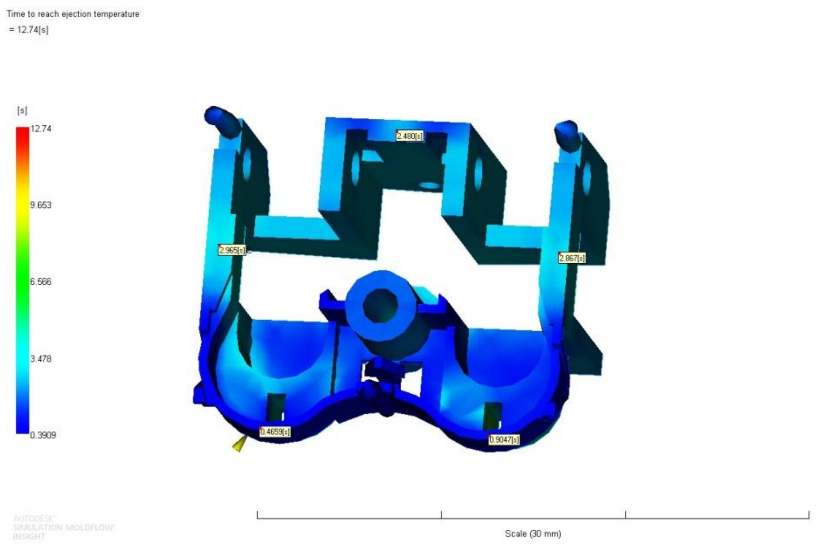

Fig. 30 Time to reach ejection temperature (first shot)

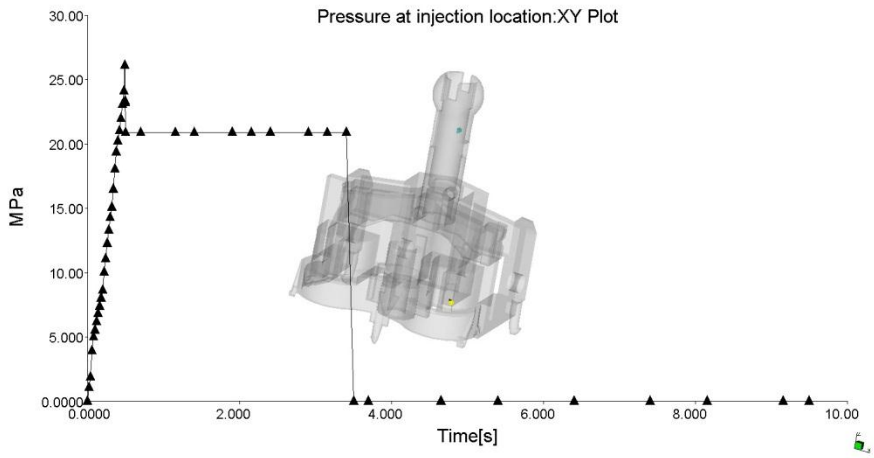

Fig. 31 Pressure at injection location (first shot)

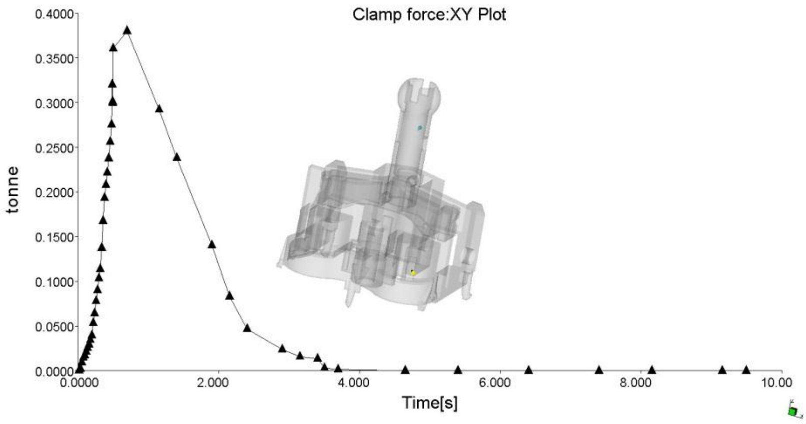

Fig. 32 Clamp force (first shot)

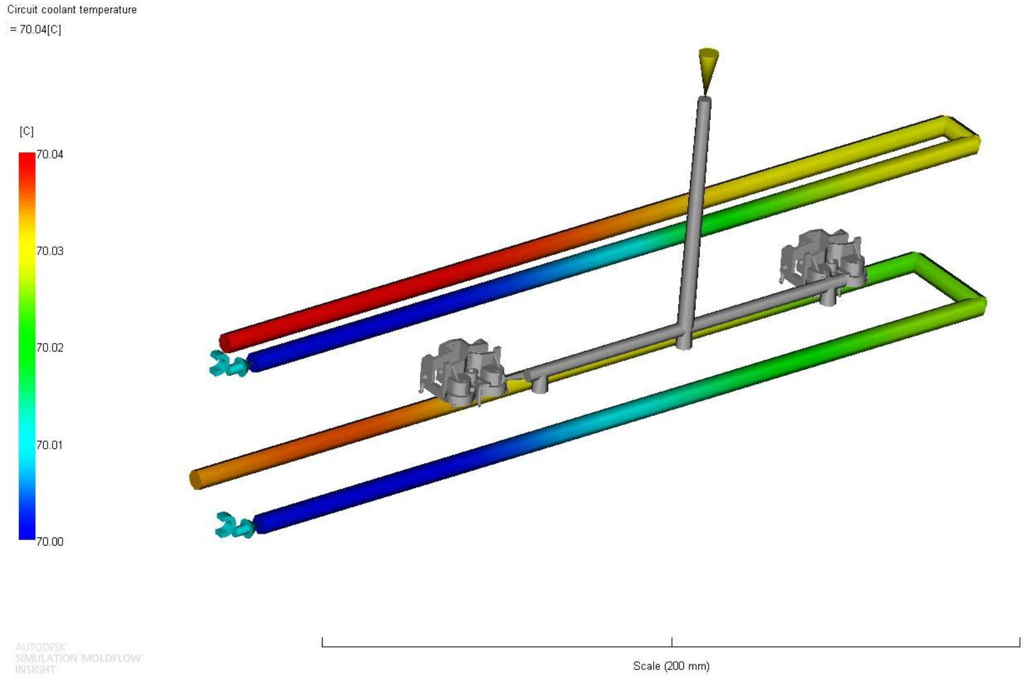

Fig. 33 Cooling circuit

10.2 Analysis results (second shot)

The injection molding process settings of the second part were: melt temperature 210ºC and 80ºC mold surface temperature. Due to cavity dimensions was within 0.4s. Thin walls ensured quick part cooling, therefore ejection temperature at the ejecting areas reached within 3 s.

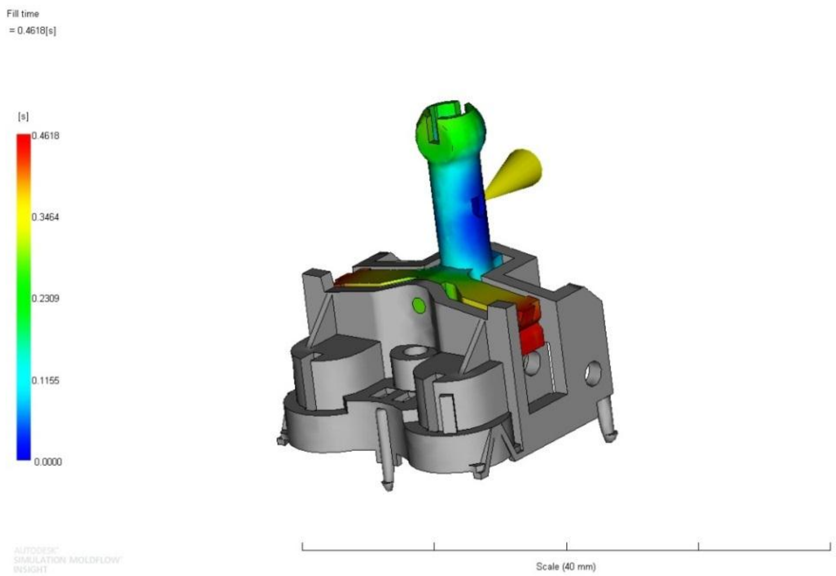

Fig. 34 Fill time (second shot)

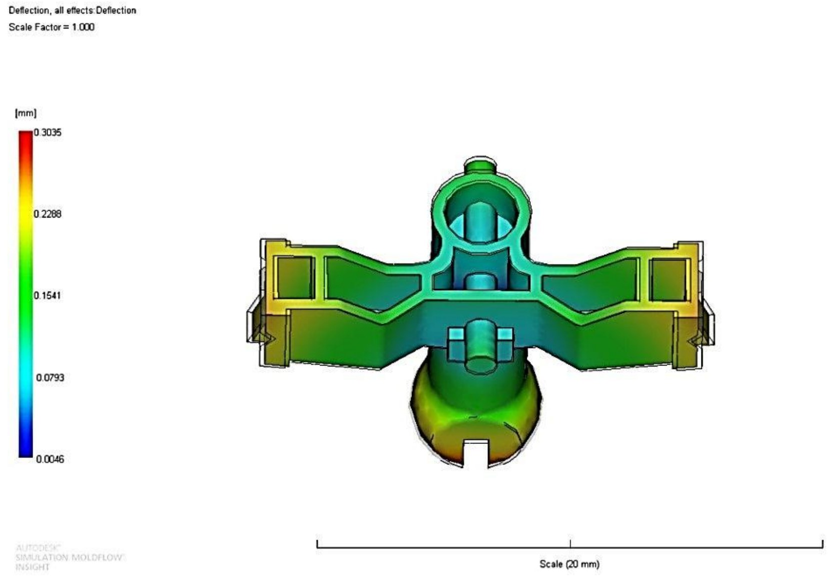

Fig. 35 Deflection – all effects (maximal deflection – 0,3mm)

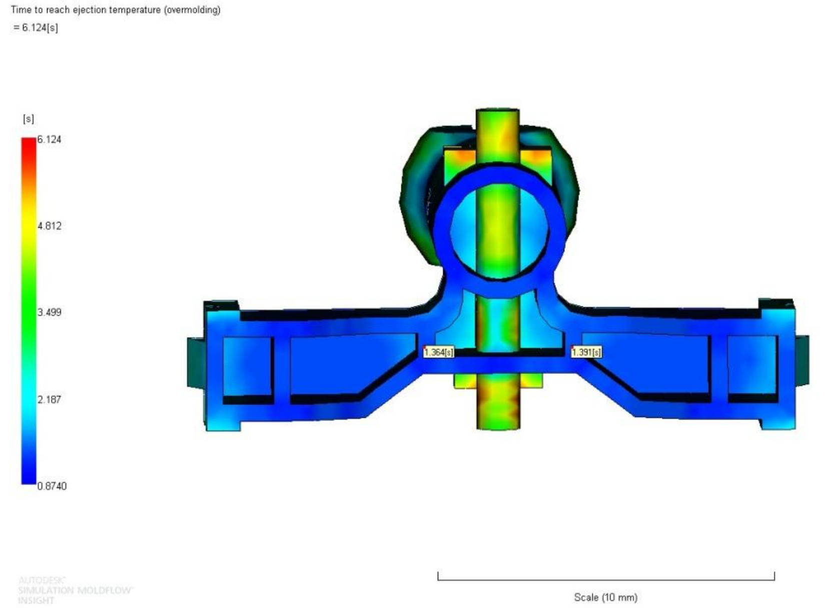

Fig. 36 Time to reach ejection temperature (second shot)

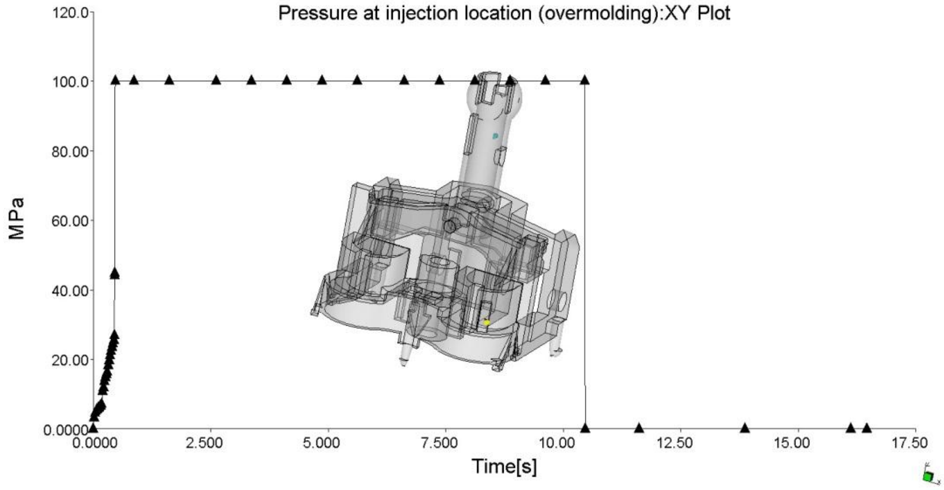

Fig. 37 Pressure at injection location (second shot)

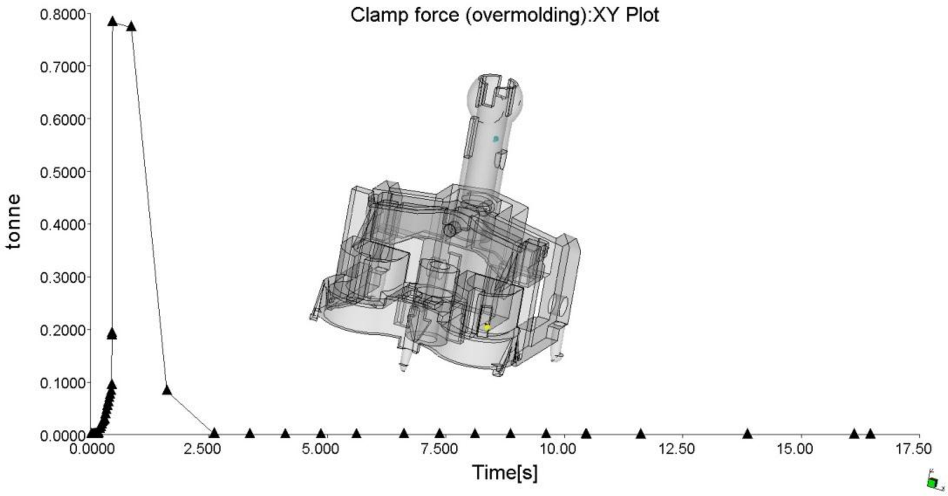

Fig. 38 Clamp force (second shot)

11 INJECTION MOLD DESIGN

11.1 Core and cavity

The injection mold is two multiple for production of two component part, therefore was needed to create four core inserts and four cavity inserts. Each cavity is negative of the part shape and enlarged by the shrinkage value of the material.

Core insert

All four core inserts have the same design. Dimension is 50 x 50 mm and thickness 27 mm. There are first shot and second shot runner drain, holes for ejectors, 6mm drill hole for cooling system. Insert is hardened 60 HRC (0,2 mm).

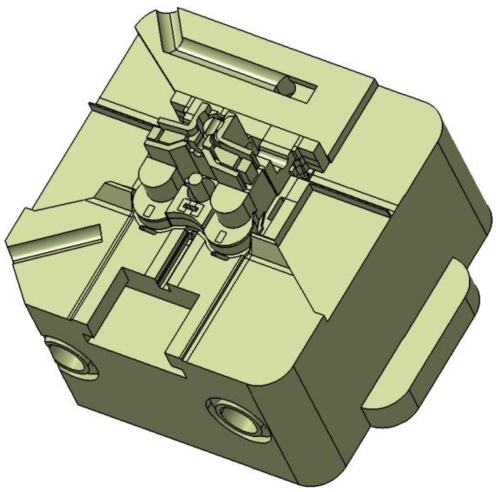

Fig. 39 Core insert

Cavity insert

Dimension of all cavities is the same 50 x 50 mm and thickness 36 mm. Cavity insert for injection molding of the first shot part are different. Dimension is 50 x 50 mm and thickness 36 mm. Because cavities are on the right side (non-movable), then upper two are designed for first shot and have just own runners and gate and bottom two are only for second shot material. Inserts are hardened 60 HRC (0,2 mm).

a) b)

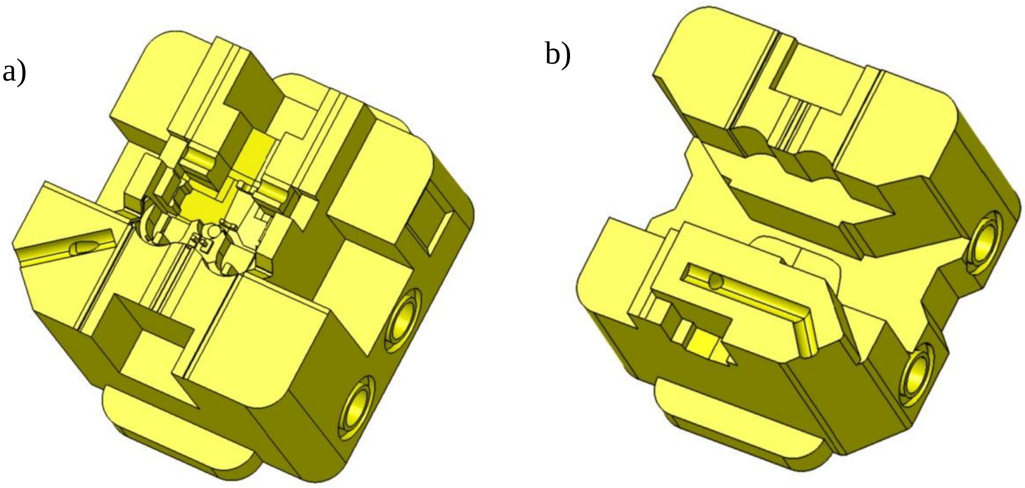

Fig. 40 Cavity inserts: a) the first shot, b) the second shot

11.2 Undercuts

Molding of the two component small dimensioned part is difficult process requires accuracy within design and production. Given part contains several shapes (undercuts), which is not possible molded in the main splitting line, therefore is necessary to use sliders lifters. Slider´s motion is accomplished by angled pins (18º) with length 60, 80 and 100 mm. Sliders are moving between guiding rails and kept in right position by lockers when mold is closed. To avoid move sliders out the right position, when mold is opened was performed by spring plunger.

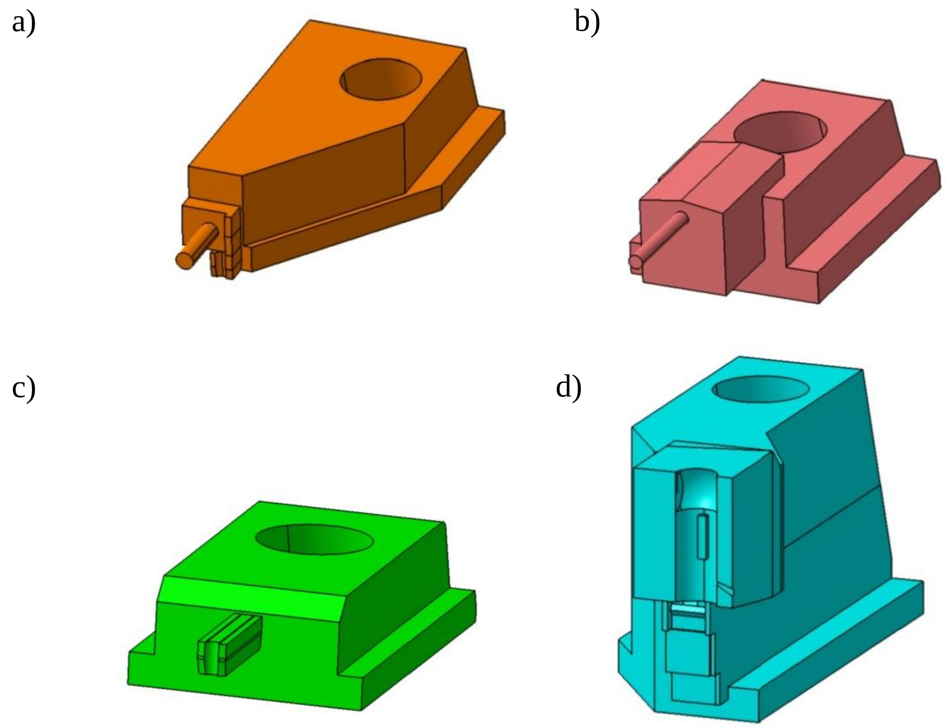

Fig. 41 Sliders lifters a),b),c) for the first shot part, d) for the second shot

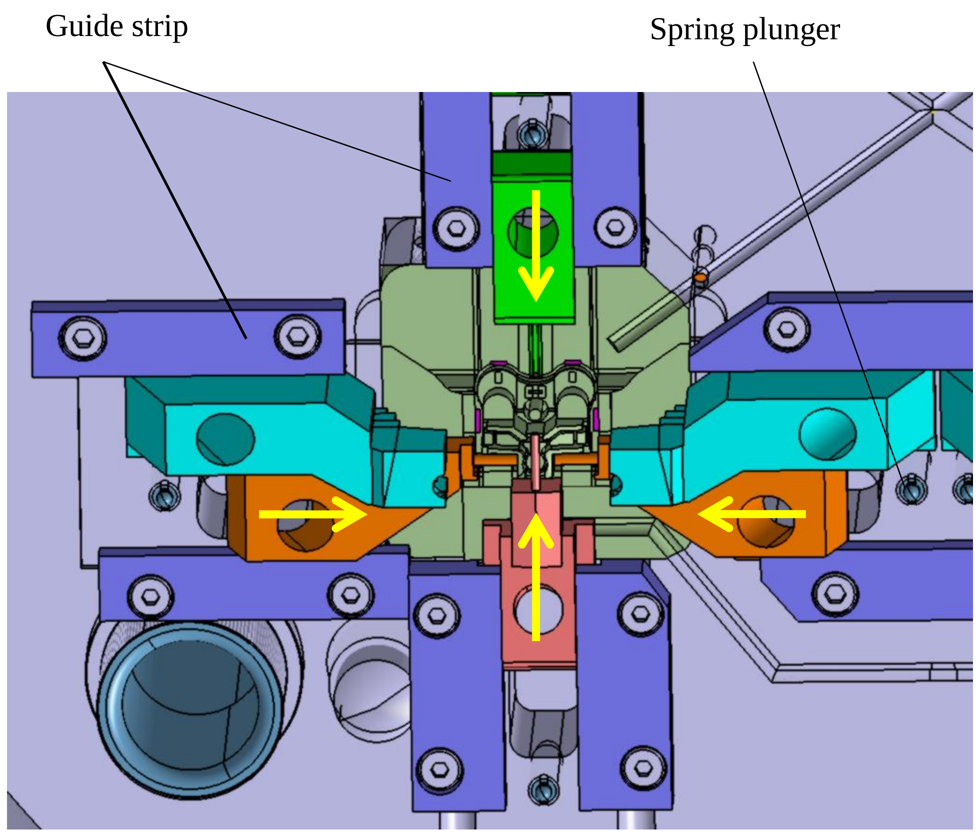

Fig. 42 View to core side, when first shot material is injected

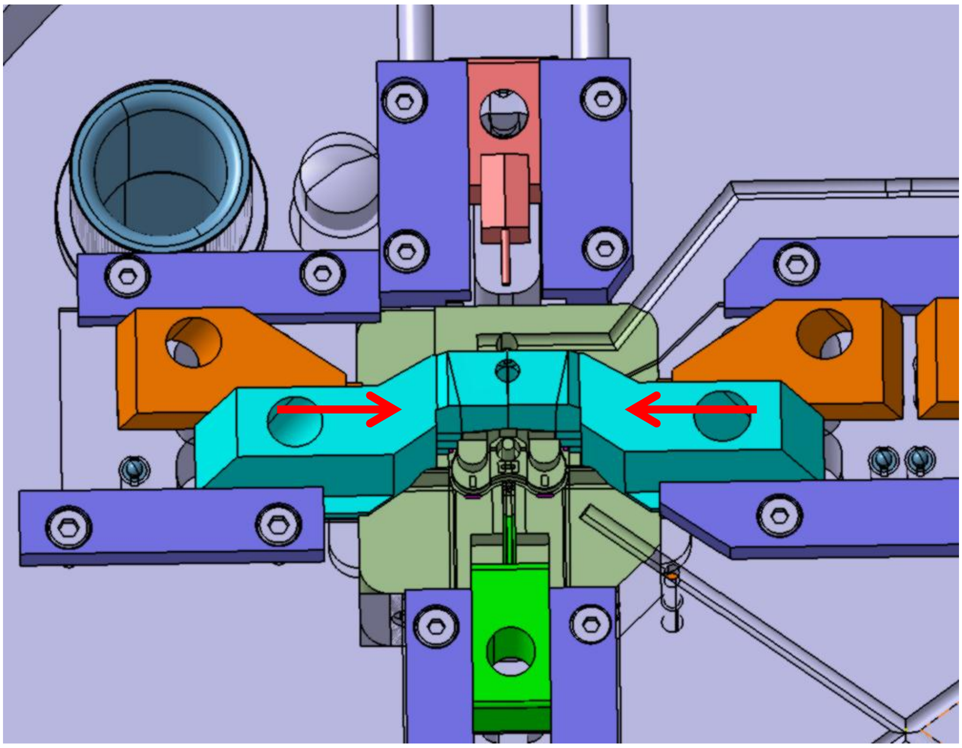

Fig. 43 View to core side, when second shot material is injected

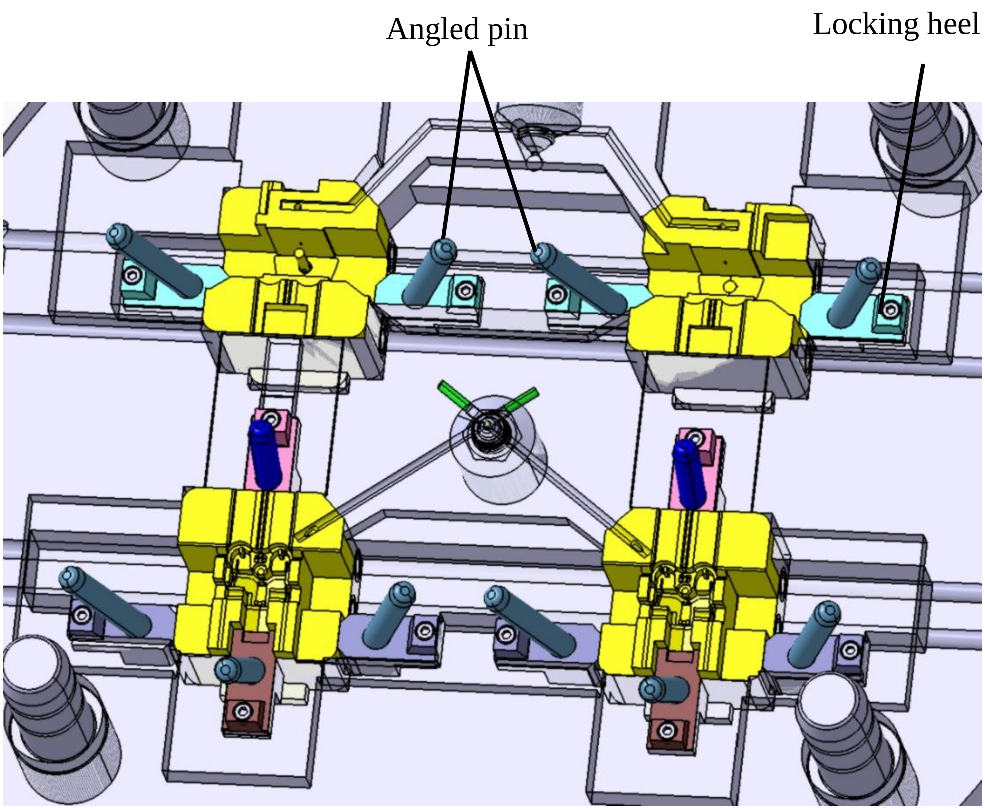

Fig. 44 Left side of splitting line design

11.3 Runner system

The first shot part has combined feed system consist of one hot nozzle Z200 and cold runners ended by submarine gate enables to cut runner system from part itself. For the second shot part is used also combination of hot nozzle Z200 and cold runner system with submarine gate.

11.4 Ejection system

For the ejection of the injected two component part and cold runner were used two the same, but independent ejection plates. Between them actuating rod with pull – push element ensures ejection movement just upper plates at the time, when both components are injected and cooled. Every cavity contains 9 ejectors: 5 flat ejectors Z465, 4 rounded Z41/2,5 x100, two for runner system ejection, therefore together 36 ejector pins, which are modified for required length.

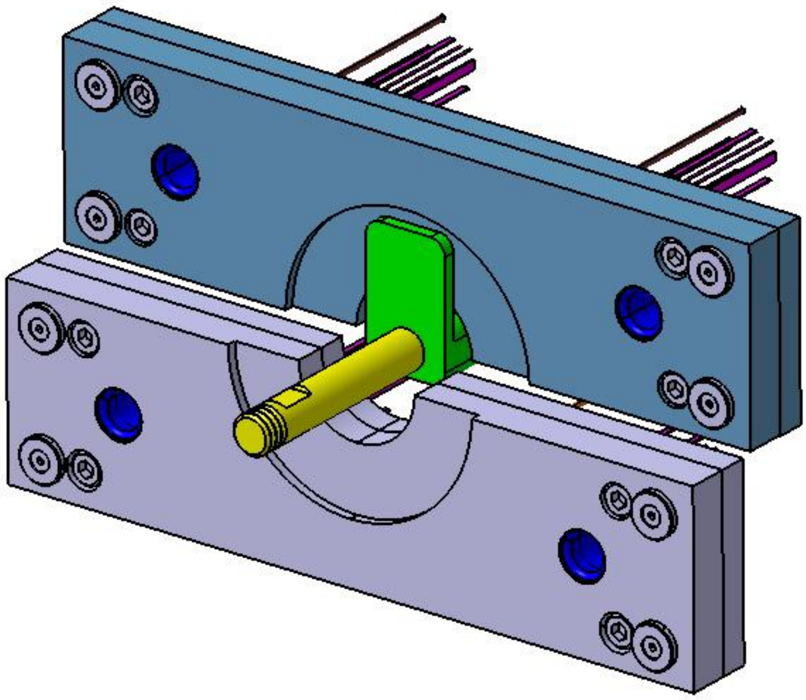

Fig. 45 Ejection system with push pull element

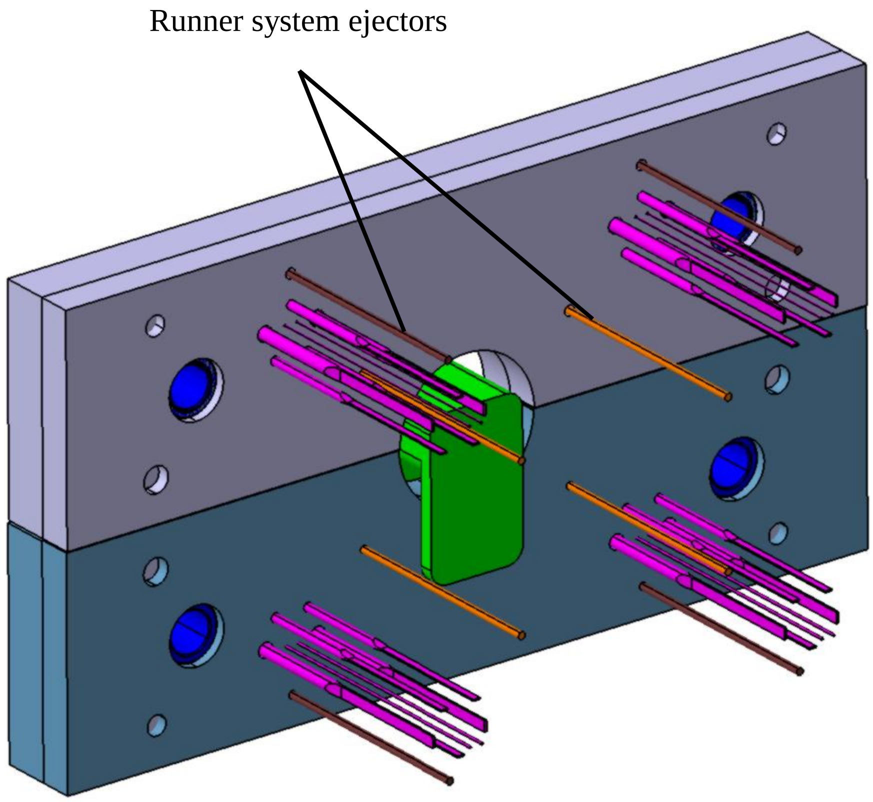

Fig. 46 Ejection system

11.5 Cooling system

Generally, it was created 4 cooling circuits, for each two cavity or core insert one circuit. Cooling system was created on the left and right side by 6 mm channels drilled into the core and cavity inserts and retainer plates. The direction of cooling water through the channels determines plugs. To avoid leaking between core or cavity insert and retainer plates ensures rubber sealing rings.

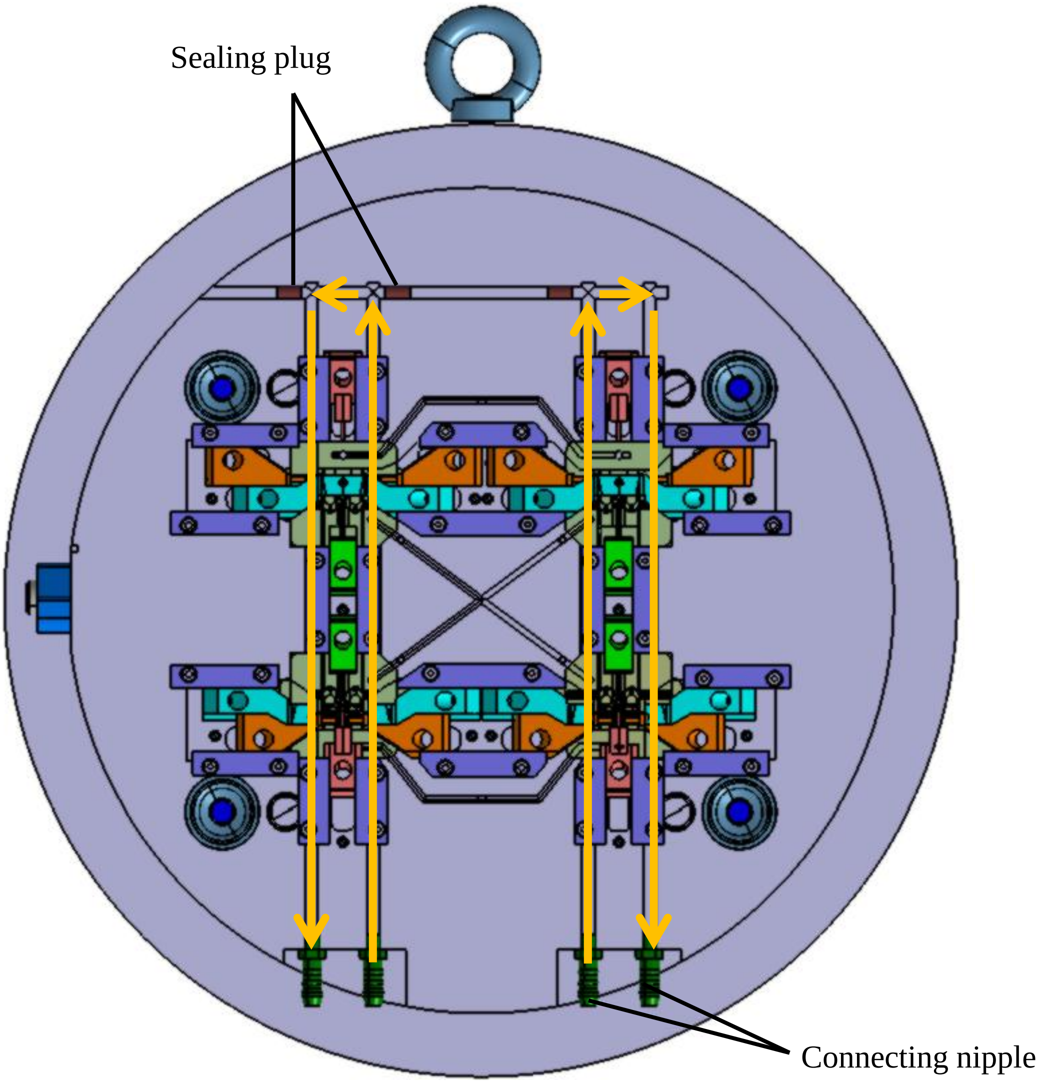

Fig. 47 Cooling of left side

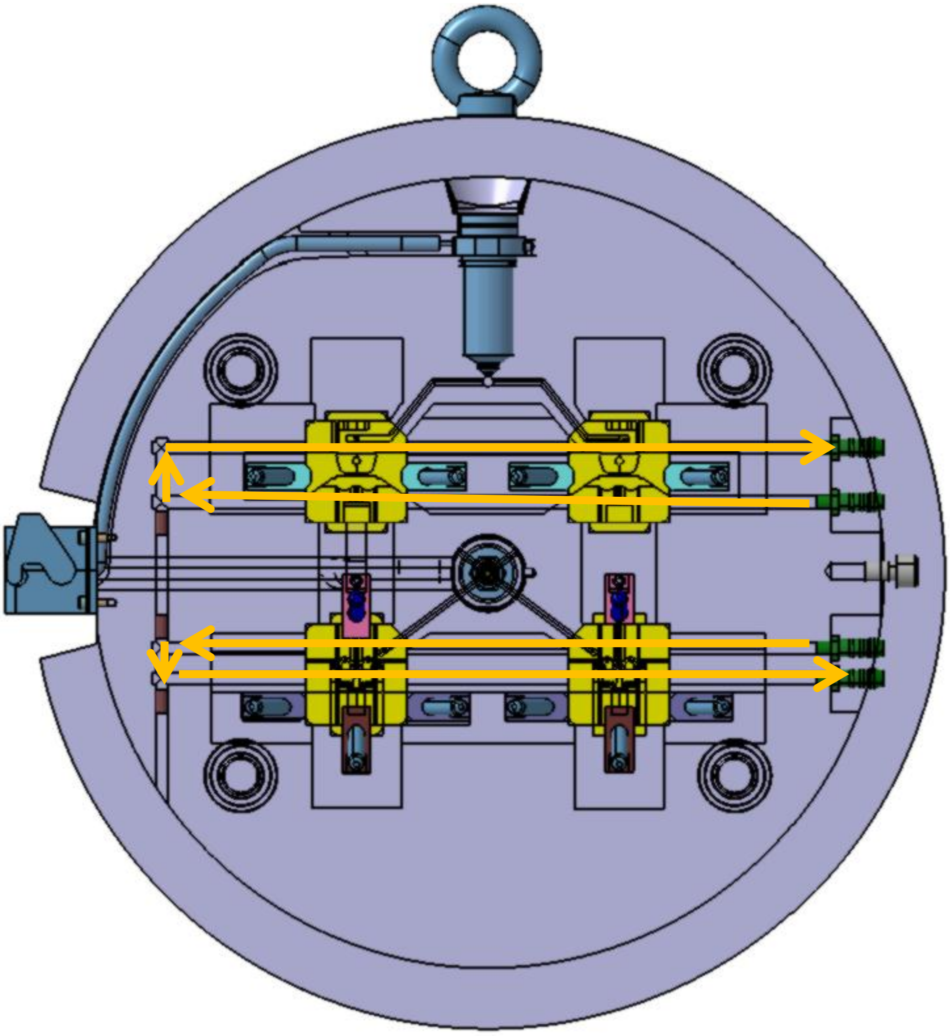

Fig. 48 Right side cooling

11.6 Injection mold

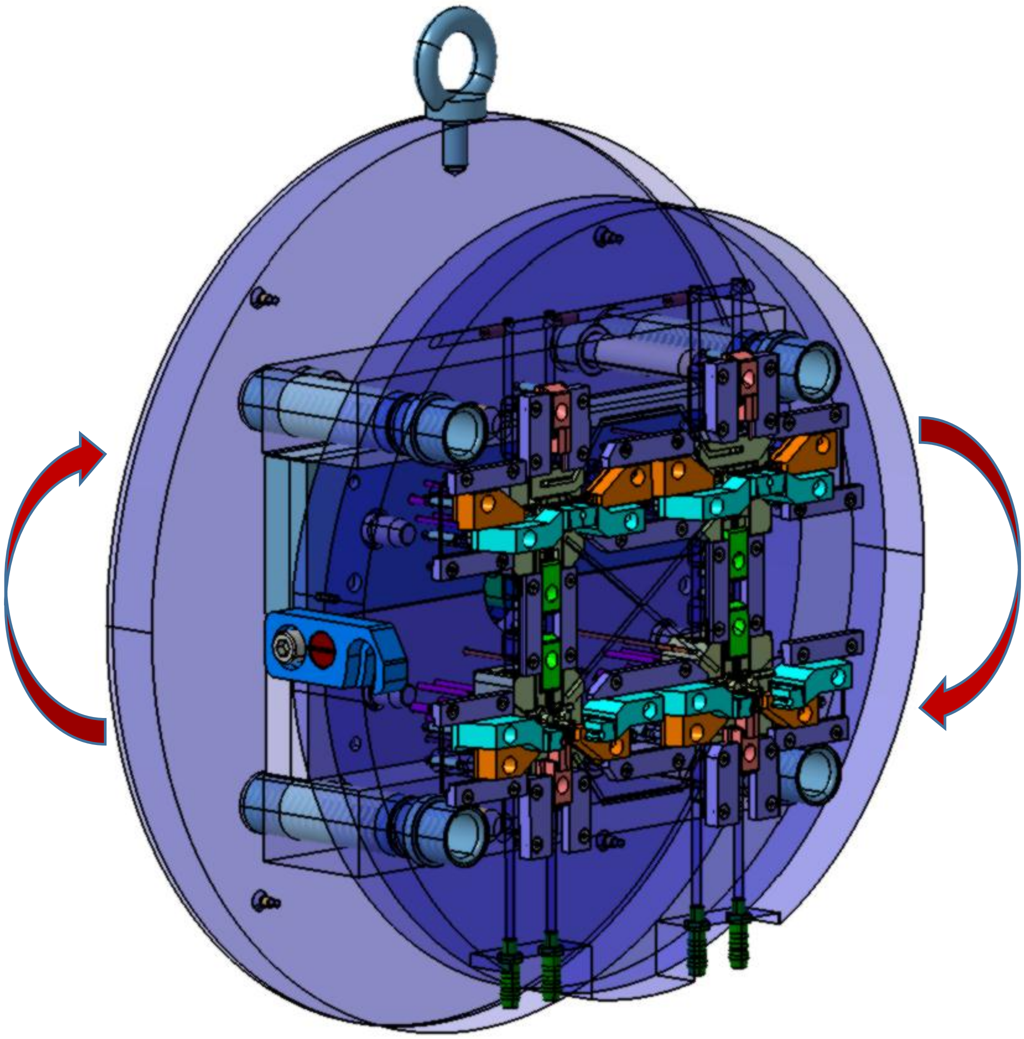

Fig. 49 Left turning side of the Injection mold

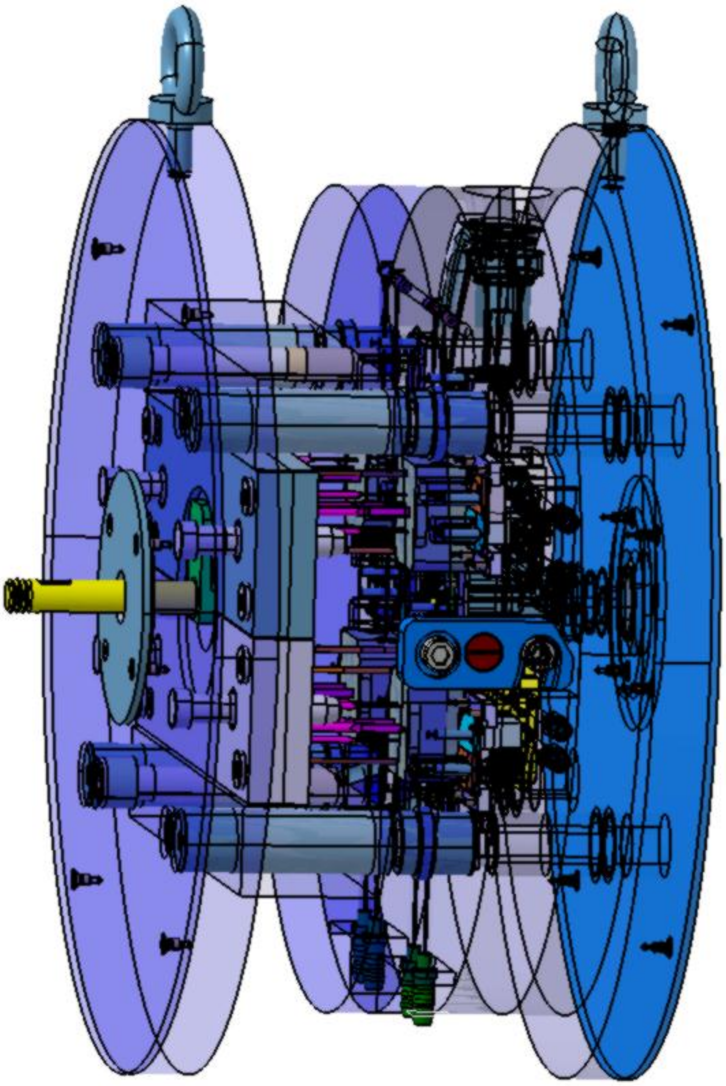

Fig. 50 Injection mold design

12 INJECTION MOLDING MACHINE

The injection molding machine choice is based on a few assumptions, which have to be fulfilled.

Size of the injection molding machine is determined by:

- Tie bars range dimension
- Clamping force
- Plasticizing unit performance

The characteristics of designed injection mold:

- Outside dimensions: 450x243mm
- Clamping force when the first shot molding: 0,4 tons
- Clamping force when the second shot molding: 0,8 tons
- Cavity volume of the first shot: 5,6 cm^3
- Cavity volume of the second shot: 5,2 cm^3

Parameters are comply with injection molding machine Arburg 470S Multi-component included. Machine properties list is attached at appendix.

Table 6 Injection molding machine ARBURG 470S Multi-component properties

Clamp force	1100 MPa
Distance between tie bars	470 x 470 mm
Horizontal injection unit stroke volume	10,3 cm^3
Vertical injection unit stroke volume	10,3 cm^3

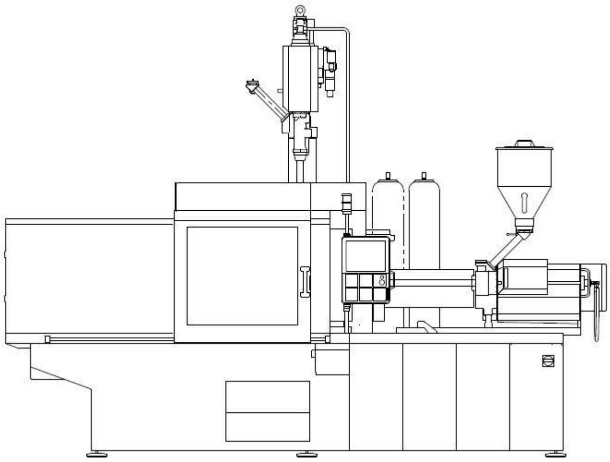

Fig. 51 Multi-component injection molding machine

According to conditions was selected appropriate turning plate Arburg 470S.

Table 7 Rotary unit Arburg 470S properties

Rotating platen diameter	558 mm
Turning diameter between tie bars	689 mm
Ejector stroke	111 mm
Electric/ hydraulic rotating time	0,9/1,8 s

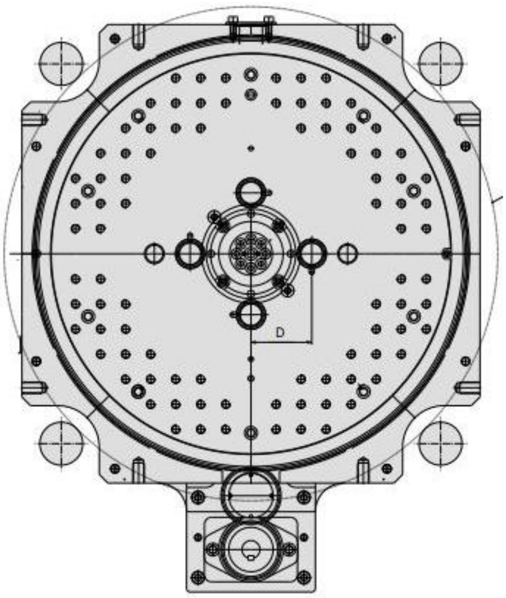

Fig. 52 Rotary unit

13 RESULTS DISCUSSION

Master thesis was created according to given task. The main part was injection mold design for two component in-mold assembly. Design was provided by software Catia V5R19. Standard parts were paste from Hasco Daco module. Design was supported by Autodesk Moldflow Insight 2014.

Injection mold was specified as a two multiple. By the first shot material polyamide 66 with higher processing temperature than second shot material which was polyoxymethylen. Core, cavity inserts and sliders were negative to the part shape enlarged by shrinkage value of appropriate material. For part molding were used a several sliders to proper molding of undercuts.

The injection of each shot was ensured by combination of hot nozzle Z200 and cold runner system ended by submarine gate. Tempering system affecting cavity filling and polymer melt solidifying. Every injection cycle is mold surface increasing temperature. Parts need to be injected in the same temperature to ensure the same conditions, therefore was designed 8 circuits of cooling system. For each two cores of cavity is used one cooling circuit. Lastly was created ejection system to release finished parts and runner system out of the splitting line.

The mold was designed as a multi-component with turning left side of mold, ensures proper molding of each shot separately. The mold plates are rounded.

Injection mold machine was selected on the basis of a several technical parameters, which are needed for high quality products and mold size. As optimal molding machine was chosen Arburg 470S Multi-component, included rotary plate Arburg 470S.

CONCLUSION

At the master thesis was described theory regarding to injection molding and overmolding. Theory about injection molding technology deals material rheology, producing cycle, design the of injection mold. Injection mold design theory is focused on feed system design rules, molding of undercuts, cooling and ejection system. At the last chapter are described the most overmolding types of this sophisticated molding field.

The practical part of the master thesis was to design injection mold for two component plastic product, which was electro technical switch. This part was designed according to real product. After that was created moldflow analysis of each part to support mold design. The injection mold was designed as a two multiple for two component part. All mold parts were described. Final part of thesis was 2D assembly drawing and bill of materials.

All designed parts and saved on the CD. 2D drawnings are attached.

BIBLIOGRAPHY

[1] ROSATO, D. V., ROSATO, D. V., ROSATO, M. G. *Injection Molding Hand book (3rd Edition).* NYC, NY, USA: Springer - Verlag, 2000. 1485s. ISBN 978-0-7923-8619-3.

[2] GOODSHIP, V. *Practical Guide to Injection Moulding.* Shropshire, UK: Rapra Tech. Ltd. and ARBURG Ltd, 2004. 202s. ISBN 1-85957-444-0

[3] ZEMAN, L. *Vstřikování plastů: úvod do vstřikování plastu.* 1.vyd Praha: BEN – technická literatura, 2009. 247 s. ISBN 978-80-7300-250-3.

[4] BOBČÍK, L.a kol. *Formy pro zpracování plastů II.díl - Vstřikování termoplastů,* 2. vyd. Brno: UNIPLAST, 1999. 212 str.

[5] BOBČÍK, L. a kol. *Formy pro zpracování plastů I.díl - Vstřikování termoplastů,* 2. vyd. Brno: UNIPLAST, 1999. 134 str.

[6] BEAUMONT, J. *Runner and gating design handbook.* 2nd ed. Munich, Germny: Hanser Publishers, 2007. 308 s. ISBN 978-1-56990-421-3.

[7] ARBURG [online]. [cit. 2014-02-08]. Available from WWW: < http://www.arburg.com >

[8] *Vstřikování Plastů* [online]. 2005 [cit. 2015-01-08]. Available from WWW: <http://www.ksp.tul.cz/cz/kpt/obsah/vyuka >.

[9] NEUHAUSL, E. *Vstřikování plastických hmot.* Praha: SNTL, 1973. 206 s.

[10] RANNAR, L.E. *On Optimization of Injection Molding Cooling*, NTNU Trondheim 2008

[11] KOLOUCH, Jan. *Strojírenské výrobky z plastů vyráběné vstřikováním.* 1. vyd. Praha: SNTL, 1986. 229 s.

[12] *Smartplast* [online]. [cit. 2015-01-18]. Available from: WWW: < http://smartplast.cz >

[13] BEAUMONT, J. *Runner and gating design handbook.* 2nd ed. Munich, Germny: Hanser Publishers, 2007. 308 s. ISBN 978-1-56990-421-3.

[14] ZEMAN, L. *Vstřikování plastů: úvod do vstřikování plastu.* 1.edit. Praha: BEN – technická literatura, 2009. 247 s. ISBN 978-80-7300-250-3.

[15] HASCO [online]. c2014 [cit. 2014-01-18]. Hasco.com. Available from WWW: <http://www.hasco.com/>.

[16] UNGER, P. *Gastrow Injection Molds 130 Proven Designs.* 4nd ed. Carl Hanser Verlag GmbH & Co. KG , 2006. 365 s. ISBN 978-3-446-40592-9

[17] *University of Cambridge* [online]. c2014 [cit. 2015-02-06]. Available from WWW: <http://www.doitpoms.ac.uk/>.

[18] FRENKLER, D.; ZAWISTOWSKI, H. *Hot Runners in Injection Moulds.* Shropshire, UK; Rapra Tech. Ltd, 2001. 364s. ISBN: 1-85957-208-1

LIST OF ABBREVIATIONS

CAD Computer aided design

CAM Computer aided machining

CAE Computer aided machining

C Carbon

H Hydrogen

λ Heat transition coefficient [W/mK]

T_f Flow temperature [°C]

T_g Glass transition temperature [°C]

T_m Melt temperature [°C]

L/D Length/diameter ratio

LIST OF FIGURES

LIST OF TABLES