Design of injection mold including cooling system optimization

Bc. Martin Pinďák

Master thesis
2012

Tomas Bata University in Zlín
Faculty of Technology

Univerzita Tomáše Bati ve Zlíně
Fakulta technologická
Ústav výrobního inženýrství
akademický rok: 2011/2012

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

Jméno a příjmení:	**Bc. Martin PINĎÁK**
Osobní číslo:	**T10384**
Studijní program:	**N 3909 Procesní inženýrství**
Studijní obor:	**Konstrukce technologických zařízení**
Téma práce:	**Návrh vstřikovací formy včetně optimalizace temperace**

Zásady pro vypracování:

1. **Vypracujte literární studii na dané téma.**
2. **Proveďte analýzu plnění zadaného.**
3. **Navrhněte vstřikovací formu.**
4. **Pomocí simulačního sw optimalizujte chladící systém.**
5. **Vyhodnoťte navržená řešení.**

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: **13. února 2012**
Termín odevzdání diplomové práce: **18. května 2012**

Ve Zlíně dne 2. února 2012

Roman

doc. Ing. Roman Čermák, Ph.D.
děkan

UNIVERZITA TOMÁŠE BATI VE ZLÍNĚ
Fakulta technologická
L.S.

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

Příjmení a jméno: Pinďák Martin   Obor: Konstrukce technologických zařízení

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 15. 5. 2012

....

---

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).

[3] 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

Cílem diplomové práce je provedení a vyhodnocení simulací vstřikovacího cyklu a konstrukce vstřikovací formy pro plastový dílec, poskytnutý společností Hella Autotechnik, s.r.o. Mohelnice.

V teoretické části diplomové práce je popsána technologie vstřikování, vstřikovací stroje a konstrukce vstřikovací formy.

V praktické části práce je analyzováno šest typů vtokových systémů ve dvou variantách průřezu studeného rozvodného kanálu. Pro zvolený vtokový systém byla v programu Catia V5 vytvořena konstrukce vstřikovací formy, která je doložena 3D modelem, výkresem sestavy a kusovníkem. Pro navrženou formu byly pomocí Autodesk Moldflow Insight vyhodnoceny temperační systémy, vyrobitelné konvenčními a nekonvenčními metodami. Výsledky analýz a navržená řešení jsou vyhodnocena v závěrečné části diplomové práce.

Klíčová slova: vstřikování, vstřikovací forma, vtokový systém, tokové analýzy, temperace

ABSTRACT

The aim of this Master thesis is performance and evaluation of injection cycle simulations, and design of an injection mold for plastic part provided by a company Hella Autotechnik, Ltd.

An injection molding technology, injection molding machines and mold constructions are described in a theoretical part.

Practical part deals with six types of runner system which were analyzed, all in two cross-section variants of cold runner. The injection mold which contains its 3D model, an assembly drawing and parts bill, was designed for selected runner system in Catia V5 software. For the mold was in Autodesk Moldflow Insight software analyzed and evaluated cooling system which is makeable with conventional technology and conformal cooling system in addition. Results and designed solutions are evaluated in a closing stage of this Master thesis.

Keywords: injection molding, injection mold, runner system, flow analyses, cooling

ACKNOWLEDGEMENTS

I would like to express my thanks to doc. Ing. Anežka Lengálová, Ph.D. who had provided me knowledges of academic writing. My gratitude is also extended to Ing. Michal Staněk, Ph.D. and Ing. Tomáš Drga, Ph.D. for skilled guidance and professional advices throughout my work on the thesis.

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

CONTENTS

INTRODUCTION

The increasing interest in producing a dimensional precise, shaping complex and at the same time lightweight article has heightened the need for injection molding of thermoplastics. Injection molding is one of the most expanded technologies to produce this type of plastic parts. Nevertheless, the complexity of injected molded products always requires the new design of injection mold, including its runner and cooling system, for every new article. Moreover, every product to be molded needs the different settings of the molding process and often also other molding machine. The literature review in the theoretical part of the thesis brings the introduction to the thechnology, overview of injection molding machines and conception of injection molds construction.

Knowledge of injection molding simulation has a great importance for the process optimizing; thereby it can prevent appearing deformation and other errors of molded part. In a response to growing interest in process simulation, the costs for mold prototypes decreased. The aim of the anlysis is to design six types of runner system for given plastic part, run and evaluate the simulations of filling the mold cavities. In addition, several types of cooling system have been designed, including the one designed for Direct metal laser sintering technology. All analysis results are evaluated and compared. The purpose of evaluation of filling and cooling simulations is to find the most suitable mold construction and optimizing the temperation system of the injection mold for given plastic part. The thesis gives complete design of injection mold including three-dimensional model and two-dimensional drawings of all components.

I. THEORY

1 INJECTION MOLDING OF THERMOPLASTICS

1.1 Technology

Injection molding is an ideal way how to process polymers in form of granulates into geometrically complex, thin walled technical parts. It is the most used polymer processing technology which has been developing since the end of the 19th century when the first injection molding machine was built. In the beginning cellulose nitrate articles were produced thanks to invention of this machine, which consisted of a closed mold, steam-heated chamber and hydraulically operated plunger. Plastic processing technologies including injection molding underwent significant progress in the twentieth century. It grew up to be the process for production of complex three-dimensional shaped parts based on composites, foams, rubbers and thermosets, in addition to thermoplastics. To improve the efficiency and flexibility of the process and to extend the scope of molded products, many innovations have been developed so far, such as gas and water assisted injection molding or micro-injection molding. With progress in computer systems, the technology has been enhanced with better process simulation, control and optimization, and also product quality. [1]

Injection molded articles appear in our everyday life, but they are also becoming common in less obvious applications. The main representatives of this technology can be found in automotive parts, packing and household articles, electronics, toys and many others. There are a lot of possibilities how to improve mechanical, chemical or esthetical properties by additives (glass fibres, glass balls) or by radiation effects. Thus, improved polymers can replace other materials in applications where earlier only aluminium or steel were used. Significant progress in injection molding has reflected in products and at present thermoplastics is processed by this technology as an inseparable part of industrial production. [1]

1.2 Injection molding cycle

Injection molding of thermoplastics is a cyclic process of conveying, melting and cooling polymeric material. Initially, plastic granules are fed to the machine through the hopper. The first area where the granules are in contact with the machine is the screw in the barrel, where they are melted. After entrance into the barrel, the rotating screw moves the granules forward in the channels, where they are forced against the heated wall of the barrel, and melted due to dissipation heat and conduction of the heat generated from the cells along the barrel. Both conveying and melting the polymer take place in first two of three sections of the screw called plasticizing stage. In the last section, the molten material is conveyed to the tip of the screw, develops the pressure against the closed nozzle, and the screw moves backward to accumulate the molten polymer at the front of the screw barrel (Fig. 1). The screw rotation stops when the required amount of melt is obtained and the injection stage of the process compounded of four phases (filling, packing, cooling and demolding) can begin (Fig. 2). [1]

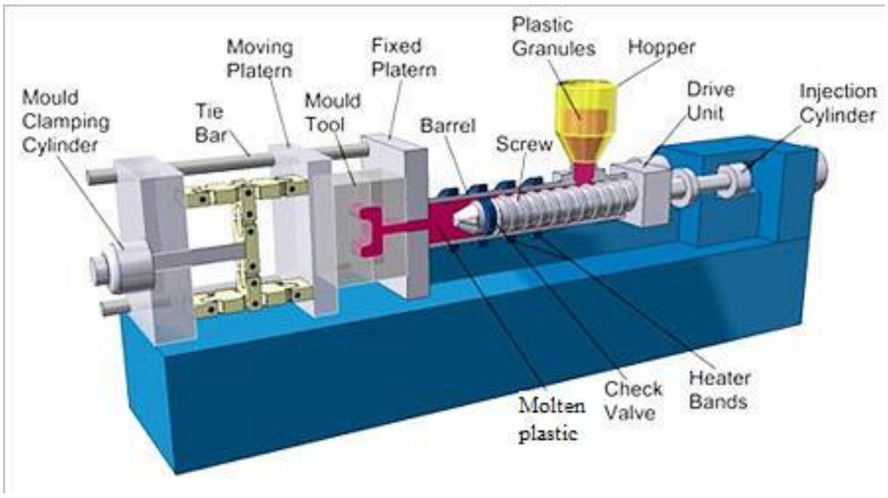


Figure 1 Injection molding machine - devices assembly [18]

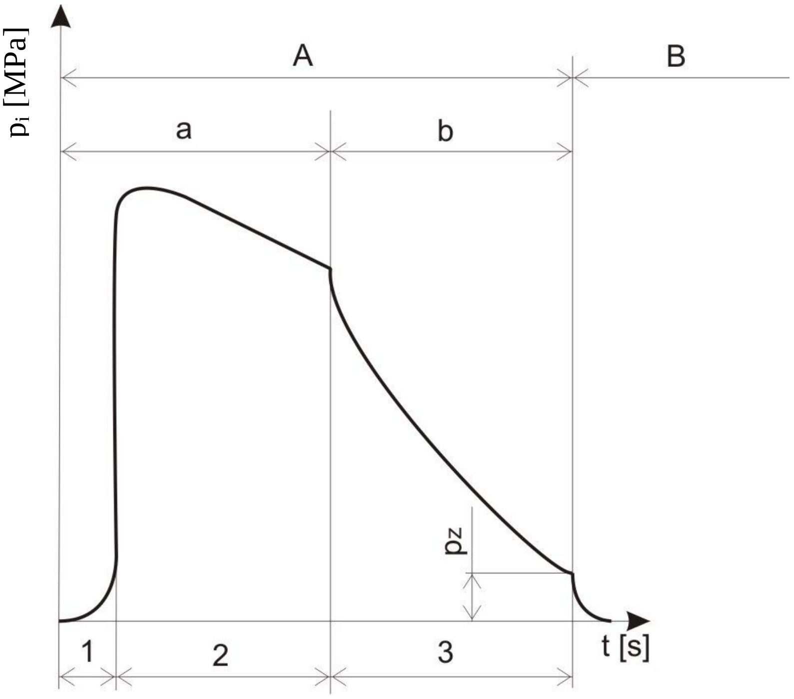


Figure 2 Pressure versus time relation in the mold cavity;

1 - injection, 2 – packing, 3 – cooling, A) closed mold,

B) opened mold, a) injection time, b) cooling time

p_z *– residual pressure [1]*

Cooling phase can be divided into cooling at injection pressure and cooling at descending pressure. While cooling stage is taking place, the molten polymer is affected by shrinkage. To prevent formation of sink marks, it is necessary act by additional pressure on the melt into mold cavity – package. The value of pressure can be constant for whole time of the package phase, or it may be less after a few seconds and cooling takes place at reduced pressure. That is why the pressure is divided into isobaric and isochoric. The specific amount of material has to stay in front of the screw tip in order to pursue package. There are further descends of pressure in the cavity right to residual pressure value (p_z) at cooling stage. [2]

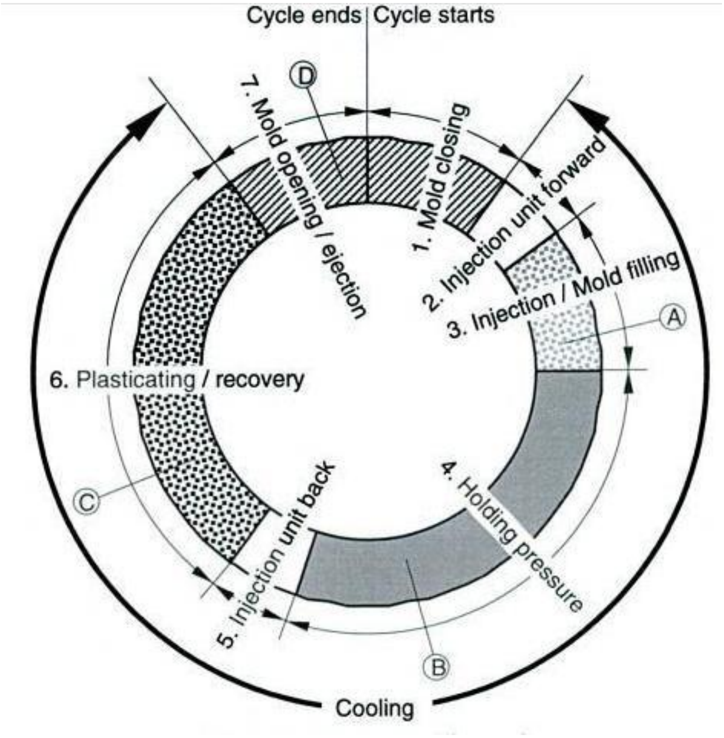


Figure 3 Injection molding cycle [2]

1.2.1 Filling phase

In the step of filling the screw moves forward and the molten polymer is forced to fill the mold cavity while clamping unit holds the mold closed. This phase is labelled A in the injection molding cycle (Fig. 3).

For filling stage, important parameters are injection rate and injection pressure. They are of great importance to the final result, especially when considering factors such as warpage (orientation effects) and surface finish (skin formation). Furthermore, injection rate can be neither too high nor too low. The former can cause jetting and degradation of polymer and this may reflect on mechanical properties. The latter can lead to a thicker frozen layer and short shots and also to incomplete filling of the mold cavity. In other words, too low injection rate affects pressure requirements. [2]

Figure 4 illustrates types of cavity filling process. There are two general types of cavity filling in the figure. The first type (a) is called jetting. It consists of two steps: 1) the polymer flow at high injection rate entraces into the cavity and 2) the free space in the cavity is filled. The second type (b) demonstrates gradual filling of the cavity, where number 3 illustrates the front of the melt, and the area marked 4 is the beginning of melt cooling. The last part of the figure (c) shows the melt cooling in the mold cavity, where the intensity of cooling is given by sequence of numbers. Number 5 illustrates the plastic core of the melt. [3]

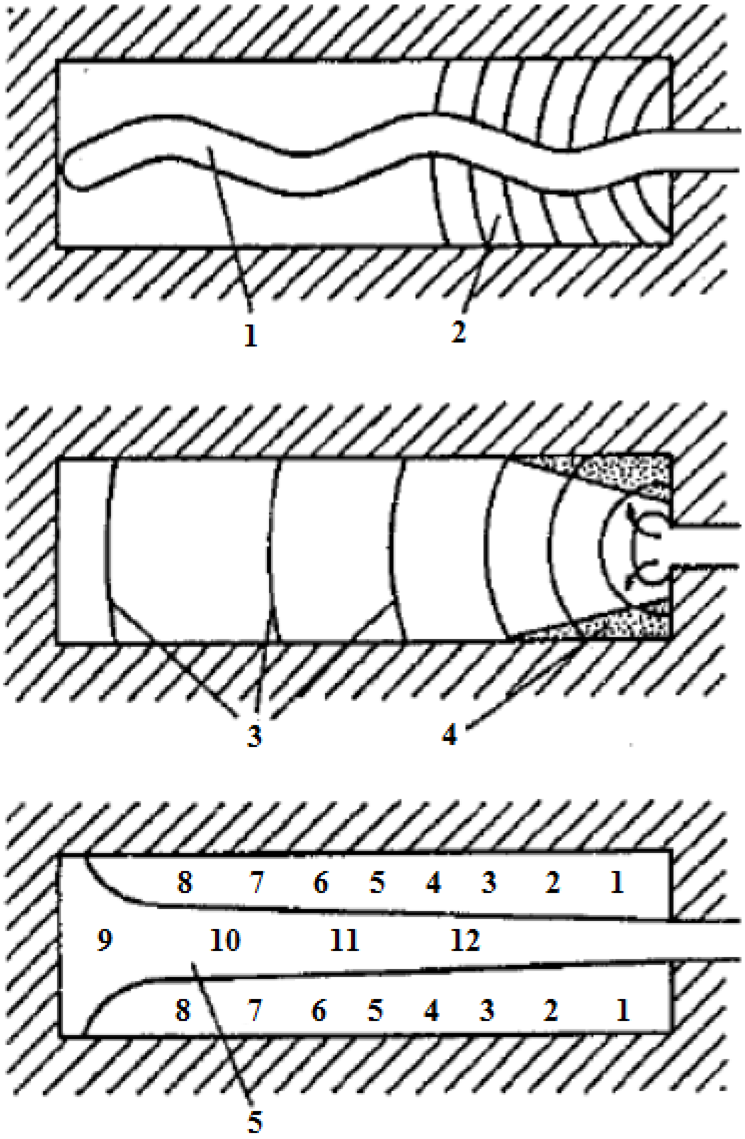


Figure 4 Process of mold cavity filling [1]

1.2.2 Packing phase

When the mold has been filled up, the screw is kept in the forward position or slightly moves to maintain a holding pressure, while the polymer cools down and shrinks. This is why the packing stage is labelled holding pressure in Figure 3. Nevertheless, due to volumetric shrinkage and decreasing density of solidifying polymer, there is a little more free space in the mold cavity, which can be compensated by delivering additional material in this packing stage. Packing is a very important part of the process from the quality point of view, because if additional polymer was not injected, the molded part would shrink, i.e. warp and sink marks would appear. [3]

During the phase of packing, precise adjustment of mold temperature, packing time and pressure are is very important. Packing significantly increases the pressure in the mold cavity, but it is not the only factor. Bad design of the mold cooling system can result in non-uniform cooling on the cavity surfaces, which can cause increased residual stresses in the product. [3]

1.2.3 Cooling phase

The purpose of the cooling stage is to ensure that polymer has solidified sufficiently to withstand the ejection forces. Meanwhile, the screw starts rotating and moves backwards, and the next plasticizing stage takes place. Once the polymer has cooled sufficiently, the produced component can be ejected and the next injection cycle continues. While one part is cooling, plasticization of the next cycle has already begun (C in Fig. 3.). [3]

1.2.4 Ejection (Demolding) phase

When the whole component and runner system (for cold runner systems) are cooled and stiff enough to withstand forces of ejection without deformations, the mold opens and the molded part is ejected with the assistance of ejection pins, tubes or stripper plates. As a result of the effort to shorten the cycle time, the ejection of component including mold opening is one of the shortest stages of the cycle (D in Fig. 3.). [3]

2 INJECTION MOLDING MACHINES

Injection molding machines are assorted according to the machine size, article weight, projected article area and clamping force. While the design of injection mold is in progress, the designer must pay attention also to the machine, including its process parameters. In this chapter the basic types of machines and their construction versions are introduced. [5]

As for the machine size – in principle, the size of the machine must be chosen in accordance with the size of the molded article. However, this is not always true, especially where small molded parts are concerned. It is usually much more economical to manufacture many small moldings simultaneously on a large machine than to manufacture just one article at a time on a small machine. [5]

The size (defined as volume) of the molded part requires a closer examination in its relation to machine selection. The heavier the article, the more molding material must be provided by the plasticizing and injection unit within a given period of time. The selection of a machine for the production of a given product is primarily determined by the projected area of that article – the area which projects in the clamping direction. [6]

The plasticized material is injected into the mold under high pressure, up to 2 000 bar or 200 MPa. The mold must be held closed against the injection pressure so that no melt escapes at the parting line between the mold halves (flash). This clamping force is provided by the clamping unit. Generally, injection molding machines are classified into sizes which correspond to this clamping force (120 – 80 000 kN). [5]

Processing of thermoplastics usually involves the use of horizontal injection molding machines (Fig. 6). Because parting plane between the mold halves is arranged vertically, the completed articles can drop into a container once they are released from the mold (Fig. 6). [6]

In the vertical injection molding machine, the parting plane between the mold halves runs horizontally (Fig. 7). As a result, this machine is especially well suited to the production of insert moldings (e.g. electrical plugs). Most articles made of elastomers are produced on vertical machines. [6]

Figure 5 Horizontal injection molding machine [7]

Figure 6 Vertical injection molding machine [7]

Another type of injection molding machines is rotary table machines; here several clamping units are assigned to a single plasticizing unit. As a result, this machine is best suited to molding articles requiring a long cooling or heating period (Fig. 8). [6]

Figure 7 Rotary table injection molding machine [7]

2.1 Types of injection molding machines

Basically, injection molding machines vary in the straight or two-stage plunger and they also differ in the screw version. Articles can be produced on a single injection molding machine, but each part requires its own different mold. The individual structural units of injection molding machines and their functions will be now described in greater detail. For simulation of the process is important to know how the machine is used for each mold. [6]

- The straight plunger machine

The method of plasticizing thermoplastic material by plunger was originally adaptation of the technique used for molding rubber. The method of plasticizing by plunger is now obsolete and is only used for variegated pattern moldings, such as cosmetic containers. [6]

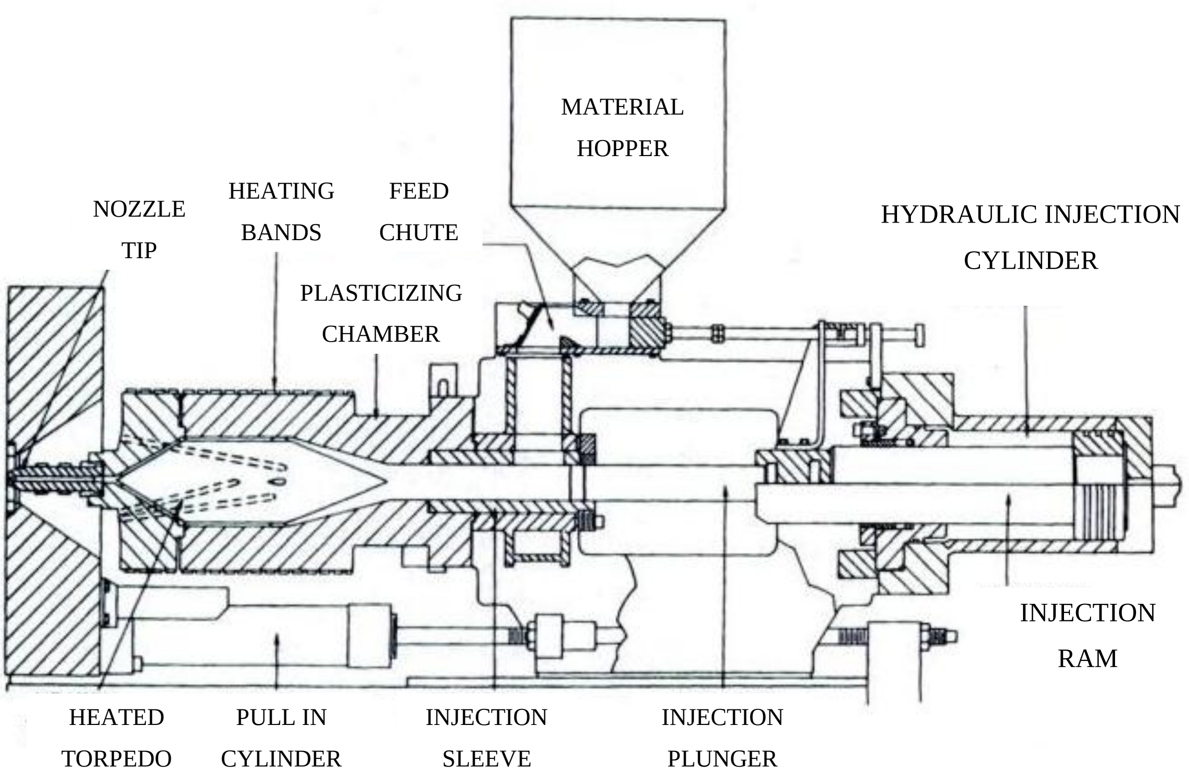


Figure 8 Injection end of a single-stage straight plunger injection molding machine [8]

- The two-stage plunger machine

The obvious disadvantages of the single-stage plunger were overcome by putting one plunger on top of the other and so separating the two functions. The top plunger plasticizes the material, which is pushed in front of the piston of the second cylinder, forcing it back a predetermined distance that corresponds to the proper amount of hot melted material for the next shot. The second cylinder then shoots the material into the mold (Fig. 8). This approach is called preplasticizing. [8]

The two-stage plunger-plunger (Fig. 9) machine is obsolete and has been replaced by the two-stage screw-pot. [8]

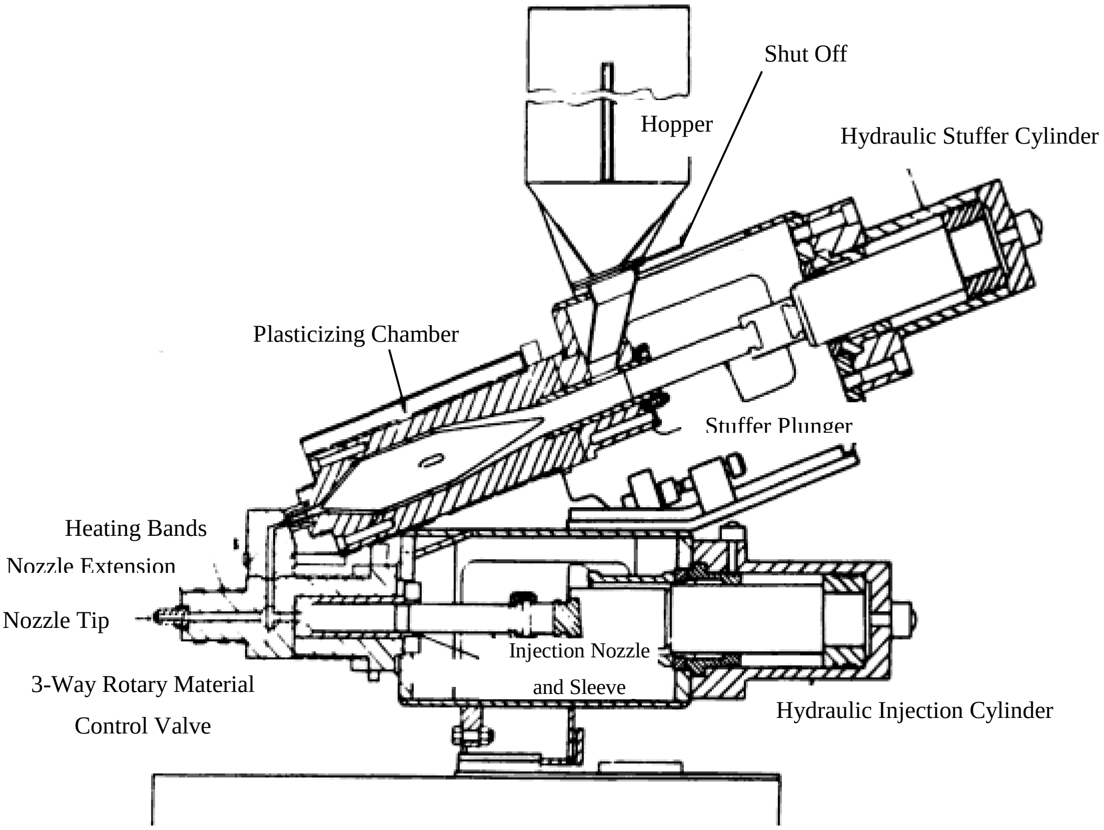


Figure 9 the Two-stage plunger unit [8]

- The reciprocating screw machine

The injection end of a reciprocating screw machine is shown in Figure 11. The extruder screw in the barrel is turned most often by a hydraulic motor instead of electric motor attached to a gear system. [6]

As the screw turns, it picks up material form the hopper. As it progresses down the screw, the resin is compacted, degassed, melted, and pumped through the nonreturn flow valve assembly at the injection side of the screw. This is the check valve which allows material to flow only in one direction. As the material is pumped in front of the screw, it forces back the screw, hydraulic motor, and screw drive system. It also moves the piston and rod of the hydraulic cylinder used for injection. Increasing the resistance requires higher pressures from the pumping section of the screw, and results in better mixing, a slower cycle, and greater energy consumption. [6]

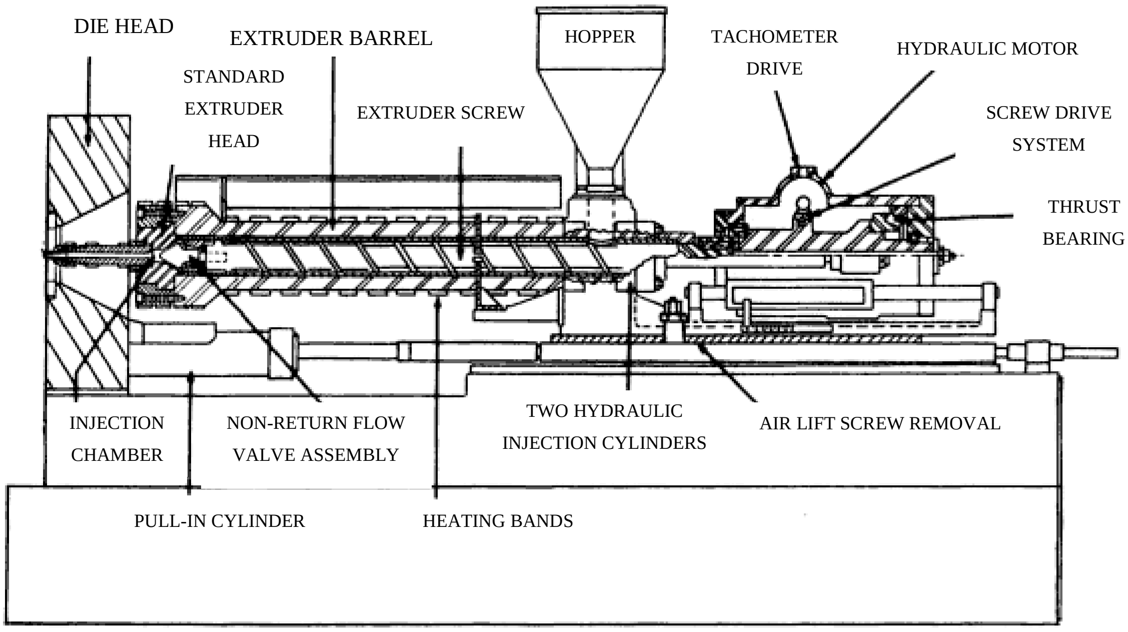


Figure 10 Schematic drawing of a reciprocating screw unit [8]

- The two-stage screw-plunger machine

Using a two-stage screw-plunger machine may overcome limitations of single stage machines. The two-stage screw-plunger machine forces the material to go along the full length of the screw through a rotary valve into the injection or shooting chamber. Here, the injection plunger is forced back until it reaches a predetermined point, at which time the screw stops. The rotary shutoff valve is rotated so that when the injection plunger advances, the material is injected into the mold. [8] This is the essential difference from previous types of injection machines. [6]

2.1.1 Gas- or Fluid- assisted injection molding machines

Gas-assisted injection molding technology (GIT):

As the geometry of molded parts becomes more complex and development in the field goes further, the technology of gas-assisted injection molding is becoming increasingly widespread. The injection molding machine requires special equipment for injecting gas into the mold cavity (Fig. 12). The inert gas, which is always nitrogen, then forms cavities in the interior of the melted polymer. Except gas injection equipment (special nozzle, gas feed unit), gas-assisted injection molding technology can use injection molding machines without restrictions. However, often the gas is introduced directly into the part in the mold. GIT can be used with advantage in the processing high-thickness parts. As for material, GIT is suitable for thermoplastics, thermosets and rubber types.

The essential part of GIT method is delivery of gas; here the following methods may be used:

- Direct connection of the gas cavity from a gas bottle with control valve,
- Storage of a supply of nitrogen in a refrigerated, liquefied state, and supply for injection via a means of pressurization,
- Continuous nitrogen generation by means of air fractionation with a molecular sieve (Fig.12). [5]

Water-assisted injection molding technology (WIT)

There is a considerable similarity to gas-injection molding technology because into the molten polymer additional medium is injected. Unlike in GIT, there is a difference in the injected medium, which is water now. This process takes place in the mold cavity. The water helps to intensify heat removal hence WIT is highly attractive. As a result of water cooling effect, the cooling time of the whole molding cycle can be shortened to 10-20 % of cooling time of classic injection molding. [9] The process of water injection can be done through the injection nozzle or independent injector (in-part injection, or in-runner injection) (Fig. 13). The advantages of WIT are reduction of clamping forces, shorter cycle time (fast cooling due to small wall thickness), and small deformation of molded parts.

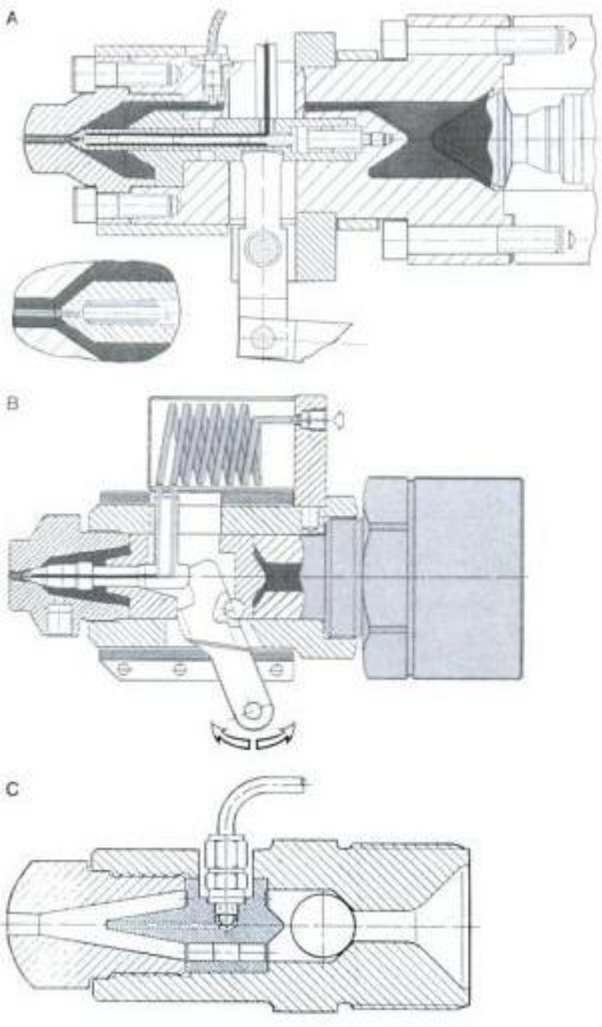


Figure 11 Gas injection nozzles; a: gas melt nozzle; b: injection nozzle; c: airmold nozzle [5]

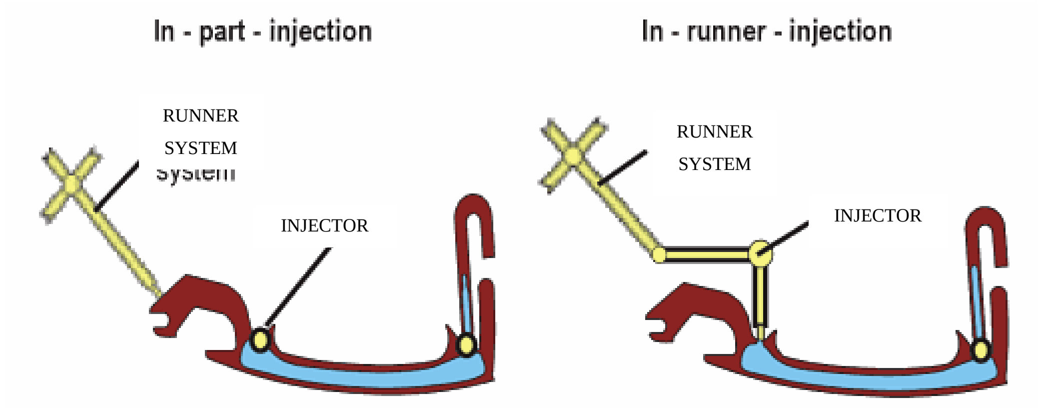


Figure 12 Location of injector [9]

2.2 Injection unit

From wide array of injection molding machines, the one with in-line reciprocating screw is the most common. Its advantage is that it can also be used as a single-stage plunger. Two-stage screw-plunger units with screw preplastication are used on special machines. Single-stage plunger units are only of importance for mini injection molding machines, when the desired shot weight can be provided by a plunger diameter of 2 to 20 mm. [10]

The injection or plasticization unit of the reciprocating screw injection molding machine has a major influence on the quality of the final molded part (Fig. 14). Its basic function is to accept and convey free-flowing solid pellets and additives, perform melting, convey the melt along the screw, mix the plastics additives, possibly devolatilize the melt, inject the melt into a shape-providing cavity, and keep it there under pressure (holding pressure). Its functionality is very similar to that of a single-screw extruder, except that the screw moves axially during the injection and plastication phase. [10]

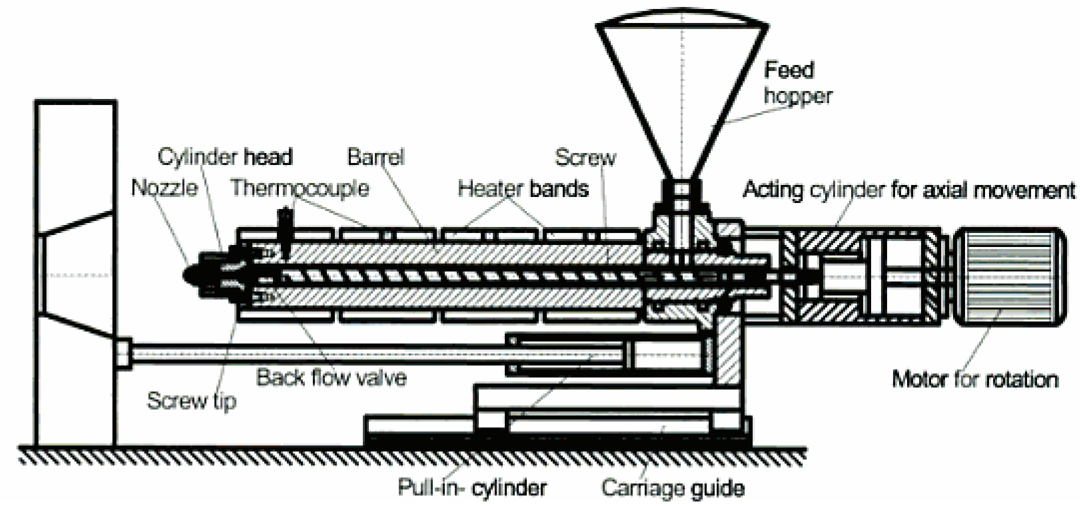


Figure 13 Injection unit and its essential components [10]

The heart of the machine is the screw (Fig. 15). This complicated-to-design component consists of three basic sections:

- The feed zone – through which the plastic passes to the screw barrel
- The compression zone – where the plastic is melted
- The metering zone – where the melted plastic is transported ahead of the screw tip and mixed

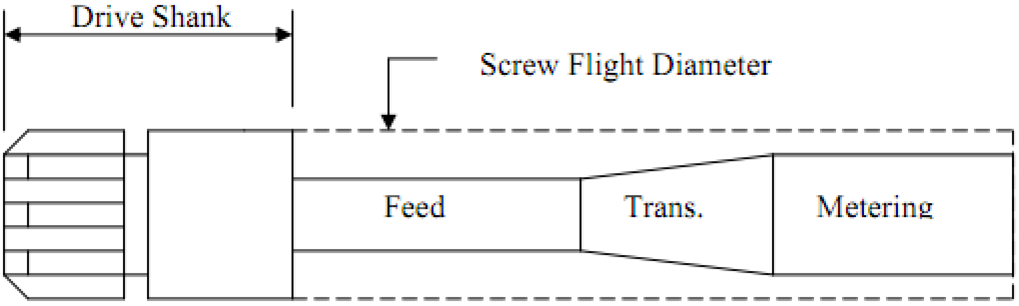


Figure 14 Basic sections of the screw [11]

The screw barrel is fed with polymer granulate through the hopper. The granulate is moving in feeding zone and is heated by heater bands and by friction with screw flights. With the turn of the screw the melt pool starts to appear and develops on the forward edge of the screw flight. As the screw attempts to move the pellets, the melt pool becomes larger and unmelted granulate portion becomes smaller. This trend continues until the metering zone is reached. Material into the compression zone of the barrel is heated by friction and electric heat so that it is melted. Hence into the metering zone of the screw enters material which consists of melt pool and in an ideal case minimum percent of plastic granules. Despite the fact that the compression zone has not uniform volume between every one spiral of the screw, the metering zone has standard flight screw, which serves to complete the process. In other words, the final section of the screw is designed to accurately pump a specific amount of material forward. The relative lengths of each section are variable. Molten material is accumulated in front of the screw tip and in this state is ready for the next step of injection molding cycle.

2.3 Clamping unit

The two mold halves from which the injection mold consists are held and mounted at the clamping unit. To enable effective molding it must provide sufficient clamping force during both injection and cooling steps. The mold must also be smoothly open and close so as to enable ejection of the molded part and the beginning of the next molding cycle. Injection molds can be fit with hydraulics, hydraulic and toggle combination or by electrical clamping unit. [13]

2.3.1 Hydraulic system

A direct hydraulic clamping system is shown in Figure 16. 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 moving platen retracted. [13]

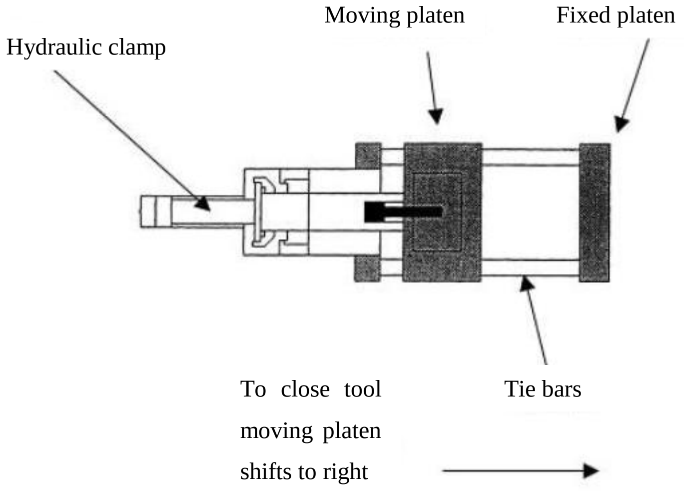


Figure 15 Direct hydraulic clamping unit [12]

2.3.2 Hydraulically-toggle system

Figure 17 shows a toggle type clamping unit. This design enables the force to be amplified. One end is attached to the stationary plate, the other to the movable platen. When open, it forms a distinctive „V“configuration and when closed, the bars form a straight line. The advantage in this design is that a much smaller force from the hydraulic cylinder is required. [13] While hydraulic system requires the application of constant line pressure, the toggles when once extended they remain there until retracted making them self-locking. Nevertheless, it is more problematic to control the force and speed of toggles. If the different depth of mold is required, the toggle must be adjusted for full extension (Fig. 18).

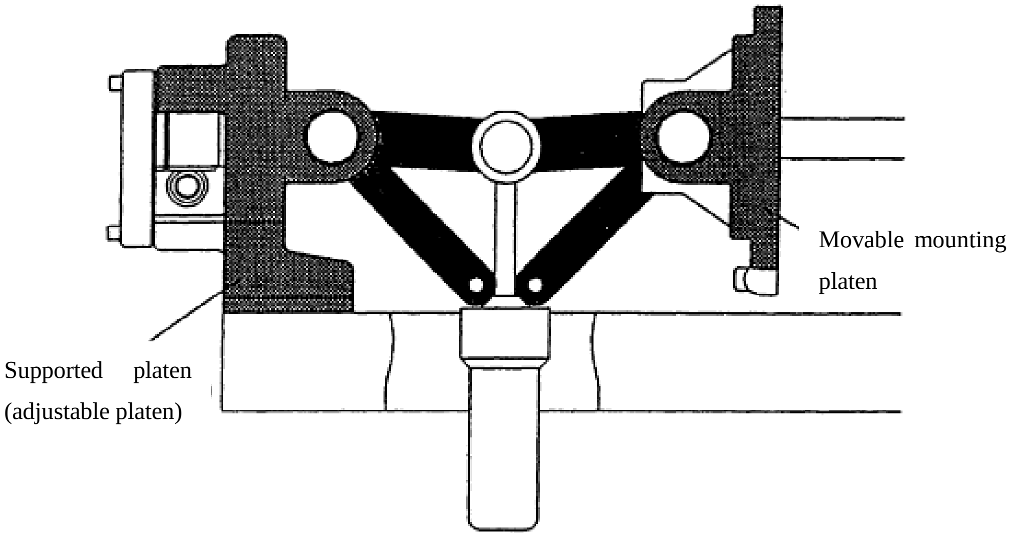


Figure 16 Toggle type clamping unit [12]

An important value of a mold characterization is the clamping area. It is the largest rated molding area the machine can hold closed under full molding pressure and plastic injection. The clamping unit provides accurately controlled motion and force to close and open the mold. When the clamp is closed in a horizontal direction with the platen vertical, the system is referred to as a horizontal clamping system. When the clamp is closed in the vertical direction, it is a vertical clamping system. [2]

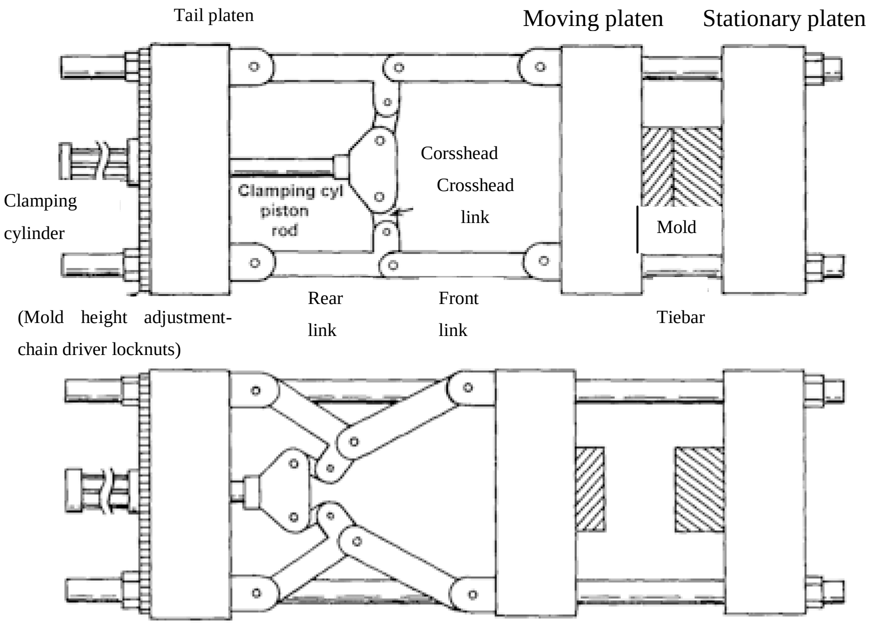


Figure 17 Closed and open toggle clamping unit [13]

2.3.3 Electrically-toggle system

The movement of toggles forward and backward into position using a ball screw mechanism is done through an electrically driven motor. The electric clamping system provides an energy-efficient mechanism to accomplish all injection molding machine functions. An advantage is that pressure can be adjusted independently of velocity adjustment. Furthermore, compared to the hydraulic clamping system, it has significant energy savings. [10]

The essential problem of any electrically driven clamping unit is to transform electrical power into linear motion. Figure 19 shows all-electric injection molding machine in which can be seen that to develop linear motion and transform it to clamping force, a servomotor is needed which is connected with a four-point double toggle through toothed belt. Energy savings, which can be reached with electrically driven clamping unit, are calculated with efficiency of motion transformation and sensors used to ensure accurate approach of mold halves. [10]

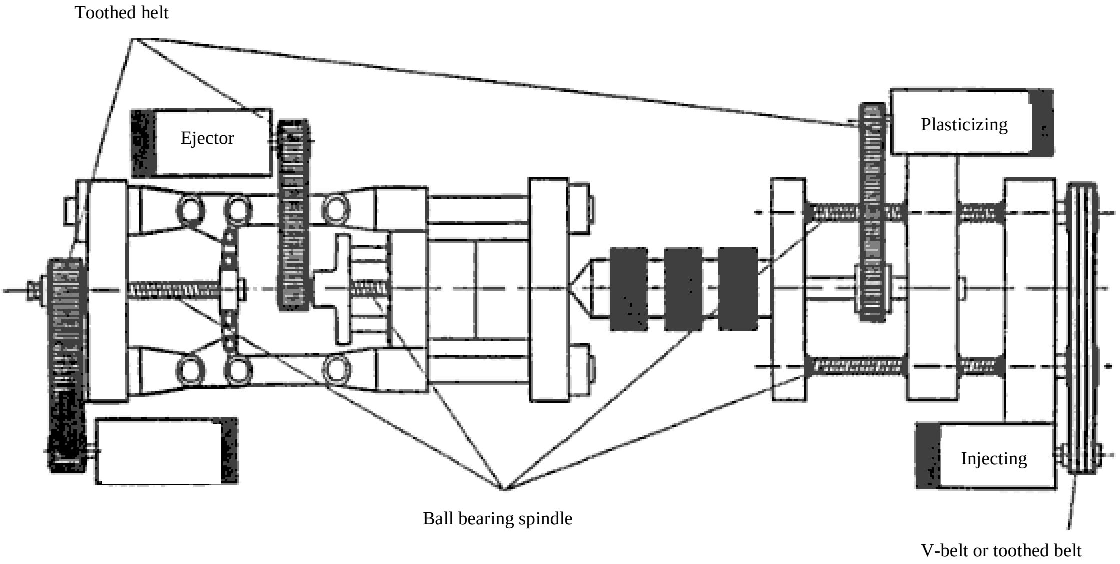


Figure 18 Functions of servomotors in the all-electric injection molding machine. [14]

2.4 Process control

The control system of an injection molding machine comprises all the equipment that controls oil and barrel temperatures, clamping forces, oil pressure, and flow rates, ensuring that they are available in the required magnitude and direction at the right time during the logical sequence of a cycle. Another parameter in the molding process is the mold temperature, which is also important for the correct function of the system. [10]

- Melt temperatures

Melt temperatures are still preferably measured with thermocouples, which are permanently installed. Sensors are always placed at the tip of the probe and, in the hook and cross bar design, they are always directed against the flow of the material (Fig. 20). Of course, they can only measure the temperature of the melt next to the wall. [10]

- Melt temperatures in the mold

To measure temperatures of the molten plastic in the cavity thermocouples are used which are installed in the wall of the cavity. Contact of sensors and cavity wall results in inaccurate values, thus the accurate value of the temperature must be determined by both the melt and the wall temperature.

Melt temperatures in the molds can be measured with IR sensors connected via fibre-optic waveguides reproducibly with sufficient accuracy and at high speed (response time ca 10 ms) in the range of 110 to 400 °C. For optical sensors directly coupled to the IR amplifier, temperatures from 70 to 260 °C can be measured. [10]

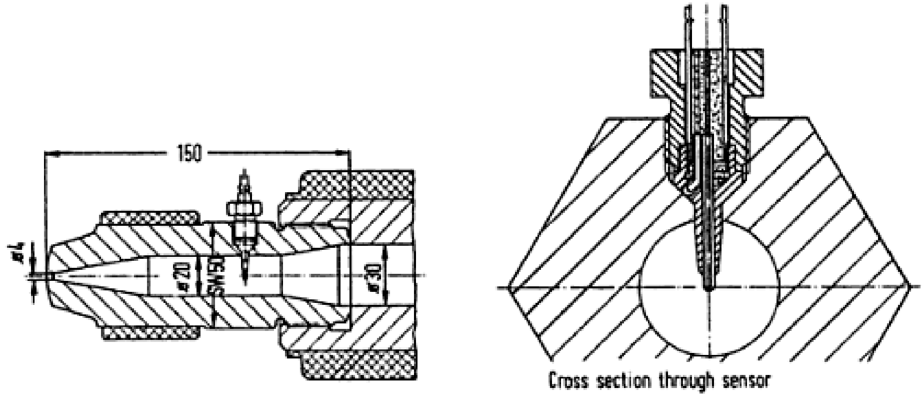


Figure 19 Nozzle with correctly installed temperature sensor and cross-section. [10]

The following temperatures are controlled in the process: [10]

- Barrel temperature
- Mold temperatures
- The temperature of hydraulic fluid in the drive system
- The coolant temperature

3 INJECTION MOLDS DESIGN

The primary tasks of an injection molding tool consist in accommodating and distributing the melt, forming/shaping and cooling the molded article and its removal from the mold. The secondary tasks - absorbing the forces, transmitting motion, and guiding molded parts - are derived from these primary assignments. The polymer granulate is prepared into the flowable state and then it is transported from the injection chamber into the mold. The cavity is the impression space into which the molding compound is injected. The thermoplastic material cools and solidifies within the cavity, hence forming the molding. In one cycle, one or more products (multi-cavity mold) can be molded. There are many different types of molds, designed to meet many different product requirements. [6]

One of the tasks is keeping the melt within the mold, and it requires two basic conditions. The first is that the mold resists the enormous forces that can tend to make the mold open. The other is that the mold contains a feed system connecting the nozzle of the molding machine to all cavities within the mold. Other conditions of the mold design are:

- Contain melt: resist displacement, guide melt
- Transfer heat: lead heat away from the part and from the mold
- Eject part: open the mold, remove the part [15]

An injection mold can have many structures to accomplish the functions required by the injection molding process. Since there are many different types of molds, the first part of the chapter is dedicated to the two-plate injection mold because it is the simplest version of the construction. Samples of injection molds construction are viewed on Figures 21 and 22. [15]

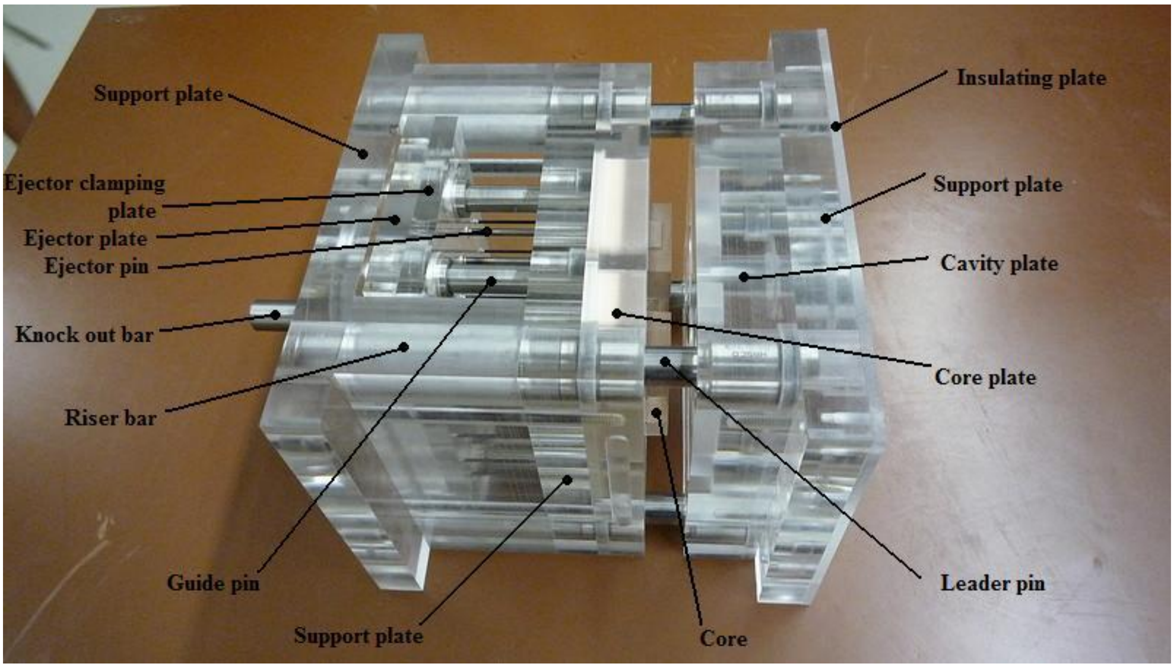


Figure 20 Sample of an injection mold design

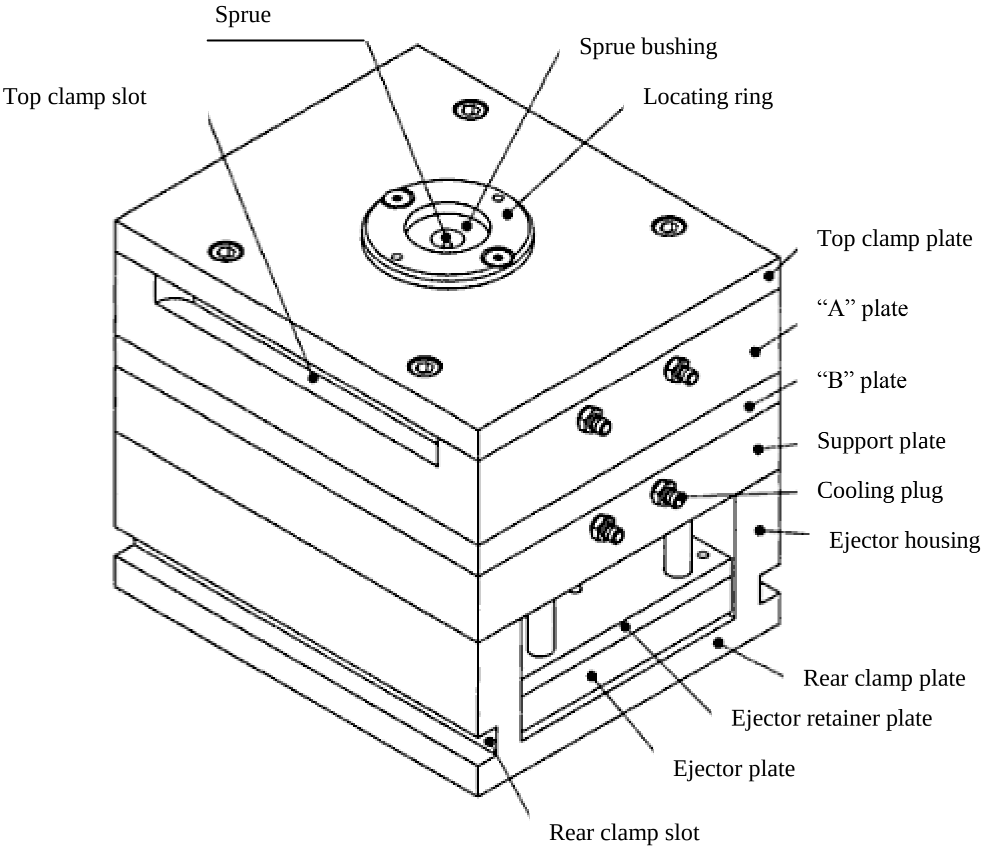


Figure 21 External view of mold [15]

3.1 Mold conception

Generally, industry identifies six basic mold types for use with thermoplastics. [2]

- The cold-runner two-plate mold
- The cold-runner three-plate mold
- The hot-runner mold
- The insulated hot-runner mold
- The hot-manifold mold
- The stacked mold

3.1.1 Two-plate molds

As shown Figure 23, the two-plate mold consists of stationary plates and moving plates and it is the simplest mold design. Mold cavities are placed in one plate only and one parting line divides two halves of the mold. In the stationary half of the mold a central sprue bushing can be placed, or it is possible to have a multi-impression mold with direct runner system. The ejection system is included in the moving half of the mold. Two-plate systems can be applied to any number of cavities, and not necessarily to cold runners only. [2]

When the mold is closed, guide pins and bushings are used to closely locate the “A” and the “B” plates on both sides of the parting plane, which is crucial to the primary mold function of keeping the melt. Poor quality of the molded parts and accelerated wear of the injection mold may be caused by improper construction of the mold components, especially unsuitable alignment of the plates. [2]

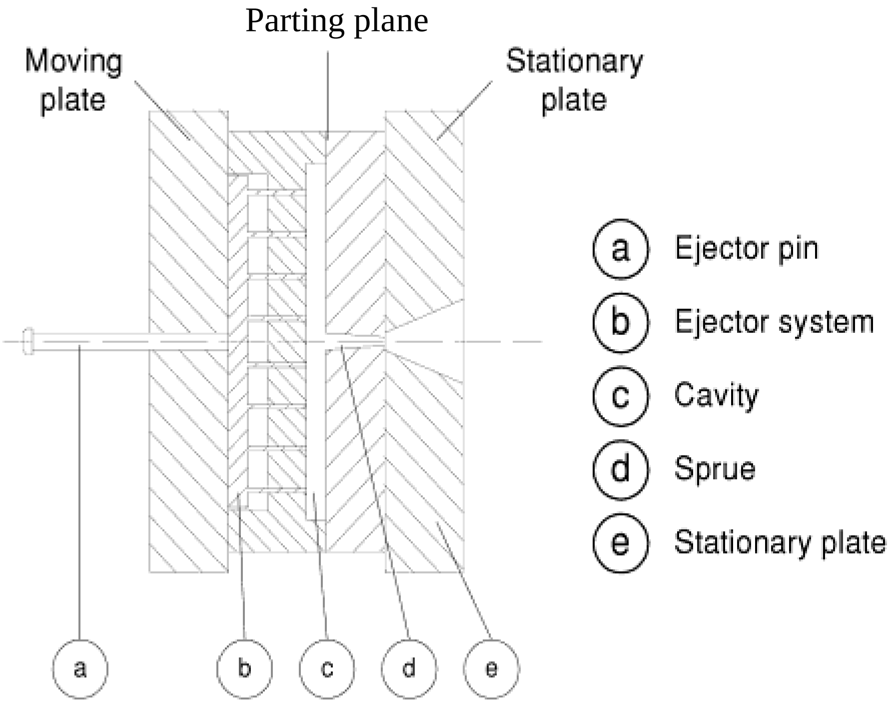


Figure 22 Schematic drawing of the two-plate injection mold [12]

3.1.2 Three-plate molds

As the name indicates, the basic three-plate mold contains three plates: [2]

- The stationary or runner plate, which contains the sprue and half of the runner
- The middle or cavity plate, which contains half of the runner and gate and its purpose is to float when the mold is open
- The movable or force plate, which contains the molded part and ejection system.

Unlike the two-plate injection mold, the three-plate mold provides a second parting plane, located between the “A“ plate assembly and the top clamp plate. During molding, the plastic melt flows from the nozzle of the molding machine down the sprue bushing, along sprues into the mold cavities, where it is shaped into the final part, while the feed system freezes.

When the mold is opened, the molded cold runner will stay on the stripper plate due to the inclusion of the sprue pullers that protrude into the primary runner. As the mold continues to open, the stripper bolt connected to the “B” plate assembly will pull the stripper plate away from the top clamp plate, stripping the molded cold runner off the sprue pullers. The stripper plate may then be actuated to force the moldings off the core. [15]

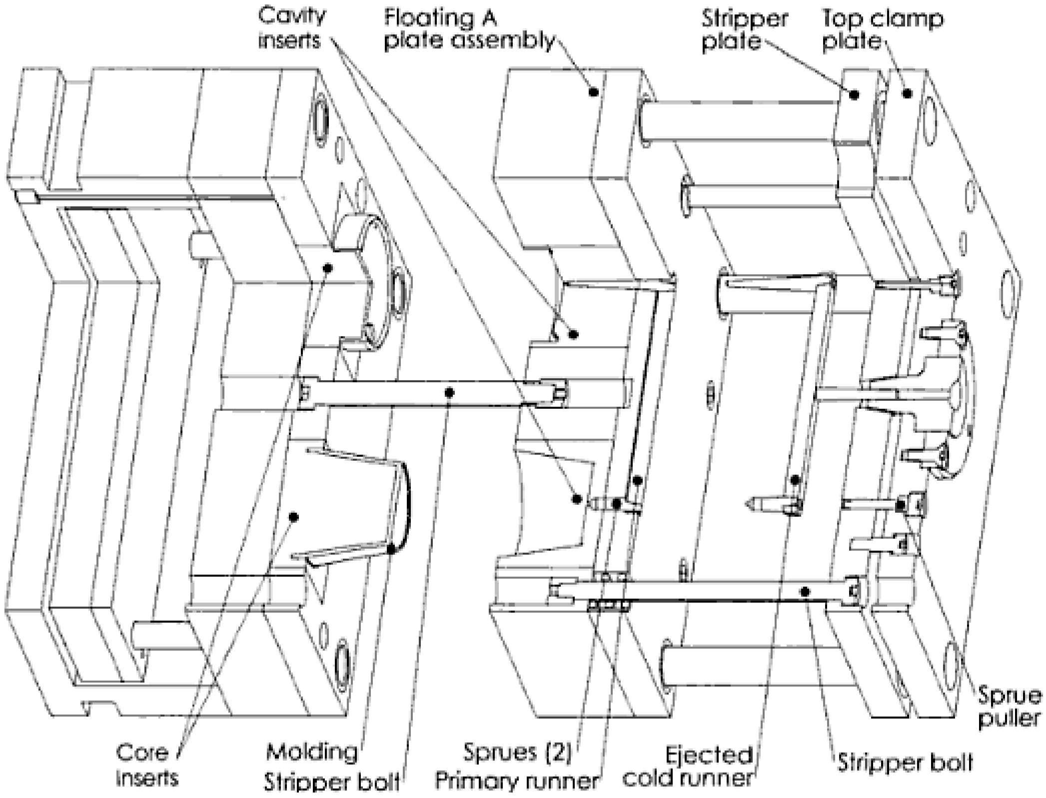


Figure 23 Section of a fully open three-plate mold with the moldings on the core inserts [15]

3.1.3 Cold runner system

A cold runner mold is defined as a mold in which the plastic material in the runner is cooled and ejected from the mold during each mold cycle. As shown in Figure 25, molten plastic material is injected through the runner, the gate, and then into the part-forming cavity. This molten plastic is then cooled by the tempered mold, and when sufficiently solidified, the mold opens and the runner, gate, and molded part are ejected. [16]

Among the advantages of cold runner systems over hot runner systems belongs their simplicity, thanks to which they are cheaper to build. In addition, the energy to maintain cold runner systems is less because it is not necessary to heat them. Hence, the cost to running these systems is less. On the other hand, the biggest disadvantage of the cold runner system is the fact that the unwanted stiffen runner must be dealt with, which requires the need to separate the molded parts from the runner. [16]

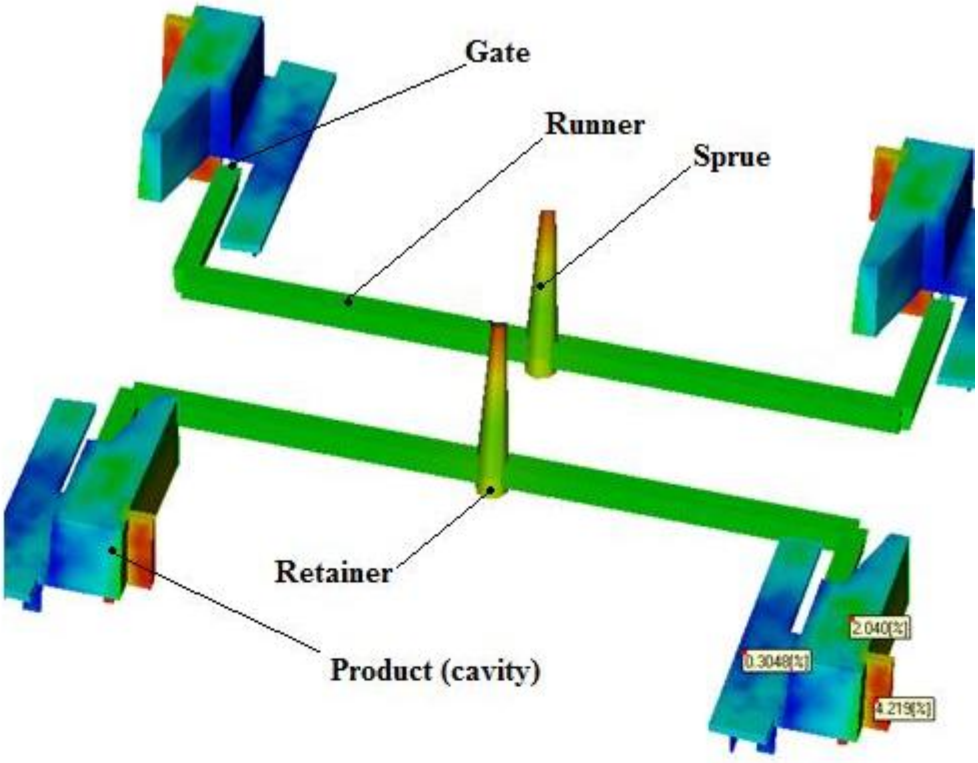


Figure 24 Sample of the cold runner system used for simulation purposes

3.1.4 Hot runner system

The hot runner technology offers the partial or total elimination of sprues and runners and, hence, a reduction in the cost of finishing and or the recycling of scrap material. In addition, the technology is suitable for long production runs and runs which do not require frequent colour changes. By contrast it involves molds which are more complex and, as a consequence, more expensive. Furthermore, the technology requires accurate temperature control and very skilled production and maintenance staff.

Beside the above mentioned, the advantages of hot runners are following: [16]

- Reduced energy (plastication, filling pressure, granulators)

  The hot runner requires less material to be plasticized (no runner to fill) and eliminates the use of granulators.

- Improved automation

  The hot runner system eliminates the need for automation equipment that would be required to handle a cold runner.

- Reduced injection pressure

Flow channel can often be larger than cold runners.

- Cleaner work environment

  Eliminating the need to handle the runners and resulting regrind.

Among the most significant disadvantages can be included cost of the system and maintenance, and more process sensitive issues, such as gate freeze, gate drool, balancing of flow and melt conditions to each gate, thermal control and over-packing. [16]

The hot runner technology uses heated sprues, nozzles and manifolds. For a multi-cavity molds, externally heated manifolds with attached nozzles are used, which can be heated internally, or externally (Fig. 26).

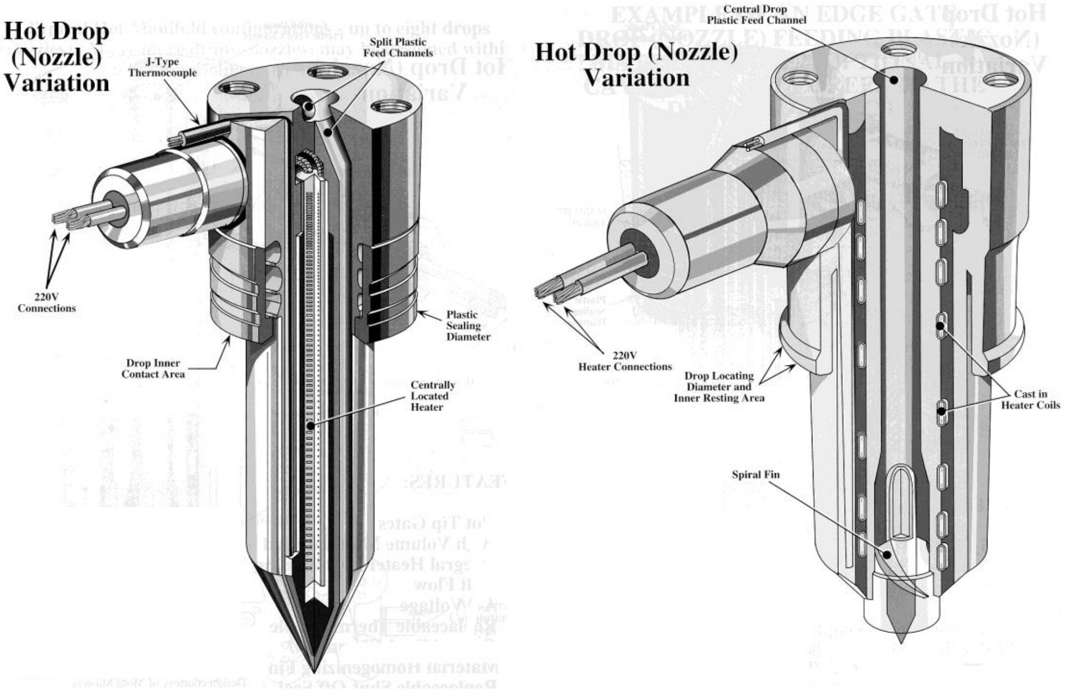


Figure 25 Internally heated nozzle on the left side;
externally heated nozzle on the right side [17]

Figure 27 shows complexity of a hot-runner mold design. The complexity of the mold is more noticeable in case of combination of hot and cold runners. In this case, molten material passes through the hot-runner (hot sprue, potentially hot manifold and hot nozzle), then enters the cold sprue and runners and finally flows through the gate into the part-forming cavity. Combinations of hot and cold runners are commonly used for multi-cavity molds. [16]

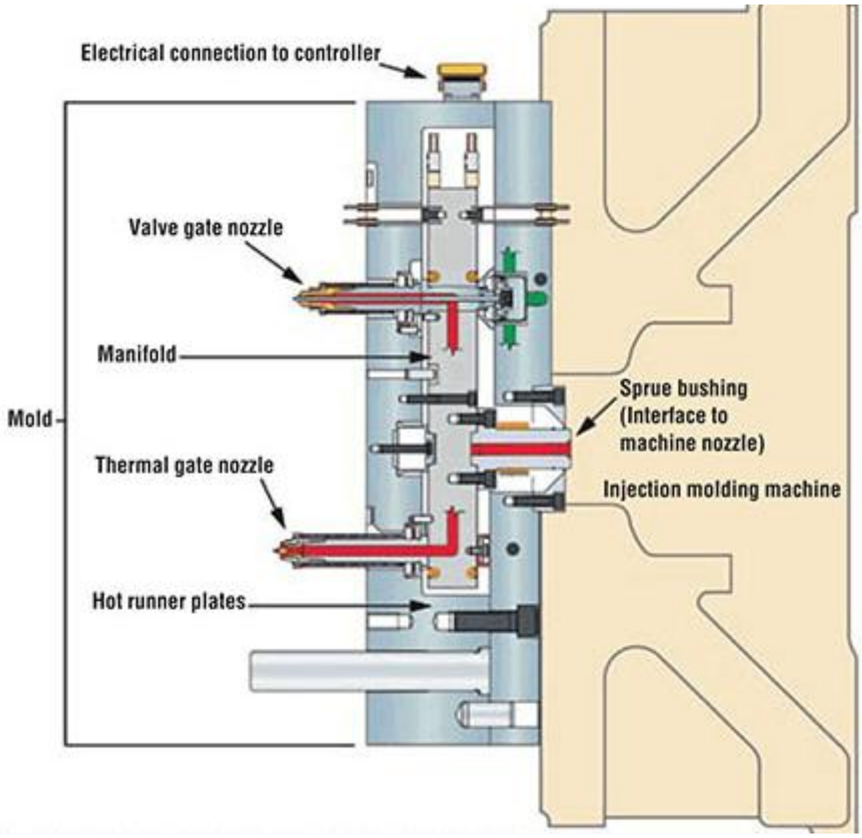


Figure 26 Section view of the hot-runner system [18]

3.2 Runner system

This chapter provides a brief orientation of runner and gate types. The following facts can be helpful for further considering and deciding about runner design and location.

In many cases, the mold design dictates the runner position, although ideally the optimum gate position should be determined on the base of the part requirements and afterwards the mold design should be selected to provide for the gate desired position. Available runner positions and gate designs depend significantly on whether the runner travels along the primary parting plane of the mold or whether it does not travel along this plane. [16]

Runners can be located in various ways: [16]

- Primary parting plane runners

  This type of runner is used in two-plate cold runner molds. The mold opens and closes on the primary parting planes (parting lines), and hence it allows ejection of the molded part and/or the runner.

- Sub runners

  They are generally placed parallel to the primary parting plane, but not along it. The sub runner type can be in either a cold runner or a hot runner mold.

- Hybrid sub-runner and parting line runner

  In this case, the mold contains both above given runners (most common when a hot runner is used). Here, the hot runner would deliver the melt to a cold runner or gate along the primary parting line.

The link between the part and the runner system is the gate. It has a significant influence on the quality of molded parts, especially its size, shape and location. The cross-section of the gate needs to be relatively small to allow separation of the part from the runner system. In contrast, a gate design that is too small might inhibit packing of the cavity and cause overshearing of the material, jetting, and other gate-related defects. [16]

Gates placement:

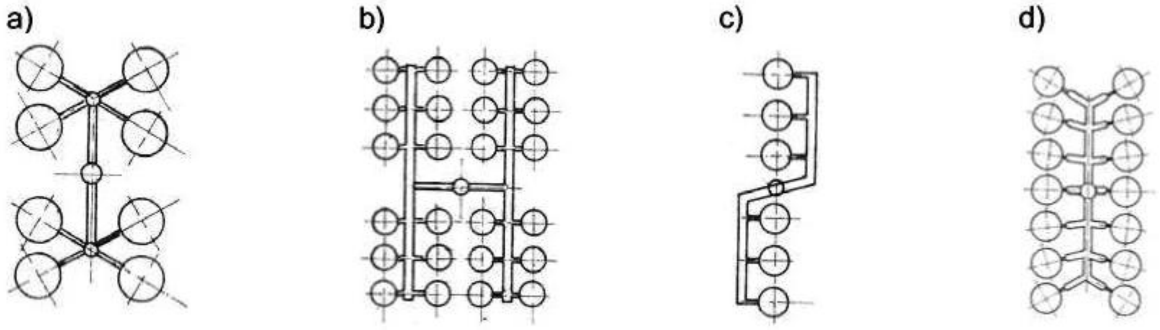


Figure 27 Line placement of the runner system; a) cavities with equal length of the melt flow, b) - d) cavities without equal length of the melt flow [17]

As shown in Figure 28, the placement of cavities is very important because of the length of the melt flow within the runner. In an ideal case, all cavities should be filled in the same time. This is why there is an effort to optimize the length of the runners, which would prevent from overfilling of the cavity and other effects. To ensure this, symmetrical placement of the runner system is used (Fig. 29). [16]

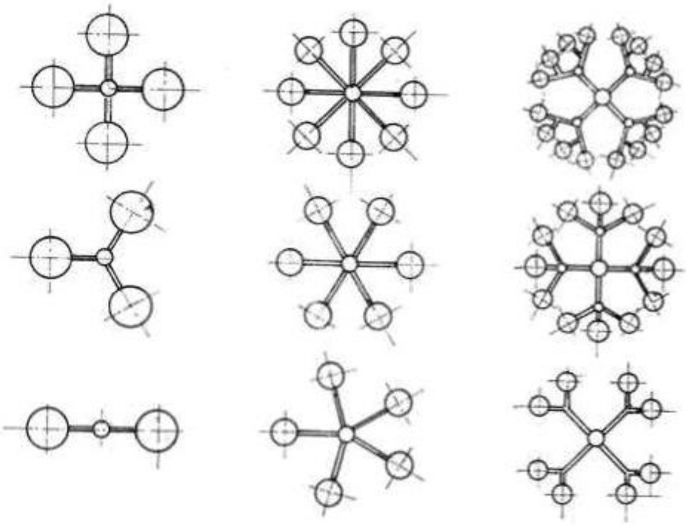


Figure 28 Symmetrical placement of the runner system [17]

Gates can be of different types, depending on the shape (Fig. 30): [17]

- Sprue gating

  the part is gated directly from the sprue. It is used in single-cavity molds, and the melt is delivered to the centre of the cavity.

- Film gating

  can be used for flat designs or large areas where warpage must be minimised, and also can be considered an extension of fan gating.

- Tunnel gating

  enables automatic degating of a part from the runner system during the ejection cycle. Recommended minimum diameters for tunnel gates are 0,8 mm for unreinforced and 2 mm for reinforces parts.

- Disk gating

  is suitable for cylindrical parts requiring good concentricity and weld line strength. A post-molding operation – degating – is required to remove the disk. A maximum land length of 0.5 – 1 mm is recommended.

- Fan gating

  is a special type of edge gate, used to feed flat, thin sections. It spreads the material flow uniformly across the cavity, helps reduce warp, and is suitable for rectangular parts. The gate area should always be less than the cross sectional area of the runner.

- Tab gating

  can be used for lenses and flat parts. It reduces gate blush and residual stress in the gate area.

- Ring gating

  enables free flow of material around the core prior to moving down to fill the mold, as a uniform extrusion. It is used for cylindrical shapes.

- Spoke gating

  also called spider gate, is used for cylindrical parts where tolerances are not critical. It can be used in combination with either a ring, or fixed cone-gate.

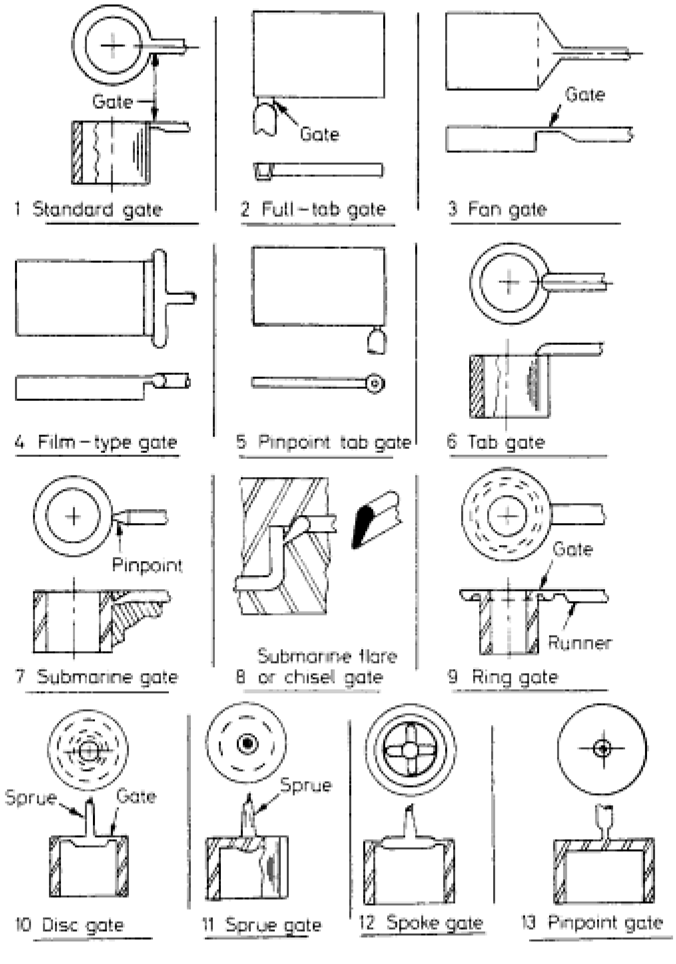


Figure 29 Types of gating [16]

3.3 Cooling system

The cooling system has a significant influence on the economics and operation of the designed mold, and it is one of the most under-engineered systems in injection molds. The cooling system which is desirably designed for the molded part can prevent several defects. On the other hand, the results of improperly designed cooling systems have often two undesirable outcomes. First, the cycle time is too long and second, there are significant temperature gradients within the mold, which results in differential shrinkage and warpage of the moldings. A carefully designed cooling system, which has heat flow throughout the mold without incurring complexity or cost, leads to effective operating injection molding. [17]

The main purposes of the cooling systems are:

- Maintain uniform wall temperature of the mold cavity
- Reach optimal phase for stiff and cool of the polymer
- Reach optimal injection time.

Table 1 Mold and melt temperatures of some polymers processing [9]

Polymer	**Melt temperature [°C]**	**Mold temperature [°C]**
ABS	190-250	50-85
HDPE	180-270	20-60
LDPE	180-270	20-60
PA 6	230-290	40-120
PC	280-320	85-120
PMMA	200-250	50-80
POM	180-220	50-120
PP	170-280	20-100
PS	180-260	55-80
SAN	200-260	50-85

The cooling system can be manufactured in two ways. The first, for conventional molds processing thermoplastics, the area for cooling medium is fabricated by conventional working technologies, such as drilling. The other, used for inserts, uses direct metal laser sintering technology. Holes trajectories for cooling medium are randomly shaped, thus the cavity insert can be cooled as much as possible. A disadvantage of this relatively new technology is its cost and the fact that it can be used for cavity inserts only. [1]

Cooling agents are:

- Active: fluids, air, electric heaters
- Passive: Its physical properties influences the heat remove from the mold, for example material of the cavity inserts [1]

Design of the cooling system must take the following principles into consideration:

- Cool fluid is led from the most heated areas
- Channels are not located near the corners of the part
- Channels diameter is greater than 5 mm
- Channels are located in an optimum distance from the cavity, while stiffness is retained
- Volume rate of flow is regulated according to the hotter and colder areas. [1]

The cooling system is obviously located both in the injection side of the mold and also in the ejection (moving) side. It is created by a complex of channels, which are used to allow the cooling medium to pass through and to take heat away. The distance of the channels from the cavity must be optimal, with consideration of stiffness and strength of the mold. The coolant channels are located uniformly around the cavity (Fig. 31). [19]

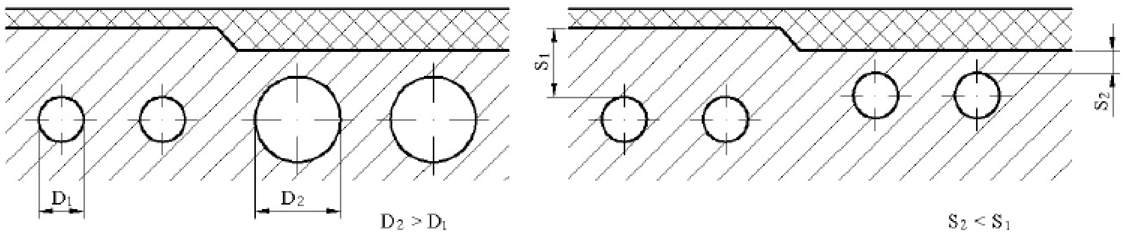


Figure 30 Location, distances and diameters of cooling channels

D_1, D_2 = diameters of channels; S_1, S_2 = distances from the cavity [9]

The developed computer-aided engineering could be very useful for injection mold design in obtaining an optimal configuration of the cooling system in terms of radii and locations of cooling channels. As well as determining the optimal processing conditions of the cooling stage in terms of the inlet coolant bulk temperature and inlet coolant volumetric flow rate of each cooling channel by minimizing certain objective functions related to the part quality and/or the productivity in the injection molding process. [19]

Cooling of injection molding tools is crucial to their performance, influencing both the rate of the process and the resulting quality of the parts produced. However, so far cooling circuit design and fabrication have been confined to relatively simple configurations, primarily due to the limits of the fabrication methods used to make molds, but also due to the lack of design methodology appropriate for cooling circuits. [19]

3.3.1 Conventional cooling systems

Conventional cooling is realized with cooling channels, spirals, baffles (Fig. 32), or thermal tubes. Channels are prepared by conventional ways of machining, such as drilling and milling.

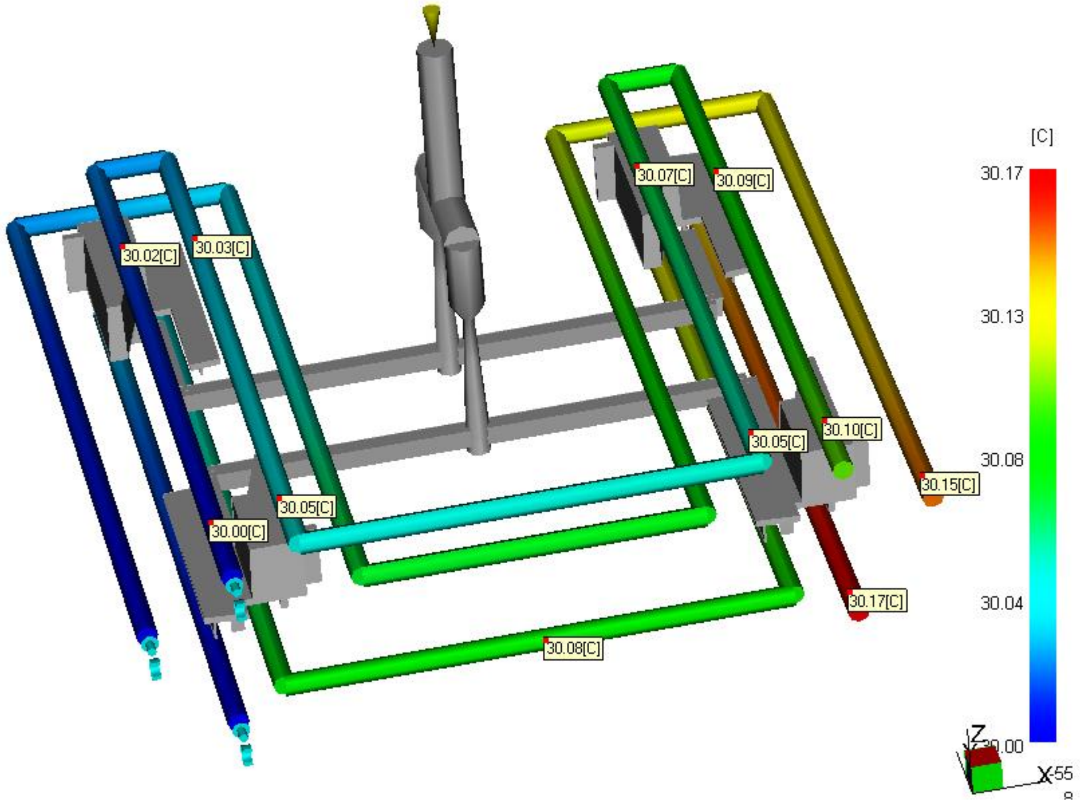


Figure 31 Example of drilled cooling circuit, inlet and outlet temperatures are shown

One of the biggest advantages of drilled cooling circuit is its simplicity and cost. In an ideal case, the designed cooling system has both ideal heat transfer in terms of optimal cooling time and simplicity of manufacture, which means also low cost. For monitoring thermal process in injection molds are used simulation softwares, which can ease designers deciding about location, distances and diameters of cooling channels, thus they can make cycle time shorter and influence the product quality. [19]

3.3.2 Conformal cooling systems

Conformal cooling is defined as the ability to create tempering configurations within a tool that essentially follows the contour of the tool surface or deviates from that contour as thin/thick sections of the part may dictate for optimal thermal management. In general, the objective is to cool the part uniformly, as well as the conventional cooling system does. Conformal cooling provides a tremendous advantage in mold tooling through significant reductions in cycle times. Beside the obvious piece-cost savings, other benefits include mold and equipment savings.

Recent studies show that conformal cooling may reduce cycle times between 30 to 60 % over conventionally cooled molds. This saving is very much geometry-dependent. It should be noted that in many cases, resurfacing the mold might not be necessary to enable the prototype tool to be used in production. Depending on the types of parts being produced, such as volume, surface requirements or abrasiveness of materials, the prototype mold may be sufficient for production as built.

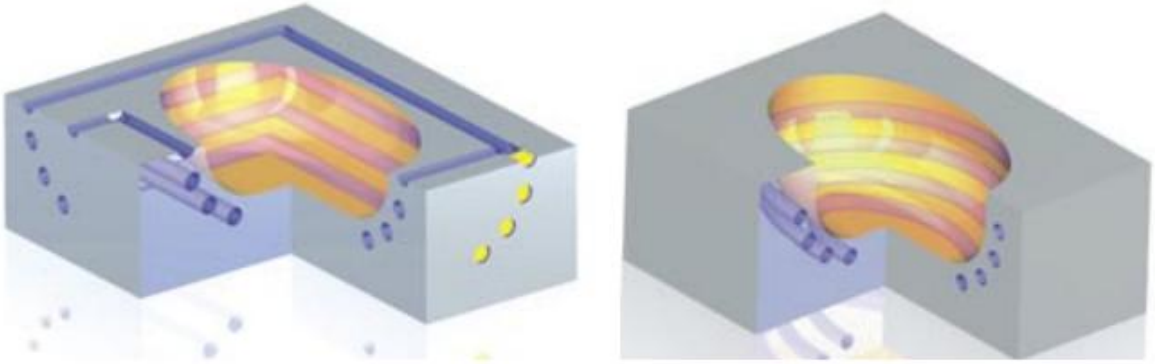


Figure 32 Conventional cooling channels (left), conformal cooling channels (right) [20]

As can be seen on Figure 34, unlike conventional cooling system, conformal cooling channels can be led in random trajectories. [19]

3.4 Ejection system

Once the component has solidified and cooled down, it needs to be removed from the mold cavity. In an ideal case, this is done by gravity, i.e. the part falls down. However, some components with design features such as adhesion or internal stresses, undercuts or brush surfaces may have to be removed from the mold by robots or manually.

The ejector system is normally housed in the movable mold half and is usually actuated mechanically by the opening stroke of the molding machine, or it can be done pneumatically or hydraulically.

Mold opening causes the mechanically actuated ejector system to move towards the parting line and to eject the molding (Fig. 35). The result of this procedure is that the molding stays on or in the movable molds half. This can be achieved by undercuts or by letting the molding shrink onto a core (Fig. 38). [12]

In general, the molding can be ejected from the mold by following types of ejection systems: [2]

- Ejection pins
- Stripper plates
- Tube ejectors
- Inclined ejection
- Two-stages ejection

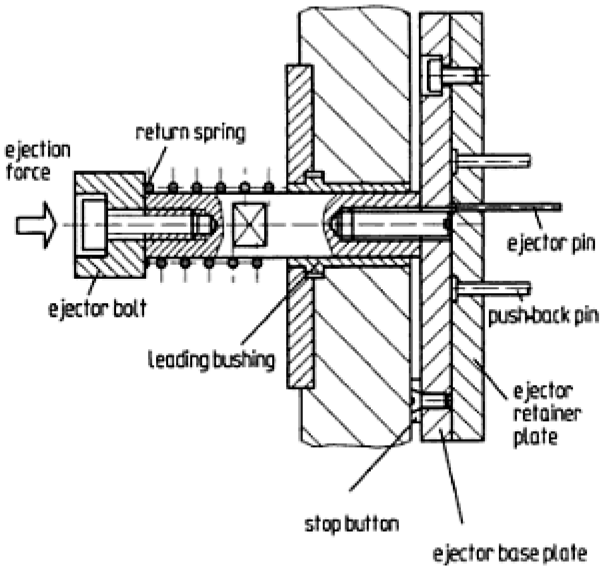


Figure 33 Ejector system [6]

The requirements ofn the ejector systems are: [2]

- No visible marks should be left on the part
- The part should be ejected without destroying it
- The ejector system should be coordinated with the cooling system

- Parts should be loaded equally during ejection
- Ejector pins should have a set position.

Parts can be ejected by many different systems besides ejector pins, as given above. For parts with a central cylindrical core, ring ejectors are often used because they give a better transmission of ejecting force. In addition, they do not leave visible marks on the part.

Ejector pins are used for ejecting the part from the mold after the injection process is complete (Fig. 36). If the pins are too long, the plastic will flow around them and cause a depression in the part. [2]

Stripper-plate ejection is generally used when ejector-pin marks would be objectionable on the part and maximum ejection surface is required. Stripper plates are used for single- and multiple-cavity molds. An angle of aproximately 5° is machined in the stripper plate and on the plunger, which prevents scoring of the plunger as the stripper plate moves in and out over the plunger. [2]

Tube ejectors act on the molded part by annular area, which means that unlike with the pin ejector, the part is strained by the same force but greater cross-section.

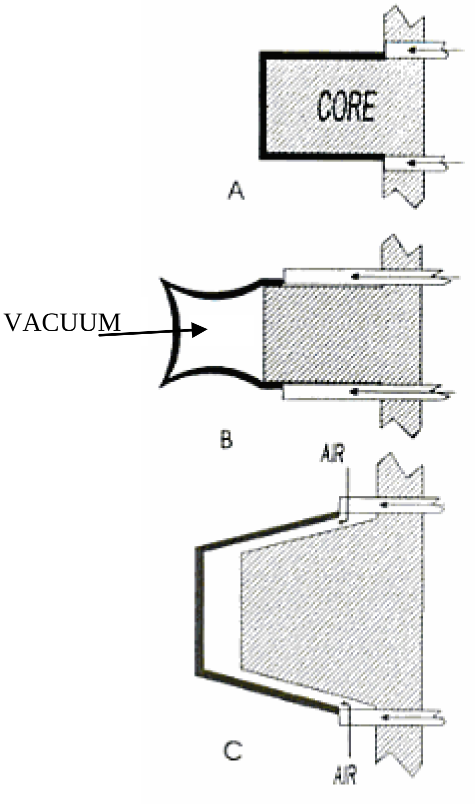


Figure 34 Using of ejector pins. A), B) Difficult ejection due to vacuum formation, C) remedied by tapering inside and outside walls [2]

Inclined ejectors are viewed in Figure 37.

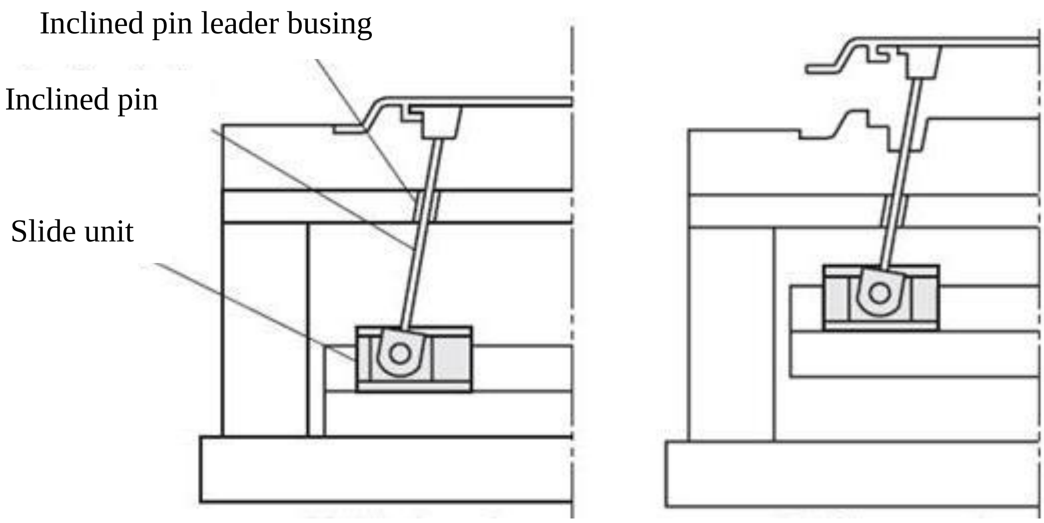


Figure 35 Inclined ejectors within closed (left) and opened (right) mold [21]

Two-stage ejection is the combination of two mechanical ejection systems in one mold, acting on the same product, and can be combinations of these motions: [21]

1. Ejector - ejector

2. Ejector – stripper
3. Stripper – ejector
4. Unscrewing – stripper
5. Unscrewing – ejector
6. Side core – ejector.

Any of these systems can be actuated by the following:

1. Machine ejector (usually hydraulic)
2. Fixed (bumper) ejector (rarely used today)
3. Cam
4. Air operator (cylinder), internal or external
5. Hydraulic operator, where large forces are required. [21]

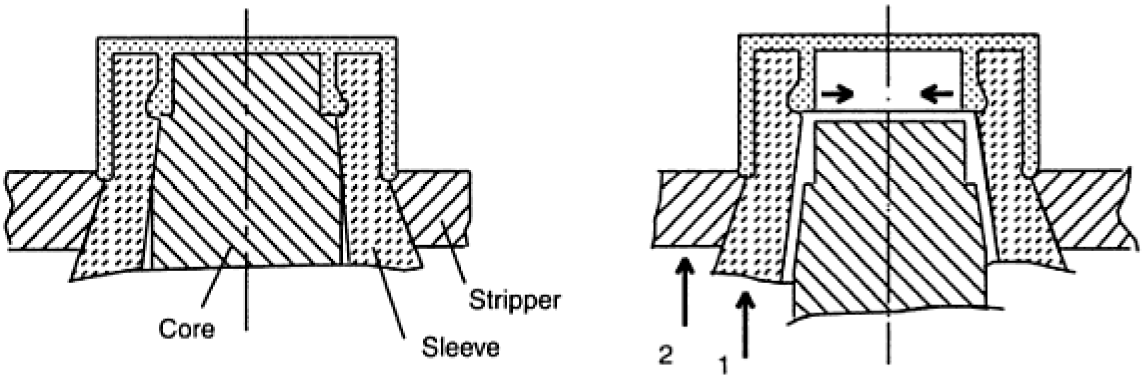


Figure 36 Example of product with undercuts, which must be ejected in two-stages: 1. - sleeve and stripper lift the product off core, 2. stripper continues to push the product out. [21]

3.5 Mold venting

Air contained within every mold must be removed or displaced as the mold is being filled with a plastic material. It is very important to allow air to escape freely during injection. At high injection rates, mold venting may produce a considerable compression of the air, with consequent slow mold filling, premature plastic pressure build-up, and, in extreme cases, burning of the plastic material.

Venting is realized by small gaps or vents provided in the mold parting lines, or other small channels in the mold, for example around ejector pins or cores. Vents must be provided at the end of the flow path. A center-gated mold cavity, for instance, must be vented all around, whereas in an edge-gated cavity the vents must be provided at the point where the flow path is expected to end (generally the cavity end). [2]

To overcome the trapping of air or gas in a cavity, in locations that are difficult to vent effectively, molds may be designed such that all cavity vents feed into a space that is sealed from the outside of the mold (when closed) by an O-ring seal, and is connected to a vacuum reservoir through a vacuum line obtaining a solenoid-operated valve. In operation, as soon as the mold is closed and the transfer plunger enters the pot, the aforementioned solenoid valve is automatically opened, causing the cavities to vent rapidly into the vacuum reservoir before the molding compound has entered or filled the cavities. [2]

To provide a very large venting area, porous metal can be used. The key to the process is that the porous metal is directly cooled by the water behind it, which keeps it from overheating (porous metal has very low heat conductivity) and plugging with hot plastic. Coolant does not leak out, because it is held at sub-atmospheric pressure. [2]

II. PRACTICAL PART

4 DETERMINATION OF THE WORK AIM

In this Master Thesis was determined following aims:

- a theoretical review formation of given topic
- creation of cavity and core 3D model in Catia V5
- set up six versions of runner system and execution of a mold cavity filling analysis in CAE (Computer Aided Engineering) software Autodesk Moldflow Insight (thereinafter only Moldflow)
- selection of the most suitable runner system according to simulation results, and design of the injection mold in Catia V5
- set up one conventional (drilled channels) and one conformal (Direct metal laser sintering technology) cooling system for the injection mold
- execution of cool and deformation analysis (cool + fill + pack + warp simulations) Moldflow
- evaluation of results and designed solutions

The theoretical part is divided into three chapters. Firstly, there was described an injection molding technology including an injection molding cycle characterization. Secondly, types of injection molding machines were introduced and emphasis was putted on injection and clamping unit. At last, in the third chapter was, as one of the main purpose of my thesis, described injection mold construction. A mold conception, runner, cooling and also ejection systems were characterized in detail.

For purposes of my practical part, which were determined above, was provided a technical plastic part from the HELLA Autotechnik, Ltd. Mohelnice, which is a daughter company of a German concern HELLA KGaA Hueck & Co. The three-dimensional model of the cavity and core for this part was created in Catia V5 software. This was made with pay attention to purpose of use of plastic part including the mold construction requirements.

Next step was a preparing of six injection mold sides for purpose of prepares runner systems dimensions. Runner systems were analyzed in variety of two and four impression molds, and either within cold and combination of hot and cold runner system. Each of these was designed with trapezoid and circular cross-section of cold runners. Filling simulations were evaluated according to the most testify results which could be chosen from Moldflow software. For the most suitable runner system was design an injection mold with all its necessities from HASCO standard components, such as ejection, clamping and guiding elements.

Two ways how the mold can be cooled was realized on this selected and designed mold. In the Catia V5 software was created trajectories of runner and cooling system, which were putted to Moldflow simulation software for obtain analysis results of cooling, filling and packing phases including warp and deformation simulations.

5 PRODUCT CHARACTERIZATION

A plastic part was obtained, for the purposes of analysis and design of the injection mold, from the company HELLA KGaA Hueck & Co., is a holder and simultaneously a cover, of a light-emitting diode (LED). The part is located and mounted in the housing of a passenger car head-lamp. A product was provided in form of 3D data both in CATIA part and in data file of a 3D model with suffix *.STL for use in Moldflow as a model composed of tetrahedral, three-dimensional elements shown in the Figure 37.

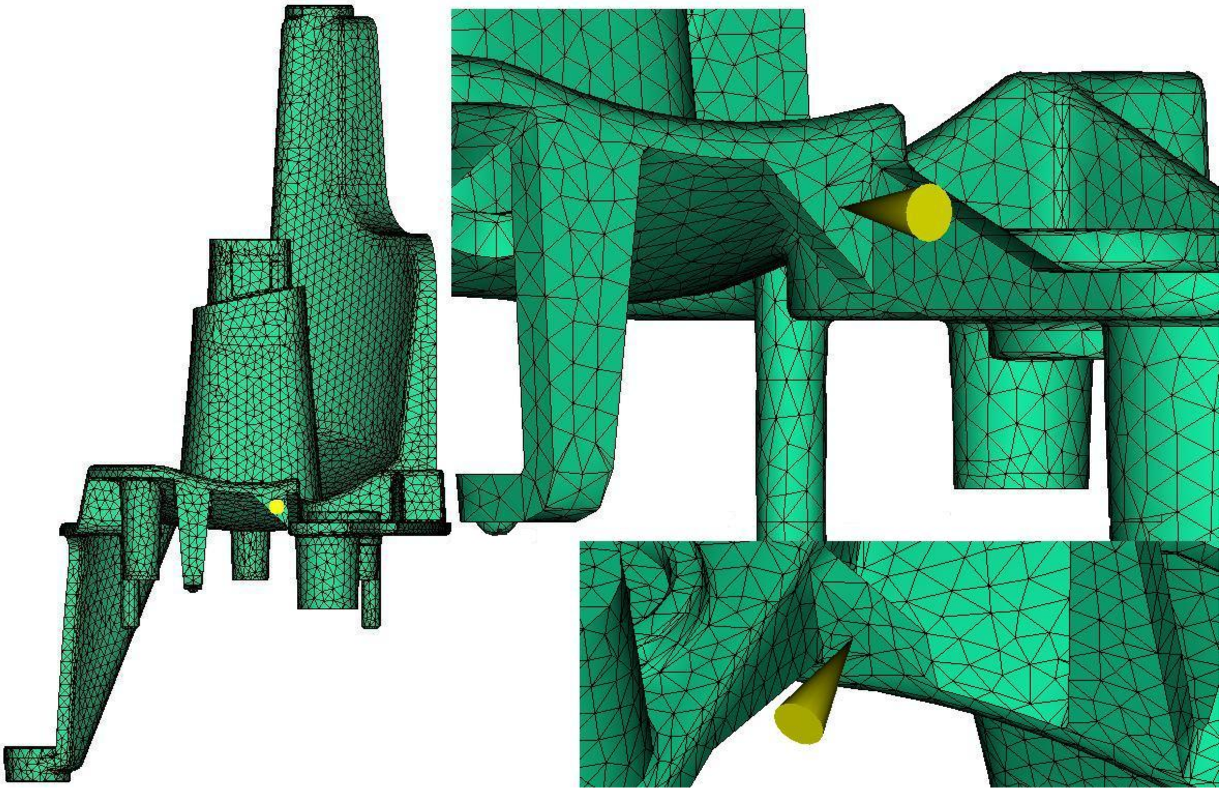


Figure 37 Definition of injection location

5.1 Product construction and description

Shape of the product is very complex with many details, clamping and supporting faces which are requested to be shapely precise and these shapes are in the mold represented with shape inserts which can prevent air traps from happening.

In Figure 38 can be seen surface of the part. It reveals section for holding a light-emitting diode (center passage of plastic part), of which surface is smooth, pure transparent and without marks of high roughness as on a whole surfaces of this side – cavity side. The faces in which will be a

LED device mounted is suggested to be transparent. It is simultaneously the only one piece of a whole product, which can be seen on the headlamp.

Figure 38 Cavity side of the LED holder

Figure 39 shows the core side of product with many clamping shapes and with roughness marks from machining. There is also visible marker of material and production company, remnant from ejector pins and runner.

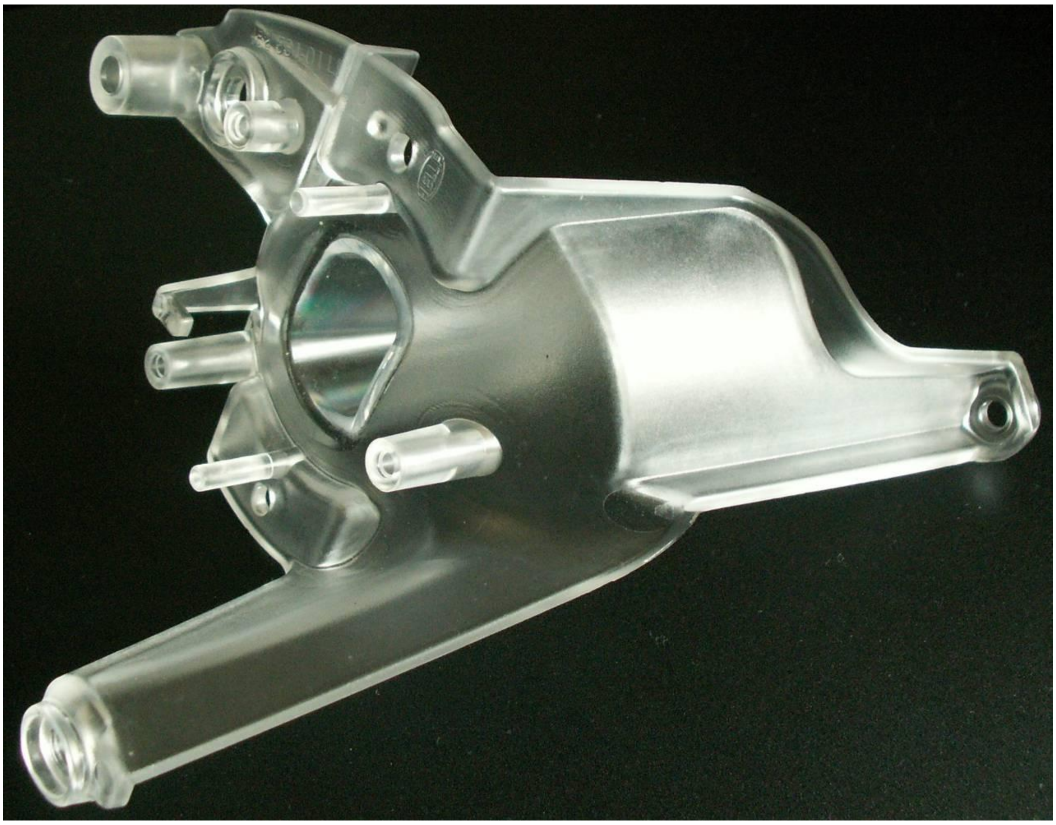


Figure 39 Core side of the LED holder

The part is, in itself, very tough and stiff, which is partially determined by a complex construction with ribs and partially by selected material.

5.2 Product material characterization

Material of the part was exactly determined as Polycarbonate (PC) Markolon Al 2447 from Bayer Material Science which characteristics properties are: [25]

- melt viscosity ratio (MVR, 300°C/1.2 kg) = 19 cm^3/10 min
- melt temperature = 280 – 320 °C; Moldflow setting = 300°C
- mold temperature = 80 – 120 °C; Moldflow setting = 100°C

Used material has amorphous structure, low viscosity, is UV (ultra-violet) stabilized, easy to release and is available in transparent colors only. Polycarbonates are used for purposes, where high stiffness, rigidity, chemical and temperature resistance are supposed. Polycarbonates found their application in automotive lighting as a headlamp lenses for automobile forward lighting and also as a technical products in interior. In addition, they are used for lenses of glasses, fibers, CDs, films, and so on.

6 RUNNER SYSTEMS AND FLOW ANALYSES

6.1 Process parameters

Flow analyses were ran with these parameters:

- mold surface temperature = 100 °C
- melt temperature = 300 °C
- filling control by: injection time of 2 s
- velocity/pressure switch-over by: % volume filled at 98%
- packing/holding control by: % filling pressure vs time (Tab. 2)
- other setting was kept as a default

Table 2 % Filling pressure vs time

Duration [s]	% Filling pressure [%]
0	80
10	80

6.2 Runner system structure

Before design of injection mold, were defined and analyzed various types of runner systems according to assignment, furthermore all these types was analyzed with two different versions of cold runner. At first, with trapezoidal, and secondly with circular runner. Thereinafter, this two runner versions will be described and compared. Exact concepts are shown in Table 3.

Table 3 Concepts of runner designs

Mold multiplicity / Runner type	**Variant of runner system**	**Variant of runner system**
2 impressions	Cold	Combination of cold and hot
4 impressions / Type H	Cold	Combination of cold and hot
4 impressions / Type X	Cold	Combination of cold and hot

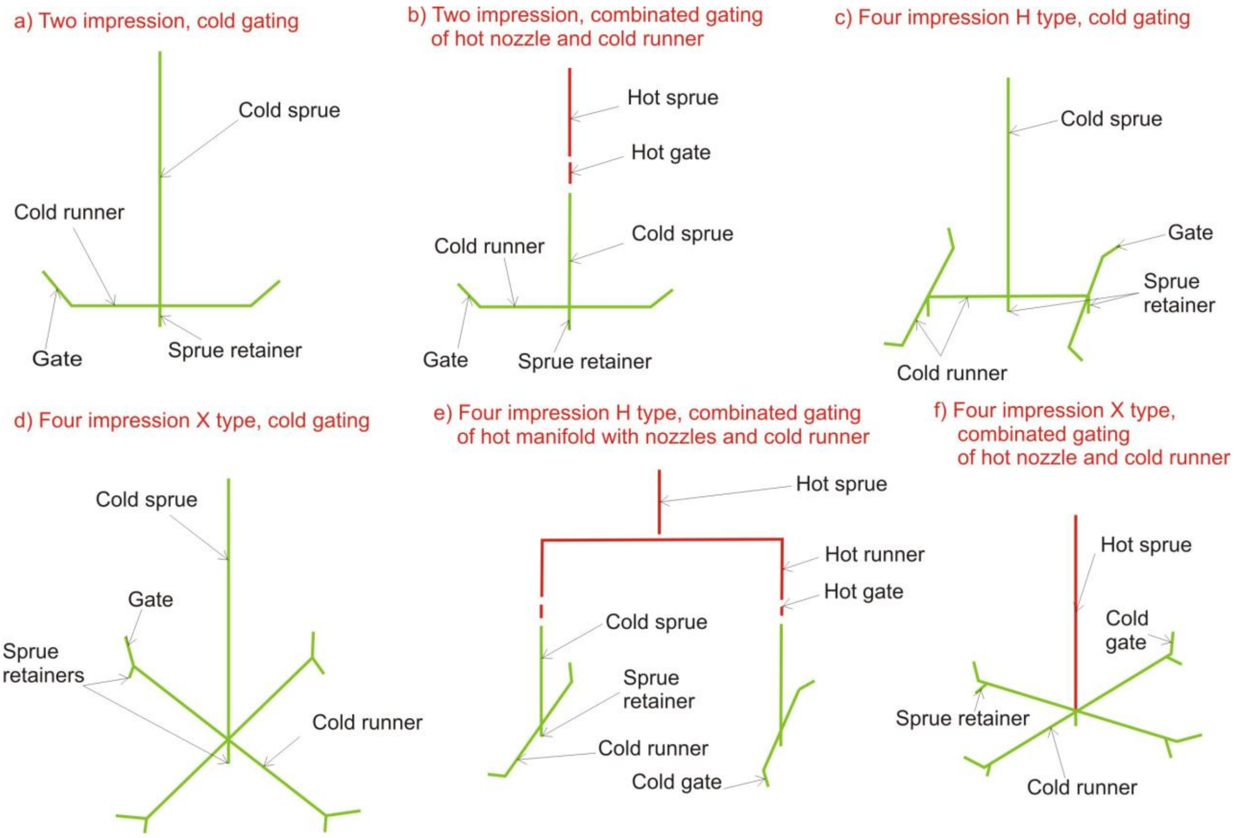


Figure 40 Types of designed runner systems

To design all these type of runners is necessary to have their trajectories, which are imported to Moldflow then. In addition, diameters, or dimensions of sprues, runners and gates has to be known. For this purpose was drafted six injection mold sides with HASCO standard components, and dimensions of trajectories were exactly measured. These trajectories was saved as data file with a suffix *.IGS and loaded subsequently to Autodesk Moldflow, where they were transformed into beams of tetrahedral elements. The only dimensions, that all runner systems have the same, are diameter of cold runner and two diameters (start and final) of gates.

Figure 40 shows conceptual organization of hot or cold runners, which are represented with hot runners and sprues, and cold runner systems (cold sprues, runners and gates).

6.3 Flow analyses

For all types of flow analysis was used a Finite elements method (FEM) software Moldflow, which enables simulations of injection molding process. In this chapter is only described phase of mold cavity filling.

6.3.1 Runners comparison

Each type of runner was simulated with two different cross-sections of cold runner, which enables comparison of pressure and shear rate within the runner.

Dimension of cross-sections is shown in the Figure 41.

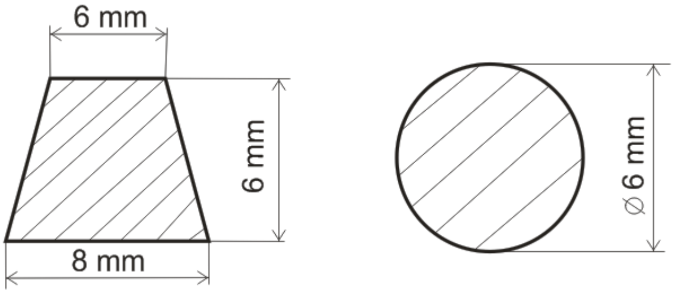


Figure 41 Runners cross-sections

An area of trapezoidal runner is: $S_t = \frac{a+c}{2} \cdot v = \frac{8+6}{2} \cdot 6 = 42 mm^2$ (1)

An area of circular runner is: $S_c = \frac{\pi + d^2}{4} = \frac{\pi + 6^2}{4} = 28{,}27 mm^2$ (2)

The pressure difference from one location to another is the force that pushes the polymer melt to flow during filling. The maximum pressure always occurs at the polymer injection locations and the minimum pressure occurs at the melt front during the filling stage. Generally, the maximum pressure should be less than 100 MPa for parts with runners. [25] Between the injection location and the melt front is a runner, which also has influence on pressure distribution during the process.

The shear rate result shows the rate of shear strain, in other words, the velocity gradient through the cross-section. High shear rate tend to occur in the runner system, where the greatest velocities takes place. Thus, the shear rate is a suitable indicator for deciding about runner types. Increase the cross-section or decrease the flow rate cause to reduce the shear rate. [25]

Trapezoidal runner

Figure 42 shows trapezoidal cross-section of cold runner and shear rate within it. This analysis belongs to four cavities, H type combination version of runner. Maximum detected value of shear rate is 16 362 s^{-1}. Maximal shear rate of used material is 40 000 s^{-1}.

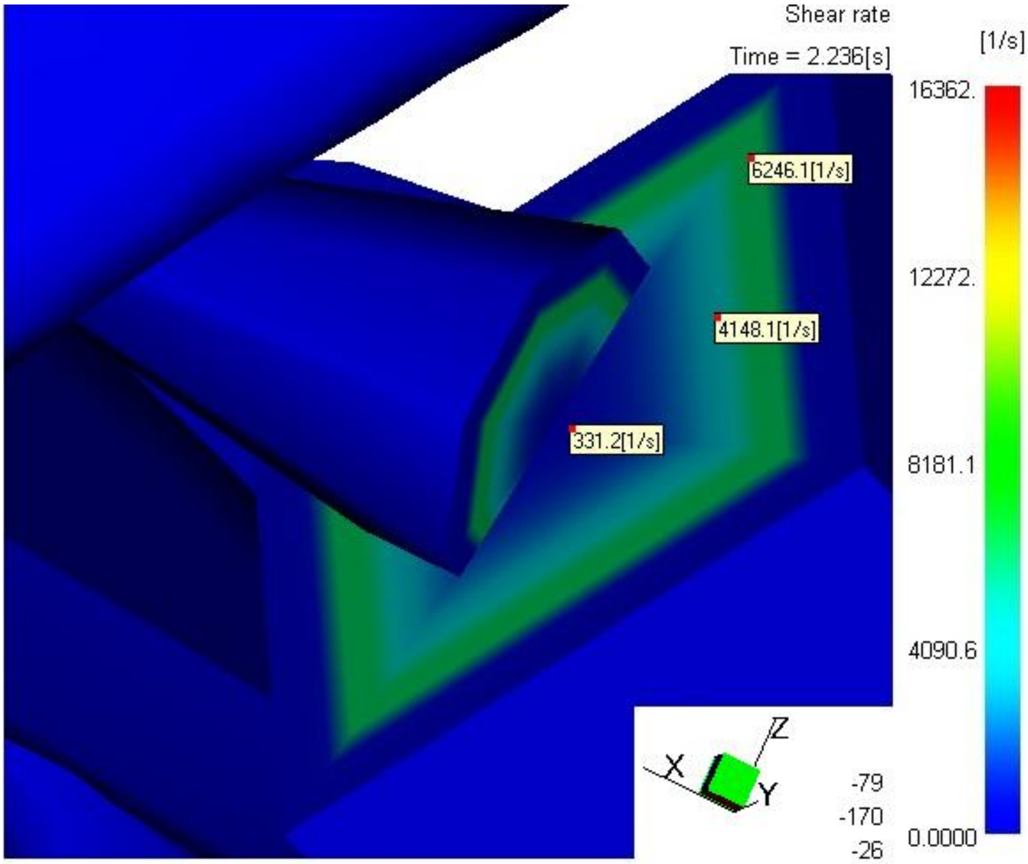


Figure 42 Shear rate in trapezoidal runner

Figure 43 reveals the pressure distribution in one point marked with black dot, and is located in cold runner. As can be read from the graph the pressure is gradually increasing until the point of switch to the packing phase after two seconds. Maximum measured pressure in this part of runner was 41,79 MPa.

The pressure result shown in Figure 43 is an expression of the pressure distribution through the flow path inside the mold cavity and runner system.

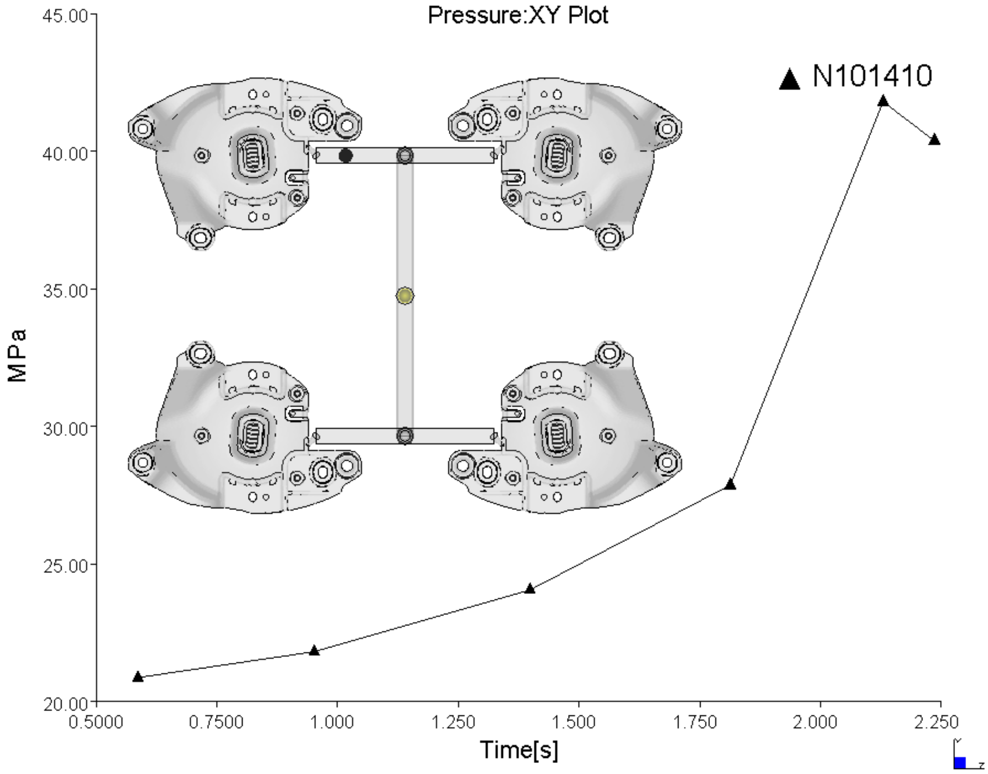


Figure 43 Pressure distribution inside the trapezoidal runner in view of time

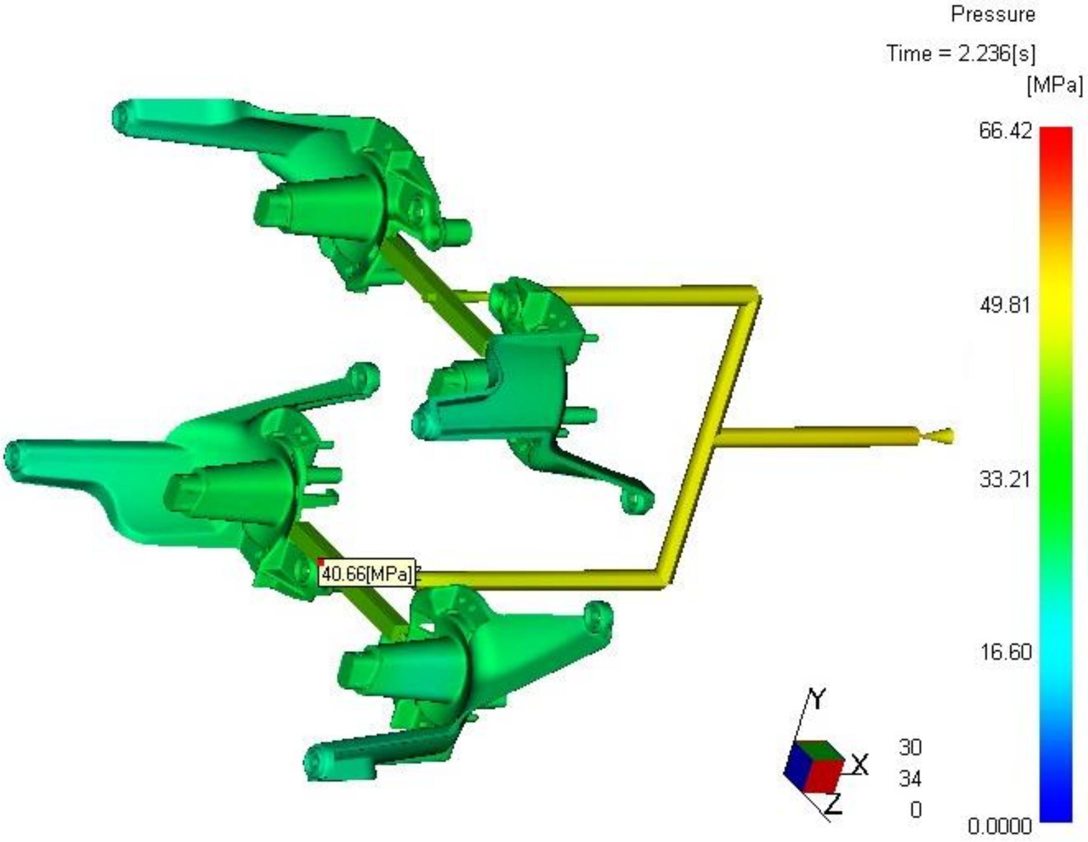


Figure 44 Pressure distribution inside mold cavities and runner system

Circular runner

In the Figure 45 can be seen the mold cavity at about one half filled state. An accuracy of the mesh created from tetrahedral elements is finely visible in the Figure. The Figure is divided into two parts, and on the right side of it, is viewed formation of weld line and air traps closing. On the left side of the figure is clearly visible ending of the flow path within thin and relatively long part of the product, where the air traps can occur. The air traps will be eliminated with shape inserts inside mold cavity, through which the air can escape.

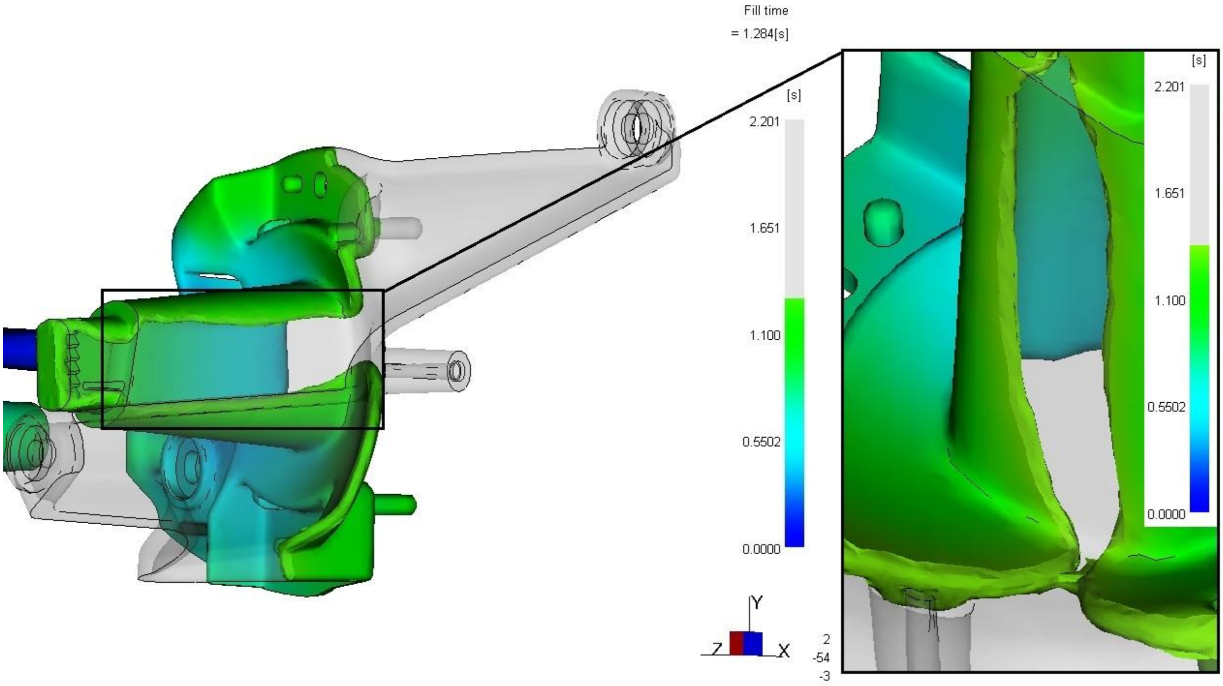


Figure 45 Fill time of cavity through circular runner

Figure 46 pointing on shear rate inside a circular cold runner, which is lower than inside trapezoid runner with greater cross-section. The maximum shear rate occurs in the area with greatest velocities are, like runner from runner to mold cavity. For circular cross-section is maximum shear rate 33 675 s^{-1}, and for trapezoid is 16 362 s^{-1}. Values of shear rate are conform in both trapezoid and circular case. Maximum shear rate of used material is 40 000 s^{-1}. The difference corresponds with almost one half decrease of cross-section area. Very high shear rate can lead to degradation of polymer. This problem can be also solved with decrease flow rate which passes through the runner system.

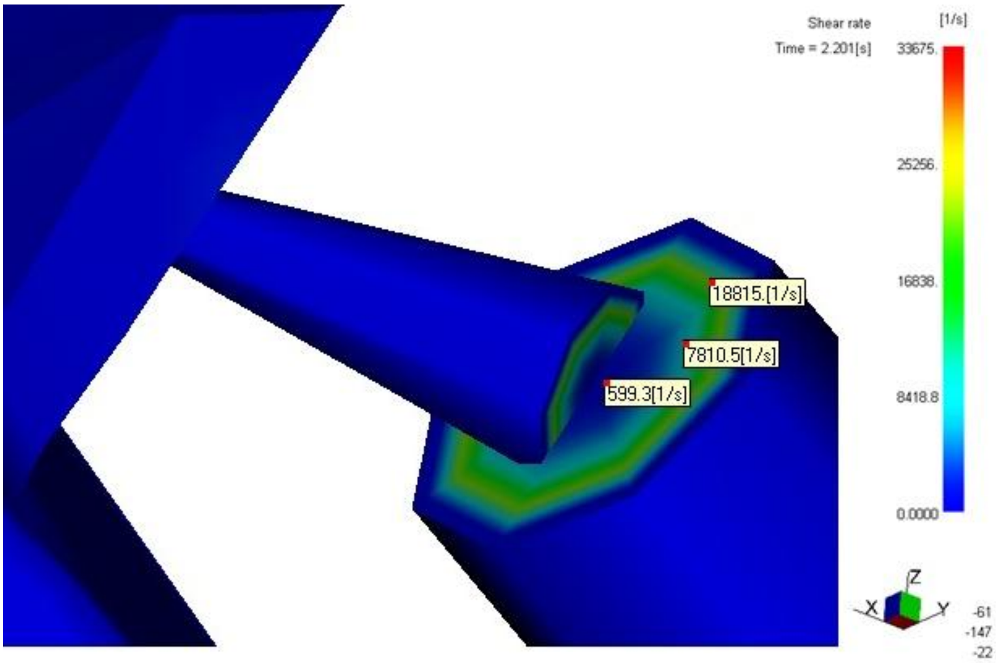


Figure 46 Shear rate inside circular cold runner

Figure 47 illustrates the pressure of melt inside the runner system and mold cavities.

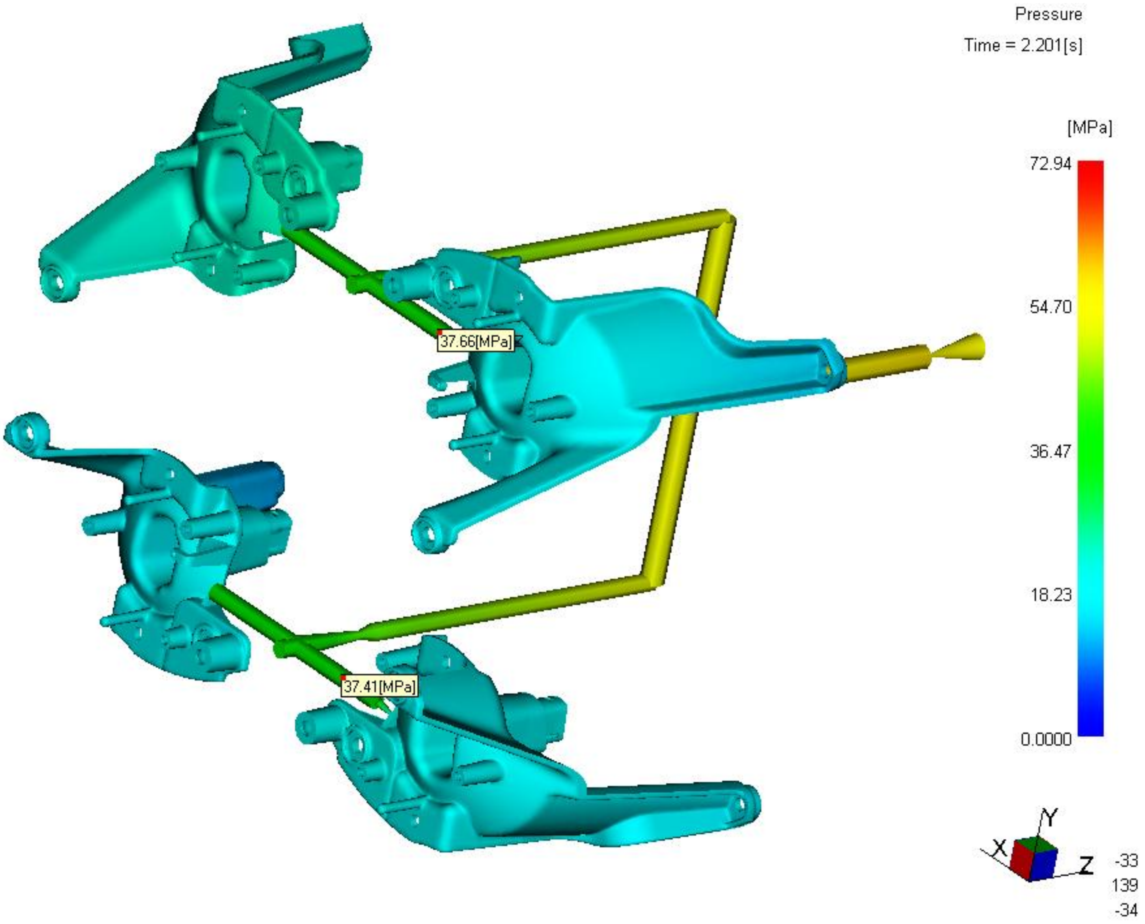


Figure 47 Pressure distribution inside mold cavities and circular runner

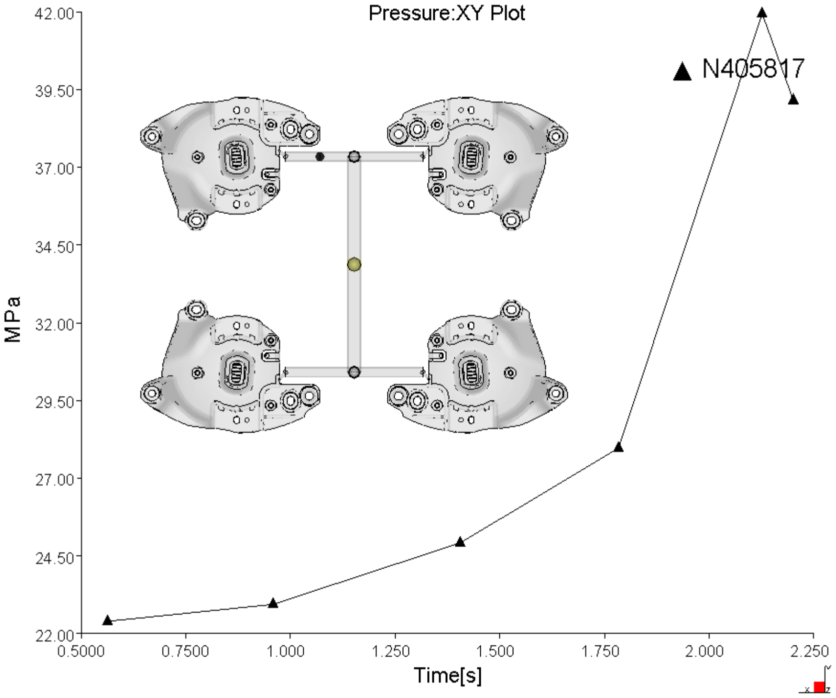


Figure 48 Pressure distribution in one point inside the circular runner, in view of time

From the Figure 48 can be read the pressure increasing process until reach the maximum value of 41,94 MPa in 2,127 s. Then is a cycle switched on to the packing phase and pressure in the marked point decrease up to the end of filling.

According to simulation results and manufacture reasons, the runner with circular cross-section was selected for further process of given tasks. Although, shear rate was reduced, thanks to use trapezoidal runner with increased cross-section, circular runner, from the manufacturing point of view, is cheaper and simpler solution.

Thus, the emphasis is putted on the runner system with circular cross-section of cold runner thereinafter.

6.3.2 Two - cavity, cold runner version

In the Figure 49 can be seen fill time of two cavities with cold runner. The result reveals that all cavity areas were filled by polymer melt within 2,172 s. The ends of the flow paths were checked, to eliminate an option of short shot. Thanks to fill time result, the area of weld lines or air traps appear can be predicted.

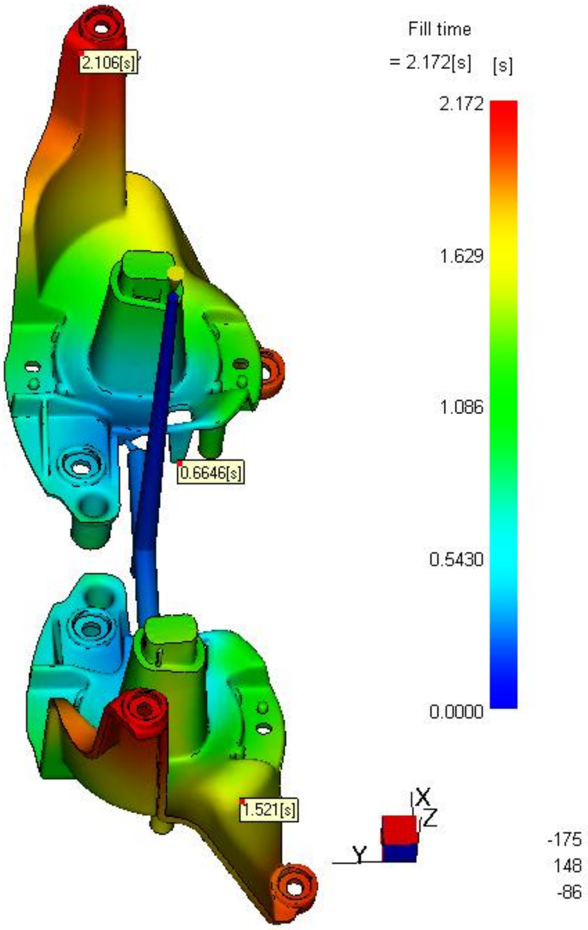


Figure 49 Fill time

An outcome of a simulation of mold cavity filling was almost the same in every case. Among all runner systems there was a fill time difference about one tenth.

6.3.3 Two - cavity, hot nozzle version

Figure 50 shows Temperature at flow front outcome, which suggests the temperature of polymer in the center of the plastic cross-section, while the flow front is reaching specified areas of runner system and mold cavities. As has been proved, the flow front temperature variation in the filling phase is under 10 °C, which is desirable. Larger difference can be indicator of too low injection time, or the hesitation areas presence. The problem of too high flow front temperature can be solved with increasing the injection time. Too low flow front temperature may cause excess shear in the gate, which can resulting in degradation and surface defects. To prevent this cause and subsequent possible problems the injection time has to be decreased. In the case of two cavities fed from hot nozzle through cold runner is maximum temperature at flow front 307,9 °C and minimum is 297,7 °C.

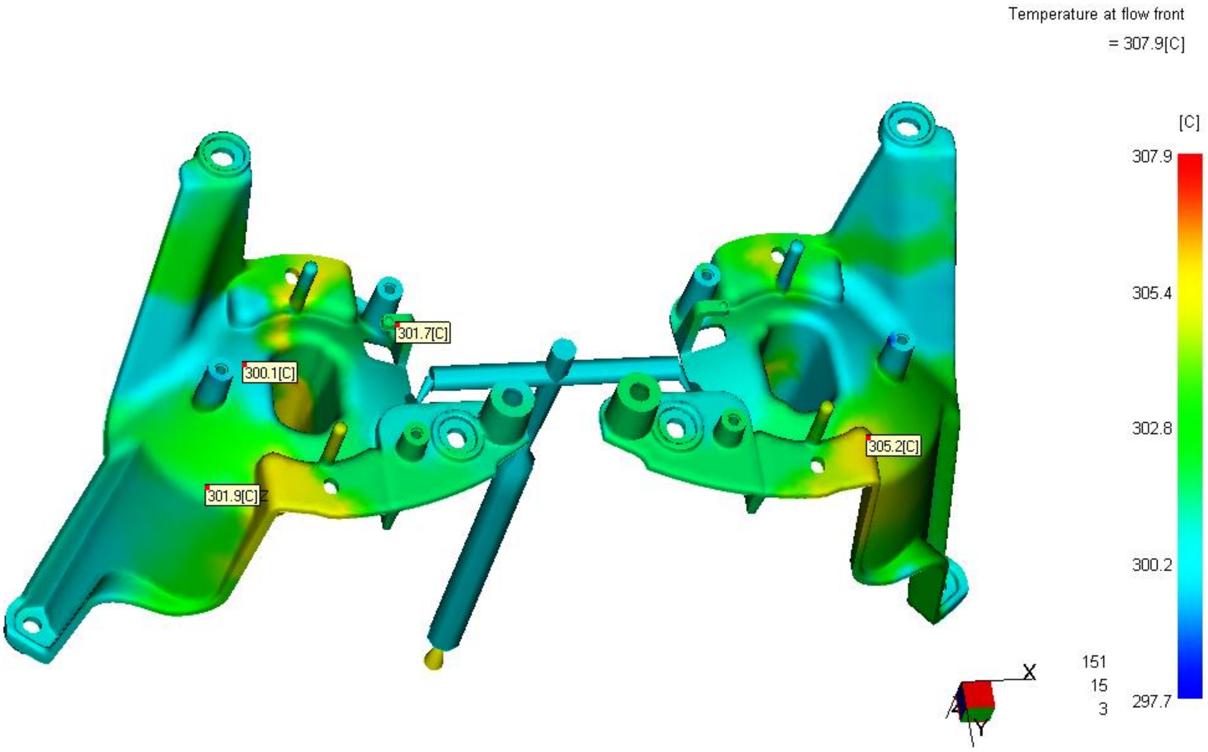


Figure 50 Time to reach ejection temperature

6.3.4 Four – cavity, H – type, cold runner

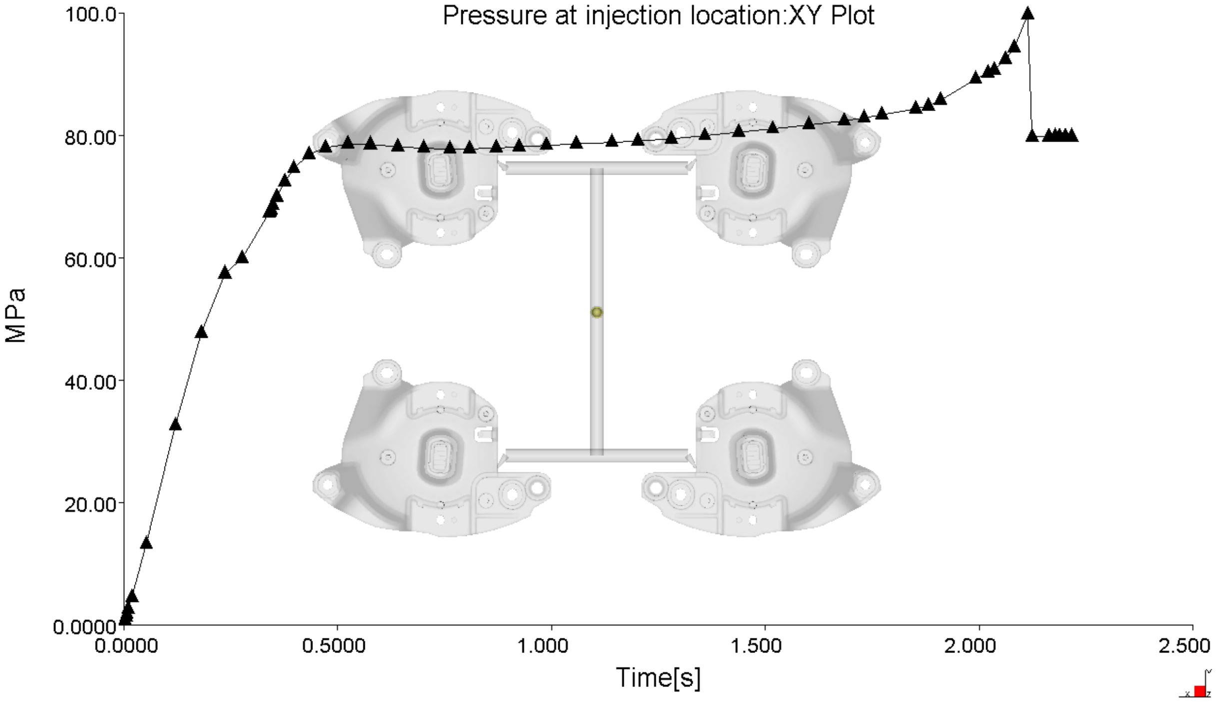


Figure 51 Pressure at injection location

Figure 51 provides information about distribution of the pressure at the injection location at various times during the filling and packing phases. From the graph can be read a linear pressure increase from start of filling phase up to switch on the packing phase. This gradual shape of the pressure curve is result of well balanced and distributed pressure in the mold cavity. The only one pressure spike, which is normally a sign of imbalance, is the maximum reached pressure value is 100 MPa.

6.3.5 Four – cavity, H – type, combinated runner

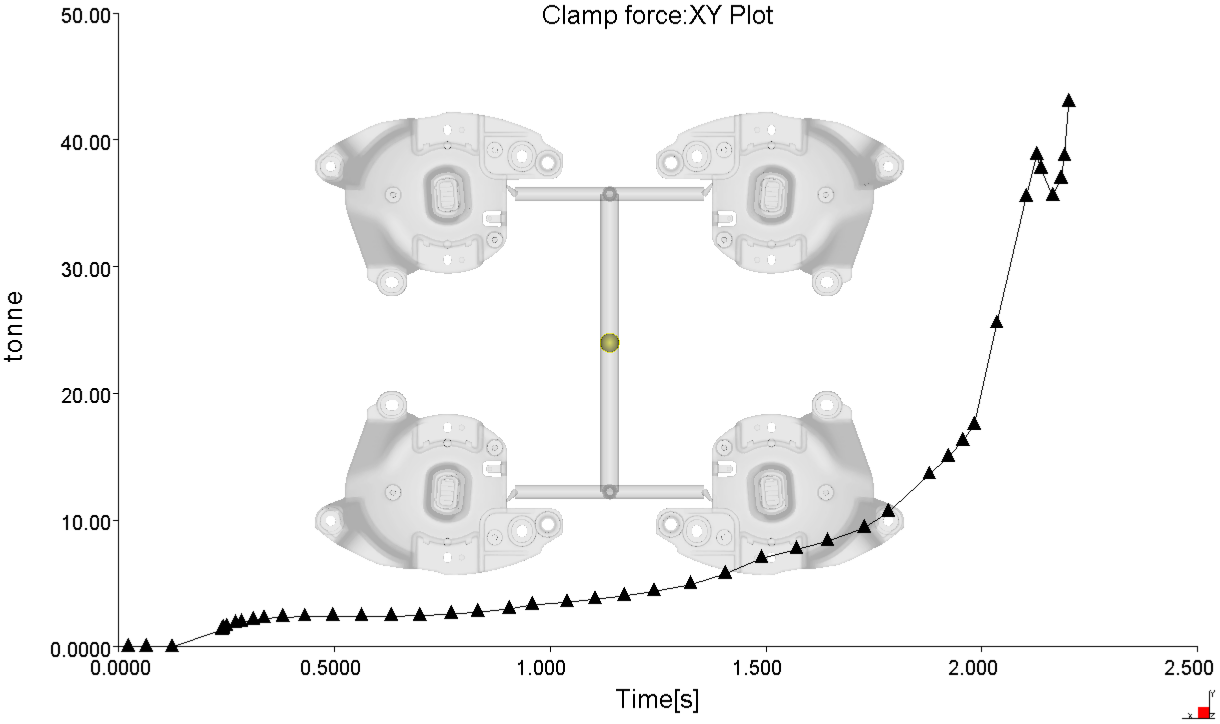


Figure 52 Clamp force

Figure 52 demonstrates a force of the mold-clamp over time and shows the pressure distribution over entire part. The filling and packing pressure acts to open the mold and the clamp force is the resultant force of these. According to maximum value of reached clamp force, which is 44,5 MPa, will be selected injection molding machine with suitable properties.

Figure 53 shows an air traps result, which reveals the area of appearing the air, which is closed by at least two melt flow fronts. Air traps has a great influence to a final appearance of molded part and also to the structural uniformity of the plastic material. The marked spots are the places of future application of shape inserts inside the mold cavity and core. Small gap between the shapes inserts and the cavity allows closed air to pass through the cavity outside of the mold and this can lead to improve the final part quality. Shape insert will be showed and described thereinafter.

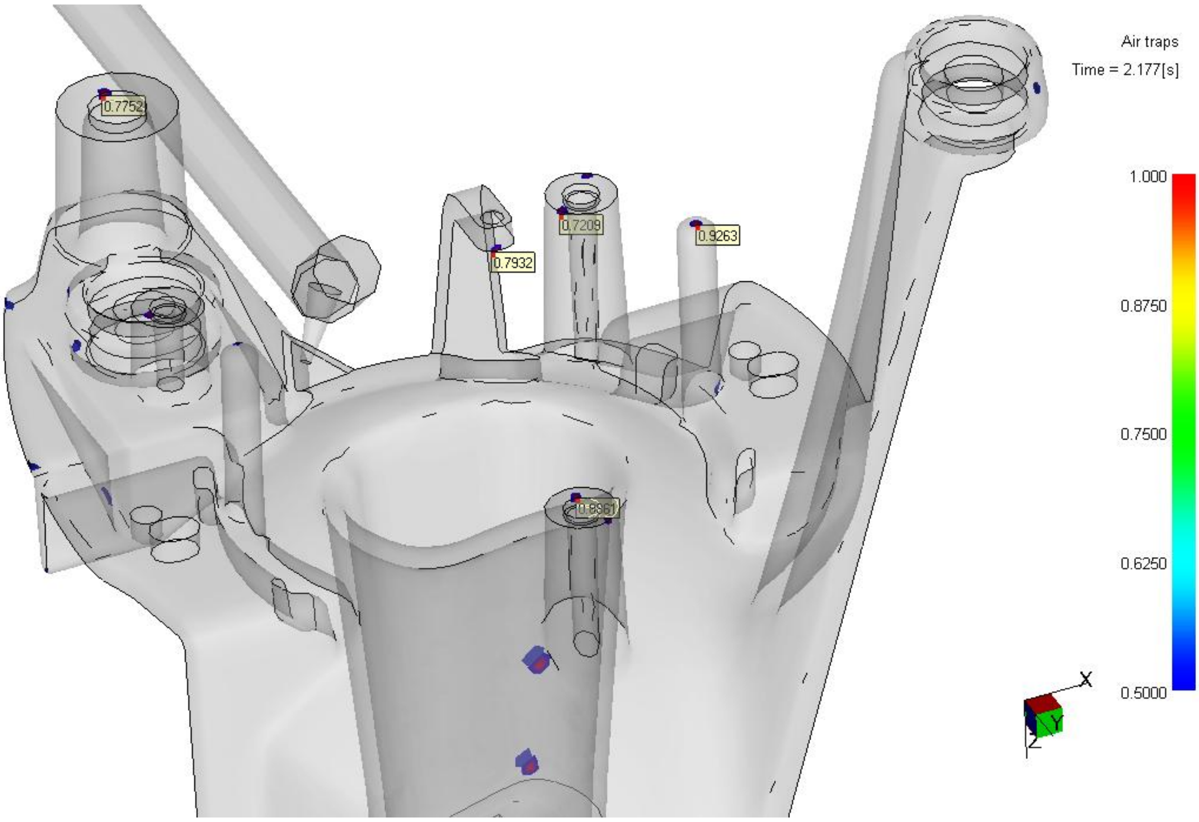


Figure 53 Air traps

6.3.6 Four - cavity, X – type, cold runner

As can be seen in Figure 54, the maximal amount of time required to reach the ejection temperature is 42,54 s. But this value was measured on the hottest and the least cooled areas. For detecting time to reach ejection temperature at the places, where the ejection forces takes place, the marks with exact values of required time was added. The Figure reveals that the longest time was detected at the place where the sprue puller acts. Times to reach ejection temperature under ejecting pins and sleeves fluctuates from 0,77 to 2,2 second. Due to use the 3D (three-dimensional mesh created from tetrahedral elements) mesh on the model, time values are at the individual nodes, which mean the flow path is very accurate. This outcome is very important for deciding about runner system. Well-designed runner system in association with good part design including the wall thickness has a particularly influence on the cycle time.

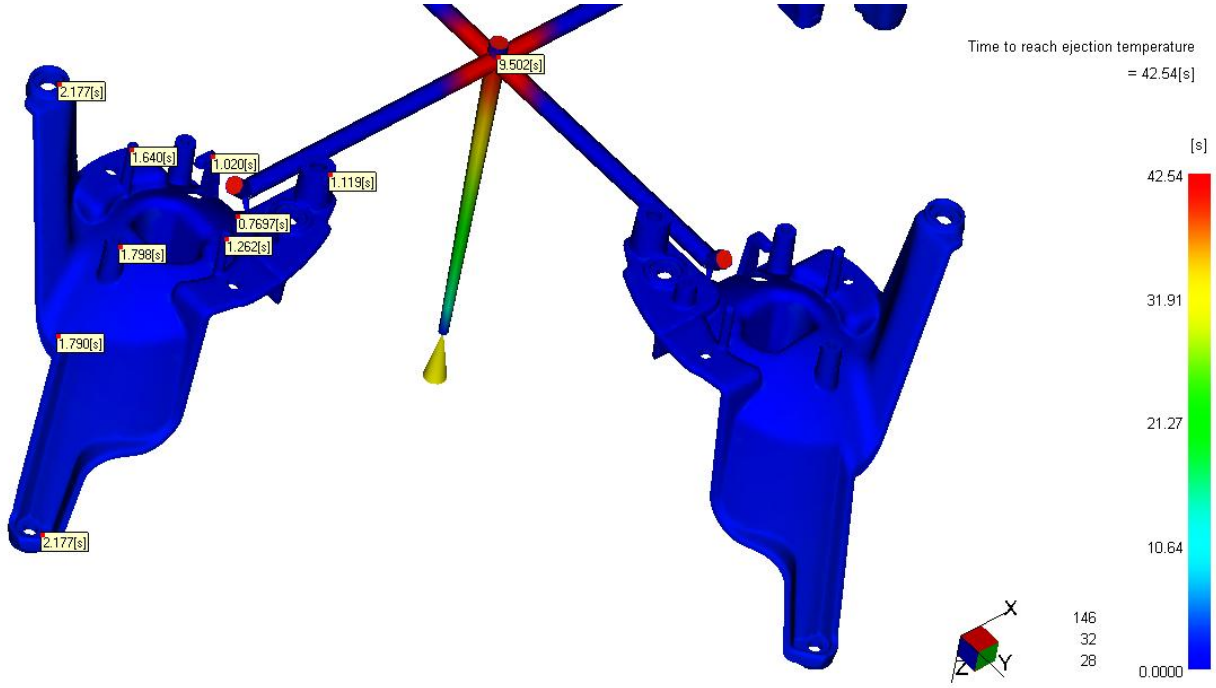


Figure 54 Time to reach ejection temperature

6.3.7 Four – cavity, X – type, combinated runner

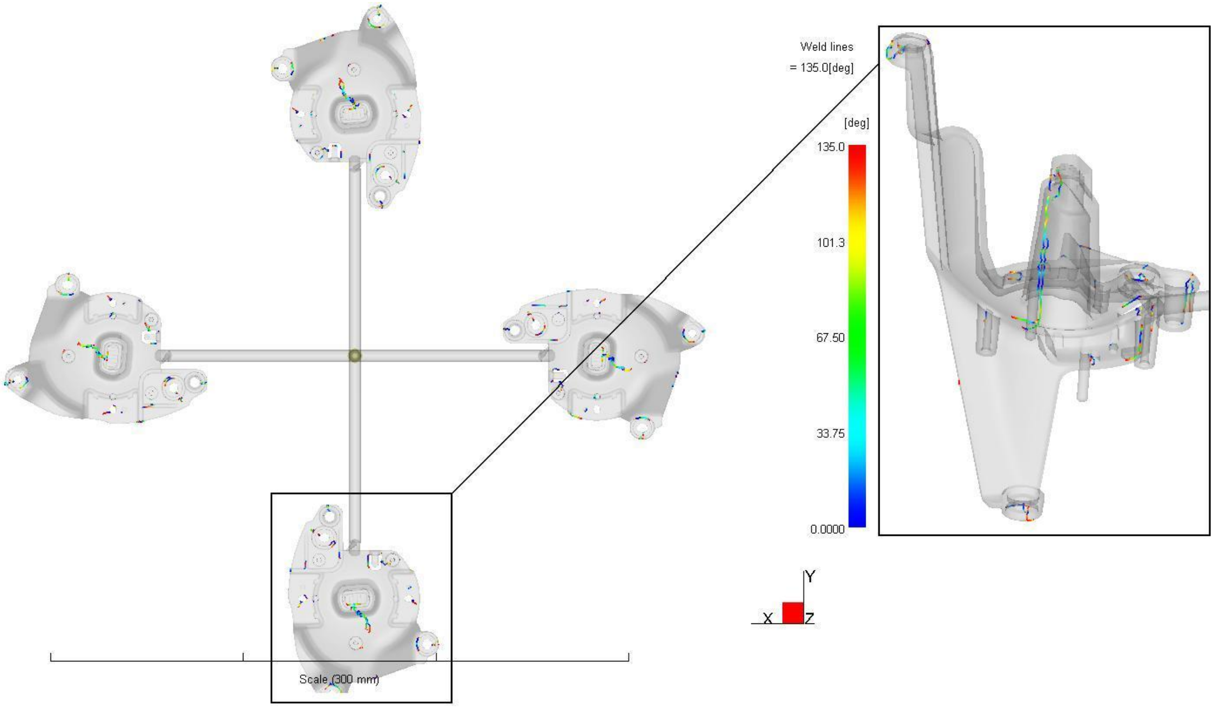


Figure 55 Weld lines

Last runner system shown in the Figure 55 is used for showing area on the part, where the weld lines takes place. Also, from the Figure can be read the angle of convergence as two flow fronts meet. As can be seen weld lines takes place on the problematic places, such as around the hole for clamping, or in the center area of the part, where a structural weakness and surface blemish are not desirable. The most of these places will be the area of application shape inserts which will be used as a vent. The most effective change to move the weld lines, or improve its quality is movement of injection location and possibly optimizing the design of the runner, which means reduce runner dimensions and maintain the same flow rate. The injection location was exactly assigned and the change of runner dimensions has almost the same result. In the other words, runner with circular cross-section reached almost the same results as a runner with much greater cross-section. Which means, although, shear heating was utilized to increase the melt temperature at the flow front, it had no visible result to a large degree.

6.3.8 Runner systems comparison

On the basis of previous simulations are various runner systems compared according to selected results. The results were chosen with respect to the fact, that some of the outcomes may be changed after the cooling system will be added for run the complete cool + fill + pack + warp analysis. A few results from filling analysis will be different after addition of cooling system, such as Time to reach ejection temperature, Shrinkage and so on. Some of the results can be used now. Table 4 presents the selected results of runner systems with the cold runner with trapezoid cross-section.

Table 4 Some results of the runner system with trapezoidal runner

Results / Runner systems	2 - cold	2 - combination	4H - cold	4H - combination	4X - cold	4X - combination
Fill time [s]	2,184	2,192	2,181	2,236	2,180	2,189
Pressure at V/P switchover [MPa]	75,32	59,33	87,96	66,42	87,62	74,09
Temperature at flow front [°C]	308,3	304,9	309,5	306,0	308,7	308,9
Pressure at injection location [MPa]	78,4	60,5	85,1	68,8	87	75,2
Clamp force [tonne]	35	35,9	70,2	75,5	70,1	75,5

Table 5 summarizes the selected results of runner systems with the cold runner with circular cross-section. The results were selected according to data validity in response to their further changes after addition of cooling system.

Table 5 Some results of the runner system with circular runner

Results / Runner systems	2 - cold	2 - combination	4H - cold	4H - combination	4X - cold	4X - combination
Fill time [s]	2,172	2,227	2,216	2,201	2,177	2,187
Temperature at flow front [°C]	309,1	307,9	312,2	308,3	313.2	310,3
Pressure at injection location [MPa]	80	64,1	100	74,2	100	72,5
Clamp force [tonne]	20	25	68,1	45	64,5	55,6

The injection mold for assigned plastic part will be designed on the basis of the results of filling analyses. The shortest filling time and necessary clamp force has the two-cavity mold with cold runner system. But volume of wasted material (runner system) has too much great volume against to the volume of molded part. The volume of the part is 2,803 * 10^{-5} m^3 and mass of 34 g. The volume of cold sprue, runner, sprue puller and gate for two-cavity mold is 3,077 * 10^{-6} m^3 and mass of 4 g, which is 10,52% of injected shot for one part.

The smallest cold runner system, which will be transformed to the wasted material, has a version with hot manifold and hot nozzle. The plastic melt is kept warm in hot manifold and nozzle, and is ready for next shot while the cavities are filled. This property of hot runner system makes the waist of material of the cold runner smaller. Its value is 8,7 % of whole injected shot. In other words, the hot runner requires less material to be plasticized (no runner to fill).

From the economical point of view, the four-cavity molds are more profitable, thanks to less number of strokes, more products from one shot and lower costs for maintenance of the injection mold. From four-cavity molds has H-type much smaller dimensions than X-type, thanks to its location of core and cavity inserts. It also makes the mold cheaper.

On the basis of above mentioned reasons was selected four-cavity mold with combined runner and with cavities located in the shape of the letter H. For this type will be designed a whole injection mold. The analyses of cooling will be processed and evaluated in two versions – conventional and conformal – also for this one type of mold.

7 DESIGN OF INJECTION MOLD

The injection mold with selected runner system, described in detail in a Chapter 6, was designed in a Mold tooling design of Catia V5R18 with the assistance of HASCO standards added from the Hasco Dako catalogue. The mold was designed for a plastic part, which is described in a Chapter 5 and which was provided as a Catia part from the Hella Autotechnik, s.r.o. company.

As a first step of work were designed cavities and cores. Once, they are mounted inside the injection mold, a shape insert can be created and added to the mold. A content of next step was adding of leading and supporting elements. Then was finished an injection side of the injection mold, including a hot manifold with two hot nozzles, both was added from HASCO catalogue. An ejection system was created according to the requirements of the part construction and the cavity vent. Ejectors were designed carefully for safety ejection of the molded part. At last, both a conventional and conformal cooling system was designed.

7.1 Parting plane determination

After the parting plane is considered, core and cavity inserts are to be created. Cavities and cores were designed with assistance of Core & Cavity and Generative shape design modules of Catia. Since, the runner system for 2 + 2 multiple mold was selected, cores and cavities were mirrored and mounted to requested location inside the injection mold. Finished parts are to use as a components for left and right headlamps that is the reason why the Mirror function has to be used.

Requirements for the core and cavity design:

- changeable shape inserts: Basically, the reason for using this inserts is better demolding and vent. Places of inserts location were chosen according to filling analyses results, especially the Air traps outcome was used.
- shrinkage of material: Shape cavity inserts has to be created with respect to shrinkage properties of used material. After the filling phase is completed and cooling of the mold takes pace, the material in cavities is cooled and the volume of original injected shot decreases. This is why into cavity dimensions has to be added also shrinkage percentage of the material.

- core and cavity attachment: Changeable cavity insert is mounted on the cavity plate with screws through the support plate. On two bottom edges of the cavity inserts are chamfer for better locate and remove.

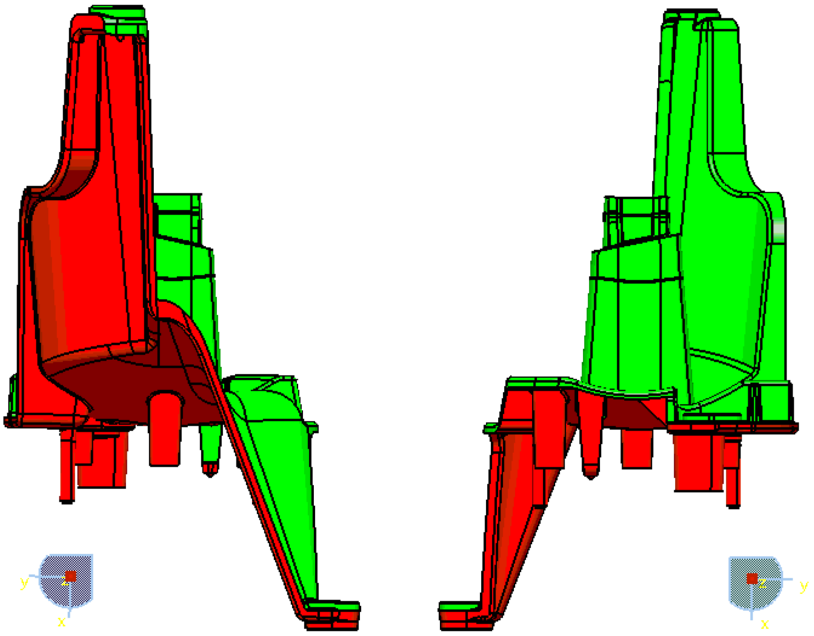


Figure 56 The parting plane considering; Green color shows impression shapes on the cavity. Red color is impression shapes on the core side.

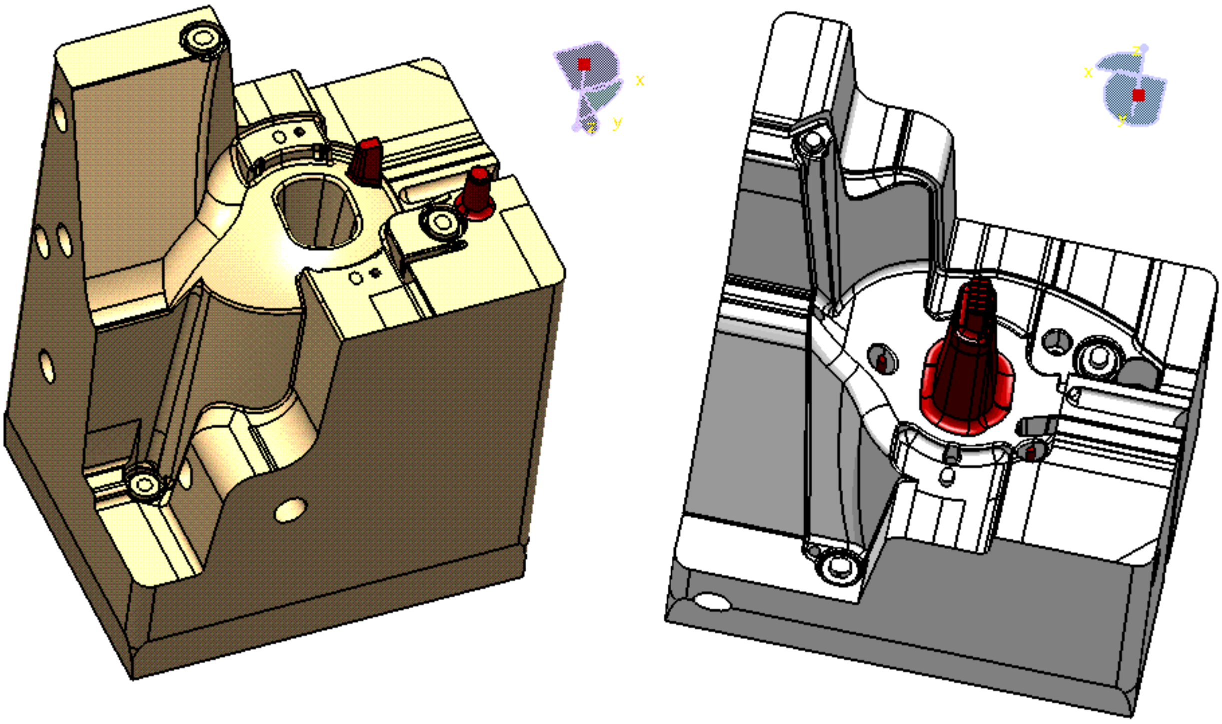


Figure 57 On the left side of the picture is the cavity insert and on the right side is shown the core insert; Shape inserts are colored with red

Figure 56 shows a model with colored shapes of cavity and core. A Consideration of parting plane was done in Catia, Core & Cavity module. After the parting line was defined, shapes of final parting plane could be created.

In the Figure 57 can be seen core and cavity inserts with mounted shape inserts colored with red. Final parting plane was created with assistance of shapes shown in the Figure 56 and with use of trim and pocket functions in Catia.

7.2 Mold vent

While the mold cavities are filled by molten polymer, an air inside the cavity is closing by flow front. This effect can be a cause of few unwanted consequences, such as an optical defect on the part, burned places or degradation of material. Normally, the closed air is exhausted through the clearance between parting planes. Occasionally, ejectors can also allow the air to pass through their clearance with the mold plates. In this case, the air traps were located thanks to previous analysis, and so the shape inserts could be created to serve as a vent. In the Figure 58 can be seen location of the air traps. To prevent the creation of these, the shape inserts were used. Their location on the part and the location of their application is shown in the Figure 59.

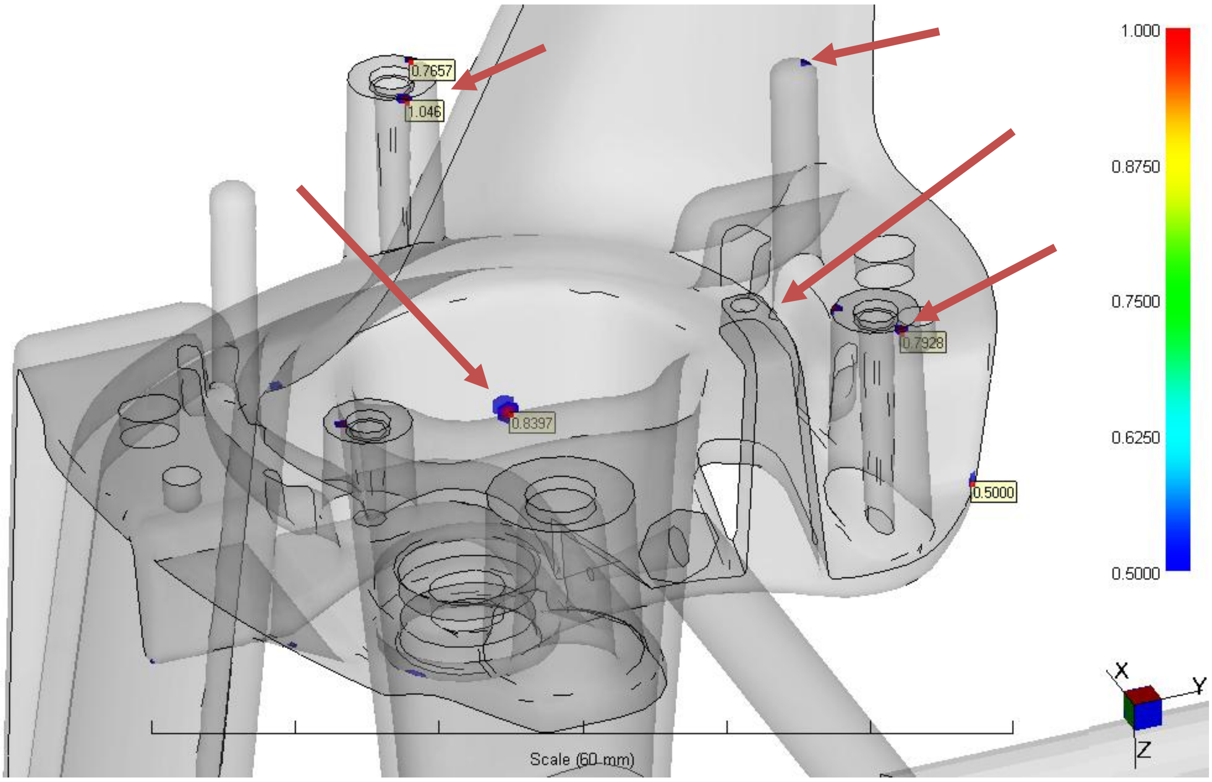


Figure 58 Air traps location

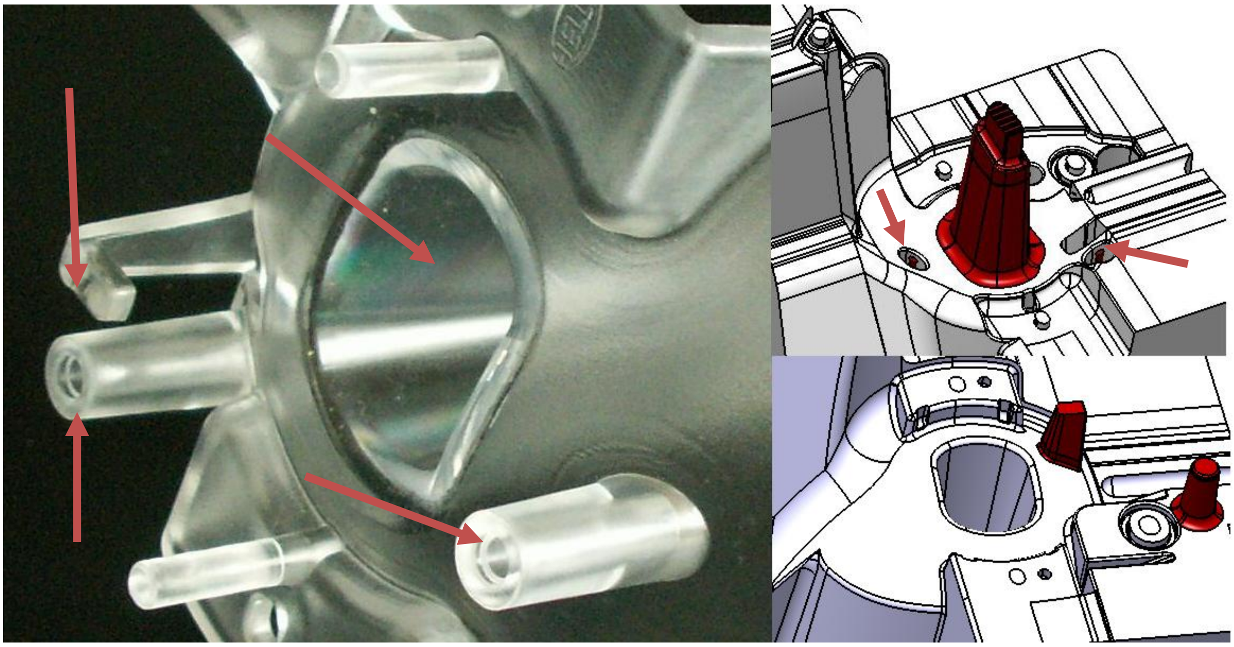


Figure 59 Location of shape inserts on the part marked with arrows; on the right side of the picture is their location on the mold core and cavity

7.3 Runner system

Location of gate is shown in the Figure 37. From twelve runner systems was selected the one with HASCO standard hot manifold and two nozzles, through which the melt is flowing further to the cold sprue and runners (Fig. 60). Then, the polymer melt is reaching four mold cavities.

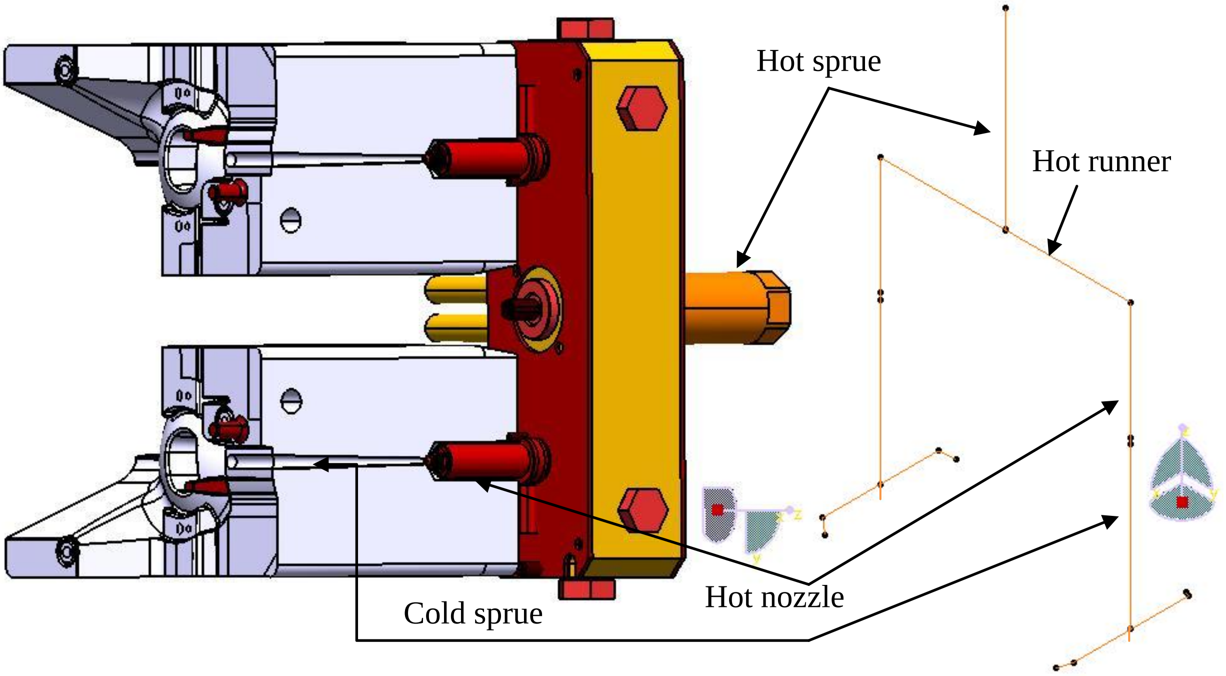


Figure 60 Runner system description; left side-Catia model, right side-IGS trajectories

Diameters of runner and gate were choosen according to standards after consultation with a mold expert from the Hella Company. Cold runner system is created from the cold sprue, runners, gates and also sprue retainers of “Z” shape for hold a runner with sprue on a movable side of the injection mold.

7.4 Ejection system

Figure 61 shows ejection system which is composed of ejection pins, sleeves, shaped ejectors and sprue retainers (all colored with blue). The ejection system has a sufficient stroke to demold the molded part safely out of the core. On the part will be visible marks from the ejectors after demolding. In consideration of this fact, ejectors location was selected on places with small visibility. In addition, ejection forces acts on the core, coarsen side of the part. Ejectors are mounted inside the supporting and foundation plates. Length of ejectors is variable according the surfaces on which they act. On the right side of the Figure 57 can be seen surfaces on which the ejection force is acting during an ejection phase of the cycle. On the left side of the same figure is shown the location and mounting of the ejectors. These are used for the only one part ejection. For ejection are used ejectors of various shapes. The part is ejected thanks to four ejection pins, two ejection sleeves and one shaped ejector.

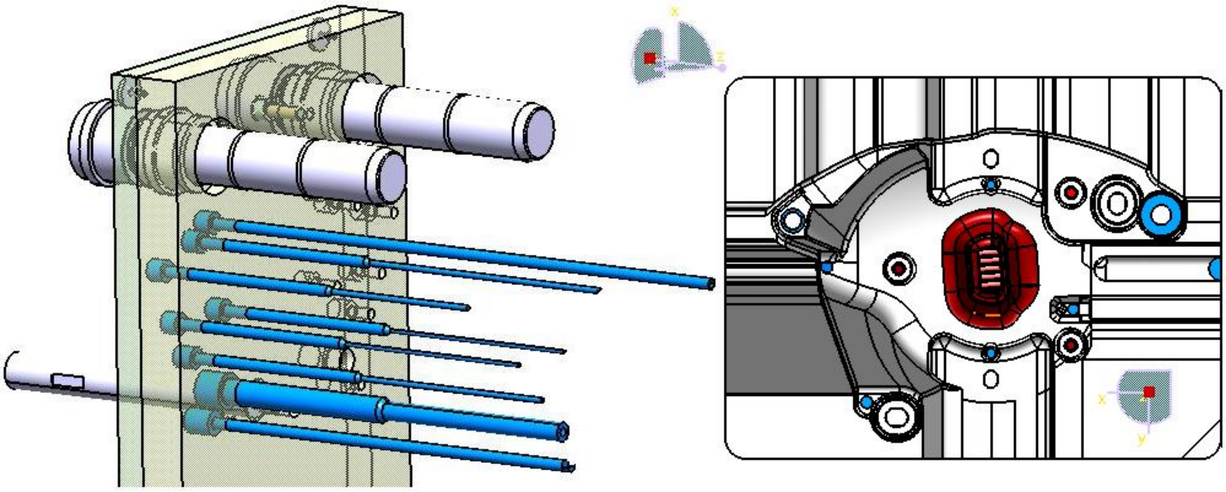


Figure 61 Ejection system

7.5 Cooling system

A mold surface temperature of used material (PC Makrolon Al 2447 – more in Chapter 5.2) is set on 100 °C. According to Moldflow habit and in consideration of convection heat transfer the temperature of coolant inlet was set on 95 °C. As a coolant medium can be used a water, which will circulate in cooling channels under pressure of 3 bar (300 kPa). An effort was putted on the effectiveness of heat transfer from the mold cavities. For this purpose was designed two cooling systems with the same process parameters (mentioned above), but with different trajectories and also technology of manufacture.

7.5.1 Conventional cooling system

Figure 62 illustrates first type of cooling system, which utilizes common drilled holes and channels, leaded in the core and cavity inserts and plates under different angles with respect to rules of cooling system rules and habits. The cooling system contains four independent circuits with four baffles, added from HASCO standards, together. In other words, every coolant circuit on the cavity side contains two baffles of diameter 16 mm. Diameter of odd channel is 10 mm.

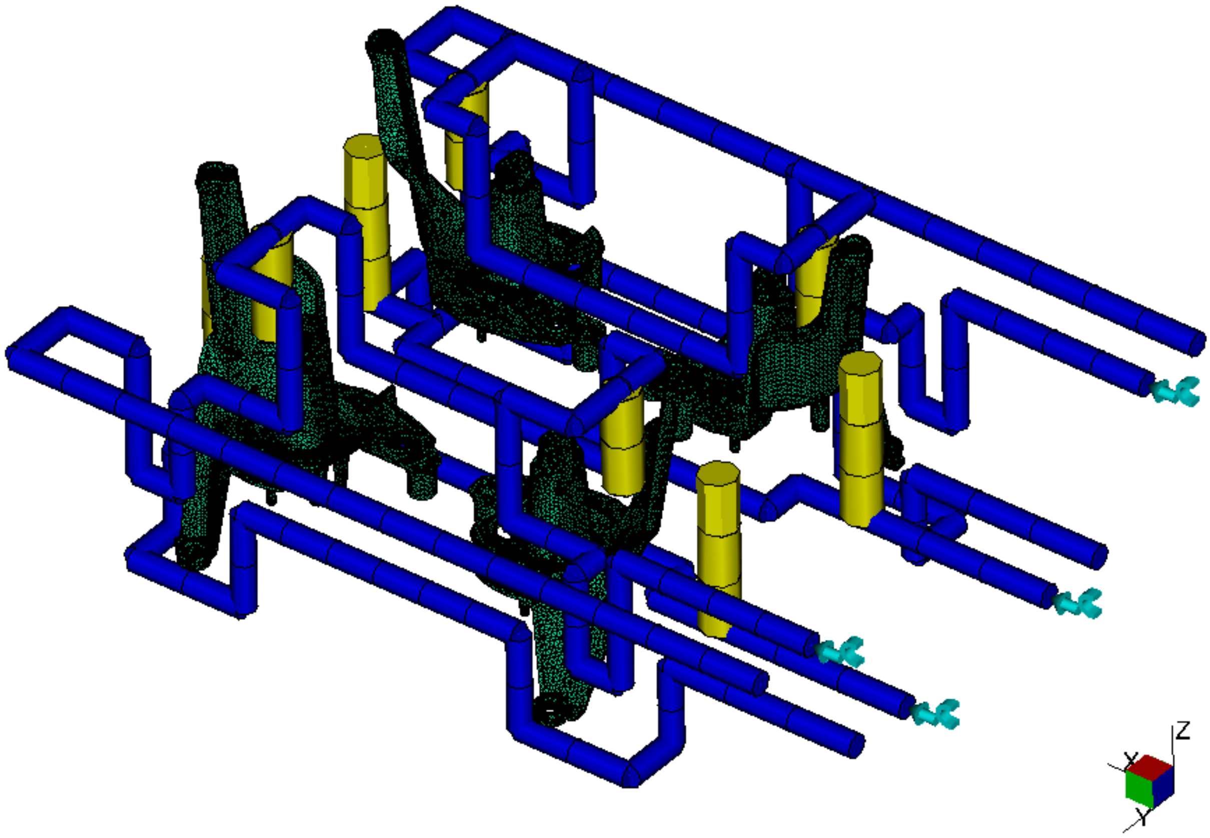


Figure 62 First type of cooling system

7.5.2 Conformal cooling system

Figure 63 shows second type of cooling system which takes advantage of a DMLS technology, which means Direct Metal Laser Sintering of shape very complex parts. With this technology can be produced core and cavity inserts with already implemented cooling channels. The process uses a high-power laser to sinter layer by layer of fine metal powder, to create fully dense complex part. Neither core circuit nor cavity coolant circuit contains baffles. Diameter of channels in both cases is 8 mm, which is the smallest dimension, which can be used for cooling. It must be decreased from recommended 10 mm, because of very complex trajectories and small distances form the mold cavity.

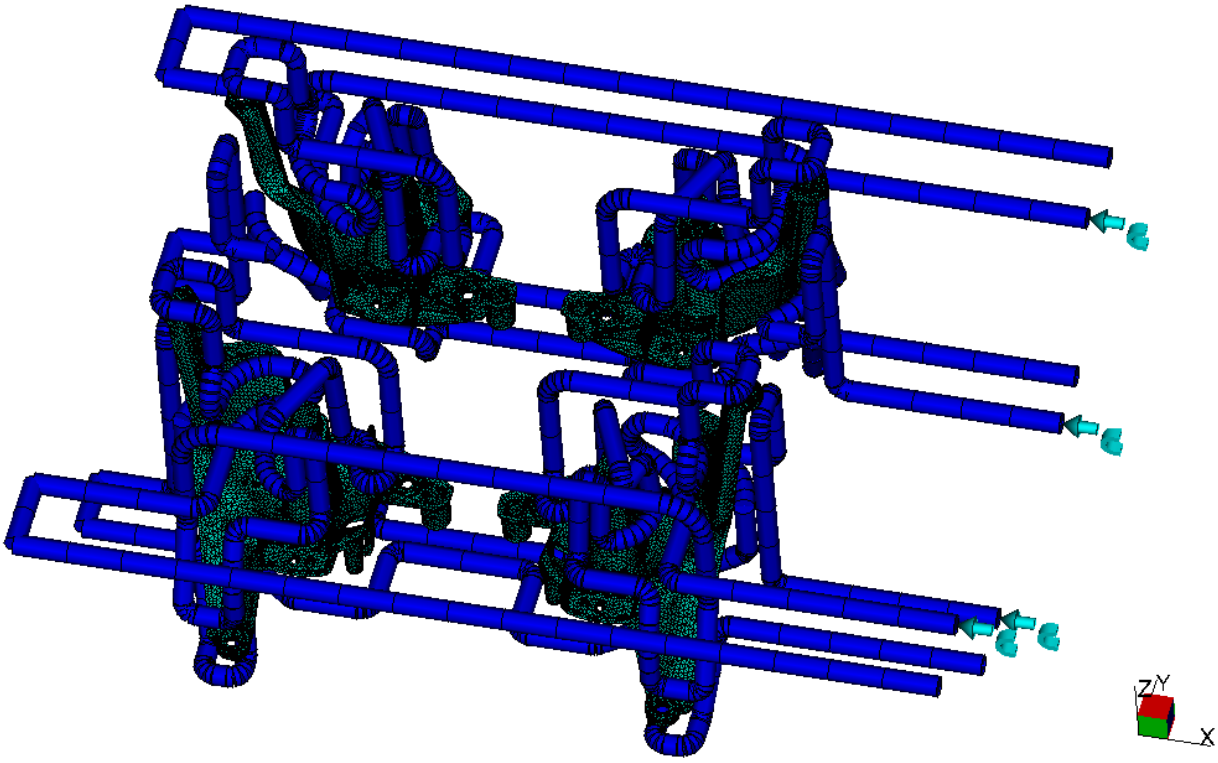


Figure 63 Second type of cooling system

7.6 Injection mold assembly

The injection mold, designed for one specific part with selected runner system, as mentioned above, is shown in the Figure 62. The mold has four cavities and utilizes a two-plate opening system, and so it has one parting plane. A design of the mold contents a three-dimensional model of injection mold, which is assembled of 277 components, and two-dimensional drawing of the mold assembly including a parts bill. Largest dimensions of the mold are 396 x 446 x 581 mm and for relocation movements it can be hanged up by two lifting loops. In the Figure 60 can be seen an injection side of the mold with all its necessaries such as plugs for centralize, hot manifold joined to a 16-socket, leader pins for leading a movement of an ejection side, cavity inserts with internal shape inserts, clamping slots for mounting on an injection machine and cavity cooling channels (transition between inserts and plates are secured with O-rings). Figure 61 illustrates an ejection side of the mold. Among others, the ejection side comprised a whole ejection system, a core cooling system, core inserts including internal shape inserts and so on.

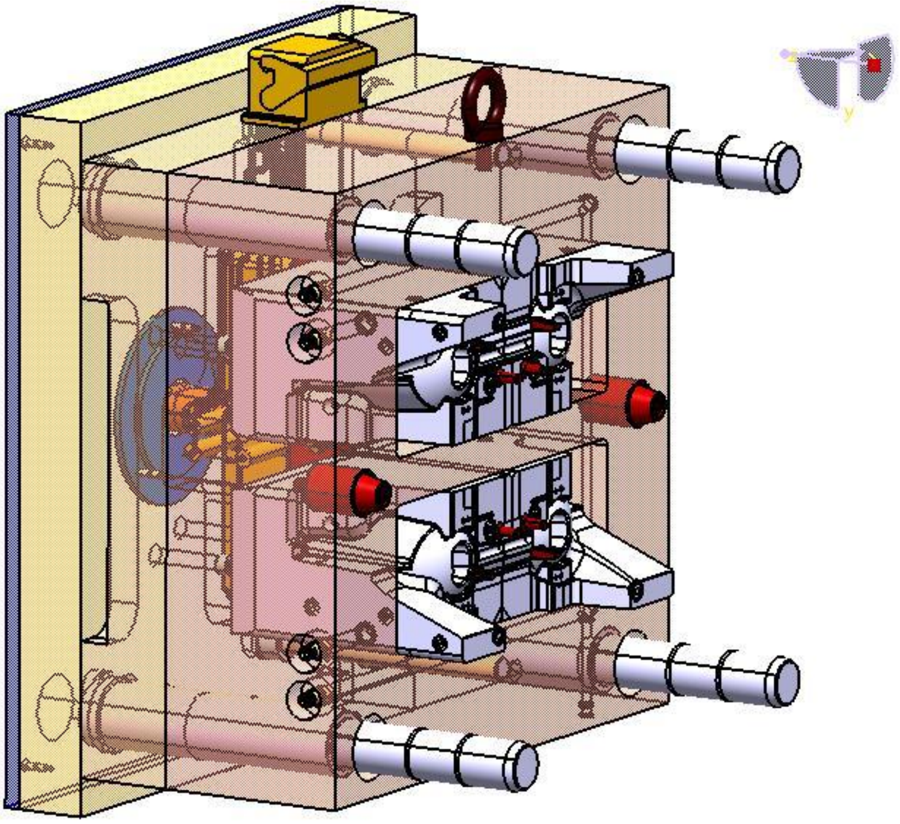


Figure 64 Injection side of the mold

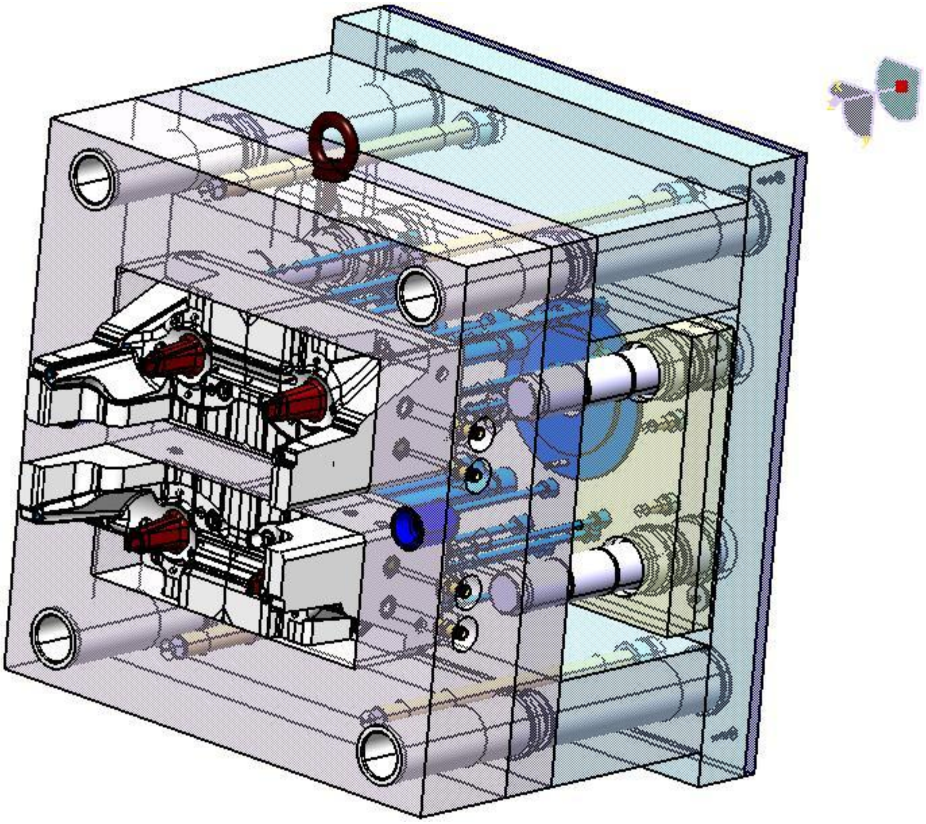


Figure 65 Ejection side of the mold

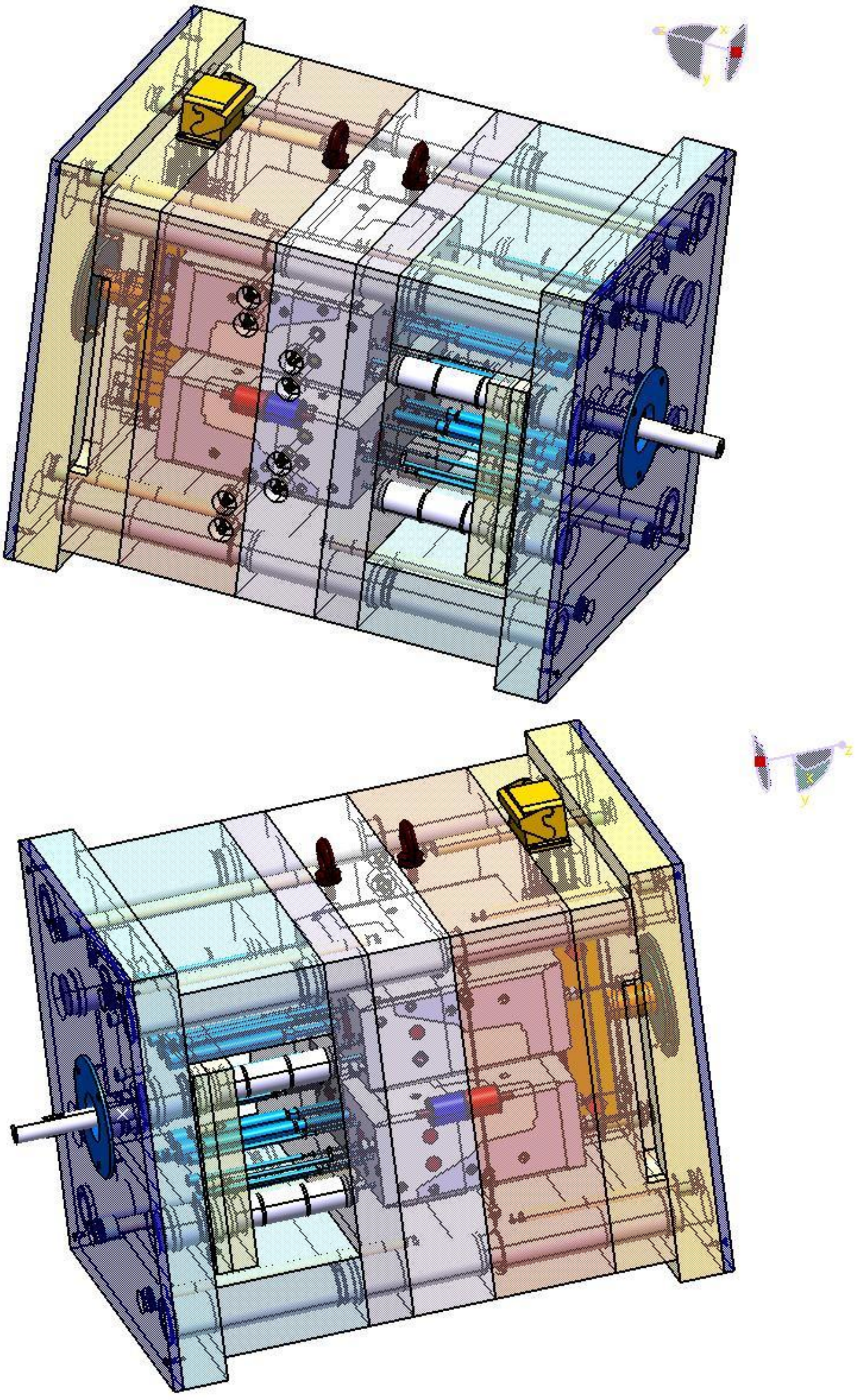


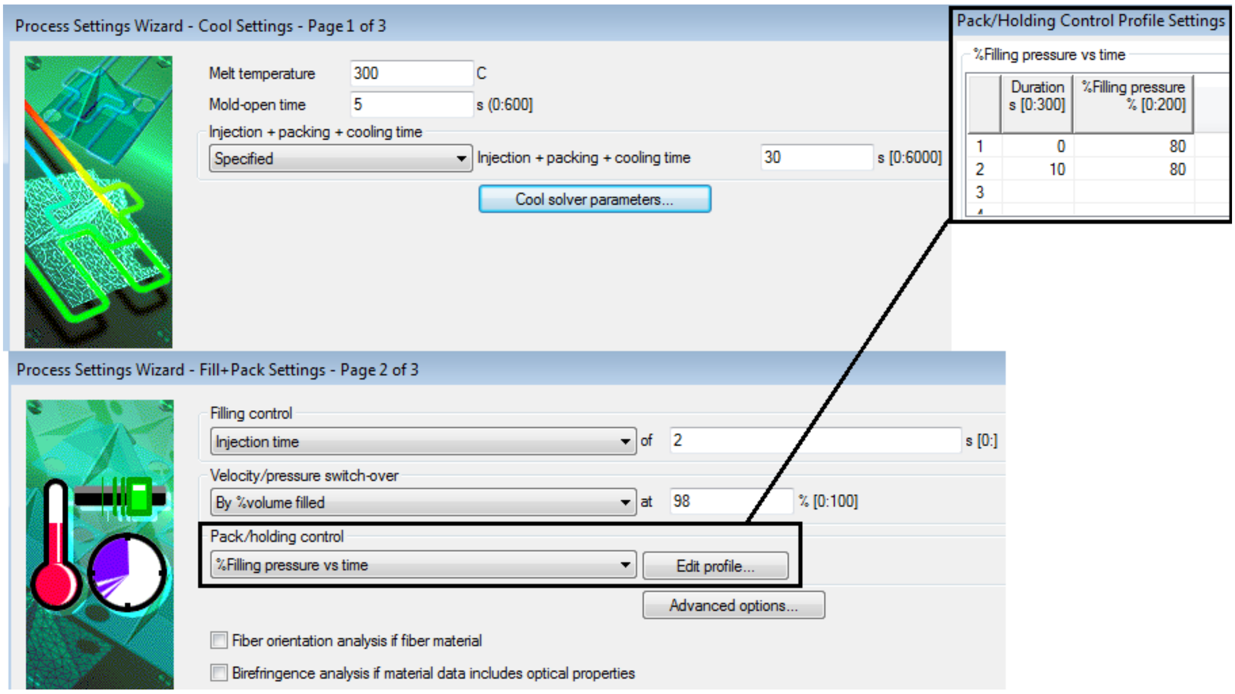
Figure 66 Assembly of the injection mold for a specific part

8 COMPLETE ANALYSES OF DESIGNED INJECTION MOLD

The analyses for designed injection mold were processed in Moldflow in two versions of cooling system. Both of them were processed under the same process parameters. The results of analyses, which will be mentioned thereinafter, are divided into three chapters – flow, cool and warp analyses. Unlike cool and warp results, the flow results are described only for one type of cooling system, because they are independent of cooling system.

8.1 Process parameters

In the Figure 67 are shown process settings from Moldflow.



Process Settings Wizard - Cool Settings - Page 1 of 3

Melt temperature 300 C

Mold-open time 5 s (0:600]

Injection + packing + cooling time

Specified — Injection + packing + cooling time 30 s [0:6000]

Pack/Holding Control Profile Settings

%Filling pressure vs time

	Duration s [0:300]	%Filling pressure % [0:200]
1	0	80
2	10	80
3		

Process Settings Wizard - Fill+Pack Settings - Page 2 of 3

Filling control

Injection time of 2 s [0:]

Velocity/pressure switch-over

By %volume filled at 98 % [0:100]

Pack/holding control

%Filling pressure vs time

☐ Fiber orientation analysis if fiber material

☐ Birefringence analysis if material data includes optical properties

Figure 67 Process parameters for complete analyses

8.1.1 Injection molding machine

In view of the fact that the clamp force, required injected shot and dimensions of the injection mold were known specifications, the injection molding machine with suitable parameters could be selected in Moldflow.

Required parameters:

- dimensions of the mold: 396 x 446 x 581 mm, important to know dimensions among leading columns of the injection molding machine

- clamp force: According to fill analysis the clamp force of the machine has to be at least 125 tons (1250 kN)
- one injected shot: 144 g including a technological waste in a form of cold runner, which makes 11,52 % (8 g) of a whole shot.

Selected injection molding machine:

Arburg Allrounder 520C 176 tons, (45mm)

Specification of the machine:

- Distance among leading columns: 520 x 520 mm
- Clamp force provided: 1300 – 1500 kN
- Weight of shot: 141 – 232 g

Table 6 Required and provided values comparison

	Required	Machine provides
Mold clamping dimensions	396 x 446 mm	520 x 520 mm
Clamp force	1250 kN	1300 – 1500 kN
Wieght of shot	144 g	141 – 232 g

Tabe 6 shows comparison of values required to successful production of the part and values provided by selected injection molding machine.

8.2 Flow analysis

Unlike next types of analyses, flow analyses are independent of cooling system and hence they are desribed with the only one picture and its specification. A flow analysis predicts polymer flow and freezing inside the mold cavity and runner system. Because some of the results of this specific runner system were described above, they will not be repeated in this chapter.

8.2.1 Fill time

The result of filling time is useful for reveal good balanced flow pattern in cavity. It can be noticed when all flow paths finish at the same time (which is important especially in the case of four impression mold), or when the contours are evenly spaced. Figure 68 illustrates that cavities were filled, which is worth of the process condition and well-designed part with runner system. If some unfilled sections appear, it would be rectified by decreasing of a pressure or temperature of the polymer. On the other hand, also an overpacking effect can takes place there. The effect happens when on flow path finishes before other do, and can be a reason of high-part weight or undesirable warpage which can lead to structural or visual defects.

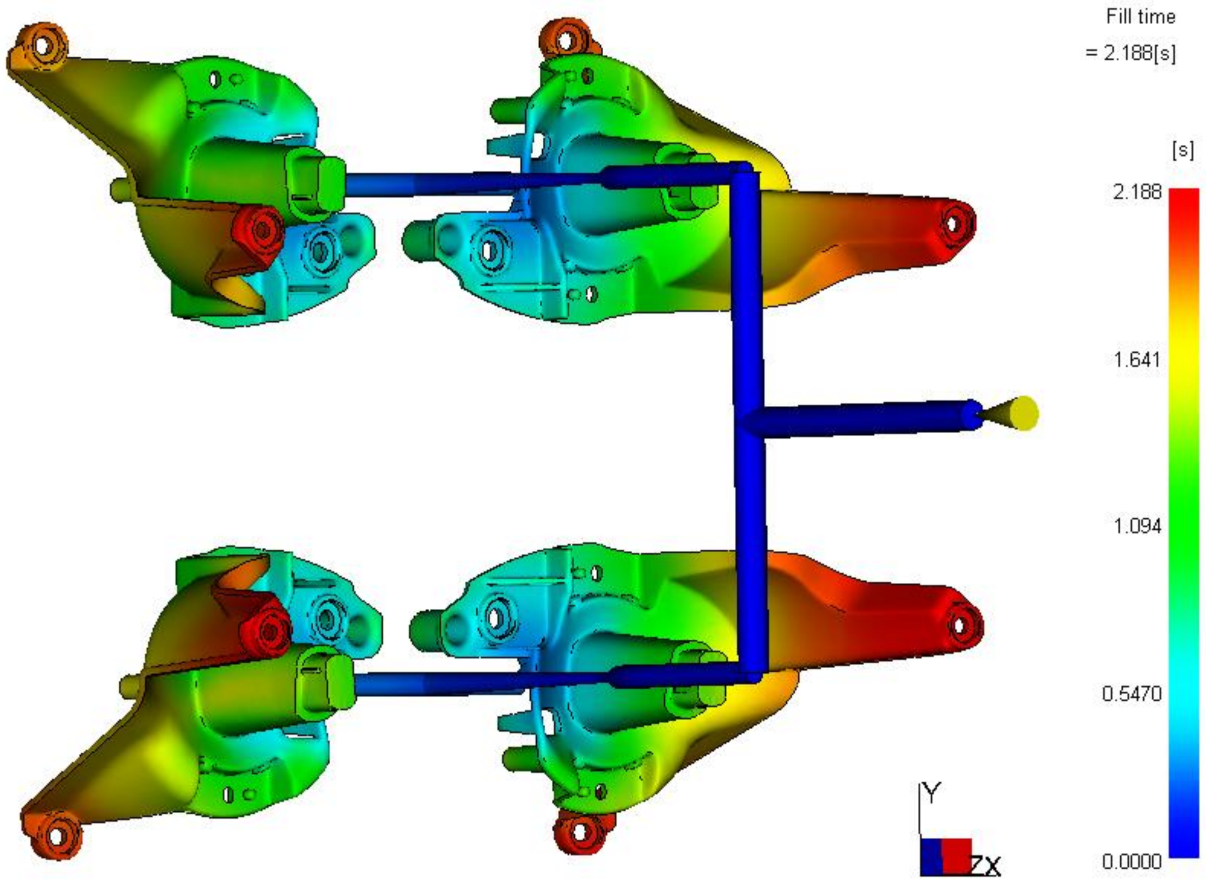


Figure 68 Fill time

8.2.2 Pressure at the injection location, XY plot

A very useful result to ensure the packing profile is correct and also helpful to checking whether there are any imbalances or pressure peaks in the cavity. The result shows the pressure distribution in time exactly. Figure 69 reveals a pressure distribution and packing profile as it was set up. According to settings of process parameters the packing profile uses only 80 % of the whole delivered pressure at switch over for 10 seconds. Maximum pressure at injection location was detected in 2,126 second and the value is 82,67 MPa. Then, the pressure was reduced at 66,14 MPa.

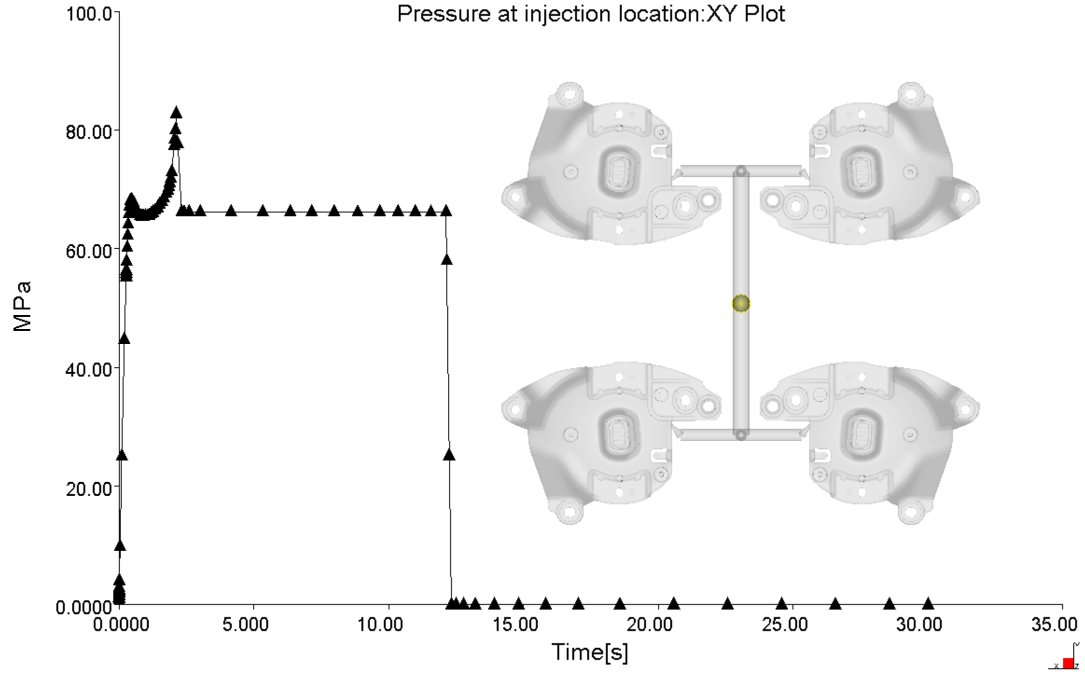


Figure 69 Pressure at injection location

8.2.3 Temperature

Figure 70 illustrates a temperature result of the part and takes advantage from 3D analysis technology. On the right side of the Figure is shown, in detail, mold cavity filling at the time of 1 second, where can be clearly seen difference among temperature distribution across part thickness. On the right side of the Figure is shown and marked temperature distribution in cavities and runner system. This result can be utilized to looking for early freezed areas, which is a cause of a low injection rate or melt temperature. Next effect, which could be useful to looking for is a hot spots. Hot spots can be indicators of shear heating, which usually takes place around the gate, or on the last filled place. Maximum temperature of the melt, shown on the Figure 65 is 378,5 °C, which is measured in the hottest spot of the polymer melt. On a part surface is measured and marked temperature 96,4 °C, which is acceptable in view of material and process settings.

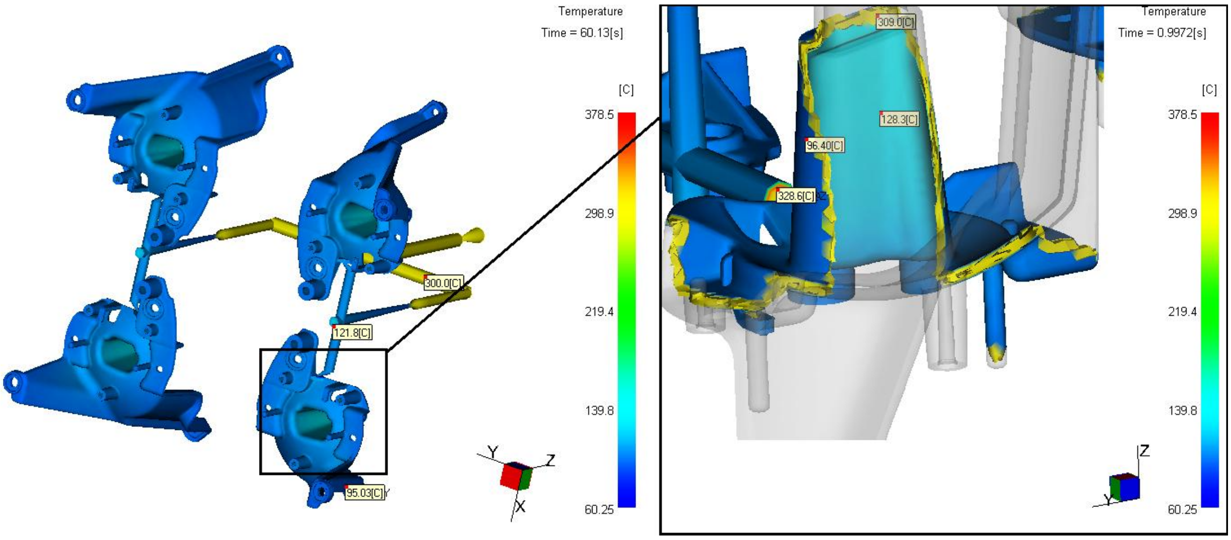


Figure 70 Temperature

8.2.4 Temperature at flow front

For demonstrate a temperature difference between the temperature on the surface of the cavity and in the center of a polymer melt, an outcome named Temperature at flow front was added. The result in the Figure 71 reveals an exact temperature in the center of the plastic cross-section, when the flow front reached a specified point. On the left side of the Figure are shown cavities and runner, with marked temperatures of various values. But unlike these almost usual temperatures, on the right side of the Figure is marked the temperature value, which is nearly highest. The problematic place is a rib colored with red, where was detected 310 °C. This place may be cause of hesitation, material degradation or visual defects. Hesitation or too low injection time were also indicated thanks to larger changes of a flow front temperature during filling phase. Unlike detected change of more than 10 °C, an optimum is about 2 or 5 °C. A resolution is increasing of the injection time. In addition, in the Figure can be seen also very low temperatures about 301 or 302,4 °C. If they were detected in a thin area, it would be a result in a short shot. For improve very low temperatures at flow front, changing of the following process setting may help: decrease injection time, increase mold temperature, increase the melt temperature in material processing boundaries.

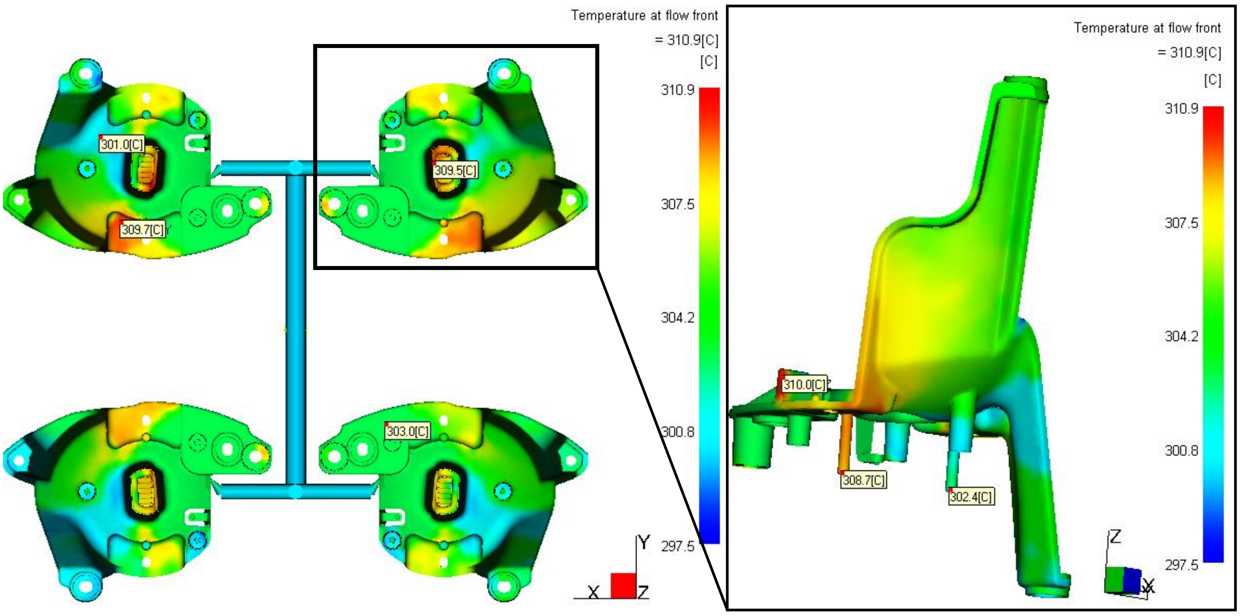


Figure 71 Temperature at flow front

8.3 Cool analysis

Selected outcomes of cool analysis are presented thereinafter, every result in two types of cooling for effortless orientation (first conventional, then conformal version – always in this order). As a coolant temperature was selected a pure water under pressure of 3 bar (300 kPa), which according to analysis log gives a flow average of 21 l/min. A coolant temperature is 95 °C. A diameter of conventional cooling channel is 10 mm, and diameter of conformal cooling channel is 8 mm. The smaller dimension was selected due to possibility of cooling of more part details and more complex trajectories of cooling channels.

8.3.1 Circuit coolant temperature

The temperature of the coolant inside the cooling circuits was detected by the Circuit coolant temperature result shown in the Figures 72, 73. In Figures are marked inlet and outlet temperatures and also in the middle of each circuit. Changes between coolant temperature of inlet and outlet should not be greater than 3 °C to ensure uniform cooling of the part. In the first case of conventional cooling is difference of 0,09 °C. The highest temperature of the conventional system was detected in the center of the cooling circuit and its value is 95,08 °C. That small deviation is caused by a convective heat transfer from melt through the steel to the coolant.

Figure 68 shows the temperature of coolant inside the conformal system. The deviation of coolant temperature in this case is 0,35 °C, which is caused by a smaller distance between cooling channels and mold cavities.

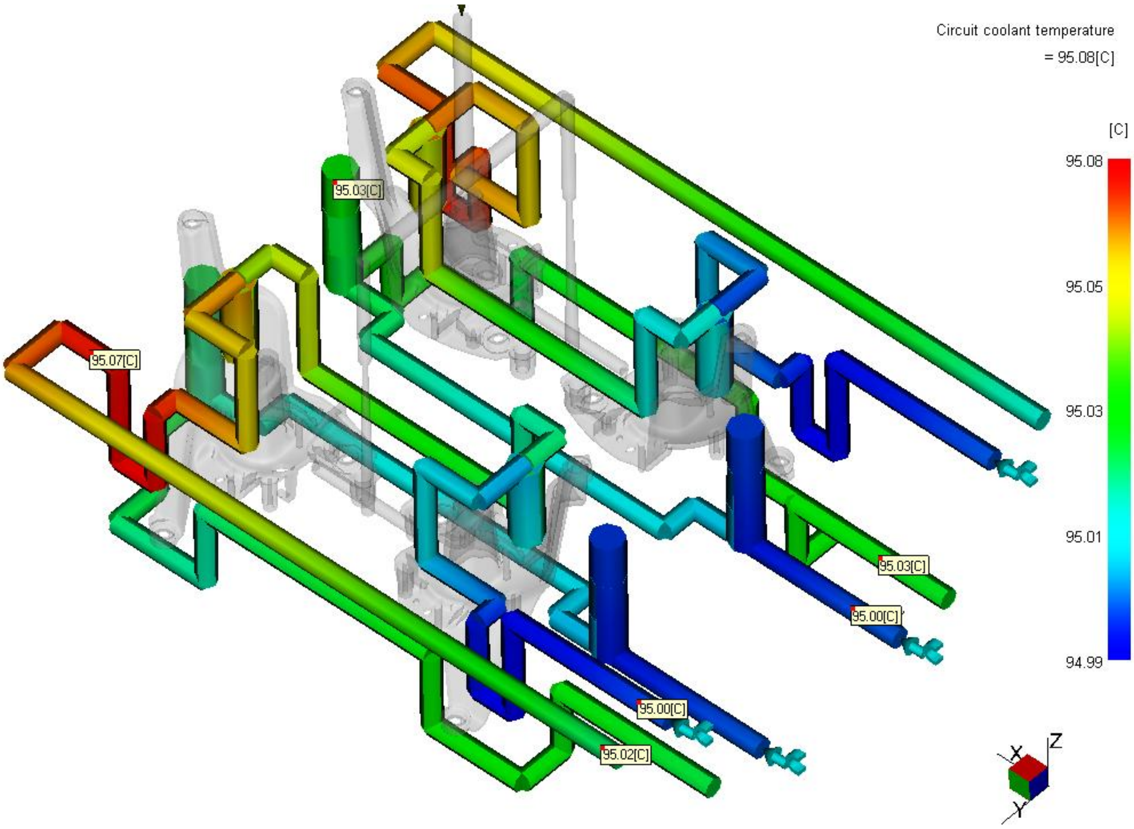


Figure 72 Circuit coolant temperature, conventional system

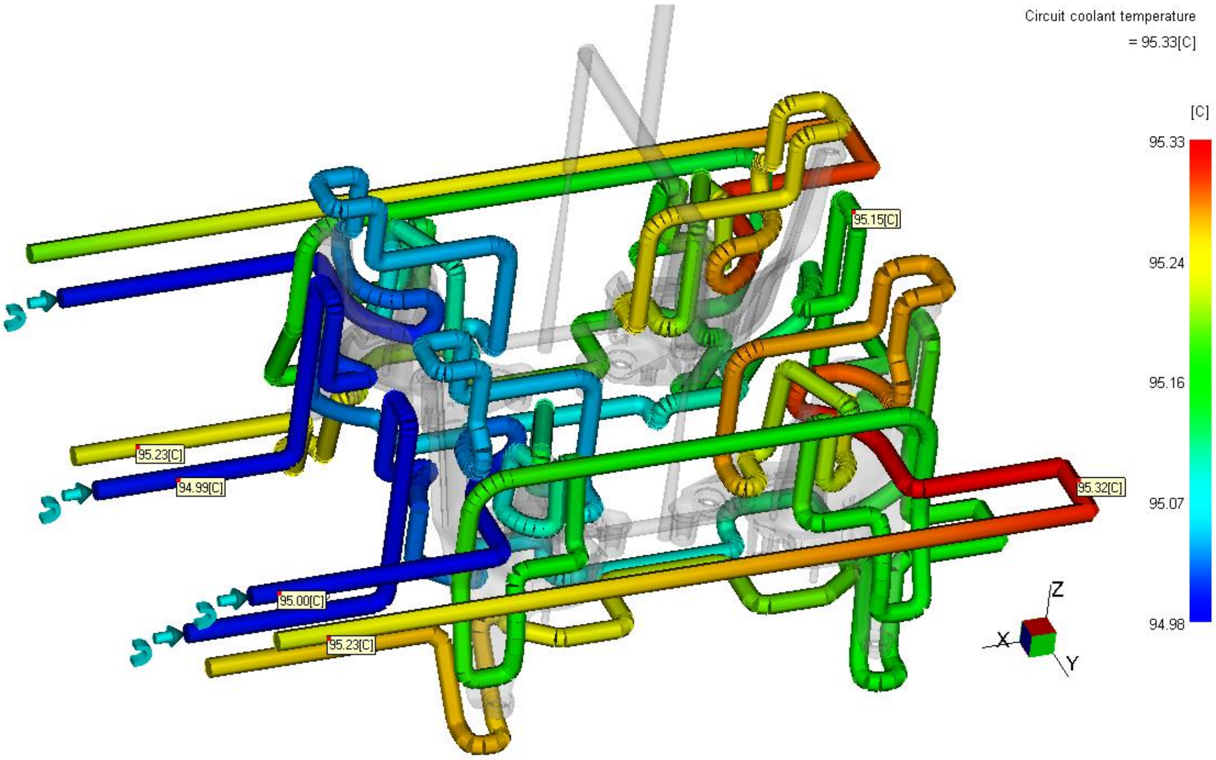


Figure 73 Circuit coolant temperature, conformal system

The conventional and conformal cooling system proves small difference between above mentioned values, which indicates a narrow mold surface temperature range. It means internal strain of the plastic part is of consideration. In the case of wide coolant temperature variance, trajectories of cooling channels should be changed.

8.3.2 Circuit heat removal efficiency

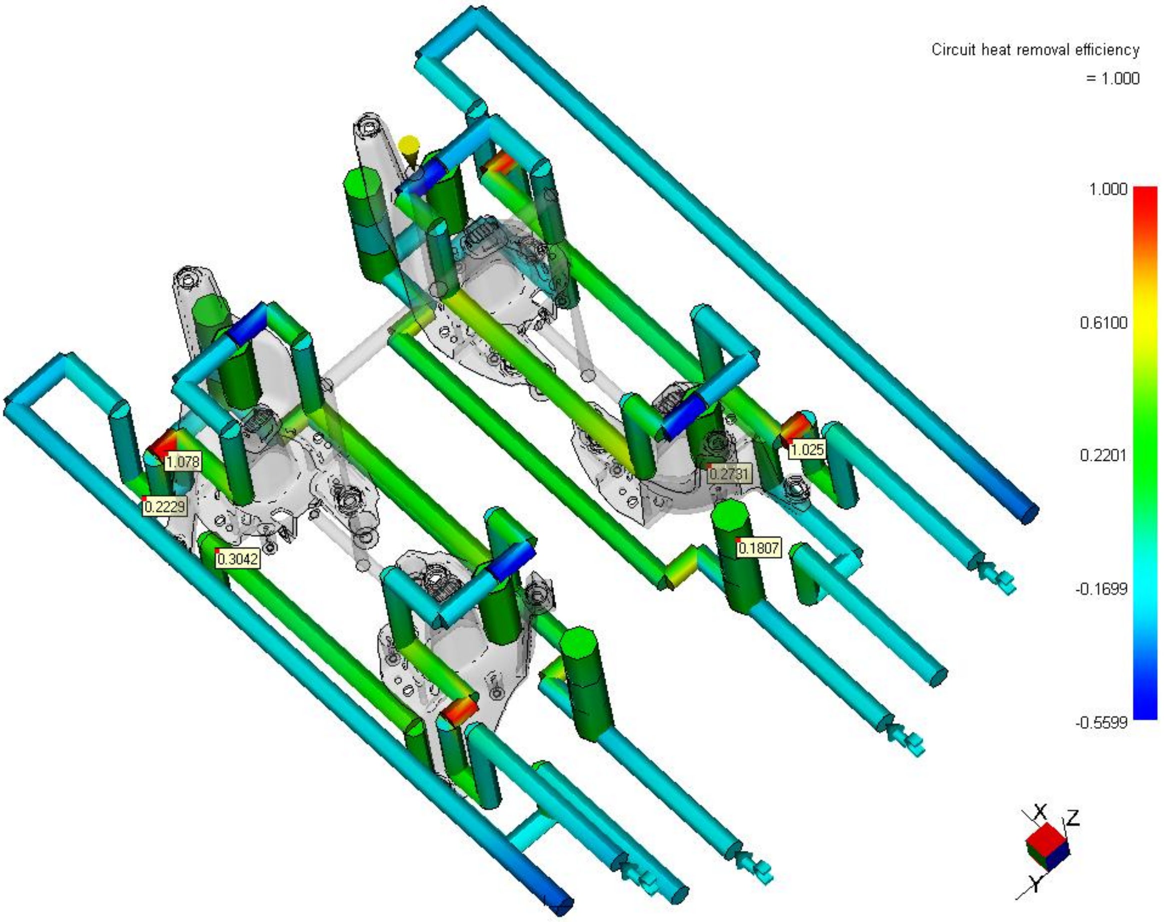


Figure 74 Circuit heat removal efficiency, conventional system

The extracted heat from the mold during a cooling cycle is demonstrated in the Circuit heat removal efficiency result. Figures 74 and 75 shows effectiveness of heat transfer (in percentage) from the mold cavity to flowing coolant inside each cooling channel. The result helps to identify which channel section removes more heat relative to the others. Sections of cooling channels which are red colored has the best heat removal efficiency and they are marked with a value 1. On the other hand light blue colored sections marked with a value nearby 0 have the worst heat removal efficiency.

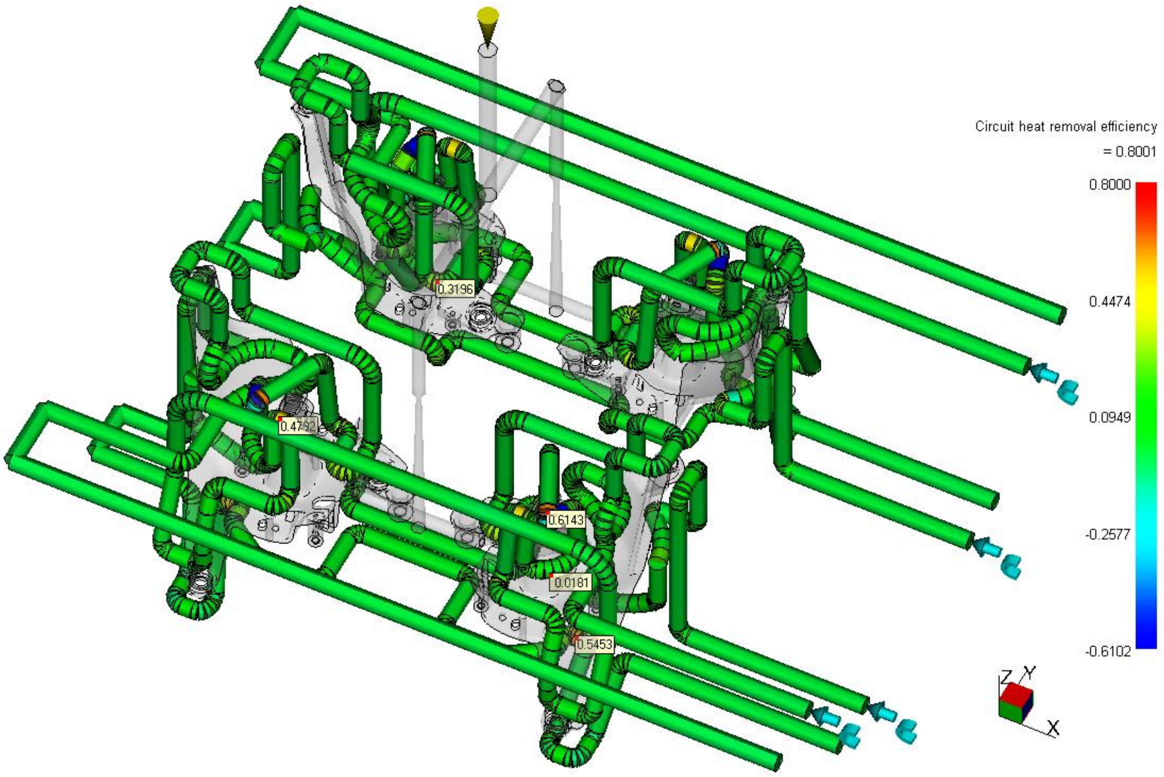


Figure 75 Circuit heat removal efficiency, conformal system

Sections colored with dark blue have a minus value and conversely, they give the heat from coolant to the mold. The heat removal efficiency is relative to a distance between cooling channels and a part, circuit Reynolds number and temperature difference between the coolant and the cooling channel. Even though, conformal cooling has a distance between channels and cavities a good deal less than a conventional cooling system, the effectiveness of heat remove is greater in this case. The heat removal efficiency of the conventional system was detected in average from 10 to 50 %, speaking about sections around the mold cavities (Fig. 74). The average detected heat removal efficiency of the conformal system was from 10 to 40 %. The lower difference could be caused by the smaller diameter of cooling channels in conformal cooling system. As can be seen, even much more complex trajectories of cooling channels, cannot compensate higher volume of coolant, contained inside cooling channel with greater dimension.

8.3.3 Circuit Reynolds number

The coolant Reynolds number inside the cooling circuit is close related to the heat removal efficiency. The Reynolds number greater than 10 000 is a guarantee of turbulent flow of the coolant which is contributing to greater remove of heat from the mold. The greater is a cross-section of a channels, the lower is the Reynolds number of the coolant. The greater a coolant flow rate, the greater Reynolds number. Conventional channels, shown in the Figure 76, have a diameter 10 mm. As can be seen the Reynolds number is much lower inside a baffles with diameter 16 mm. The lowest detected value is 113 900 which is sufficient for turbulent flow.

As for conformal cooling channels, they have a diameter 8 mm and circuits do not contain baffles. As can be seen in the Figure 77, minimum value of Reynolds number is 96 500 and maximum is 126 700. The difference of 32 000 between each channels has no influence on turbulent flow inside channels, because values exceed 10 000 criteria. And so effective cooling of the parts is ensured.

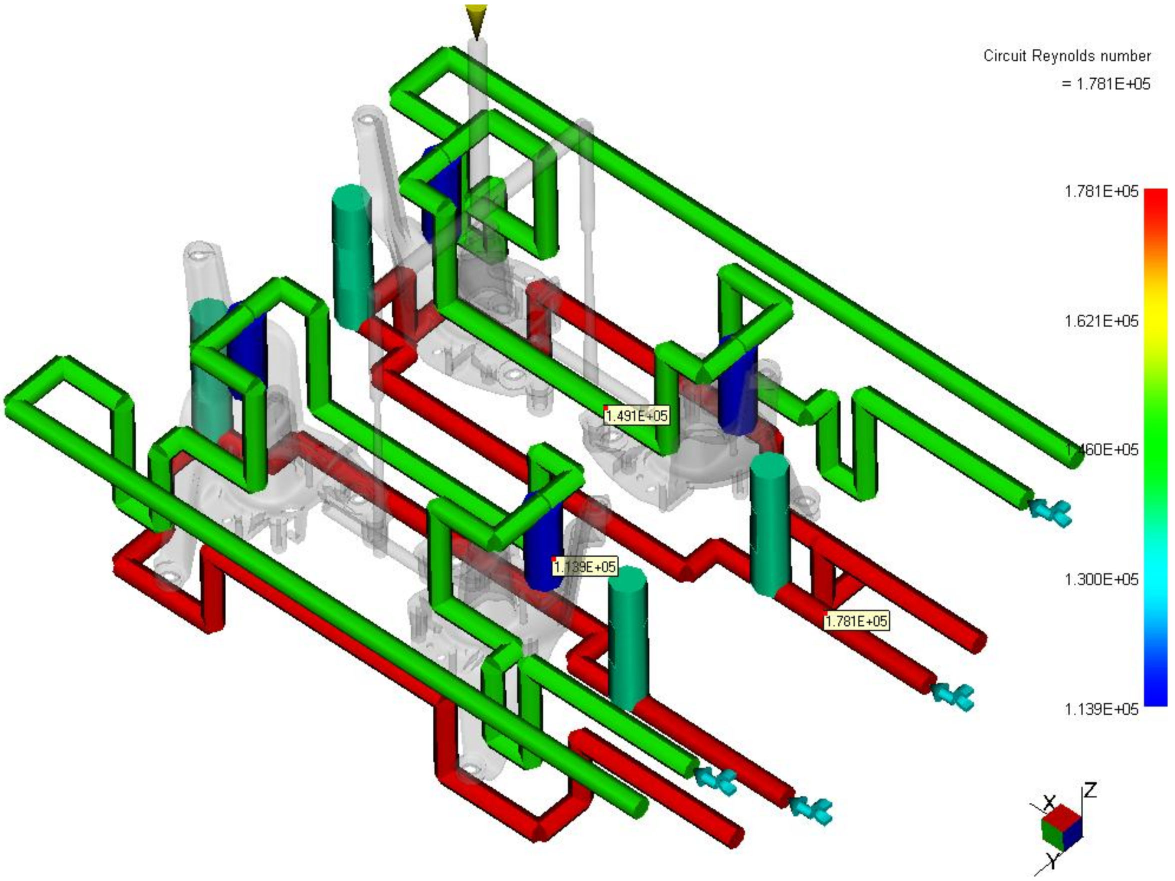


Figure 76 Circuit Reynolds number, conventional system

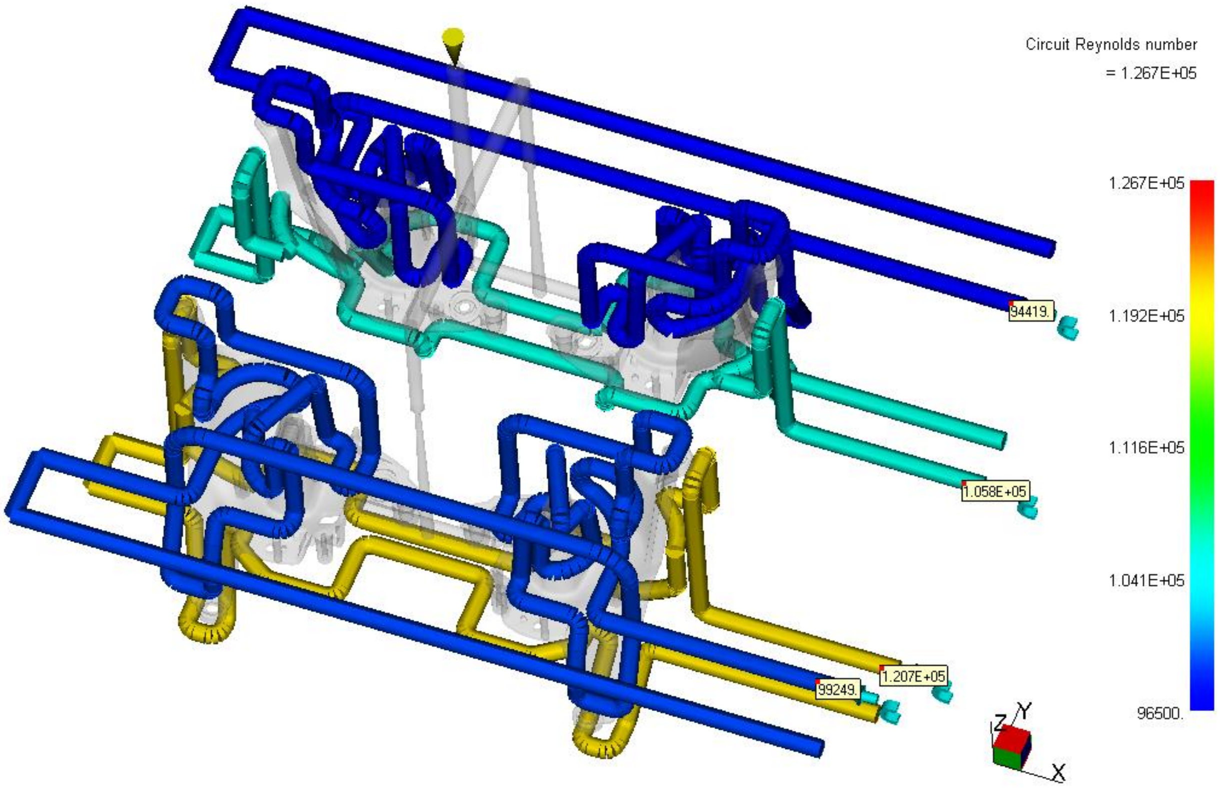


Figure 77 Circuit Reynolds number, conformal system

8.3.4 Circuit pressure

Figure 78 illustrates the pressure of coolant inside of conventional cooling channels. As can be seen in the Figure 78, delivered pressure of 300 kPa was almost complete spent until coolant reached an outlet from the channel. It was caused due to baffles contained in the circuit. Behind baffles is a significant pressure drop. A marked value 290,3 kPa on the inlet was quickly reduced to 249,5 kPa, after a coolant passed through the baffle. In one circuit is two baffles which is satisfied to a rule of maximum number of baffles within one circuit (the number is five). The pressure inside a last discharge branch is lamentable. Nevertheless this branch is not useful for the cavities cooling.

Figure 79 illustrates conformal cooling channels which have diameter 8 mm and do not contain any baffle. The pressure drop inside conformal cooling circuits is much more gradual and slow, as inside conventional channels. But decrease of pressure is still very noticeable. The changes of delivered coolant amount can have bad influence upon heat removal effectiveness of coolant.

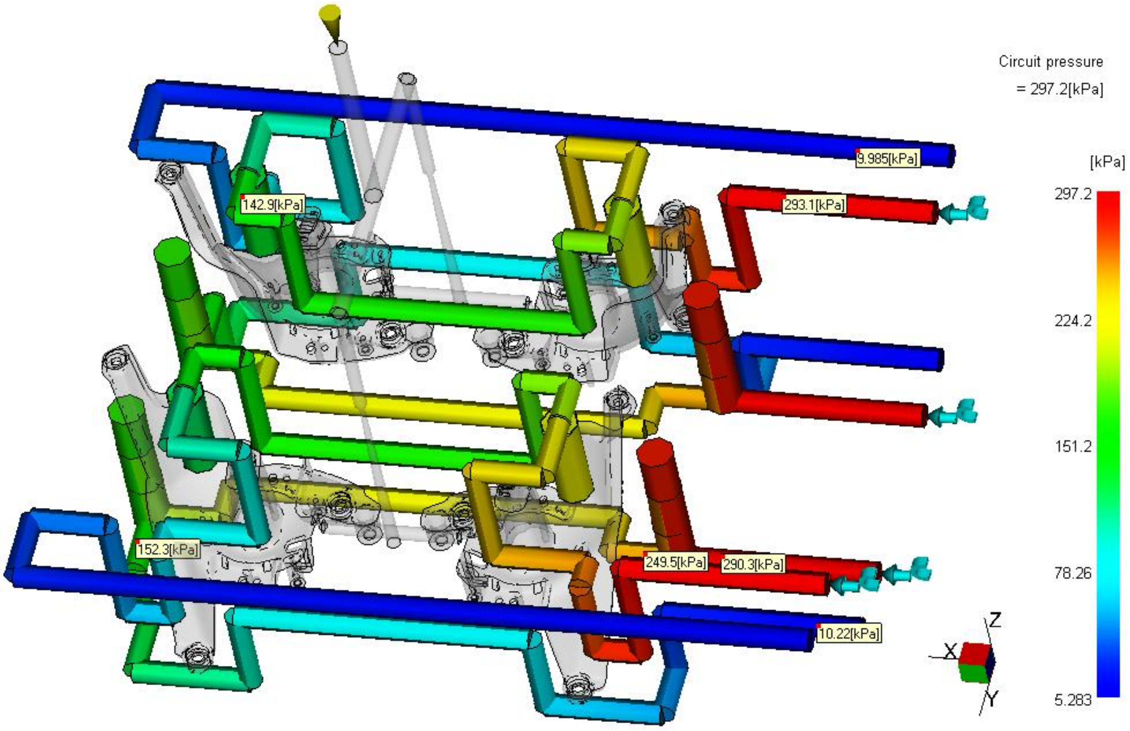


Figure 78 Circuit pressure, conventional system

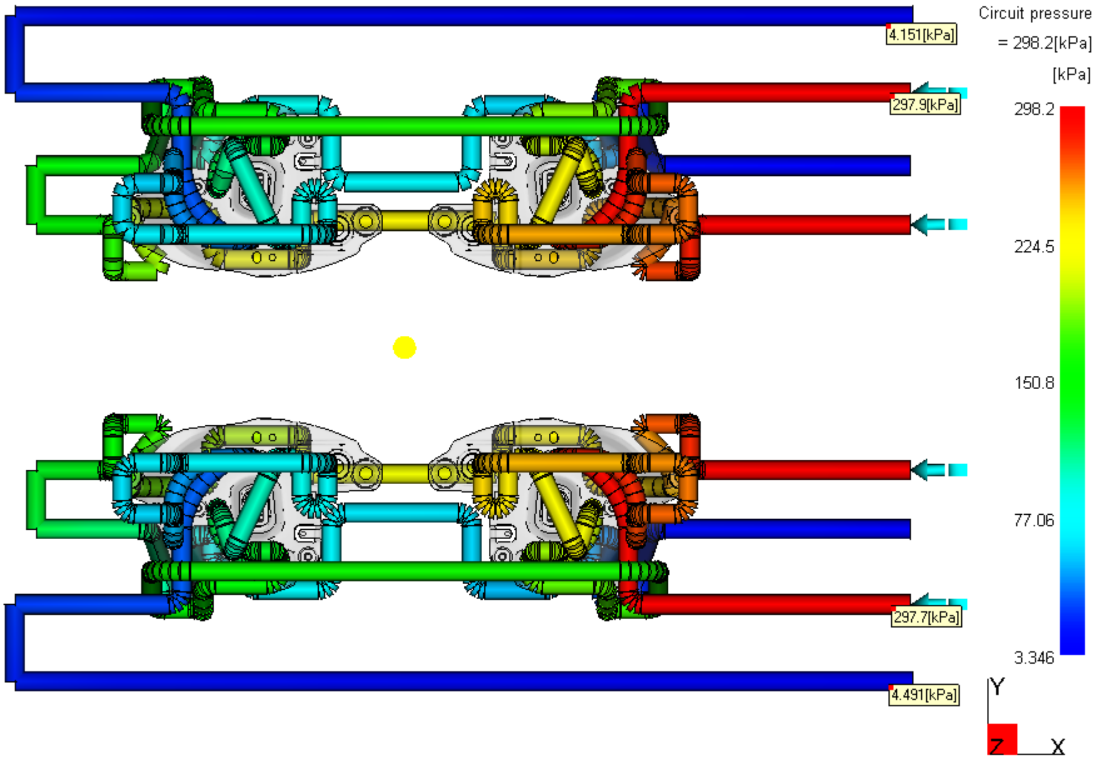


Figure 79 Circuit pressure, conformal system

8.3.5 Time to reach ejection temperature, part and cold runner

This result shows a time required for reach ejection temperature of used material in the mold cavities. Time is measured from the start of the cycle and is different for the part and for the cold runner. The result is very important from the economical point of view. Reduction of freezing time leads to reduction of a whole cycle time which brings a savings for a part price.

Figure 80 shows times for conventional type of cooling system. In the Figure were marked a points where an ejector system acts. As can be seen values are from 1,9 to 2,7 second for a part and 50,3 second for a cold runner – in the area, where a sprue retainer is located. The longest time for the part is 30 second and for the cold runner it is 53,24 second.

Figure 81 illustrates times for conformal type of cooling system. As can be realized after comparison of Figures, times to reach ejection temperature on the part have a difference of about five tenth, with dependence of measured place. Conversely, cold runner of conformal cooled system needs about 5 second less time to reach ejection temperature, on the place sprue retainer location. This fact has a particular influence to the injection cycle time.

The result can be improved with change of process parameters, such as, increase of flow rate in cooling channels to ensure turbulent flow and efficient heat extraction, or decrease an inlet coolant temperature to reduce mold surface temperature. The comparison of both, conventional and conformal cooling system can be seen in Table 7.

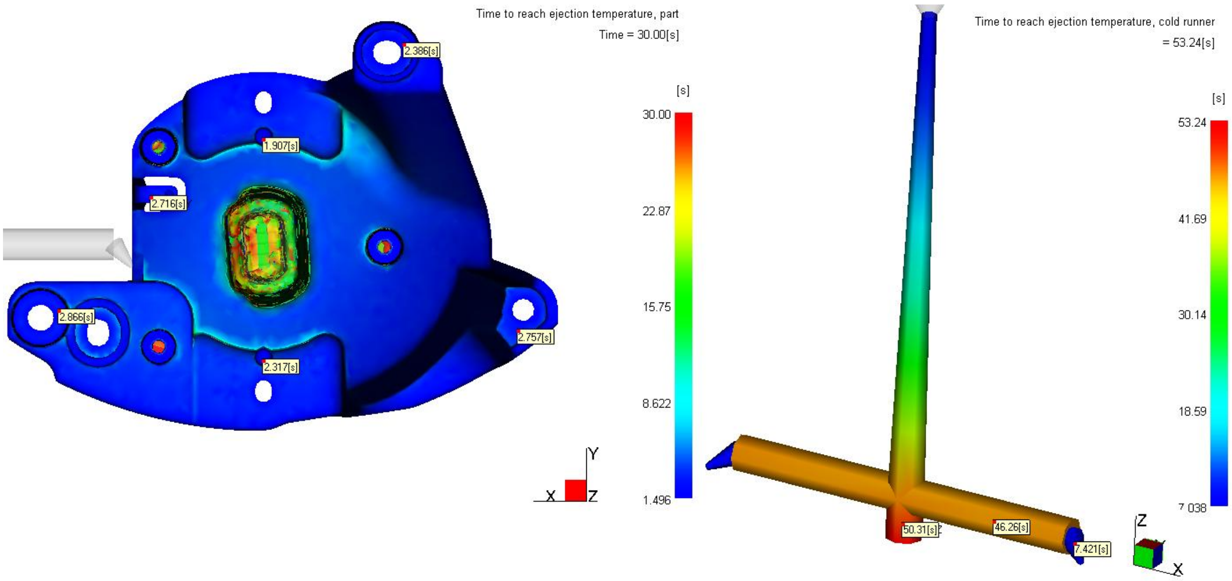


Figure 80 Time to reach ejection temperature of part and cold runner, conventional system

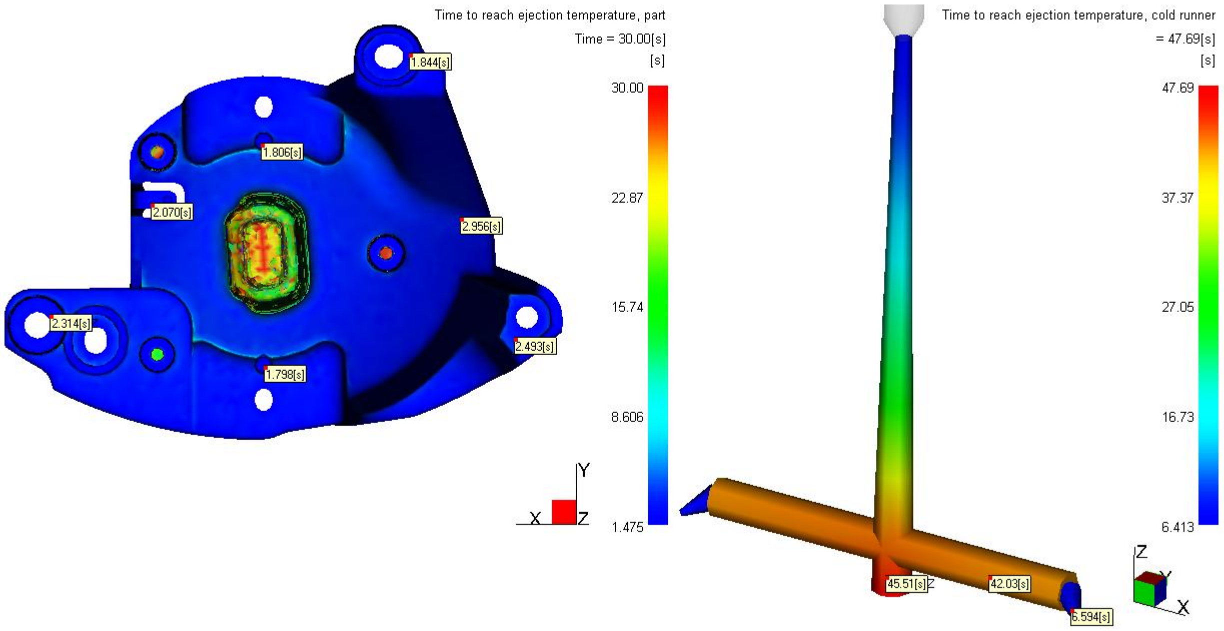


Figure 81 Time to reach ejection temperature of part and cold runner, conformal system

Table 7 Comparison of times to reach ejection temperature

	Part [s]	Cold runner [s]
First cooling	30	53,24
Second cooling	30	47,69
Difference [%]	0 %	2,95 %

Cycle time reductions of 5,55 second are not that significant to compensate a DMLS technology cost, and so, the conventional cooling system is considered as better and more suitable for temperature treatment of the injection mold.

8.4 Warpage analysis

The analysis of warp shows a total deflection at each point of a part.

8.4.1 Deflection, all effects

This result contains all effects of deflection, for example corner effect, and shows molded parts deflection in direction of X, Y and Z axis together.

Figure 82 reveals a deflection in five-multiple scale with details of areas which are the most impacted by shrinkage and warpage effects. As can be seen the most impacted area is upper hole of the part, which is predicted to be clamped by with other product. Deflections on this very important place are 0,8 mm which is at the same time the greatest deflection of the whole part. The warpage is influenced by cooling system – distance of cooling channels, pressure, type of flow and used medium of the coolant, material used to fill the cavity and material used to cavity insert, runner location, packing and time of packing phase. The shrinkage value was considered into cavity design. In other words a cavity dimension was increased according to shrinkage value of used polymer.

Figure 83 shows a deflection of the part with details of the most impacted areas, in greater scale. Deflections of conformal cooled part are smaller in comparison with the molded part cooled with conventional system. It can be caused by smaller diameter of cooling channels.

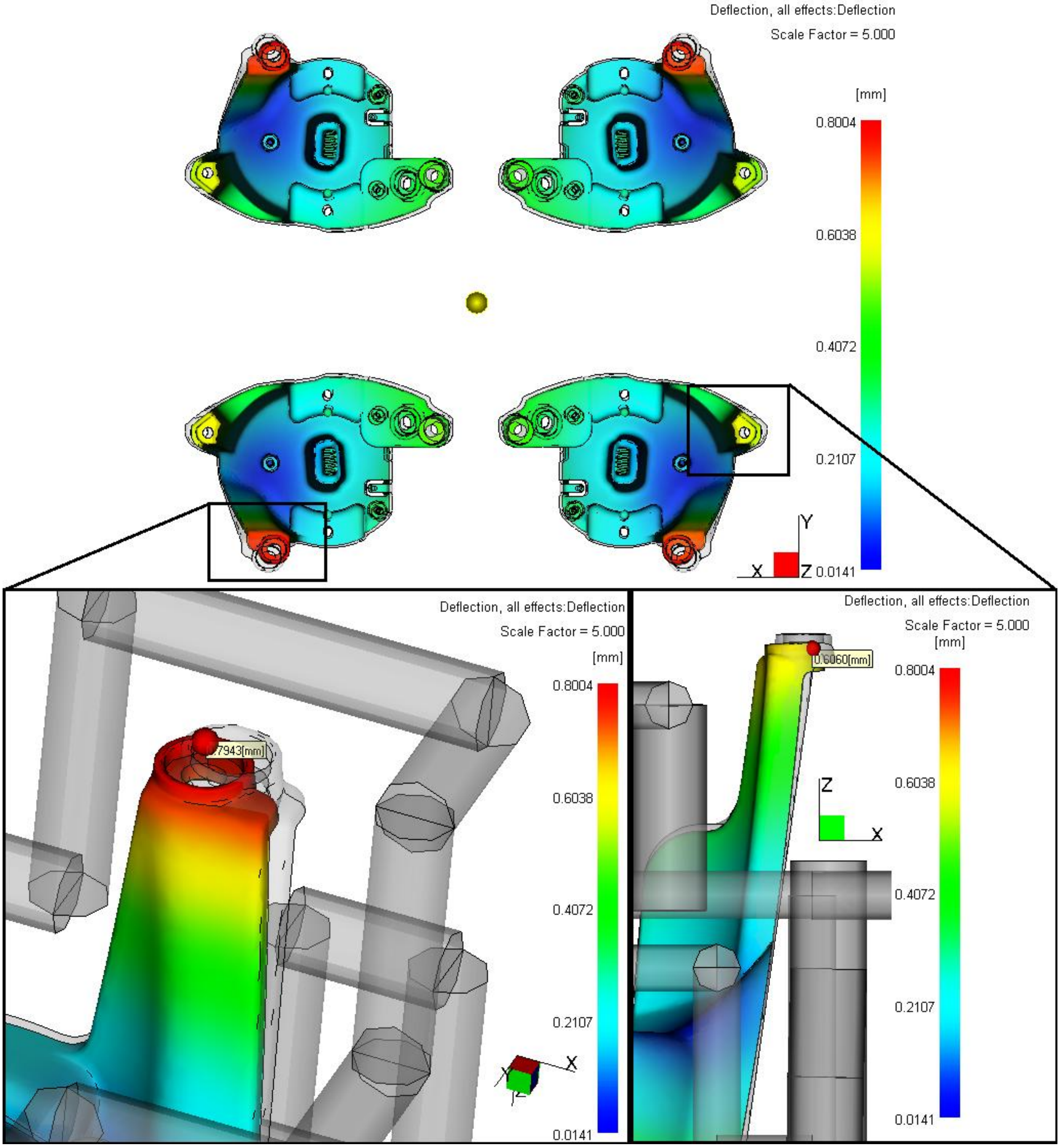


Figure 82 All effects of deflection, conventional system

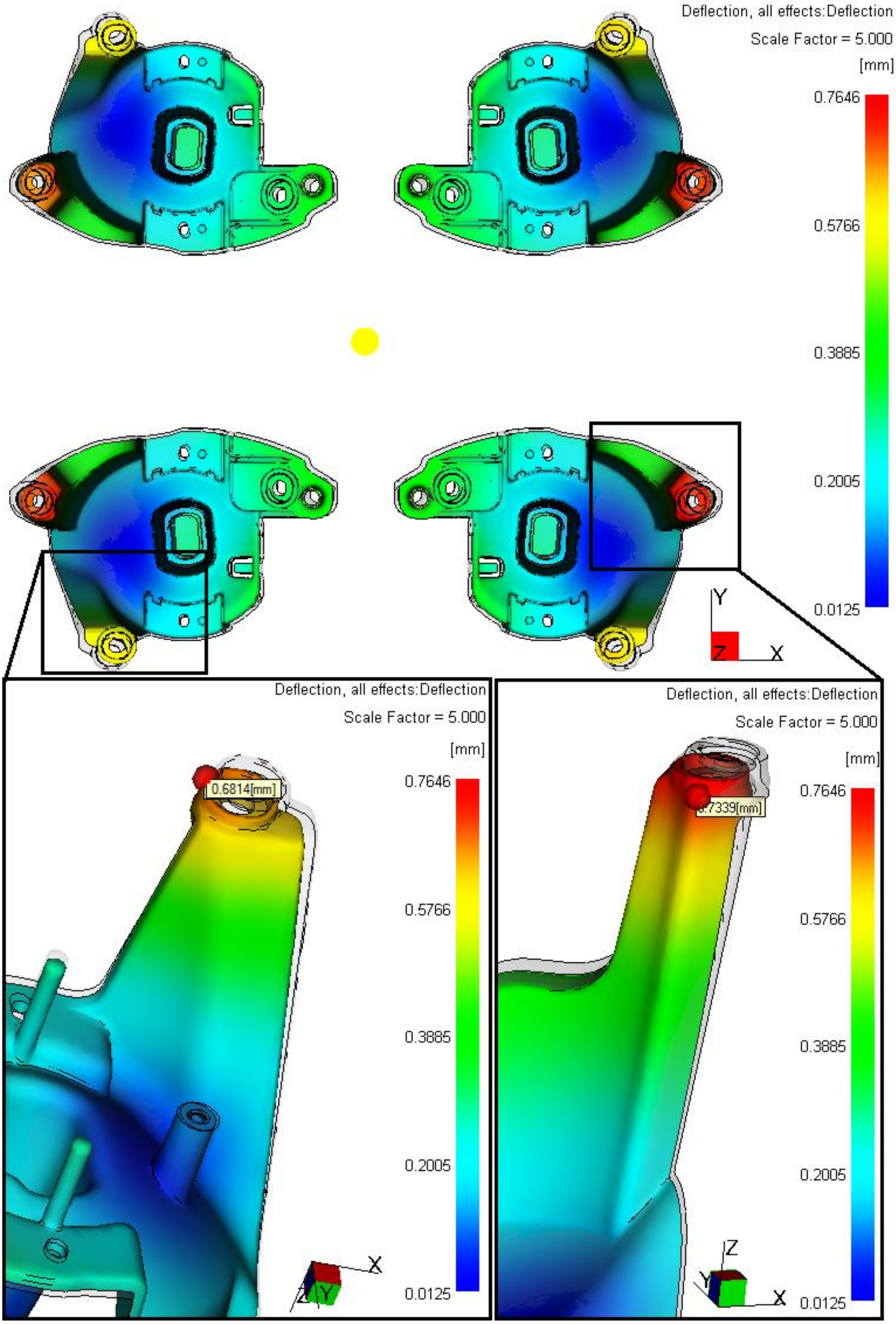


Figure 83 All effects of deflection, conformal system

9 RESULTS DISCUSSION

9.1 Results of filling analyses

The filling analyses for given plastic part were launched in the software Moldflow. To a possibility of launch filling simulations of the injection cycle, trajectories of each runner system were necessary. After considering and designing of various injection sides of the injection mold, to ensure the correctness of dimensions, the trajectories were built in software Catia V5. Specifications of runner systems are discussed in the Chapter 6. Filling analyses of the injection molding cycle were necessary to decide about the most suitable runner system for the mold.

Every type of six runner systems was designed in two versions of cold runner – with trapezoid and circular cross-section. After specification of process parameters in Chapter 6, was putted emphasis on a comparison of two cross-sections according to Shear rate and Pressure result. In Table 8 can be seen a difference among each values, with divided sizes of cross-section.

Table 8 Cold runner comparison

	Trapezoid, greater cross-section	*Circular, smaller cross-section*
Shear rate [1/s]	13 362	33 675
Pressure [MPa]	41,79	41,94

Shear rate is a suitable simulation outcome and index for consider a possibility of material degradation and hot spots appearance. A pressure result has no noticeable changes between the results. Nevertheless, Figure 42 and 46 show its gradual increase and further more changes, which is interesting from the process settings point of view. The difference in shear rate of circular runner corresponds with almost one half decrease of cross-section area. Very high shear rate can lead to degradation of polymer. This problem can be also solved with decrease flow rate which passes through the runner system.

According to simulation results and manufacture reasons, the runner with circular cross-section was selected for further process of given tasks. Although, shear rate was reduced, thanks to use trapezoidal runner with increased cross-section, the circular runner, from the manufacturing point of view, is cheaper and simplified solution.

Further description of filling analyses results was only related to runners with circular cross-section of cold runner. There was selected only one result to describe each runner system because results were very similar. Amongst the most useful results counts a Pressure at injection location, which was fluctuated from 64,1 MPa up to 100 MPa. Next useful outcome is clamp force which could be used to determination about an estimated value of injection machine clamping force to be needed for complete analysis. Clamp forces result from filling analyses were vary from 20 tons to 68,1 tons (200 – 681 kN).

The most suitable runner system had to be selected. From all possibilities and evaluated results was chosen the one runner system for four cavities located like H letter with combination of hot and cold runner. Unlike the cold runner, combined runner systems have a particular advance in a hot material inside hot runners, which leads to cycle time savings. Cavities located into shape of the letter H have an advance, against X shape cavities, in smaller dimensions of runner, which caused in smaller mold dimensions.

9.2 Solution of the injection mold design

The injection mold was designed for a plastic product provided by the Hella Autotechnik, s.r.o. company. A plastic part is very shaped complex, but the right way how to define a parting line was found. According to the parting line were created 3D models of cavity and core inserts. With respect to Air traps results from filling analyses was created a shaped inserts to exhaust air from the mold cavities. Cavity and core inserts were mounted to mold plates according to the selected runner system, and further design of a whole injection mold was continue. Runner and ejection system were added as well as clamping and leading elements.

The injection mold design was processed with use of Hasco standards. In a whole Catia product of the injection mold is contained 277 parts. A three-dimensional assembly, assembly drawing and parts bill were added into an appendix of the thesis.

9.3 Cooling system analyses

An introduction of the Chapter 8 contains a specification of process parameters and selected injection molding machine. Following results evaluation deals with flow analysis and four outcomes are demonstrated there.

9.3.1 Flow analysis

Results are described only for one type of cooling system there, because changes in cooling analysis does not influenced this type of analysis. Table 9 is well arranged summarization of these results.

Table 9 Flow analysis results

	Result
Fill time [s]	2,188
Pressure at injection location [MPa]	82,67
Packing phase pressure [MPa]	66,14
Temperature [°C]	378,3
Temperature at flow front [°C]	310,9

To ensure that all cavities was filled and ends of the flow front reached complex details of the mold at least at estimated same time, the fill time result was checked. Mold cavities were filled by polymer within 2,188 second.

Graph of the Pressure at injection location gives information about packing profile. In this case can be seen that cavity is well pressure balanced and pressure decrease on 80 % of delivered after switchover on the packing phase.

Between temperature and temperature at flow front can be seen a significant difference. The Temperature result shows the plastic temperature through the part thickness, whereas the Temperature at flow front reveals the temperature in the center of the plastic cross-section when the flow front reached specific point. That is, the Temperature result is much higher because it is maximal value of the polymer after the flow front pass the specific point, and so in this point is furthermore heated with surrounding melt. Temperature at flow front is good index to see, where the hot spots, or freeze sections can take place. In addition, it has influence upon the weld lines creating.

9.3.2 Cool analysis

Table 10 summarizes selected results from cool analysis which are organized for each cooling system independently, including a result deviation.

Table 10 Cool analysis summarization

	Conventional cooling	Conformal cooling	|Difference|
Circuit coolant temperature [°C]	95,08	95,33	0,25
Circuit heat removal efficiency [%]	1	0,8	0,2
Circuit Reynolds number [-]	113 900	96 500	17 400
Circuit pressure (pressure drop) [MPa]	291,917	292,854	0,937
Time to reach ejection temperature (cold runner) [s]	53,24	47,69	5,55

From Table can be read which cooling system is more suitable for temperature treatment of the injection mold. Furthermore, the only one better result of conformal cooling system is Time to reach ejection temperature, which could be an important index to make an injection cycle shorter. Nevertheless, price of use DMLS technology and costs saved by cycle time reduction should be compared. Five seconds of injection cycle time reduction is surely useful property of conformal cooling, but it cannot compensate a cost of use DMLS technology. As a result of this and other results is cooling system makeable with conventional methods considered as a better and more suitable for temperature treatment of the designed injection mold.

9.3.3 Warp analysis

In a closing stage of the Chapter 8 are evaluated deflections of the part, which are shown in two Figures with details of places the most impacted by warp effects. The greatest influence on deflection has shrinkage, which is primarily affected by intensity of temperature treatment, except shrinkage properties of used material and so on. In other words, a type of used coolant, diameters and trajectories of cooling channels and delivered pressure has significant influence upon deflection. Table 11 offers comparison of these deflections.

Table 11 Deflection comparison

	Conventional cooling	Conformal cooling	|Difference|
Deflection, All effects [mm]	0,8	0,7646	0,0354

Note

Values from Moldflow analyses are very useful and benefical, but they are not exact. The simulation result gives orientational values, which only indicates tendency of polymer and mold material behavior while the injection cycle takes place. An accuracy of result depends on quality of input data. A good example of very useful result utilized during the practical part of this thesis is air traps outcome, which predicted areas of their possible occurrence. After evaluation of this result could be created shaped inserts to venting the cavity and prevent air traps effect from happening.

CONCLUSION

The aim of the thesis was to conduct research in the field of injection molding technology, which was subsequently used for the practical design of a runner system, design of an injection mold, and optimization of a cooling system. The theoretical part contains a characterization of the injection molding technology including the molding cycle, the injection machines, and the mold design description. This overview was very useful for further work in the practical part.

The first step of the practical part was to design, launch, and evaluate six different types of runner systems in two cross-section variants. The circular and trapezoid cross-sections were evaluated according to shear rate and pressure results. The results were dissimilar as a result of their very different content. Six types of runner systems were presented with various results. Results of analyses were important for consideration of the runner systems. In other words, the runner systems were designed and analyzed to select the optimum one. Finally, the four-impression mold with combined runner and circular cross-section of the cold runner was selected and used for the injection mold design.

Even though the results of Moldflow analyses cannot be taken as exact values, they are very useful for predicting the behavior of the polymer inside the runner system and cavity. These results were taken and evaluated as an indicator of tendency. The outcome depends fully on the precision of the input data. This is the only possible limitation of results utilization.

The injection mold was designed for submission plastic part and the assembly of the mold contains 277 parts. The part was provided by the company HELLA KGaA Hueck & Co. and its purpose of use is holding and covering a light-emitting diode inside a headlamp of passenger car. This part has a very complex shape and shaped inserts inside the mold cavities have to be used for its production. The design of injection mold, using the software Catia V5, started with parting plane consideration and cavity and core inserts creation. Subsequently the injection mold could be created with all necessities, such as a hot manifold and nozzles on the injection side, or an ejection system on the ejection (movable) side of the mold. Hasco standards were used in this design. The injection mold is multiple with four impressions of the part, which means that one injection cycle is capable of producing two identical parts and two mirrored parts.

In the next step, two types of cooling systems for the injection mold were created. The first type is makeable by the conventional method of drilling holes. Second type of cooling system is conformal cooling and takes advantage of DMLS (Direct Metal Laser Sintering) technology.

First, exact trajectories of the cooling channels, located around the cavities, were designed and subsequently loaded to Moldflow. In Moldflow were these trajectories meshed and process parameters of cooling phase were set up.

The complete flow, cool, and warp analyses were launched in Moldflow. From their summarization and evaluation is obvious that runner and cooling system were designed well. Startlingly, the conventional cooling system had better results than the conformal system, in view of the fact that the cycle time is shorter with use of the conventional cooling system. One of the most influencing factors is a smaller diameter of the cooling channels, used in the conformal cooling system for the reason of denser channels pattern and more complex trajectories. Furthermore, the second type of temperature treatment is the most expensive for reducing the cycle time. On the other hand, the second type of cooling system has a little better result from the warpage analysis. As could be seen, deflections of the part are a little bit smaller when using the conformal cooling system which could be caused by smaller circuit heat removal efficiency.

The first type of cooling system makeable with conventional methods has a smaller pressure drop and coolant temperature, which helps to maintain balanced temperature distribution. The higher results are heat removal efficiency and Reynolds number, which is better for turbulent flow of coolant. This is a consequence of greater channels diameter, and more effectively designed cooling system. The part has a complex shape and conformal cooling system is not capable to maintain more useful cooling effect than conventional system does, even with such dense trajectories. This is why the first type of cooling system, makeable with conventional methods, is considered to be better for providing temperature treatment for the designed injection mold.

BIBLIOGRAPHY

[1] J. Krebs, Teorie zpracování nekovových materiálů, Skriptum TU Liberec: VŠST, 1991.

[2] D. Rosato. Dominick Rosato, Injection molding handbook - 3rd ed., Norwell, Massachusetts, USA: Klower Academic Publishers, 2000. ISBN 0-7923-8619-1.

[3] T. X.-Jhun. Fan. Rong Zheng, Injection Molding-Integration of Theory and Modeling, Berlin: Springer Heidelberg Dordrecht London New York, 2011. ISBN 978-3-642-21262-8.

[4] Ruthand plastics [online], Available from WWW: <http://www.rutlandplastics.co.uk/>.

[5] F. Johannaber, Injection molding machines, Munich: Hanser Gardner Publications, Inc., 2008. ISBN 978-1-56990-418-3.

[6] W.Michaelli, Training in injection molding-2nd ed., Ohio, USA: Hanser Gardner Publications, Inc., 2001. ISBN 1-56990-302-6.

[7] Arburg für effizientes Spritzgießen [online]. Available from WWW: <http://www.arburg.com/>.

[8] SPI plastics engineering handbook of the Society of the Plastics Industry, Inc, Norwell, Massachusetts, USA: Springer, 1991. 869 p. ISBN 0-412-99181-0.

[9] Příručka – Formy pro zpracování plastů: II.díl – Vstřikování termoplastů, Brno: 1999, 134 p.

[10] M. R.Kamal, Injection molding-technology and fundamentals, Munich: Hanser Publications, 2009. ISBN 978-3-446-41685-7.

[11] F. Johannaber, Injection molding machines-a user´s guide - 4th ed., Munich: Hanser Publications, 2008. ISBN 978-3-446-2581-7.

[12] C. G. Zehev Tadmor, Principles of Polymer Processing - 2nd, Hoboken, New Yersey: John Wiley & Sons, Inc., 2006. ISBN 0-471-38770-3.

[13] Rapra, Practical guide to injection molding, Rapra Technology Limited and Arburg Limited, 2004.ISBN 1-85957-444-0.

[14] Bee Jay Molding [online], Available from WWW: <http://www.beejaymolding.com/>.

[15] Book of papers-Injection molding-Advance technology for optimisation of operational performance, Sawbury: Rapra Technology Ltd., 1994. ISBN 1-85957-023-2.

[16] D.O.Kazmer, Injection mold design engineering, Munich: Hanser Publications, 2007. ISBN 978-3-446-41266-8.

[17] J. P.Beaumont, Runner and gating design handbook - tool for successful injection molding - 2nd ed., Ohio, USA: Hanser Gardner Publications, Inc., 2007. ISBN 978-3-446-40765-7.

[18] R. Wimberger.-Fried, J. Greener, Precision injection molding-process, materials, and applications, Munich: Carl Hanser Verlag, 2006. ISBN 3-446-21670-7.

[19] P. Kennedy, Practical and Scientific Aspects of Injection Molding Simulation, Eindhoven: Technische Universiteit Eindhoven, 2008. ISBN 978-90-386-1275-1.

[20] TCT Magazine [online], Available from WWW: <http://www.tctmagazine.com/>.

[21] H. Rees, Mold Engineerind-2nd ed., Munich: Hanser, 2002. ISBN 3-446-21659-6.

[22] H. GASTROW, Injection molds : 130 proven designs. Edmund Linder and Peter Unger . 3rd edition., Munich : Hanser, 2002. 313 s. ISBN 3446214488..

[23] Society of Plastics Engineers [online], Available from WWW: <http://www.4spe.org/>.

[24] M. Gordon-Jr., Total quality process control for injection molding-2nd ed., Hoboken, New Yersey: John Wiley and Sons, Inc., 2010. ISBN 978-0-470-22963-7.

[25] Autodesk Moldflow Insight [online], 2011 – Available from WWW: <http://www.moldflow.com/>

LIST OF ABBREVIATIONS

Symbol	Meaning	Unit
a	Length of bottom	mm
ABS	Acrylonitrile butadiene styrene	
c	Length of top	mm
d	Diameter - internal	mm
D	Diameter - external	mm
HDPE	High-density polyethylene	
LDPE	Low-density polyethylene	
MVR	Melt Viscosity Ratio	cm^3/10 min
PA 6	Polyamide 6	
PC	Polycarbonate	
PMMA	Polymethyl methacrylate	
POM	Polyoxymethylene	
PP	Polypropylene	
PS	Polystyrene	
S	Content	mm^2
SAN	Styrene acrylonitrile	
S_c	Content of circle	mm^2
S_t	Content of trapezoid	mm^2
v	Height	mm

LIST OF FIGURES

LIST OF TABLES

APPENDICES

APPENDIX P I – 3D Assembly of injection mold

APPENDIX P II – Drawing documentation

APPENDIX P III – DVD disc contains:

- Master Thesis
- Injection mold files: 3D assembly, drawing documentation, Catia and Moldflow files of the part
- Filling analyses of designed runner systems
- Complete analyses of injection mold with two types of cooling system
- Support files

APPENDIX P I: 3D ASSEMBLY OF INJECTION MOLD

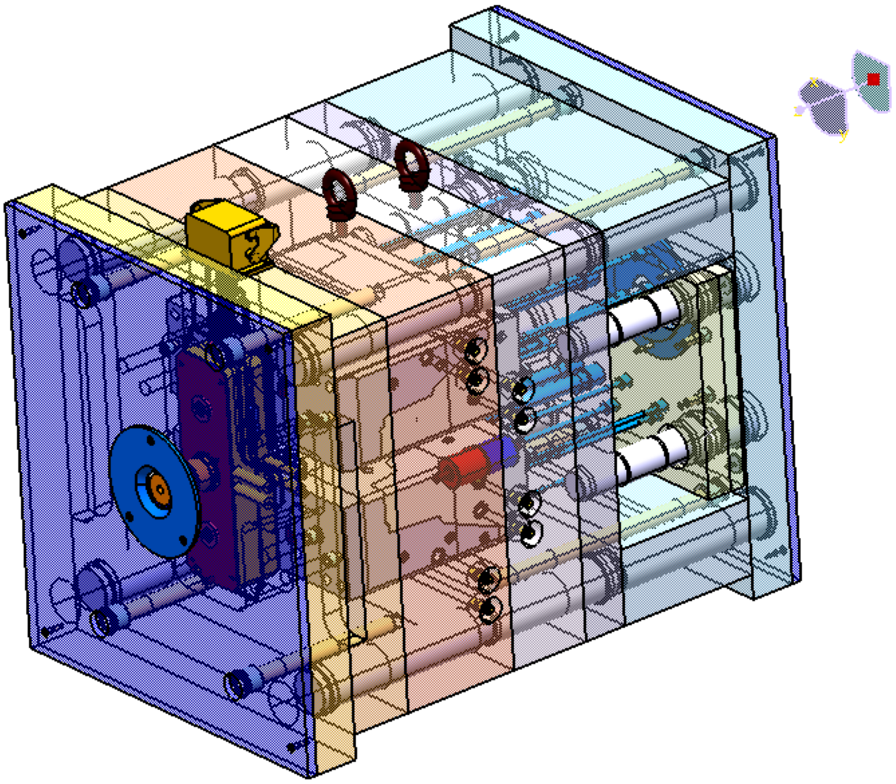


Figure 84 Mold assembly, view from the injection side

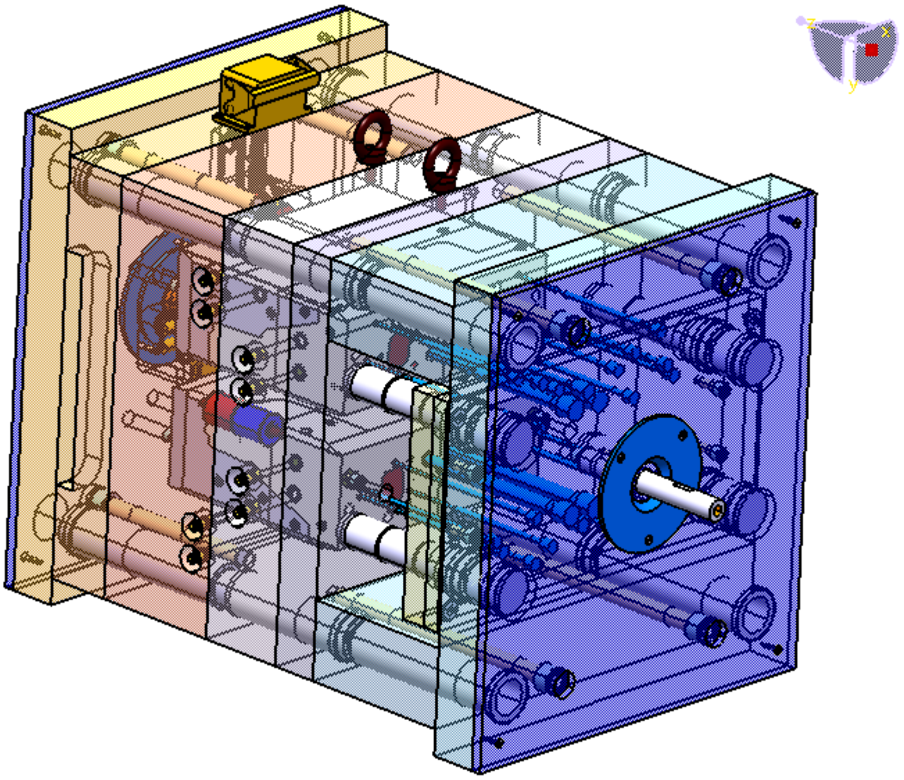


Figure 85 Mold assembly, view from the ejection side

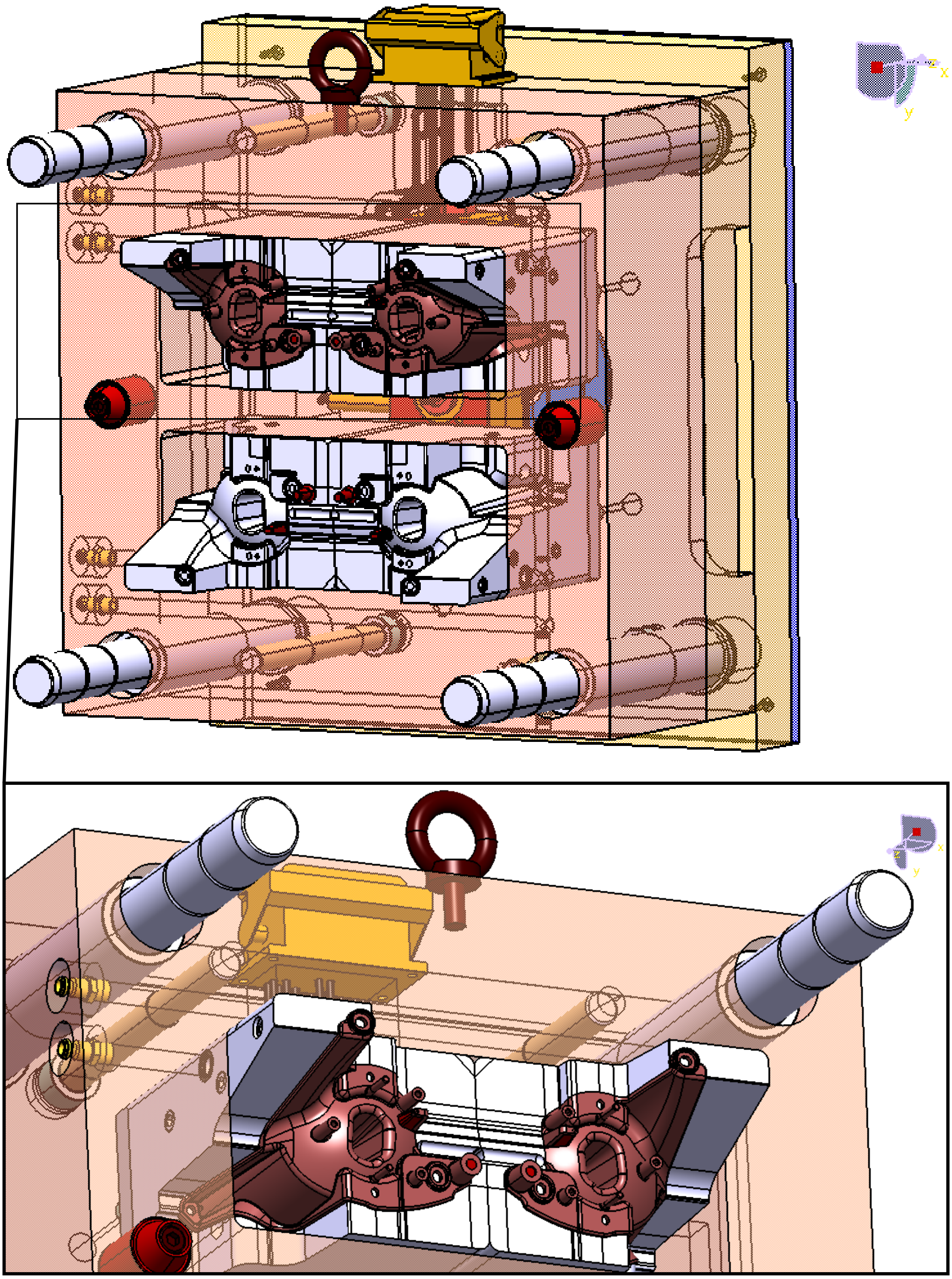


Figure 86 Injection side with mounted plastic parts on upper cavities

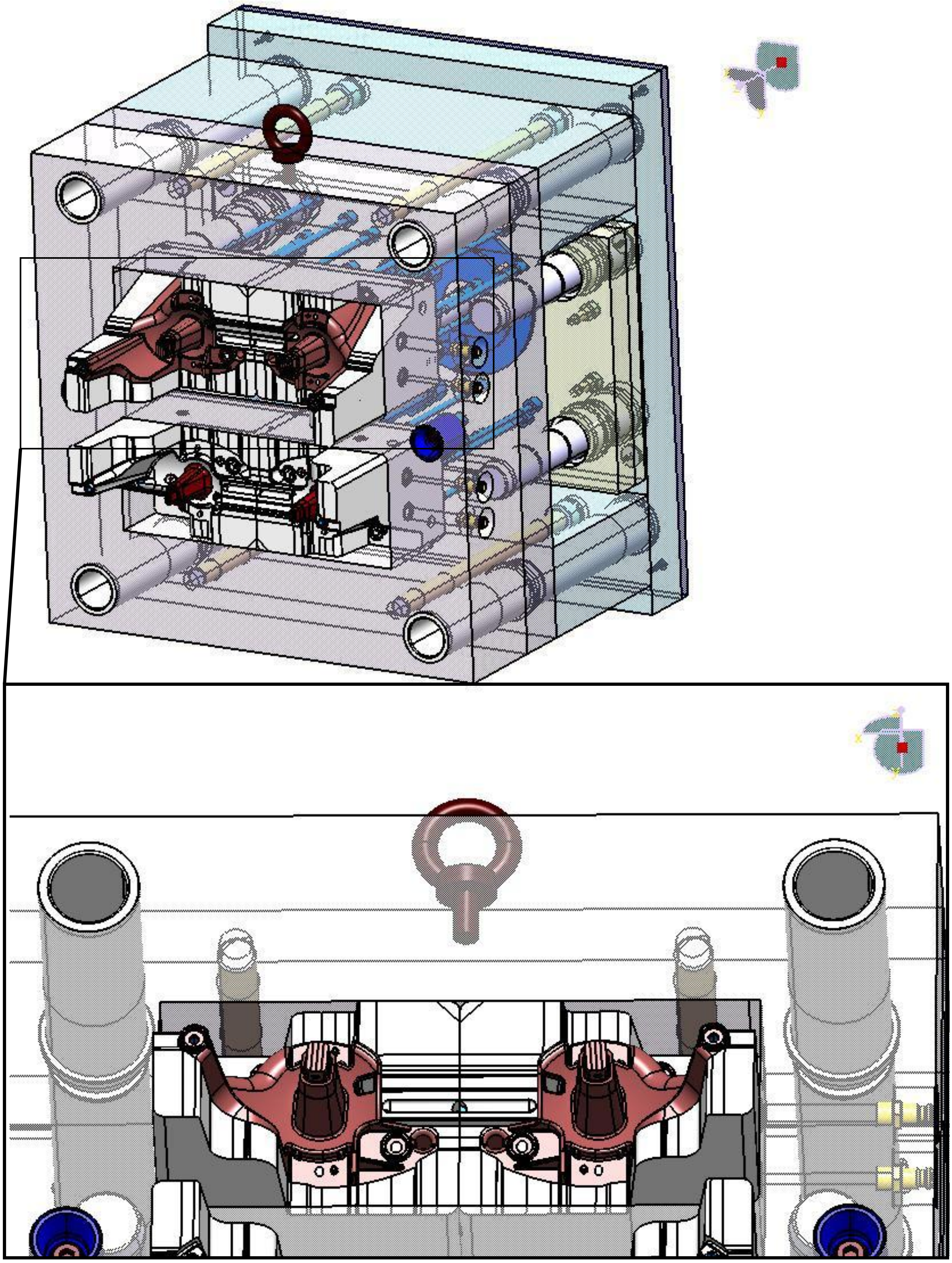


Figure 87 Ejection side with plastic parts mounted on upper cores

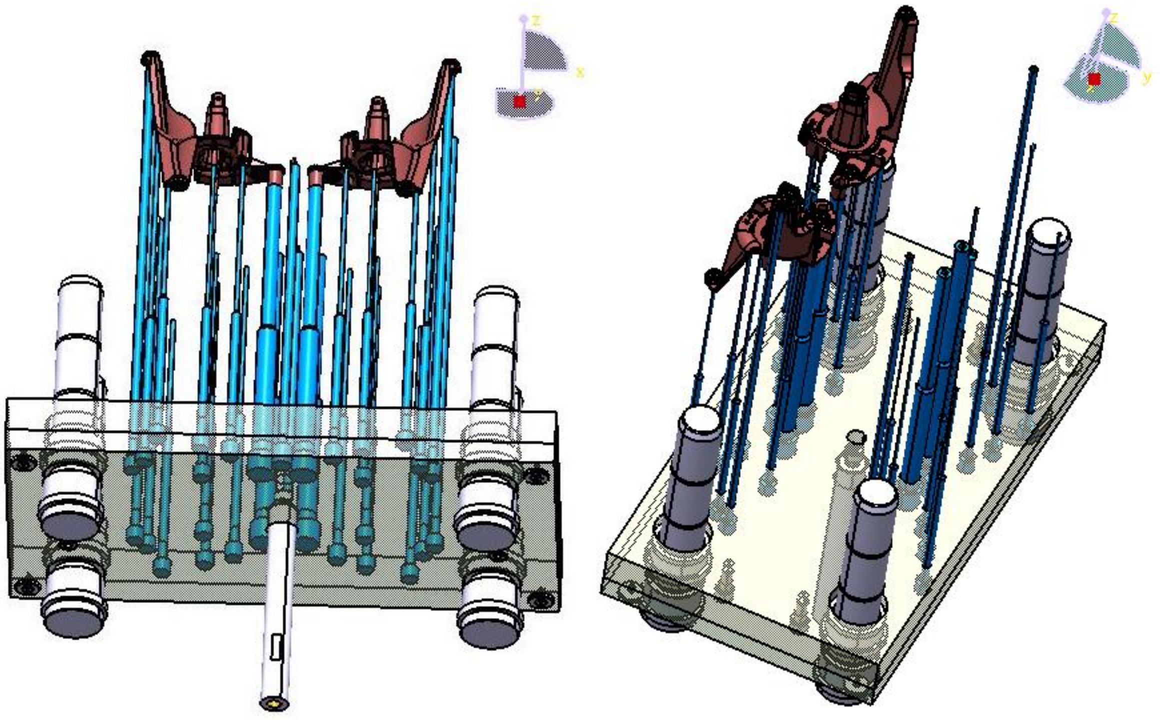


Figure 88 Ejection system with two plastic parts to demonstrate ejectors location