Journal of Modern Processes in Manufacturing and Production, Volume 13, No. 3, Summer 2024

DOI: 10.71762/kkq9-yj18

Research Paper

Prediction of Defects in the Plastic Injection Process by Mold Flow Software Based on the Experimental Data

Ahmad Afsari[1*], Seyed Ahmad Behgozin[2], Mohammad Ramezani[1], Seyed Alireza Hamidi[3]
[1]Department of Mechanical Engineering, Shiraz Branch, Islamic Azad University, Shiraz, Iran
[2]Department of Mechanical Engineering, National University of Skills (NUS), Tehran, Iran
[3]Department of Mechanical Engineering, Mashhad Branch, Islamic Azad University, Mashhad, Iran
[*]Email of the Corresponding Author: Ah.Afsari1338@iau.ac.ir
Received: August 23, 2024; Accepted: October 6, 2024

Abstract

Several factors influence the quality of the final parts of the plastic injection process, as many variables play a role in controlling this process. These factors can include the machine, mold, operator, raw materials, and working environment. An extensive study revealed that molding machines significantly impact quality compared to other factors. Adjusting and optimizing the machine parameters makes it possible to achieve parts with the desired or acceptable quality. The main goal of this project is to develop an application system that selects the regulatory parameter values for machines handling polycarbonate and other polymers. Additionally, the defects will be predicted in injected parts, and their properties will be analyzed using Moldflow software. Another software, based on practical data, will take initial user input and provide the necessary machine parameters to the operator from a reliable information source. During the production stage, if a defect occurs, the software will generate instructions tailored to the defect type and the conditions and parameter values. If the defect persists after following the provided instructions or if the nature of the defect changes, the software will adapt its guidance until defects are resolved, creating perfect parts without any flaws.

Keywords

Plastic Injection Molding, Polycarbonate, Defect Analysis, Moldflow Software, Parameter Instructor Software

1. Introduction

Injection molding, also known as plastic injection molding, is widely used for manufacturing plastic products. It is one of the most essential processes in the plastic industry today. With the advent of new technologies, materials, and market demands, injection molding continues to evolve, delivering higher-quality plastic products for industrial manufacturing and consumers. The application of this process has significantly increased across various industries, especially in automotive parts. In injection molding, plastic pellets serve as the raw material. These pellets are heated until melted and injected into a mold, where they solidify to form the desired shape. Once cooled, the mold is opened, and the finished part is ejected. Numerous studies have aimed to improve and optimize this process, ensuring the production of high-quality parts across a wide range of commercial plastic injection molding machines. Several design parameters influence the quality of the final plastic products [1].

Polycarbonate (PC) and Acrylonitrile Butadiene Styrene (ABS) are thermoplastic polymers suited for different applications. PC is typically injection molded or thermoformed, making it ideal for high-impact situations and where optical transparency is essential. In contrast, ABS is usually injection molded or extruded and is more rigid than polycarbonate. It excels in applications requiring toughness and heat resistance. Choosing between PC and ABS for manufacturing can be challenging. Polycarbonate boasts outstanding mechanical properties, making it strong and durable. Its highly non-crystalline structure allows it to absorb significantly more energy upon impact, offering superior impact resistance compared to semi-crystalline materials.

Additionally, the amorphous nature of the PC results in excellent transparency, making it suitable for applications where light transmission is a key requirement. Moreover, the PC's high glass transition temperature is ideal for elevated-temperature environments. Polycarbonate (PC) can be processed using various metal-forming methods, such as press brake bending, and is also suitable for injection molding, extrusion, Fused Deposition Modeling (FDM), 3D printing, and machining. However, due to its high glass transition temperature, polycarbonate requires elevated temperatures and specialized equipment for extrusion with 3D printers. PC is often used for prototyping because it can be quickly processed at room temperature with sheet metal machining techniques. Typical polycarbonate applications include bulletproof windows, medical devices, safety equipment (like visors, eyewear and screens), electronics and other projects requiring transparency and shatter resistance.

Acrylonitrile Butadiene Styrene (ABS) is a thermoplastic polymer known for its durability, rigidity, and good dimensional stability. ABS is a strong, rigid plastic that provides a high-quality, scratch-resistant surface finish and maintains dimensional stability across a wide temperature range, preventing warping. Its high rigidity and strength make it resistant to deformation under both tensile and compressive loads, and the stiffness of ABS can be enhanced by adding glass fibers. ABS can be dyed with various pigments, making it versatile for different applications. The most common manufacturing techniques for ABS are injection molding and FDM 3D printing. It's particularly well-suited for 3D printing because it can be extruded at relatively low temperatures, eliminating the need for specialized high-temperature equipment. ABS's beneficial properties allow it to be used in various applications. It resists warping across various vehicle temperatures, providing excellent dimensional stability. This phenomenon makes it ideal for automotive parts like dashboards and steering wheels. ABS is also used in applications with essential scratch resistance and visual appeal, such as light switches, office equipment, and children's toys [2]. The studies of Zakir et al. [3] showed that increasing the temperature of the molten polymer increases the yield strength and decreases the ductility. Changing the injection speed and the cooling time while keeping other parameters constant did not affect the polycarbonate's properties and mechanical behavior. Increasing the time and temperature of polycarbonate annealing also increases the yield stress and changes the fracture state from soft to brittle. Research on the effect of injection molding parameters on the mechanical properties of polypropylene has also shown that yield strength increases with increasing mold and molten polymer temperatures. Still, the effect of the melt temperature is much smaller and is more visible at lower temperatures. Increasing the temperature of the mold reduces the cooling rate. It creates enough opportunity for ordering the polypropylene molecular chains and creating more crystal order, thus decreasing the internal stress and increasing the strength [4]. Clamp-holding pressure in moderate values also results in the best mechanical properties. Optimization and numerical and experimental investigation of plastic injection molding process parameters using a multi-criteria decision-making system by another group of researchers [5] have shown that the use of a coherent cooling system instead of a conventional cooling system can significantly reduce shrinkage defects and also reduce the whole production cycle time. At the same time, it has a weak effect on reducing the clamping force. Since today, the industry demands the production of thinner and lighter parts with better mechanical properties, and dynamic temperature control in the injection molding process of plastic parts has been considered.

The rapid cooling of injection molding can cause many defects, such as weld lines, sink marks, warpages, and frozen layers. At low temperatures of the mold body, the viscosity or the resistance to the flow of the melt increases, and as a result, the injection pressure of the melt and the force of the clamp must be increased. To achieve proper properties and prevent defects, it is recommended that the temperature of the mold body be higher than the polymer's glass transition temperature. It was found that by changing the temperature of the mold from 40 to 70 and then 100 and 120 degrees by keeping the melt temperature constant at 230 degrees Celsius, Young's modulus and the degree of crystallization of the samples increased up to the temperature of 100 degrees and at the same time the thickness of the frozen layer decreased to zero at a temperature of 100 degrees. At temperatures above 100 degrees, the positive effects of increasing the mold temperature decrease [6 & 7]. Also, the thermal and mechanical analysis for the coherent cooling system in plastic injection molding with Ansys software and its comparison with the actual values with three different cooling methods showed the accuracy of the simulation and the strong cooling effect of the new methods compared to the conventional methods and the cooling time is reduced by 57% [8].

The effect of mold surface temperature on the properties of the final product in the injection molding of high-density polyethylene materials also showed that increasing the mold temperature increases crystallization, decreases the thickness of crystalline layers, and increases tensile and bending strength and decreases fiber orientation, impact strength and tends to create cavities [9,10]. The effectiveness of energy and mold cooling in the plastic injection process to shorten the cycle time, mold depreciation, cooling time, and environmental effects have also been studied [11]. Reducing the molding and cooling time of the mold is very important. By designing the cooling system and the thermal properties and speed of the cooling fluid, the cooling time can be reduced by 70%. The effect of a 5-degree reduction in the mold temperature is more than double the fluid flow in the same cooling time. Research has been done on developing injection molding simulation algorithms to investigate filler separation or local change of filler distribution (fine and coarse glass particles) for polypropylene, and the effect of different parameters on the distribution of filler in different percentages was obtained. The process was also modeled using Mold Flow software (MFS). [12].

In general, by increasing the speed of melt flow, the effects of separation or accumulation of glass at different points are reduced in all percentages of applied filler and disappear at high injection speeds. Separation is more noticeable at a high initial filler percentage and low flow rates. Increasing the size of filler particles also has the same effect on separation as the amount of filler; it increases the amount of separation and makes it possible to achieve non-separation at higher melt flow speeds. Considering the amount and size of glass filler in resistance to flow inside the melt, its separation can be understood more easily. Samei et al. [13] researched additive manufacturing methods to print Corax stainless steel on 420 stainless steels. They concluded that excellent corrosion resistance and high strength make Corax steel an excellent candidate for making cooling channels in plastic injection molds. During the research conducted by Mianehrow and Abbasian [14] regarding monitoring the process of plastic injection molding with hydraulic injection molding machines, they concluded that among all quantitative parameters in energy consumption, the most important parameters related to the process can be referred to the operational power and total cycle time.

The results of examining the effect of mold temperature on the orientation of particles and the accumulation of particles in ceramic injection molding indicate that the accumulation of particles and their orientation in the molding process with ceramic injection is due to the changes in the shearing rate along the workpiece length. Cooling rates complicate it and depend on material properties and process parameters [15]. Khosravani et al. [16] have researched intelligent systems to improve injection molding to optimize production volume and reduce production cost and time. This intelligent system is recommended using virtual reality additive manufacturing technology and newer production methods.

Otieno et al. [17] introduced a predictive model based on processing parameters for warpage and shrinkage defects, and the predictive capability of triangular and Gaussian functions was investigated. Plastic processing industries can use the results to predict and control such defects for quality products and maximum productivity.

In this article, we look at the core parameters in the injection molding process and the importance of each to identify defects in plastic injection technology. We then provide solutions to fix the defects created and continue these measures to correct the production parts using the intelligent system until achieving a defect-free part. This research aims to compile the codes of an intelligent system for selecting parameter values for polycarbonate parts and also to fix the defects of the injection process with the help of MFS. This research will examine the disadvantages and problems related to the production of polycarbonate parts. While examining the problems, proposed solutions will be presented that will produce a part in the best case in terms of accuracy and precision and with minimal defects. For this purpose, a practical method using plastic injection machines to produce samples and non-destructive tests to check and identify defects is used and compared with the defect predicted by the MFS. Software and programming language (C++) have been used to plan and control production.

2. Materials and methods

In this research, the programming language (C++) was used so that in the first stage, the codes of an expert system were compiled to select the optimal parameter values for polycarbonate parts and then to reduce or eliminate the defects created in the plastic injection parts. This research aims to check the mechanical properties and defects using the MFS and then produce the parts with the initial input specifications. Many defects may occur in the plastic injection process. The operator determines the defect of the injected part, and the software determines the dominant factors in causing the defect. And finally, according to the proposed solutions for changing input parameters by software, the defects are eliminated. Existing defects include sink marks and streaks. These streaks include brown or silver burn-type moisture streaks and color streaks. Other defects include weld lines, eruptions, burn effects, stress cracks or bleaching, and incompletely filled parts. To remove the defects, it may be necessary to change the temperature of the mold wall and the geometry of the gate, correct and optimize air evacuation of the mold, change the temperature and the inner diameter of the nozzle, and finally, change the injection speed, the maximum injection pressure and the temperature of the melt. In this research, the type of polymer material, the thickness of the mold wall, the weight of the workpiece, the weight in one stroke, the diameter of the cylinder, the cross-sectional area, and the density of the material are the input parameters that were entered into the software to produce the part. At the same time, the output parameters include melt and mold temperature, initial injection holding and back pressure, cooling time, clamp force, injection speed, advancing speed, and injection screw movement to adjust the machine. In this regard, the injection machine produced the part according to the written program. A quality control operation was performed on the produced parts, and the possibility of defects in the parts was checked. After specifying the type of defect, the suggested solutions were implemented to eliminate the defect, and again, the parts production process in the mold continued until the final part was produced without defects. Fourteen types of thermoplastic materials were considered to work in the software for the injection molding process, and their melting points are shown in Table 1. In this project, polycarbonate with a melting temperature of 104 degrees Celsius has been selected for testing and evaluation.

The plastic injection machine consists of two main parts: the injection and receiving units. The injection unit injects the plastic material into the mold, and the receiving unit keeps the mold closed during injection. The injection and receiving units are used for separate purposes and complement each other. The heart of the injection unit is a heated cylinder, usually long, round, tubular, and made of steel. The tube inside is usually lined with a complex tool steel bushing to resist the corrosion induced by injection material. Machines are classified primarily by the type of driving systems they use: hydraulic, mechanical, electrical, or hybrid, and can be fastened in either a horizontal or vertical position. Most machines are horizontally oriented (Figure 1). Based on the design, the mold uses a cold or hot runner system to carry the plastic and fillers from the injection unit to the cavities. A machine used in this research is a hot runner system, which is more complicated, often using cartridge heaters to keep the plastic in the runners hot as the part cools. After the part is ejected, the plastic remaining in a hot runner is injected into the next part.

Table 1. Common polymer materials in the plastic injection process and their melting point

S. No.	Material	Melting Temperature F (^{0}C)	S. No.	Material	Melting Temperature F (^{0}C)
1	Acrylic	180 (82)	8	Polypropylene (PP)	120 (49)
2	ABS (medium impact)	180 (82)	9	Polystyrene (PS)	140 (60)
3	Nylon (type 6)	200 (93)	10	hard PVC	140 (60)
4	Nylon (type 6/6)	175 (79)	11	Flexible PVC (PVC)	80 (27)
5	Polycarbonate (PC)	220 (104)	12	Styrene Acrylonitrile (SAN)	100 (38)
6	Low-Density Polyethylene (LDPE)	80 (27)	13	Thermoplastic Polyester (PBT)	180 (82)
7	High-Density Polyethylene (HDPE)	110 (43)	14	Thermoplastic Polyester (PET)	210 (99)

The injection unit of the machine consists of a screw motor drive, reciprocating screw and barrel, and heaters, thermocouple, ring plunger. In contrast, the three main components of the clamping unit are the mold clamping motor drive, tie bars, and hydraulic QMC. The outer surface of the heating cylinder has heaters that are fastened with a metal strap. These thermal bands work electrically and are located in the cylinder. The injection screw is a spiral-shaped rod that is inside the injection cylinder. The primary hopper is located on the upper and right sides. This hopper is a container where the raw materials are stored as grains. The primary function of the injection screw is to rotate and advance the fresh material from the hopper into the thermal cylinder.

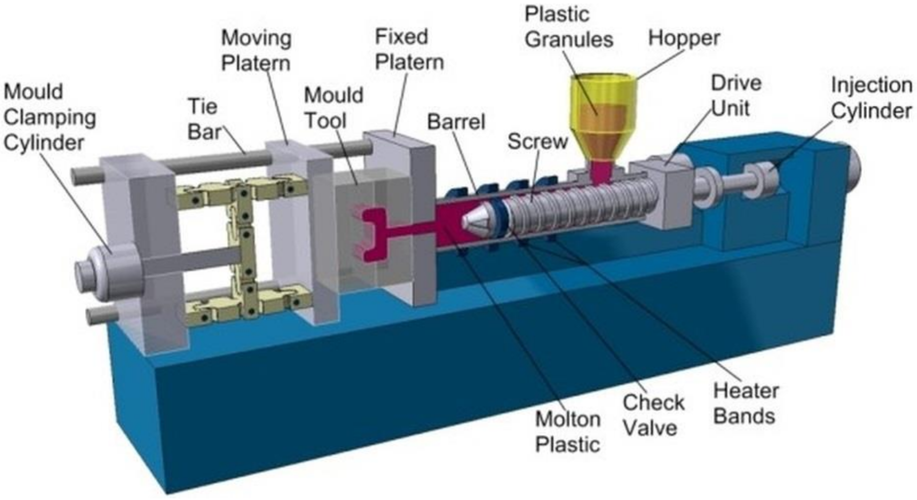

Figure 1. Plastic injection machine and injection cylinder

The second action of the screw is to mix and homogenize the plastic particles. The advance speed and amount of movement of the injection screw are essential factors affecting the properties of produced parts. A one-way ring valve is also responsible for preventing the molten polymer material from returning when the screw is moving forward. The clamping unit of the injection molding machine has the maximum clamping force, which keeps the mold closed during the injection operation. If the clamping force is less than the calculated force during the injection of plastic material into the mold, the mold is not completely shut, and as a result, the parts have burrs or incomplete filling of the parts. The injection mold or pressing unit may be seriously damaged if the applied force is too high. The injection machine used in this research is called a latching injection molding machine, and it was produced and marketed in the FT-320 series by a Taiwanese company. The specifications of the machine are presented in Table 2.

Table 2. Specifications of plastic injection machine FT-320

Injection unit	FT-320			Receiving unit	FT-320
Injection screw diameter (mm)	80	75	70	Clamping force (Tons)	320
Injection pressure (Kg/cm2)	1365	1553	1783	Shock absorber (mm)	600
Theoretical shot volume (Cm^3)	1407	1237	1078	Minimum length and width of the proposed mold (mm)	420×360
Injection weight (gram)	1273	1119	975	Mold thickness (mm)	300 -780
Injection rate (Cm^3/sec)	388	341	297	Minimum drying cycle (Sec)	3.0
Plasticizing capacity (Kg/hr.)	348	306	266	The distance between the bars (mm)	550×650
Injection screw rotation speed (rpm)	126			Mold sheet (mm)	980×890
				Throw out (mm)	190

After the design of the mold, the selection of materials and construction of the mold were done, depending on the type of production material. The choice of mold metal for a specific application depends on the characteristics of the desired piece, such as its manufacturing cost and metal availability. Hard or pre-hardened steel, high-purity aluminum, and copper-beryllium alloy are usually used to make the injection mold. The functional, mechanical, and physical properties of the mold made with the above materials are shown in Tables 3 to 5.

The two main components of each mold are the core and the mold cavity. When the mold is fastened, the space between the mold's core and cavity, where the desired part is formed, is filled with molten polymer. Sometimes, multi-cavity molds are used to form and produce several pieces simultaneously. When the mold is fastened and before the molten material is injected into the mold, there is some air inside the mold cavities. When the material enters the mold, if the air is not released, it will be compressed, and this air will burn the material and the surface of the mold due to its significant heat. As a result, burnt spots remain in the product. To prevent this defect on the part, holes must be created in the mold to allow air to escape. These ventilation holes should be small so as not to allow molten material to release from them and create surface irregularities.

Table 3. Functional properties of injection molding materials

Material of the mold	Alloy element	Capability	Reason for choosing
Hard steel	High Chrome	High coverage, good impact resistance	High surface smoothness and easy separation of parts from the mold surface
Pre-hardened steel	Very low Chrome	Wear resistance, Harder than steel	Low production volume/production of large parts
High-quality aluminum alloy	Zinc / Tin / Magnesium	Heat loss during machining and injection of materials, much less resistance than steel	Suitable for small productions
Copper-beryllium alloys	3 to 5 percent beryllium	High strength / anti-magnetism and spark, High heat transfer rate	For materials that need to transfer heat quickly

Table 4. Mechanical properties of injection molding materials

Material of the mold	Density	Modulus of elasticity	Poisson ratio	Hardness (HRC)	Melting point (ºC)
hard steel	7.85 g/cm³	190-210 GPa	0.27-0.30	50-60	1400-1500
Pre-hardened steel	7.85 g/cm³	190-200Gpa	0.27-0.30	38-45	1400-1500
High-quality aluminum alloy	2.7 g/cm³	70-80 GPa	0.35	160	580
Copper-beryllium alloys	8.94 g/cm³	117 Gpa	0.34	130	1085

Table 5. Physical properties of injection molding materials

Material of the mold	Thermal expansion coefficient	Thermal conductivity coeficient	Electrical conductivity (S)
hard steel	$9\text{-}27 \times 10^{-6}$	11.2-48.3 W/m. K	1786000
Pre-hardened steel	$9\text{-}27 \times 10^{-6}$	11.2-48.3 W/m. K	1176000
High-quality aluminum alloy	23.5×10^{-6}	173 W/m. K	37450000
Copper-beryllium alloys	51×10^{-6}	385 W/m. K	59170000

In the current research, polycarbonate was selected as the material of the production piece. Polycarbonate has characteristics such as high resistance, density value of 1.20-1.22 kg/cm³, and a transparency that can transmit more than 90% of light. However, in this article, due to the use of black granules, the passing of light is excluded. MFS was used for analysis, Also the software was written to fix the defects and determine the set parameters of the injection machine. Light weight is a characteristic of polycarbonate, which has caused many automotive industries to consider replacing glass with polycarbonate in cars. This change also increases performance and efficiency regarding strength and stiffness, energy absorption or impact resistance, and ease of installation. It also leads to lower prices. Polycarbonate is used in safety glasses police anti-riot protective shields, and the lens part of car lights. This can provide reliable protection against UV rays and have good chemical resistance against diluted acids, alcohols, oils, and grease. This material has thermal stability up to 140 ^{0}C and has good sealing properties against pressure. Many variables affect the injection molding process. New studies have identified more than 200 parameters that directly or indirectly affect the process. All parameters affecting the plastic injection process can be divided into four groups: temperature, pressure, time, and distance. Temperature is the most critical parameter for changes in properties, followed by pressure, time, and distance. Still, each of these groups is dependent on the other, and a change in one affects the rest of the parameters. Temperature change is very effective in the injection molding process.

These temperatures are melting, mold wall, cylinder, and even ambient. Plastic material is transferred from the hopper to the cylinder, then the material rotates throughout the cylinder and is transferred to the machine's nozzle. From there, the material is injected into the mold, moves along the channel system (sprue), and enters through the entrance gates into the cavity inside the machining mold. The temperature of the melt must be controlled throughout the process. Most of the shrinkage occurs in the first hours after molding. Therefore, inspection after the part is ejected from the mold is very important, but careful inspection should only be done after the part has cooled for two or three hours or more. Two zones of the injection and the receiving unit require pressure and control. The injection unit creates three main types of pressure: initial pressure, maintenance pressure, and back pressure.

The purpose of the receiver unit is to keep the mold fastened against the injection pressure. Therefore, the hydraulic cylinder's clamping force must be equal to the injection force. The biggest drawback of the hydraulic system is that if the required tonnage is nearly the working limit of the machine, the higher injection pressure may overcome the clamping force and open the mold and create a burr. The total cycle time determines the amount of time required for different activities, called the cycle time. In Table 6, the times of a complete cycle are specified. According to the obtained numbers, it takes about 20 minutes to inject a part, of which 10 minutes are related to mold preparation (rows 1 and 2).

Table 6. Average time for a complete cycle

S.No.	Time (seconds)	operation done	S.No.	Time (seconds)	operation done
1	300	The time of installing the bed of inserts on the fixture	8	300	Time of deburring the piece
2	300	Time to place the fixtures (34 nuts)	9	5	Time to open the mold parts
3	5	Time to close the machine door	10	10	Time workpiece out of the mold
4	5	Time to close the mold parts	11	5	Ejection time
5	5	Initial injection time	12	10	Turn back time
6	5	Pressure holding time	13	120	Mold inspection and cleaning time
7	2	Cooling time			

Suggested parameters to change to create a defect-free part, such as applying changes in temperature (temperature of melt, mold wall, part ejection, and cylinder), speed (injection and screw speed), inlet geometry and its diameter and other dimensions, maintenance pressure, pressure holding time, full cycle time, clamping force, injection movement, screw speed, external tension, back pressure and recommendations on using pigments with different colors and sizes, using stabilizers, color paste and changing the number of recycled materials are used in the production of parts, cleaning the air outlet channels and even changing the capacity of the machine. An example of a burr defect caused by increasing the clamping force was eliminated. The final output parameters for polycarbonate are shown in Table 8.

Table 7. Adjusting molding parameters and their effects

S.No.	parameters	Effect on properties
1	injection pressure (+)	Less shrinkage, more transparency, less distortion, easier injection
2	injection pressure (-)	More shrinkage, less transparency, more distortion, difficult injection
3	back pressure (+)	More density, more grade change, less air bubbles
4	back pressure (-)	Less density, less grade change, more air bubbles
5	melt temperature (+)	Faster flow, more grade change, more brittle, more deburring
6	melt temperature (-)	Slower flow, less grade change, less brittle, less deburring
7	mold temperature (+)	Longer cycle time, more transparency, less distortion, less shrinkage
8	mold temperature (-)	Shorter cycle time, less transparency, more distortion, more shrinkage

Table 8. Input Parameters of Injection Process

S. No.	Process	Data	S. No.	Process	Data
1	Maximum wall thickness in mm	30	4	The diameter of the cylinder in mm	60-80
2	Workpiece weight in grams	1100			
3	Shot weight in grams	1200	5	Area in mm^2	60-80

Table 9. Output parameters for polycarbonate

S. No.	Process	Data	S. No.	Process	Data
1	Melting temperature (^{0}C)	288	7	holding pressure (bar)	50-60 (% of Injection pressure)
2	Initial mold temperature (^{0}C)	104			
3	Initial injection pressure (bar)	1200-1800	8	back pressure (psi)	50-500
4	Density of matter (g/cm^3)	1.10	9	screw advance speed in glass (soft) state (m/s)	0.3
5	Cooling time (seconds)	25	10	screw advance speed in hard mode (m/s)	0.08-0.1
6	Clamping force in (tons)	1.94	11	screw course (mm)	49.23

3. Results and discussion

It should be noted that only 10% and 30% of glass fiber are mentioned in the standard production of polycarbonate granules. Therefore, these two ratios have been used when injecting polycarbonate polymer into the mold. Using MFS and specific input parameters, the output information of the software was extracted, and a comparison was made between polycarbonate with 10% and 30% glass with one material input. In addition to input data, physical properties such as specific volume, specific heat capacity, and thermal conductivity coefficient of two polymers with 10 and 30 percent glass fiber are also required to perform calculations in MFS.

3.1. Comparison of polycarbonate with 10 and 30 percent glass fiber

Specific volume changes with temperature in polycarbonate with 30% and 10% glass are shown in Figure 2. Specific volume or volume per unit weight increases with temperature and decreases with increasing pressure. Adding more glass fiber, similar to increasing pressure, causes a more significant drop in a specific volume at any temperature because the glass is more compact than polycarbonate. This compaction, if accompanied by strong connections between the glass fiber and the polymer matrix, can improve the mechanical and tribological properties of the final part. Changes in specific heat capacity and thermal conductivity with temperature in polycarbonate with 30% glass are shown in Figure 3.

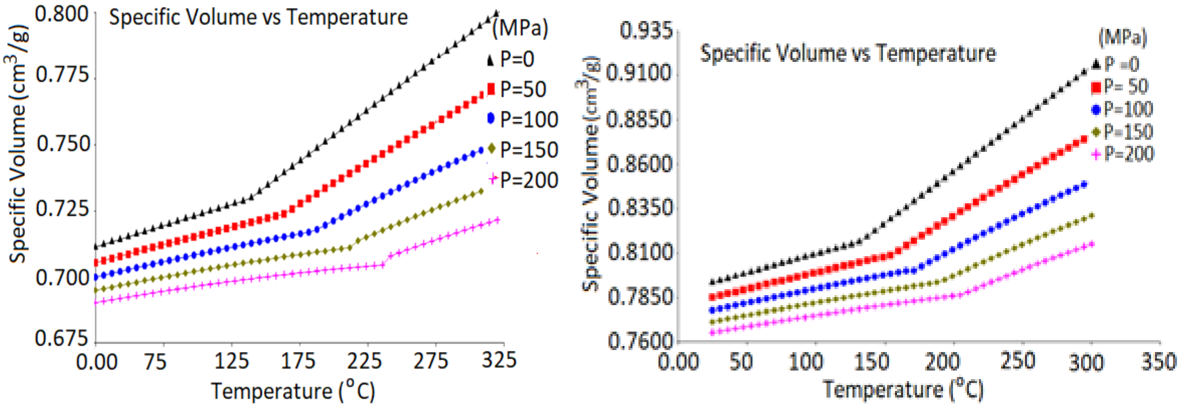

Figure 2. Specific volume changes with temperature in polycarbonate with 30% glass (right) and 10% glass (left)

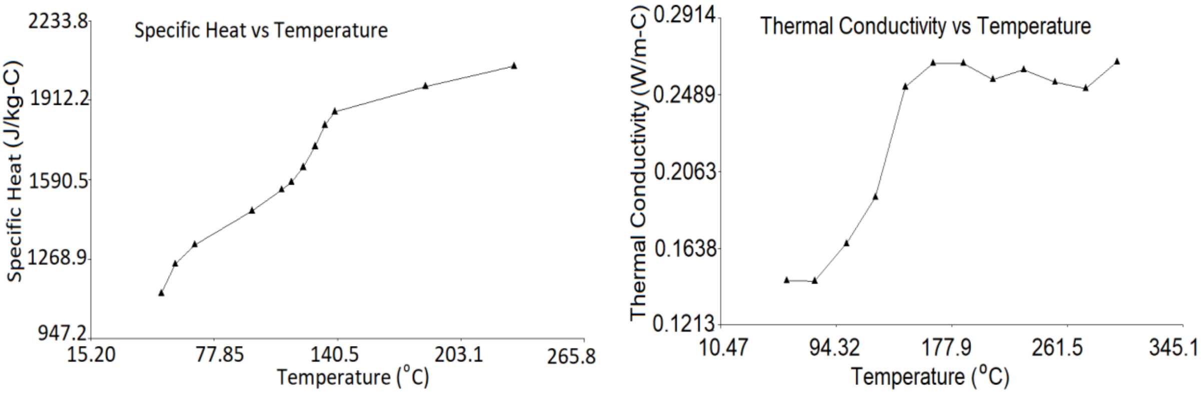

Figure 3. Variation of specific heat capacity and thermal conductivity with temperature in polycarbonate with 30% glass

The specific heat capacity or the heat required to increase the temperature of a unit weight of the object by one degree also increases with the temperature. In most common calculations, this parameter is assumed to be constant for temperature. The specific heat will increase as the glass fiber increases, but not many changes will occur. Then, by replacing glass instead of polycarbonate, due to the larger specific heat capacity of glass, more total heat will be required to increase the temperature of the glass and the polymer in the composite material. In addition, the latent heat should be given to polycarbonate to melt, but the glass fibers will remain solid. The coefficient of thermal conductivity indicates the heat transfer capability of the material. Combining this coefficient with the specific heat capacity determines the amount of heat required for the process. In general, the coefficient of thermal conductivity is not constant and increases with temperature. Adding glass increases the thermal conductivity of the composition at any temperature. Changes in thermal conductivity coefficient with temperature in polycarbonate with 10% glass are shown in Figure 4.

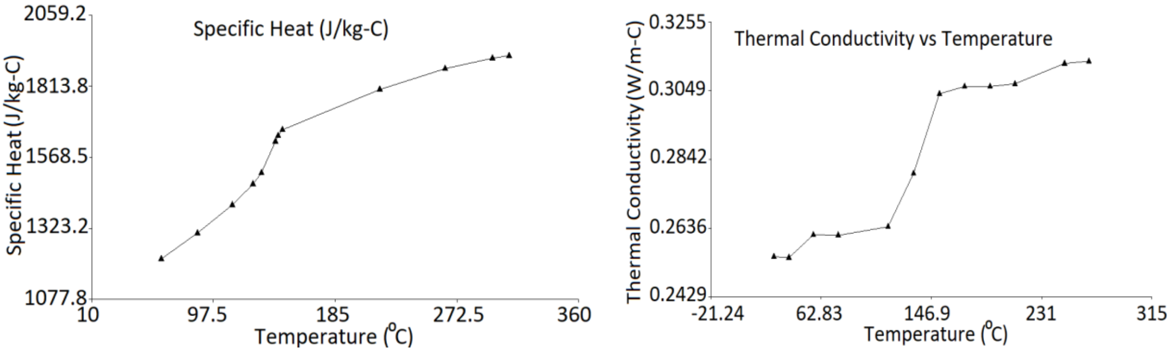

Figure 4. Variation of specific heat capacity and thermal conductivity with temperature in polycarbonate with 10% glass

Figure 5 shows the logarithmic diagram of the shear rate changes with viscosity at any temperature. In general, with an increase in viscosity or fluid resistance to flow, the shear rate of the fluid decreases, but with an increase in the shear rate, the viscosity or resistance to fluidity decreases at any temperature. An increase in temperature causes a decrease in viscosity and an increase in fluidity. The increase in glass fiber causes the viscosity to increase at any shear speed, so glass fibers interfere with the free and easy movement of the polymer fluid and increase the tension necessary for the movement of the fluid.

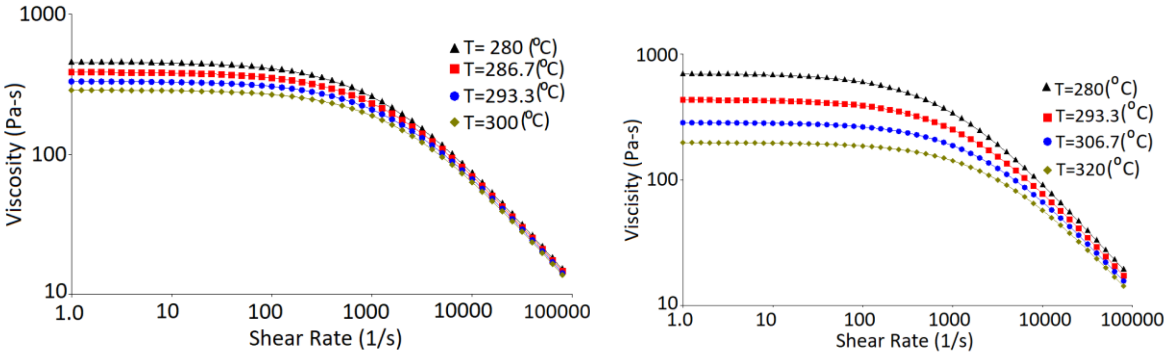

Figure 5. Viscosity changes with the shear rate in polycarbonate with 30% (right) and 10% glass (left)

At high shear rates, the effect of changes in glass fiber percentage and temperature on fluid movement and viscosity becomes weaker. Finally, due to its high strength and better resistance, the combination of polycarbonate with 30% glass was used for the final production of the part and analysis of the MFS. After selecting the glass fiber percentage, the number of injection channels should be selected. For this purpose, according to the geometrical shape of the samples, the 1 and 3 melt inlet channels were chosen to check the effect on the properties and defects. The farthest point to the left and low parts of the workpiece, relative to the injection point shown in red, need the maximum time to be filled in the mold in both 1 and 3 channels (Figure 6). In the samples that all have 30% glass fiber, choosing three channels reduced the time to fill the mold from 10.75 to 1.42 seconds, which means that the time has decreased by about 7 times, while the number of channels increases 3 times equally. Another reason for the rapid filling of the mold is the existence of higher pressures at the time of speed/pressure conversion, and materials enter at a higher speed (Figure 7).

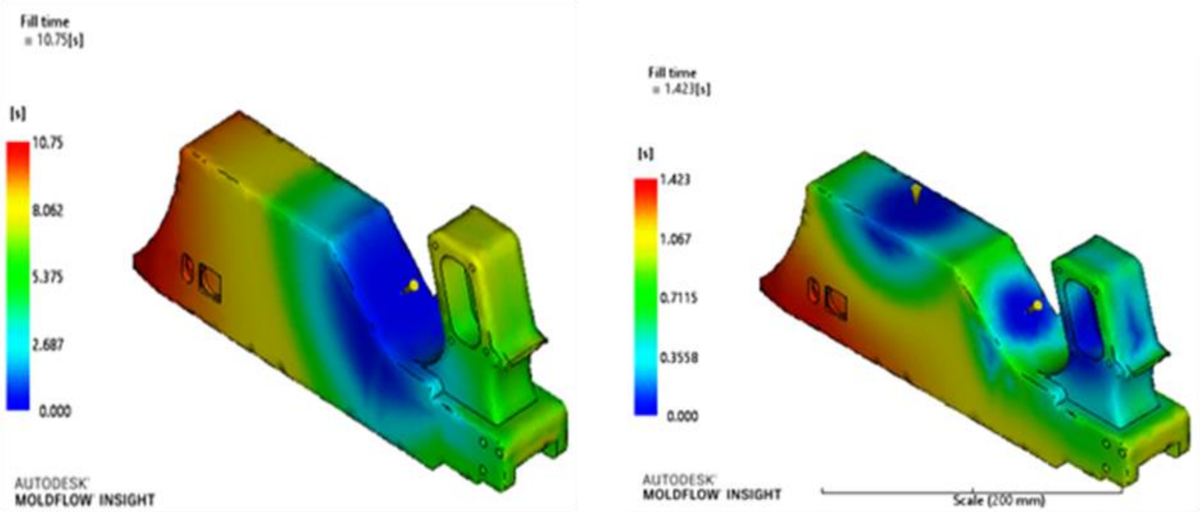

Figure 6. Filling time of different parts of the mold in polycarbonate with (Left) 1 and (Right) 3 injection gates

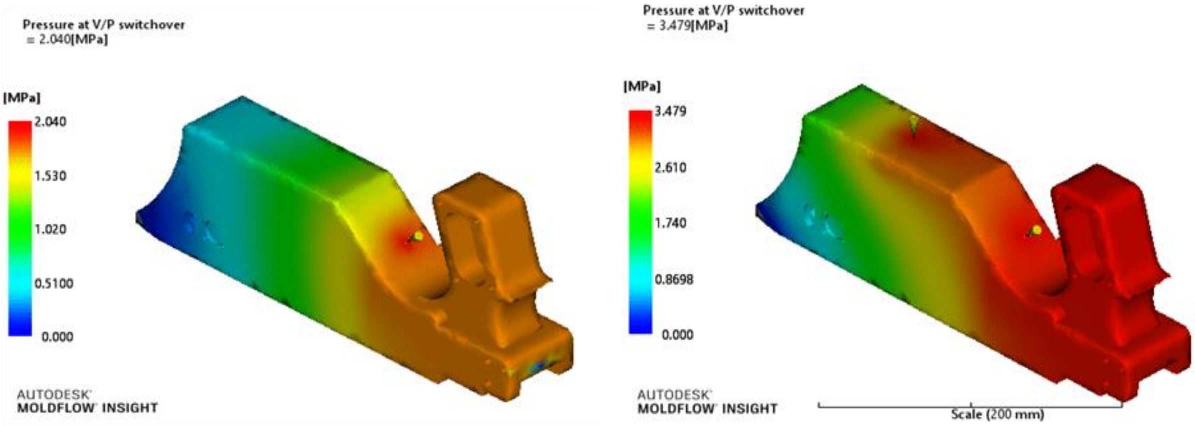

Figure 7. Pressure changes in polycarbonate with (Left) 1 and (Right) 3 injection gates

The increase in pressure causes faster flow of thermoplastic materials and reduces the mold filling time along with the presence of 3 channel gates. The parts on the right side require higher pressures to fill with melt due to their more complex shape. The maximum melt temperature suggested by the software in each case was 288 degrees in the beginning, ,and finally, modified to 300 degrees. At the moment of removing the pressure from the mold, the temperature profiles of the points in the case of 1 and 3 gates are entirely similar. However, this temperature profile was obtained faster in the three gates mode, which is 31.40 seconds, while in 1 gate, this time is 40.6 seconds. The difference between these two times is the same as the differences in mold filling time. Therefore, it seems that after the mold is filled, the temperature behavior, for example, the cooling speed, is entirely similar with 1 and 3 gates. Therefore, cooling two pieces after filling the mold until the injection pressure is removed is the same.

The maximum speed of material movement in the three gates mode has increased by more than 4.5 times due to the increase in pressure. The maximum speed of 672 cm/s equals about 24 km/h (Figure 8). In both cases, the coldest points reached a temperature of 110 degrees, which is the temperature of the mold (Figure 9).

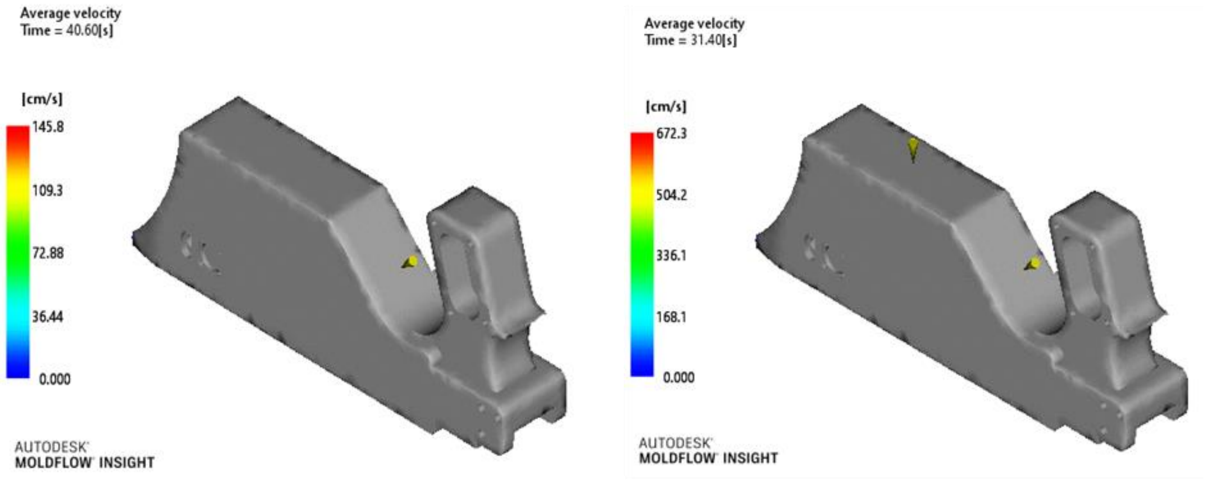

Figure 8. Average velocity of material and profile of speed variation in polycarbonate with (Left) 1 and (Right) 3 injection gates

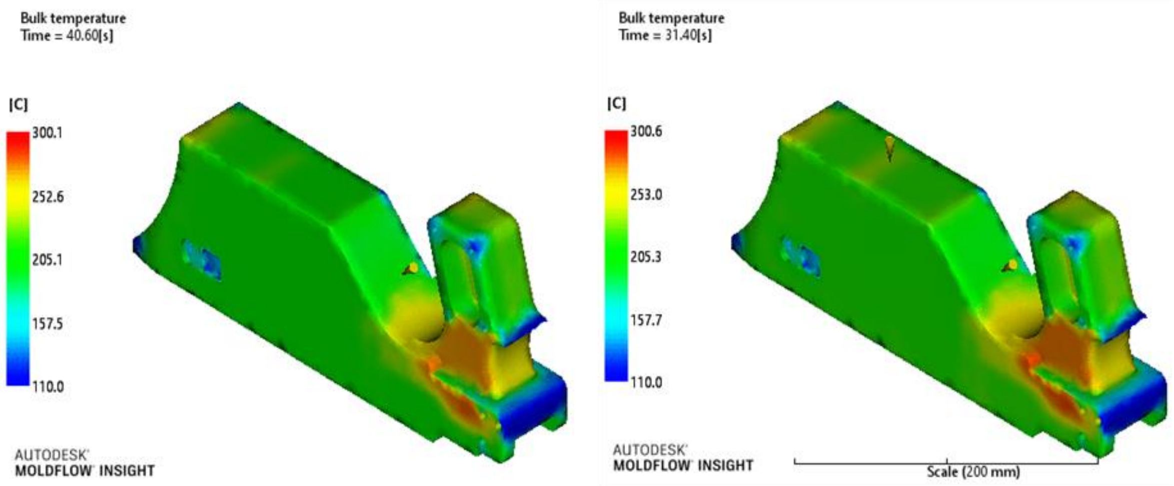

Figure 9. Times for each point of the part to reach the bulk or mold temperature of different gates for removing the injection pressure with (Left) 1 and (Right) 3 injection gates

The initial recommended mold temperature was 104 degrees, which was increased to 110 degrees after modification to create a flawless part. The part exit time is obtained from the injection pressure removal time plus 10 minutes (Figure 10). In both cases, due to the shape of the part and the design of the mold cooling system, the throat is the area with the highest temperature and the lowest cooling rate. When the injection pressure has been removed from the mold, the shear stress reaches zero at all points of the part, but the diagram shows higher stress changes in the three gates condition (Fig. 11).

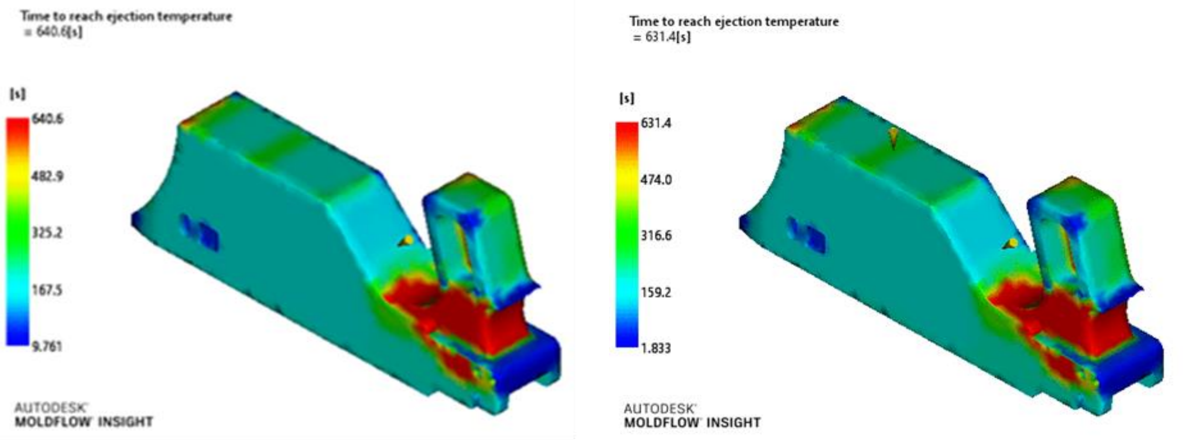

Figure 10. The time to reach the exit temperature in polycarbonate with (Left) 1 and (Right) 3 injection gates

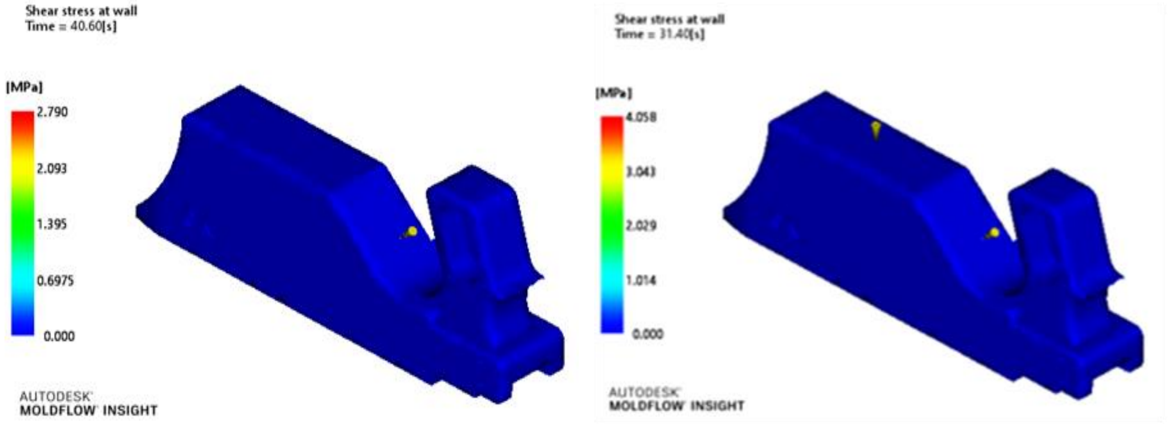

Figure 11. Shear stress changes during the injection process in polycarbonate with (Left) 1 and (Right) 3 injection gates

This shear stress corresponds more to the higher pressure in the case of 3 injection gates (Figure 11). The average shear modulus at any point also has the same maximum in both samples (Figure 12). The back points of the part, which have a more complex shape and are filled at a higher injection pressure, have a larger shear modulus than other points, which means that the fluid's tendency to shear is weaker in them, and they need more stress for shearing. The shear rate until the mold is filled is very high in the sample with three injection gates (Figure 13) due to the high pressure and very short time of filling the mold, and it is more than 2 times the sample with 1 gate.

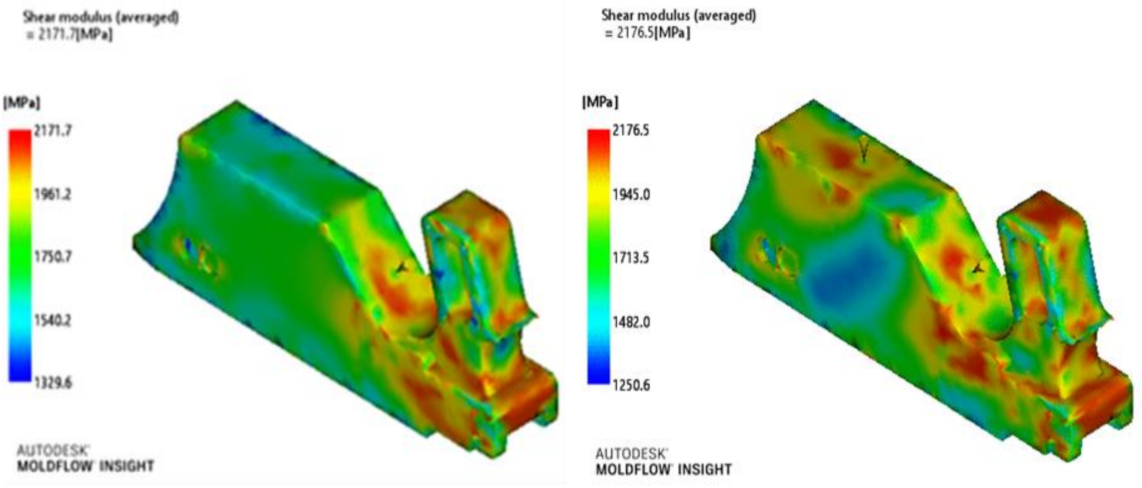

Figure 12. Average shear modulus changes in polycarbonate with (Left) 1 and (Right) 3 injection gates

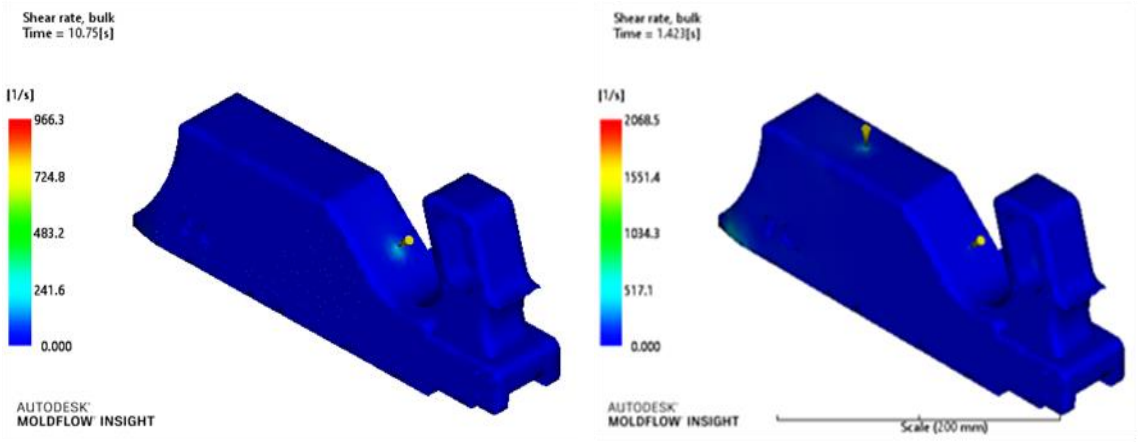

Figure 13. Changes in shear rate during mold filling in polycarbonate with (Left) 1 and (Right) 3 injection gates

The average tensile modulus behavior is similar to the average shear modulus in the three gates in the main primary direction (Figure 14). To create elastic strain in the state with three gates, a more considerable stress is needed. However, in both cases, the differences are insignificant, so it can be assumed that the change of the gates did not affect the shear and tensile modulus. At the end of the mold filling time, temperature changes can be seen in parts with 1 and 3 gates (Figure 15). The maximum temperature is the same 300 degrees set during the mold's very low filling time in the state of 3 gates, and there is no temperature drop yet. But in the case of 1 gate, it is possible to cool some parts due to the longer filling time partially. The volumetric shrinkage at the time of the part's exit depends on the temperature, and the throat part with the highest temperature has the largest volumetric shrinkage at about 9.5%. The increase in the number of injection gates due to the lack of temperature changes has not caused noticeable changes in volume shrinkage (Figure 16). Therefore, as the parts cool down more, the shrinkage will decrease. The volume shrinkage when the injection pressure is removed also has a behavior similar to when the part is removed (Figure 17), but the rate of volume shrinkage is higher. As time passes and the part gets colder, some of the shrinkage created in the part is compensated by the new incoming melt while filling the mold until the injection pressure is removed.

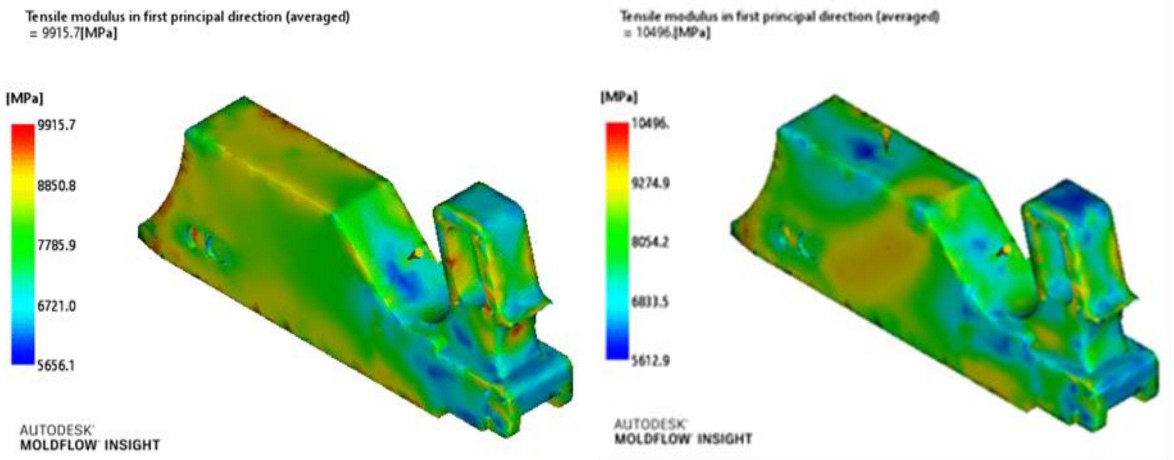

Figure 14. Average tensile modulus changes in polycarbonate with (Left) 1 and (Right) 3 injection gates

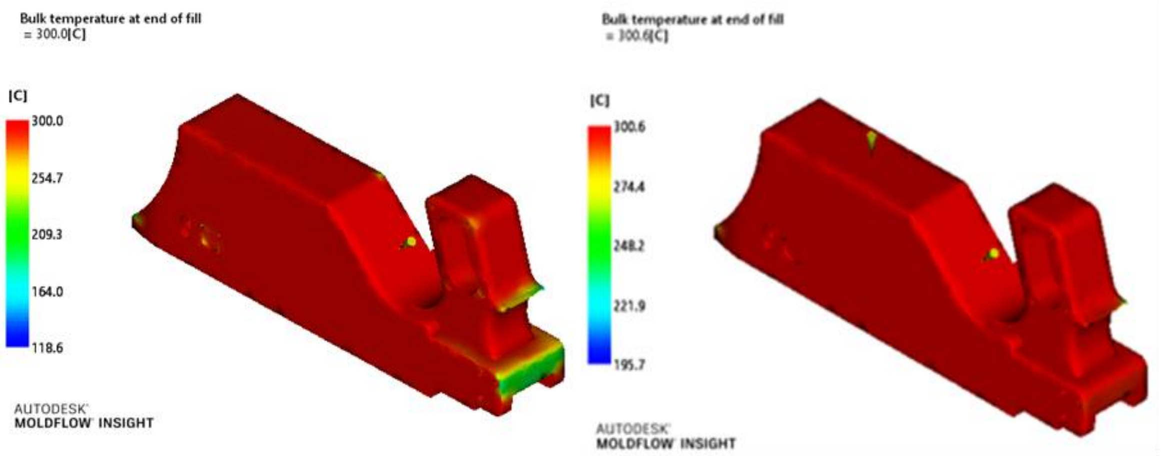

Figure 15. Temperature changes at the end of filling in polycarbonate with (Left) 1 and (Right) 3 injection gates

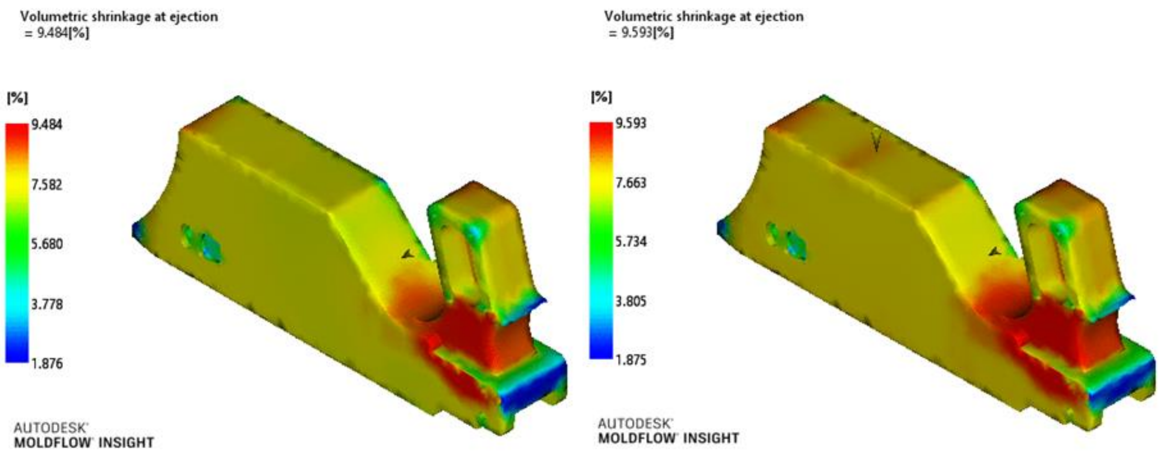

Figure 16. Volumetric shrinkage changes when the part is ejected from the mold with (Left) 1 and (Right) 3 injection gates

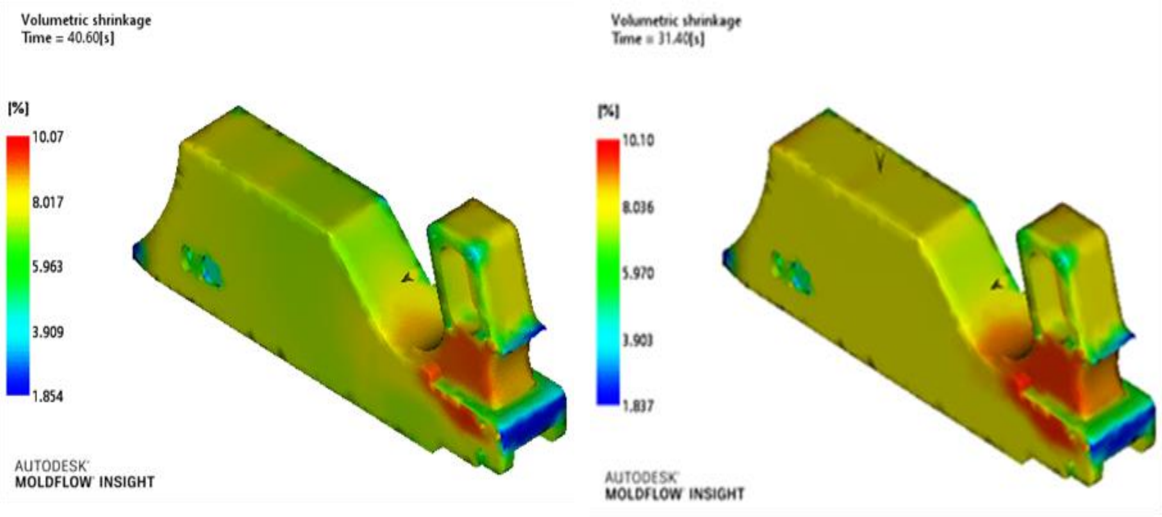

Figure 17. Volumetric shrinkage changes after injection pressure removal with (Left) 1 and (Right) 3 injection gates

Figure 18 shows the filling of the mold in the mode of 1 and 3 gates with color separation. The differences in filling times are minimal. The frozen layer defect at the end of filling time is much less in percentage in the sample with three gates (Figure 19). This defect occurred during filling due to non-wetting and incomplete contact between the melt and the mold wall. Higher flow speed and higher average temperature during filling in 3-gates mode probably eliminated this defect. The defect percentage of the frozen layer during injection pressure removal shows a similar pattern in the two parts (Figure 20). After filling the mold, due to the similar temperature profile and cooling rate, there will be no changes in the frozen layer in the sample with 1 and 3 gates. Therefore, it can be assumed that increasing the number of channels did not affect the final frozen layer. The most frozen was seen at the edges and points with the lowest volume shrinkage and the highest cooling speed. Rapid cooling and premature shrinkage, as a result, can cause the melt to separate and encourage the formation of this defect.

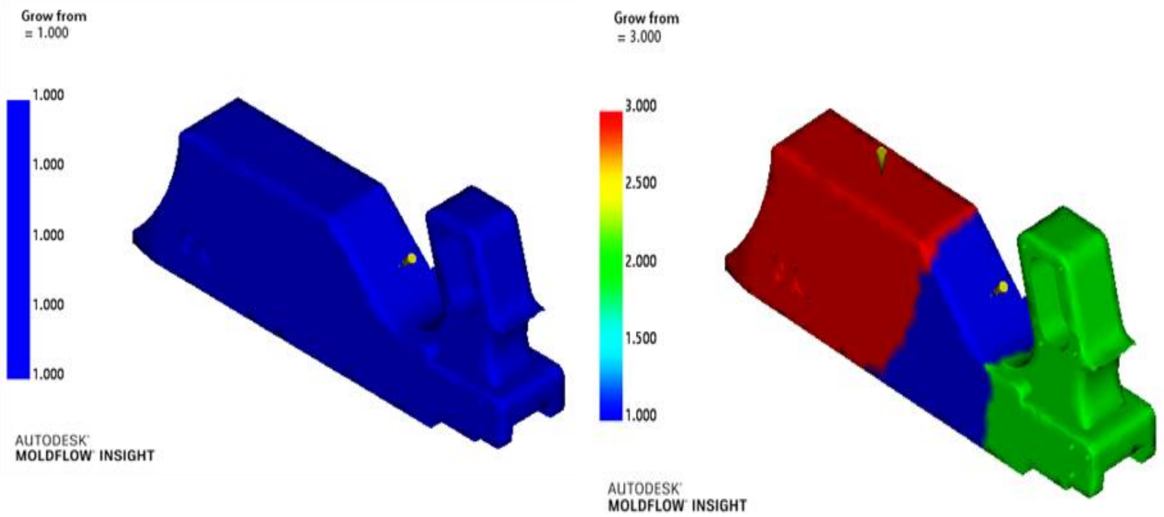

Figure 18. Mold cavity fill orientation in polycarbonate with (Left) 1 and (Right) 3 injection gates

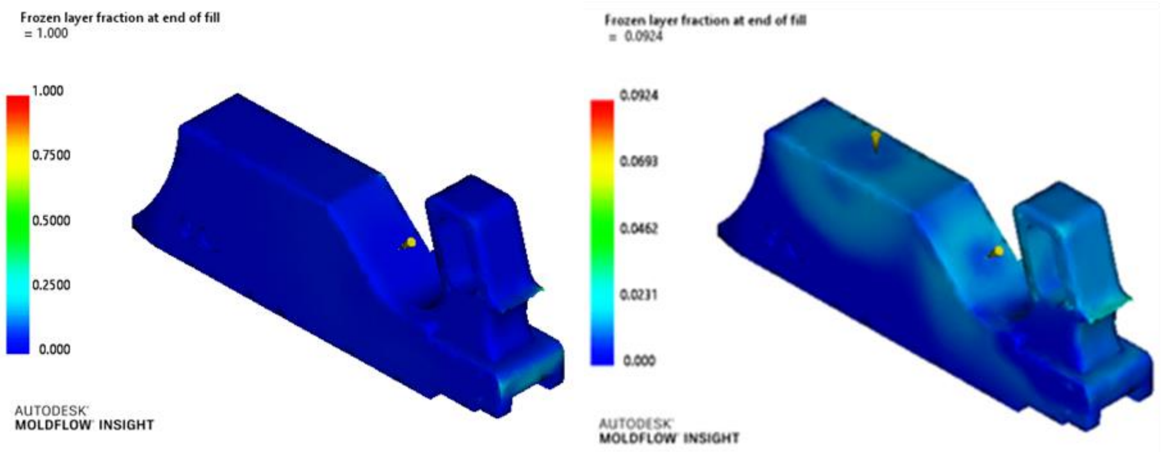

Figure 19. Changes in the percentage of frozen layer defect at the end of fill time in polycarbonate with (Left) 1 and (Right) 3 injection gates

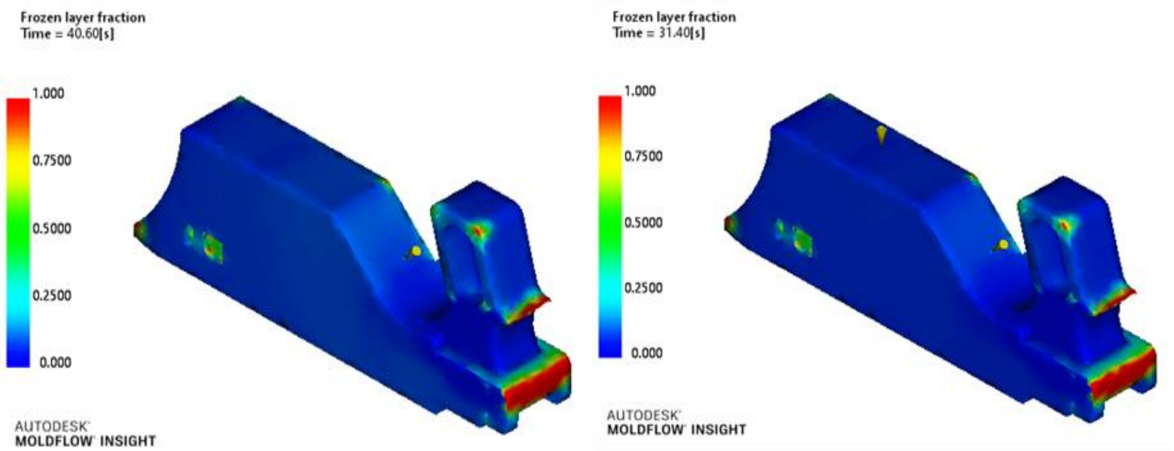

Figure 20. Variation of frozen layer defect after pressure removal in polycarbonate with (Left) 1 and (Right) 3 injection gates

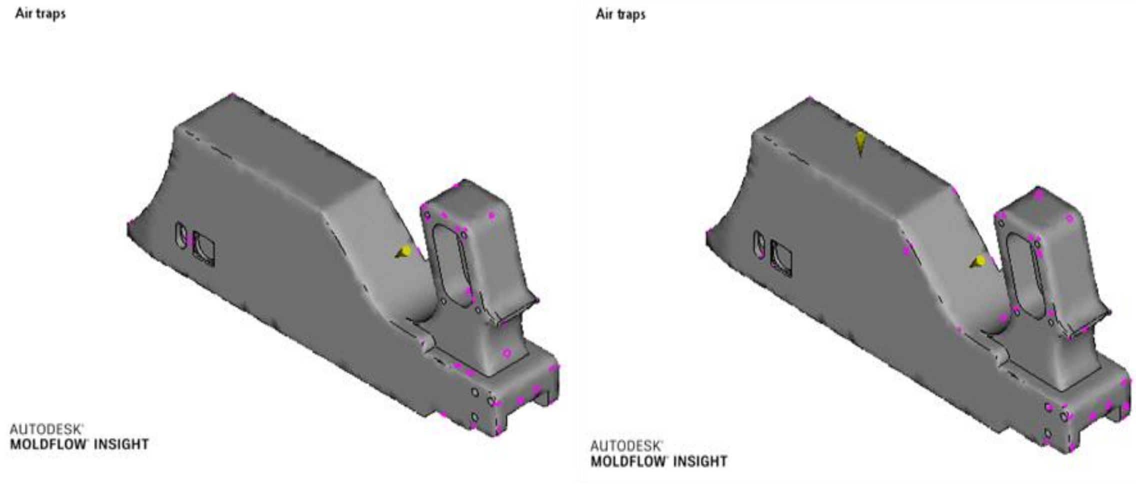

Figure 21. Possible air trap defect to create in polycarbonate with (Left) 1 and (Right) 3 injection gates

The defect of air traps or blow holes (Figure 21) can often occur in places with complex designs, and there is no remarkable difference between the two pieces with 1 and 3 injection gates. Residual stress defect (Figure 22) is similar to the frozen layer in points with lower volume shrinkage and higher cooling rate during cooling. Therefore, a faster cooling rate and less time are the origin of thermal residual stress. Residual stress is also not significantly different in the two samples.

The sink mark defect (Figure 23) can be caused by the absence of the feeding melt during the shrinkage of the piece. This occurs in the throat section of the part where the most remarkable shrinkage occurs, and this is due to the lack of proper feeding of the melt owing to the complex design of this part, as well as the high cooling rate and, accordingly, the low time available for feeding, which is predicted by the simulation program

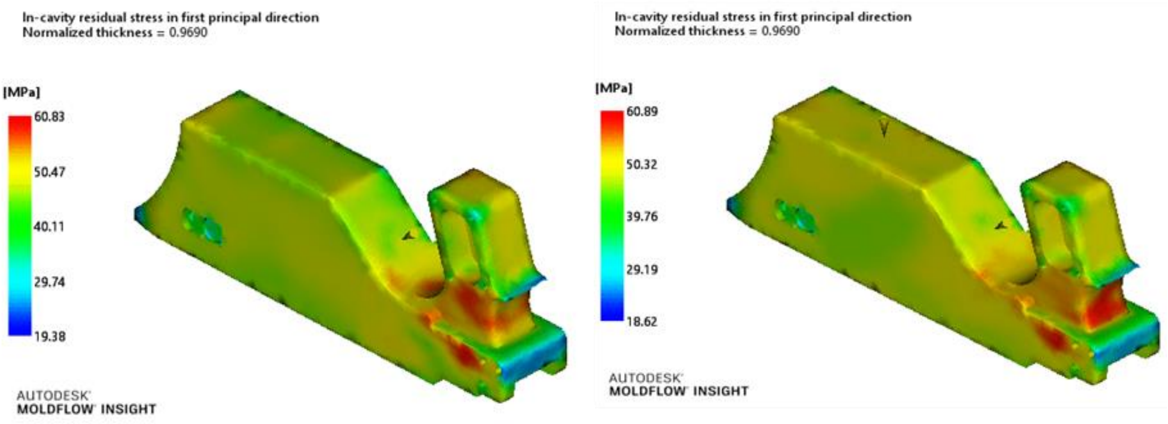

Figure 22. Residual stress defect changes in polycarbonate with (Left) 1 and (Right) 3 injection gates

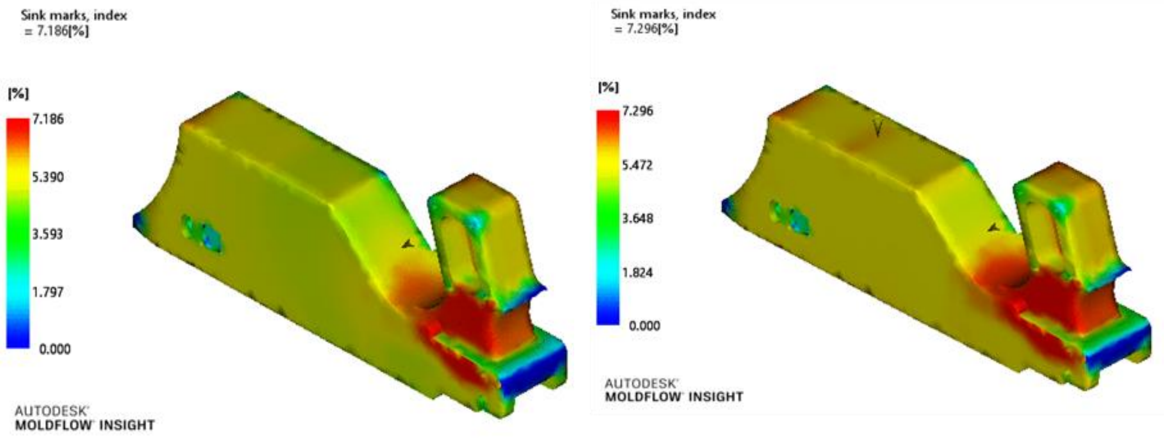

Figure 23. Variations of sink mark defect index in polycarbonate with (Left) 1 and (Right) 3 injection gates

The defect of the fusing line and possible points prone to its occurrence are also shown in Figure 24. The fusing line is usually created at the meeting place of incoming molten streams with different moving directions. It usually occurs at the edges or the meeting place of the surfaces, near it, or in the middle of the faces. Melt flows in different directions in cases with more than 1 gate, making this defect more likely. MFS can predict the settlement of the fibers on the surface or in the core of the part or in the whole part in 3D at any particular time. Figure 25 shows the location of the fibers when the injection pressure is removed. The fibers in the back part are more complex, and the sample with an injection gate has a more regular arrangement. The reason for being more irregular in the case of 3 injection gates is the creation of different paths for the melt to move.

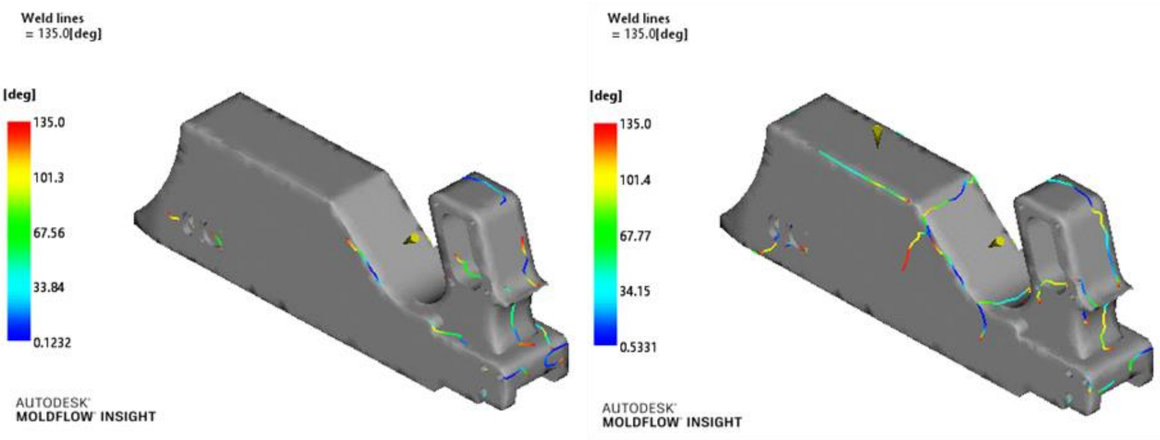

Figure 24. Welding line defects in polycarbonate with (Left) 1 and (Right) 3 injection gates

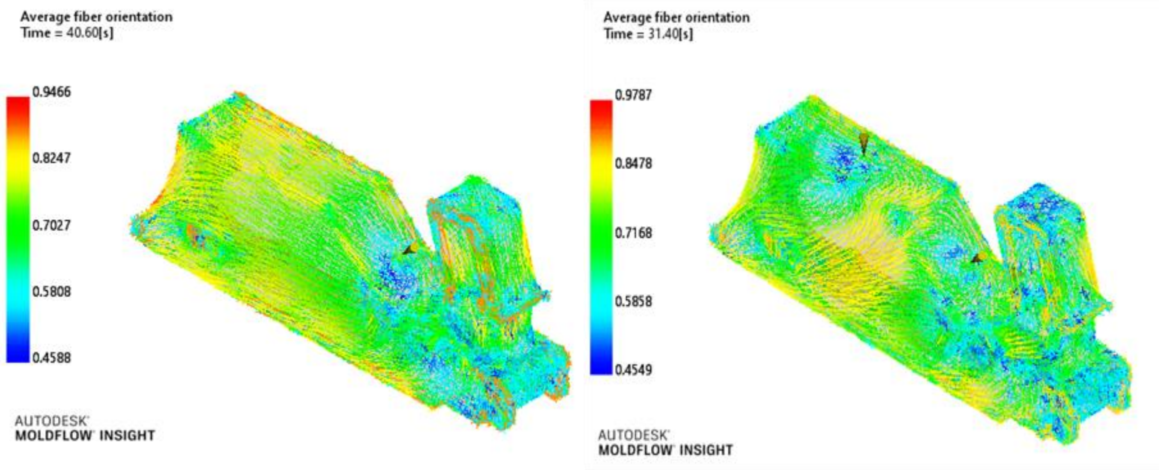

Figure 25. Changes in the fiber orientation in the workpiece after removing injection pressure with (Left) 1 and (Right) 3 injection gates

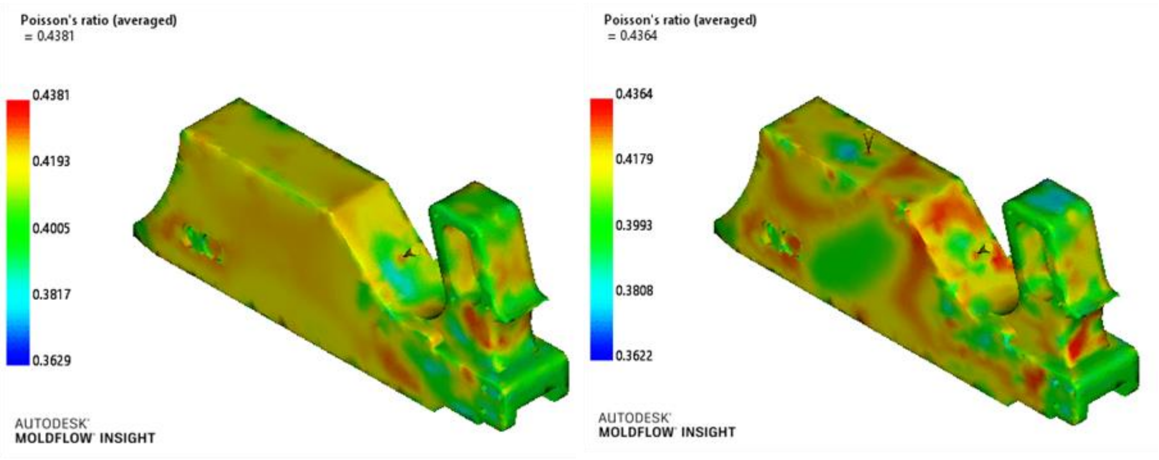

Figure 26. Variations of the average Poisson's ratio in the workpiece with (Left) 1 and (Right) 3 injection gates

Poisson's ratio also depends on the injection material type, the point's position and temperature, and the type of metal sintering. Still, the range of its changes is unaffected by the number of gates. At the same time, the Poisson ratio shows changes in the range of 0.362 to 0.437. The change of melt flow in the mode with three injection gates has caused more non-uniformity in this coefficient in the piece (Figure 26). The comparison results of polycarbonate of 30% glass with 1 and 3 gates are shown in Table 10.

Table 10: Comparing the results of polycarbonate injection with 30% glass and with (Left) 1 and (Right) 3 injection gates

S. No.	Title	1 injection gate	3 injection gates
1	Filling time of die	10.75 seconds	1.432 seconds
2	Pressure in V/P switcher (switching from speed to pressure)	2.04 MPa	3.479 MPa
3	Temperature in the flow front	300 ^{0}C	300 ^{0}C
4	Part temperature	300.1 ^{0}C in 40.60 seconds	300.6 ^{0}C in 31.40 seconds
5	Shearing rate	966.3 at 10.75 seconds	2068.5 at 1.423 seconds
6	Pressure range at the injection site	From zero to 1.6 MPa	From zero to 2.784 MPa
7	Maximum volume shrinkage during ejection	9.484 %	9.593 %
8	Time to reach the exit temperature	From 9.8 seconds to 640 seconds	From 1.833 seconds to 631.4 seconds
9	Shot weight percentage	Up to 85% in 10.75 seconds	Up to 85% in 1.425 seconds
10	average speed	145.8 cm/s in 40.60 seconds	672.3 cm/s in 31.40 seconds
11	Maximum clamping force	Up to a maximum of 6 tons	and up to a maximum of 10.8 tons
12	Poisson's ratio	from 0.363 to 0.438	From 0.362 to 0.436
13	Maximum Pressure	2.040 MPa in 40.60 seconds	3.479 MPa in 31.40 seconds
14	Maximum pressure at the end of filling	1.632 MPa	2.784 MPa
15	The highest piston speed	At 50% of the volume of the shot	At 40% of the volume of the shot
16	Shear modulus	From 1329 to 2171 MPa	From 1250 to 2176 MPa
17	Maximum sink mark index	7.186 percent	7.296 percent
18	Maximum volumetric shrinkage	10.07 %	10.10 %
19	Total weight of the workpiece	1190.4 grams	1186.7 grams
20	Maximum speed in flow front	13789 cm/s	1101.8 cm/s
21	Fusing lines	135 degrees	135 degrees
22	Time to reach lower than the initial injection temperature	40.60 seconds	31.40 seconds

4. Conclusion

1- Written software can be an effective tool for determining the selected parameters for producing polymer parts, leading to cost savings and production time. Also, using this software makes it possible to produce flawless parts. This software selects the optimal conditions for polycarbonate and obtains favorable results.

2- Polycarbonate with 30% glass has higher tensile and shear strength, greater hardness, less deformation, and better paintability than polycarbonate with 10% glass. For this reason, polycarbonate material with 30% glass was used for injection. Also, in the evaluation of the parts, the mode with one injection gate was chosen for the production of the part because the increase in the number of gates did not have a noticeable effect on the physical properties or defects. In comparison, the mode with one injection gate had a much lower production cost.

3- MFS and the final input parameters allow a better understanding of the injection molding process and mold filling and the relationship between the part properties with the amount of filler, shear rate, mold, and melt temperatures. At the same time, this software provides good numerical information about the properties and qualitative information about the prediction of common defects in the plastic injection process. Using this software and the software produced based on experimental information can create a proper understanding of the relationship between properties and defects with parameters in plastic injection molding.

4- It was found that a part with desirable properties can be achieved by changing and optimizing parameters such as melt temperature, mold temperature, clamping force, injection speed, and the number of injection gates.

5. References

[1] Zhao, N. Y., Lian, J. Y., Wang, P. F., and Xu, Z. B. 2022. Recent progress in minimizing the warpage and shrinkage deformations by the optimization of process parameters in plastic injection molding: A review. The International Journal of Advanced Manufacturing Technology. 120(1):85-101. doi: 10.1007/s00170-022-08859-0.

[2] Rohde, S., Cantrell, J., Jerez, A., Kroese, C., Damiani, D., Gurnani, R., and Ifju, P. 2018. Experimental characterization of the shear properties of 3D–printed ABS and polycarbonate parts. Experimental Mechanics. 58:871-884. doi:10.1007/s11340-017-0343-6.

[3] Dar, U. A., Xu, Y. J., Zakir, S. M. and Saeed, M. U. 2017. The effect of injection molding process parameters on mechanical and fracture behavior of polycarbonate polymer. Journal of Applied Polymer Science. 134(7). doi: 10.1002/app.44474. doi:10.1002/app.44474.

[4] Farotti, E., and Natalini M. 2018. Injection Molding. Influence of process parameters on mechanical properties of polypropylene polymer. A First Study. Procedia Structural Integrity. 8:256-264. doi: 10.1016/j.prostr.2017.12.027.

[5] Kitayama, S., Yamazaki, Y., Takano, M. and Aiba, S. 2018. Numerical and experimental investigation of process parameters optimization in plastic injection molding using multi-criteria decision making. Simulation Modelling Practice and Theory. 85:95-105. doi: 10.1016/j.simpat.2018.04.004.

[6] Kouchaki, M. 2019. Controlling the warpage of plastic parts in injection molding process using the variation in coolant temperature of core and cavity. Iranian Journal of Manufacturing Engineering. 6(5):39-46.

[7] Macedo, C., Freitas, C., Brito, A. M., Santos, G., Faria, L., Laranjeira, J. and Simoes, R. 2019. Influence of dynamic temperature control on the injection molding process of plastic components. Procedia Manufacturing. 38:1338-1346. doi: 10.1016/j.promfg.2020.01.155.

[8] Shen, S., Kanbur, B. B., Zhou, Y. and Duan, F. 2020. Thermal and mechanical analysis for conformal cooling channel in plastic injection molding. Materials Today. Proceedings. 28:396-401. doi: 10.1016/j.matpr.2019.10.020.

[9] Karagöz, İ. 2021. An effect of mold surface temperature on final product properties in the injection molding of high-density polyethylene materials. Polymer Bulletin. 78:2627-2644. doi: 10.1007/s00289-020-03231-2.

[10] Ansari, M. J. and Jabbaripour, B. 2019. Manufacture and comparison of mechanical properties of reinforced polypropylene nanocomposite with carbon fibers and calcium carbonate nanoparticles. Iranian Journal of Manufacturing Engineering. 6(5):1-12.

[11] Rashid, O., Low, K. W. and Pittman, J. F. 2020. Mold cooling in thermoplastics injection molding, effectiveness and energy efficiency. Journal of Cleaner Production. 264:121375. doi: 10.1016/j.jclepro.2020.121375.

[12] Szabó, F., Suplicz, A. and Kovács, J. G. 2021. Development of injection molding simulation algorithms that take into account segregation. Powder Technology. 389: 368-375. doi: 10.1016/j.powtec.2021.05.053.

[13] Samei, J., Asgari, H., Pelligra, C., Sanjari, M., Salavati, S., Shahriari, A., Amirmaleki, M., Jahanbakht, M., Hadadzadeh, A., Amirkhiz, B. S. and Mohammadi, M. 2021. A Hybrid additively manufactured martensitic-maraging stainless steel with superior strength and corrosion resistance for plastic injection molding dies. Additive Manufacturing. 45: 102068. doi: 10.1016/j.addma.2021.102068.

[14] Mianehrow, H. and Abbasian, A. 2017. Energy Monitoring of plastic injection molding process running with hydraulic injection molding machines. Journal of Cleaner Production. 148:804-810. doi: 10.1016/j.jclepro.2017.02.053.

[15] Bianchi, M. F., Gameros, A. A., Axinte, D. A., Lowth, S., Cendrowicz, A. M. and Welch, S. T. 2019. On the effect of mould temperature on the orientation and packing of particles in ceramic injection moulding. Journal of the European Ceramic Society. 39(10): 3194-3207. doi: 10.1016/j.jeurceramsoc.2019.03.049.

[16] Khosravani, M. R., Nasiri, S. and Reinicke, T. 2022. Intelligent knowledge-based system to improve injection molding process. Journal of industrial information Integration. 25: 100275. doi: 10.1016/j.jii.2021.100275.

[17] Otieno, S. O., Wambua, J. M. and Mwema, F. M. 2024. A Predictive modelling strategy for warpage and shrinkage defects in plastic injection molding using fuzzy logic and pattern search optimization. Journal of Intelligent Manufacturing. 1-25. doi: 10.1007/s10845-024-02331-4.