ISSN 2176-5480

22nd International Congress of Mechanical Engineering (COBEM 2013)
November 3-7, 2013, Ribeirão Preto, SP, Brazil

OPTIMIZATION OF MANUFACTURING A PLASTIC PART INJECTION

Luan Saldanha Oliveira
Universidade Federal da Bahia.
Rua Aristides Novis, 02, Federação, 40210-630, Salvador, BA.
luansaldanha@hotmail.com

Abstract. *The process of injection molding is widely used in the industries today and has grown due to the large use of polymeric materials in various parts and products. Due to this great demand and use, it is increasingly important to perform simulations and analysis before actually producing the product, because this way saves time and money. Using a software injection of polymeric material into molds were made several simulations and analysis of the injection process, such as filling, cooling, packing and warping. You can also evaluate what the best material to be injected and the main process parameters (time of filling and cooling, mold temperature, part temperature, coolant, etc.). Finally, it was possible to optimize the process parameters to yield more suitable for this application.*

Keywords: *Injection molding, Plastic part optimization, Simulation injection.*

1. INTRODUCTION

The process by injection molding is widely used in the industries today and has grown considerably due to the large use of polymeric materials in various equipment and parts. Thus, their study becomes increasingly important for those working in the area.

The injection molding is a manufacturing process of plastic parts whose main characteristics: high cost of equipment and mold, high volume production, production of several identical pieces, lot of wasted material (material that is on the sprue, system feeding, burr, etc.), pieces of different sizes and complexities, pieces ranging from 5g up to 10kg.

This process consists of the injection of the material at high pressure into a mold having the shape of the desired final product. After injection, the polymer assumes the shape of the mold cavity and cools with the help of the cooling system. To acquire its final shape, the material is extracted from the mold through extraction systems (for example, ejector pins) and, if necessary, receives a post processing before use.

In relation to the mold, this can have various sizes, types and complexities. They are constructed by two parts which are closed during the injection cycle. The mold has various systems, among which we can mention: feeding system, printing system, cooling system, exhaust system for gas, extraction system, among others.

About the materials, in injection molding can be used many kinds of polymeric materials. Depending on the functionality and characteristics of the workpiece is possible to use plastics such as ABS, polyethylene, polycarbonate, polyurethane, polyester, and a range of polymers available. The choice of material influences the parameters of mold manufacturing, injection molding machine type to be used and the design of the part and its systems.

The main operating parameters of the injection process are: temperature of the melt and the mold, injection rate and filling time, pressure and time of compression, closing force and cooling time. All these parameters must be analyzed before starting the injection, in order to optimize the process and run it the best way possible.

2. OBJECTIVES

The objective of this paper is to perform the analysis and simulation of a process of injecting a plastic part, in order to optimize the process. For this, was used Moldflow software, which is software for analyzing injection molding of thermoplastics that assists in optimizing the design of plastic parts and injection molds.

Through it is possible to obtain data for the main stages of the injection process: filling, cooling, packing and warping. Analyzes will be conducted for each step separately and then all together, to evaluate the best parameters for the injection.

To achieve the proposed objectives, the following steps are performed:

I. Selection of the piece;
II. Selection of material for injection;
III. Developing the design of the part in modeling software;
IV. Simulation of injection processes:
   a. Filling;
   b. Cooling;
   c. Packing;
   d. Warping.
V. Analysis of the parameters of each stage;
VI. Optimization of process steps injection.

3. METHOD

The use of software CAE (Computer Aided Engineering) in engineering is increasingly common. This tool assists in product development, through simulations and analysis of virtual prototypes, enabling better efficiency, quality and reliability of the product and increasing the company's competitiveness. The CAE software enables the evaluation of manufacturing conditions in order to make corrections more quickly and at less cost.

For the realization of this article was used software Moldflow (CAE), in order to perform simulations and analysis for optimization of process of a plastic part injection. Through this tool, it´s possible gain time and reduce cost of the project, because it will not need to create multiple physical prototypes to evaluate the main parameters, which can be evaluated in the software.

As mentioned in section objectives, the article was run through six main steps. Based on handouts by Pires (2012) and software tutorials, were defined, step by step, the data for analyzes of the manufacturing process.

First, there was the choice of the plastic part to be worked and what material will be used for injection. The choice of material must be performed using database and observing the characteristics of the piece required. This step precedes use of the software, but it is important because it starts from the entire analysis process.

Subsequently, starts settings in Moldflow software. Initially must be imported the piece modeled in CAD software to Moldflow and generate mesh, evaluating possible errors and refining it. The next step is insert the chosen material in Moldflow software options (if there are no material previously selected, it creates a new material entering the values of the chosen material). Then is carried out analysis of gate location to decide the best position for the insertion of the polymeric material. With these parameters defined, it is possible to assess how best machine configuration for this process. To complete this initial step, evaluate the best local and shape of runner system.

After the initial settings (part, material, mesh, gate location, machine and runner system) are carried out simulations and analyzes of the plastic part injection. This stage covers the analyzes fill, cool, packing and warpage. In sections of each analysis, it is explained the parameters necessary and how should be performed each one. At the end of each analysis is chosen the best configuration generated for the optimization of the injection of the plastic part.

4. SELECTION OF PART AND MATERIAL

4.1 Plastic part

The piece chosen was a container to hold pen and paper (Fig. 1) with dimensions 0,12 cm x 0,11cm x 0,12 cm. It was modeled and rendered in Solidworks software to then be imported into Moldflow software. The file was saved in 3 different extensions (.IGS, .SLDPRT and .STL) so that, later, was chosen the best format to be worked in software analysis (this depends on the software version and model analysis to be performed).

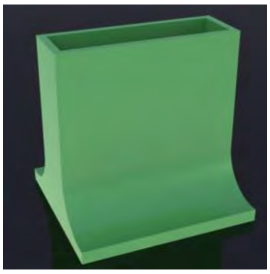

Figure 1. Container to hold pen and paper

4.2 Material

The choice of material is accomplished by observing, particularly, functionality, characteristics of the piece (relevant physical, chemical and mechanical) and cost. To accomplish this selection were used sites database of materials, such as: Campus Plastics, IDES and CES EduPack 2012.

After evaluating the parameters mentioned above, the polymer selected was polyethylene. It presents physical, chemical and mechanical suitable for this application, in addition to having a low cost. Polyethylene is a polymer "*made from ethylene (ethene). Highly resistant to moisture and chemical attack, but has low mechanical strength. It is one of the most used polymers industry, being much used in making sheets (towels, curtains, wrapping, packaging etc.), containers (bags, bottles, buckets etc), plastic pipes, toys, the insulation of electrical wires, etc*.". ([S.I.:s.n], 13 Nov 2012). The properties of the polyethylene are shown in Table 1.

ISSN 2176-5480

Table 1. Properties of Polyethylene

Polyethylene Properties	
General	
Density	960 kg/m³
Price	3,31 BRL/kg

Mechanical Properties	
Young´s modulus	0,896 GPa
Shear modulus	0,314 GPa
Elongation	200-800 % strain
Tensile strenght	44,8 MPa
Fatigue strenght at 10^7 cycles	23 MPa

Thermal Properties	
Melting point	132°C
Glass transition temperature	-15,2°C
Thermal conductivity	0,435 W/m.°C
Thermal expasion coefficient	198 µstrain/°C

Data: CES Edupack 2012

Polyethylene chosen in the program Moldflow was: Dowlex 2517 (Dow Chemical USA). Properties: Mold surface temperature: 30°C; Melt temperature: 250°C; Mold temperature range: 20 – 60°C; Melt temperature range: 230 – 270°C; Ejection temperature: 96°C; Absolute maximum melt temperature: 290°C; Maximum shear stress: 0,1 MPa; Maximum shear rate: 40000 1/s; Melt density: 0,72444 g/cm³; Solid density: 0,92573 g/cm³.

5. MESH, GATE LOCATION, MACHINE INJECTION AND RUNNER SYSTEM

5.1 Creating the mesh

The first step to be performed in software Moldflow is import of the model generated and creation of the mesh in piece (Fig. 2). Following the instructions of the program tutorials and handouts (Pires, 2012) was generated mesh. The mesh initially created by the software is not entirely correct (have any problems like overlapping, intersecting elements, orientation, not connectivity, free edges), so errors must be corrected and refines it so as not to cause problems during the simulation process and to obtain a mesh quality.

Figure 2. Creating the mesh in piece

5.2 Choice of gate location

Initially carried out a best gate location analysis. In this analysis, the software Moldflow informs the best spots to place the gate location. Through color diagram in part, Fig. 3, is reported the most suitable points (locations in dark blue) to put the gate location (the program considers the time for cooling, time to fill, possible warping, among others).

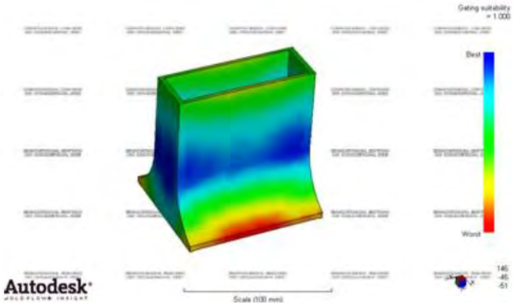

Figure 3. Best gate location

The location of the gate location to perform the injection simulations is shown in Fig. 4.

Figure 4. Location of gate location for injection

This site was chosen after being analyzed points in 3 different locations (dark blue areas such as software best points as indicated). With these analyzes, it was verified that: placing two gate location does not significantly alter the filling time of the part (difference of only 0.02 seconds); in addition, with two location gates the amount of welding lines formed is larger, because during the injection the material will meet on two fronts, generating a greater quantity of welding lines, which is not suitable for parts in the injection molding process. Thus, not justified the choice of two gates location, being just a better use. Regarding location, the gate location put on the front part shows better results for the filling (better uniformity distributes the polymeric material during the injection process throughout the piece).

5.3 Choose the injection machine to process

Three injection molding machines were used to analyze the properties of the part during the injection process. These devices are based on data from the Moldflow software. The equipment has following characteristics:

- Machine 1800: maximum machine injection stroke: 220mm; maximum machine injection rate: 270 cm^3/s; machine screw diameter: 50mm; maximum machine clamp force: 183,492 tonne;
- Machine 1500: maximum machine injection stroke: 160mm; maximum machine injection rate: 107 cm^3/s; machine screw diameter: 40mm; maximum machine clamp force: 152,910 tonne;
- Machine *default*: maximum machine injection stroke: 160mm; maximum machine injection rate: 88cm^3/s; machine screw diameter: 35mm; maximum machine clamp force: 80 tonne.

In Table 2, it is possible to compare the values obtained on each machine. After analyzing the results of the machine default and 1500, it was realized that had values that were not in the desired range. So, was selected the machine 1800 to evaluate the new results. With this equipment, the values were within the desired (shorter filling time and lower ranges from bulk temperature and flow front), which justifies the use of this.

ISSN 2176-5480

Table 2. Analysis of the parameters: Machines Default, 1500 and 1800

Analysis (Constant profile)	Default	Machine 1500	Machine 1800
Fill time	1,604 s	1,318 s	0,5332 s
Bulk temperature	186-245°C	196,1-246,2°C	222,9-252,0°C
Flow front	216,8-231,0°C	222,4-231,1°C	229,7-231,0°C
Pressure V/P	25,02 MPa	26,26 MPa	33,01 MPa
Shear stress	0,5144 MPa	0,5368 MPa	0,6568 MPa
Weld lines	135 deg	135 deg	135 deg

5.4 Runner system

After selecting the position of the gate location created the runner system (Fig. 5). By the mode *wizard* software Molfdlow, runner system was generated from two possible locations: the center of the mold and the center gate location. Taking into consideration issues of manufacturing and injection process part, the runner system was placed toward the center of the gate location and length less than the generated by *wizard* of Moldflow. Thus, it was possible to reduce the loss of material in the sprue and minimize temperature loss of the material until the mold cavities.

Other considerations about runner system: use of direct attack, because there is only one cavity in the part, this being fed directly by sprue; more of an attack can leave considerable mark and weld lines in the piece; balanced flow in the cavities to obtain parts quality.

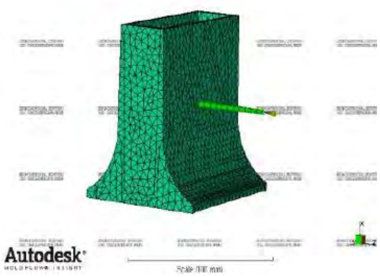

Figure 5. Runner system

6. ANALYSIS: FILL, COOL, PACKING AND WARPAGE

6.1 Analysis Fill

This is the first analysis to be performed and has to evaluate how it will be filling part and the time for this to occur. The fill analysis has results initials important as the injection time, the flow and temperature of the local part.

The following profiles were evaluated for this analysis:

- Constant profile: assessing initial behavior of the injection process and the parameters considered ideal for this part (injection time, mold temperature, variable flow);
- Variable profile: from data flow variable obtained from the initial analysis (constant profile) this analysis was performed in order to compare the results with the previous analysis;
- Profile constant with sprue 6 mm: due to the previous results are not with the recommended minimum values, this analysis was carried out to improve mainly the results of the flow front, bulk temperature and shear stress.

Data used for analysis: Mold surface temperature = 55°C; Melt temperature = 230°C; Machine 1800.

Table 3 presents the main results of the analyzes.

Table 3. Results of analysis constant, variable and constant with diameter 6 mm

Results			
Profile type	**Constant**	**Variable**	**Constant (6mm)**
Maximum injection pressure	33,0071 MPa	29,2542 MPa	17,8410 MPa
Time at the end of filling	0,5332 s	0,7322 s	0,5235 s
Total weight (part + runners)	97,4412 g	97,5435 g	99,3431 g
Maximum clamp force during filling	21,1157 tonne	8,5175 tonne	21,5174 tonne
Total volume	133,1746 cm³	133,1746 cm³	135,3215 cm³
Total projected area	279,4839 cm²	279,4839 cm²	279,4840 cm²
Bulk temperature	29,1°C	34,6°C	16,2°C
Flow front	1,3°C	1,6°C	1,1°C
Pressure V/P	33,01 MPa	18,78 MPa	17,84 MPa
Shear stress	0,6568 MPa	0,7642 MPa	0,4371 MPa

From the results, can say that the constant profile with sprue 6 mm showed the best results (within acceptable values for the process). The filling time was low (0.5235 s), the bulk temperature is in the recommended range (range should have less than 20°C), flow front also has acceptable value (it is recommended that has value less than 5°C) and pressure V/P is below the maximum injection pressure of the mold (184.25 MPa) and below recommended (it is recommended to use between 50% and 75% of the maximum pressure injection machine).

This process has been improved with the following changes:

- Increasing the mold surface temperature (30°C to 55°C). This temperature is within the range of values accepted by the workpiece material (polyethylene);
- Lowering the temperature of the melt 250°C to 230°C. This temperature is within the range of values accepted by the workpiece material;
- Definition of control stuffing through injection time (0.502 seconds);
- Increasing the diameter of the sprue.

6.2 Analysis Cool

The mold cooling accounts for over two-thirds of the total cycle time for the production of plastic molded parts. Therefore, analyzing the cooling is very important since this parameter is directly linked to the productivity of the process. In addition, a uniform and efficient cooling improves the quality, stability and dimensional accuracy of the workpiece.

The procedure for the assessment of cooling is a constant iterative optimization of the results. For this, was held three different simulations. As a first test, began with an automatic analysis via the wizard of Moldflow in order to have a first idea about the properties cooling. The next step, after interpreted the results obtained, it was applied a new cooling system, and finally, in the last analysis the obtained optimum results. Table 4 shows the results obtained in the three analysis.

Table 4. Results of analysis with systems 1, 2 and 3

Results			
System	**1**	**2**	**3**
Part surface temperature - Maximum	126,44°C	121,41°C	98,96°C
Part surface temperature - Minimum	28,29°C	22,84°C	12,23°C
Part surface temperature - Average	64,56°C	54,05°C	38,49°C
Cavity surface temperature - Maximum	126,63°C	121,75°C	99,34°C
Cavity surface temperature - Minimum	25,00°C	20,00°C	10,00°C
Cavity surface temperature - Average	61,60°C	50,90°C	35,78°C
Average mold exterior temperature	28,83°C	21,09°C	10,59°C
Cycle time	35,00 s	35,00 s	45,00 s

ISSN 2176-5480

The first cooling system is generated by wizard of Moldflow. It features four channels with simple format "snake", parallel in the x direction and separate part of a distance of 25mm. For this system, the sum of the times IPC (Injection + Packing + Cooling) was equal to 30 seconds.

The second system was designed in order to cover a larger area of the part and thus make more effective cooling. Three cooling channels were used with nominal diameter 10mm and water temperature of 20°C. The sum of the times IPC continued worth 30 seconds.

The third system is the same as above, but with some different variable values (in order to verify the improvement in cooling part). The IPC time was increased to 40 seconds and the water temperature reduced to 10°C.

Of the three tested systems for cooling the workpiece, the third showed the best results. However, it is not ideal. The average temperature of the part is below the ejection temperature of the material (there is a thermal gradient wide), there are some hotspots in part, mainly in the inner part and the time to achieve ejection temperature is 23 seconds. Due to cooling problems in the inner part, the best solution is to create a waterfall inside it, with the goal of decreasing the thermal gradient and make more uniform temperature profile. This solution helps to decrease potential problems such as warping due to temperature variations exist. Figure 6 show average temperature of system 1, 2 and 3.

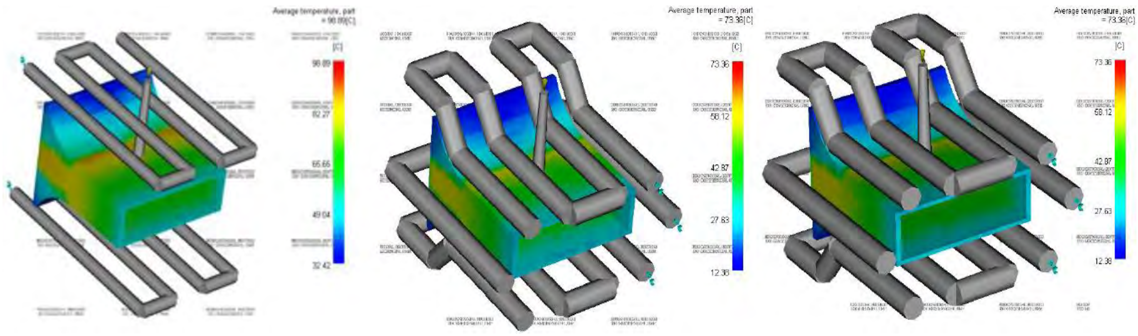

Figure 6. Average temperature: System 1, 2 and 3

6.3 Analysis Packing

After the analysis of the first two stages of the injection process (fill and cool) is realized compaction analysis (packing). This analysis aims to assess the shrinkage of the part that occurs after cooling, by changing the geometry of the part. Therefore, it is important to control this parameter so that the entire piece evenly contract.

There were performed three tests to assess the compaction of the part. The first was done with the pressure profile by default of Moldflow. The second and third analyzes were performed based on the results of previous analyzes.

For analysis packing is important to determine the maximum compaction pressure used (assuming a uniform pressure distribution throughout). This value was determined as follows:

$$P_{Máx.} = \frac{\text{Clamp force (tonnes) x 100 x 0,8}}{\text{Total projected area of model (cm}^2\text{)}} = \frac{183{,}492 \text{ x } 100 \text{ x } 0{,}8}{279{,}484} = 52{,}5231 \text{ MPa} \quad (1)$$

As this value is below the maximum injection pressure injection machine (194.25 MPa), it can vary between 80% and 120%. This is done in the analysis packing 2 and 3. The parameters of the packing analysis is shown in Table 5.

Table 5. Parameters packing analysis

	Time (s)	% Pressure Filling
Analysis 1	0	80
	10	80
Analysis 2	0	100
	18	100
Analysis 3	0	120
	18	120

ISSN 2176-5480

The first analysis was performed with the data defaults of Moldflow. The data used were those obtained in the best analyzes fill and cool. Data: Mold surface temperature: 55°C; Melt temperature: 230°C; Mold-open time: 5 seconds; IPC: 40 seconds; Injection time: 0,502 seconds; Velocity/pressure switch-over: 99% volume filled.

To perform the analysis by packing it was verified that up to a time of 18 seconds (approximately) there are still compacting the material. Thus, as in the first analysis duration was of 0 to 10 seconds that time increased to 0 to 18 seconds. Furthermore, it was increased filling pressure.

The third packing analysis was performed with the aim of improving the results obtained in previous analyzes. For this analysis, it was considered the same time interval (0 to 18 sec), however, altered the filling pressure up to 120% (range still acceptable pressure percentage). The objective is to evaluate what changes happen in the results and increased filling pressure improves the process parameters of injection.

Table 6. Results of analysis: 1, 2 and 3

Results			
Analysis	**1**	**2**	**3**
Time to reach ejection temperature	23,74 s	23,74 s	23,74 s
Volume shrinkage	[-0,06%; 16%]	[-0,33%; 12,84%]	[-0,38%; 17,69%]
Hold pressure (Maximum - Minimum)	2,20 MPa	0,85 MPa	0,85 MPa
Residual stress in the cavity	[3,69 MPa; 29,41 MPa]	[-4,77 MPa; 29,48 MPa]	[-4,98 MPa; 29,48 MPa]

Regarding the analysis of compaction, after completion of the three analyzes, it can be concluded that the best results (Time to reach ejection temperature, Volume shrinkage, Hold pressure (Maximum – Minimum) and Residual stress in the cavity (Table 6)) were obtained with the second analysis, which has duration from 0 to 18 seconds and pressure filling percentage of 100%. It is still possible to optimize the results for the injection process, but these results are already acceptable. Below, figures 7 and 8 show the principal results of the second analysis.

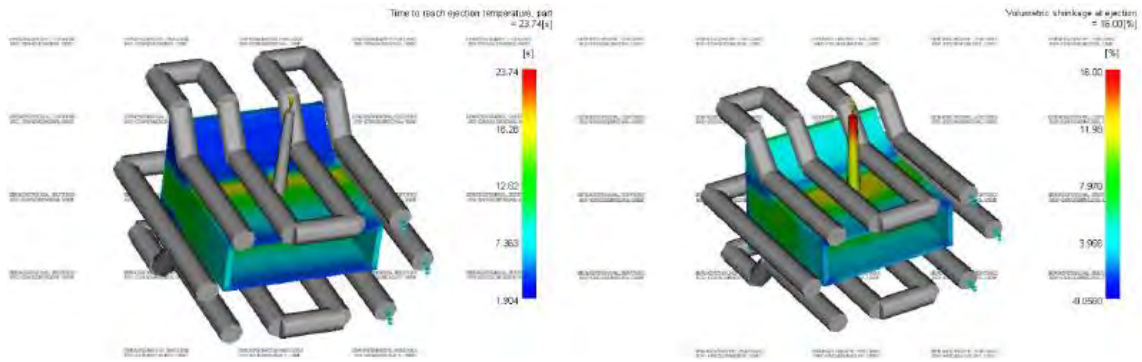

Figure 7. Time to reach ejection temperature and volumetric shrinkage (Analysis 2)

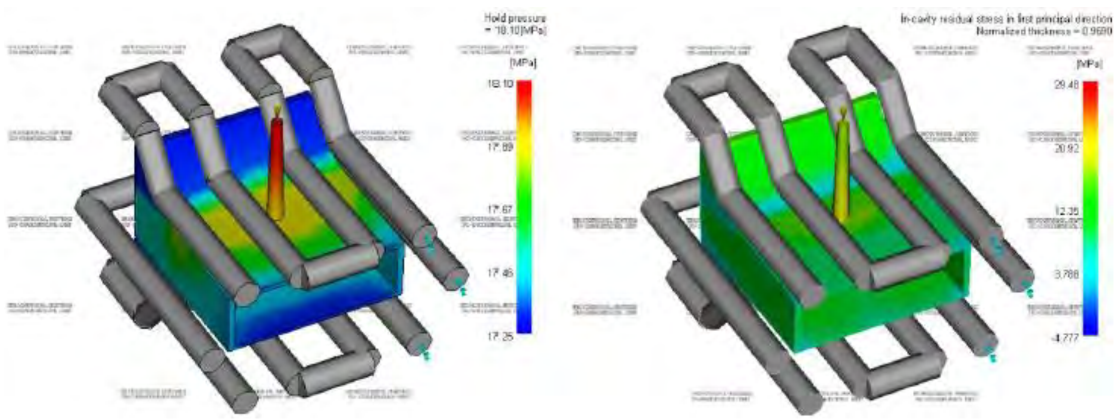

Figure 8. Hold pressure and residual stress (Analysis 2)

ISSN 2176-5480

6.4 Analysis Warpage

In general, the main cause of the warpage in part is volumetric shrinkage (caused by cooling of the injected material). If the contraction is uniform throughout the piece, do not appear warping. The piece only contracts, reducing its size. Controlling the volumetric shrinkage is critical to control warping.

There are three most common causes for a bad piece of volumetric shrinkage:

I. Non-uniform cooling: when the temperature gradient part is significant (i.e. zones are hotter than other part) may lead to folds in piece. There are indicators that must be analyzed, such as the temperature profile or temperature different plot obtained from analysis of cooling;

II. Contraction part second different directions: there are two types of contraction (longitudinal and transversal) which are determined by the orientation and efforts when the molecules solidify. For composite materials is important to know the direction of the fibers, since the contractions are more favorable in the transverse direction, for polymeric materials injected normally there is contraction in the direction of filling (longitudinal);

III. Sharp edges: in parts with well-defined edges (corners) is very common appearance of warping because the ability to "pull" heat these localized areas is also resulting in lower stresses induced by poor cooling. Moreover, the volumetric shrinkage in the thickness direction of the wall is much larger than towards "in-plane", resulting in deformation.

Other factors that contribute to the formation of warping: geometry of the part and the mold, processing conditions (mold temperature, melt temperature, injection time, time compression and compaction pressure) and the material chosen.

The warpage analysis encompasses all other analyzes made earlier. The initial step to begin analyzing the warping should be to optimize the filling, cooling and compression. Getting good parameters in these analyzes, the risk of warping is reduced.

I. Optimization of filling: the parameters that should be optimized in fill analysis are: gate location, feeding system, processing conditions, balanced flow pattern, there should be overpacking, weld lines, without hesitation, etc.;

II. Optimization of cooling: uniform heat exchange between the two parts of the mold (i.e. the cavity between temperatures of central part and mold surface and along the part). It is important that the cycle time is also optimized;

III. Optimizing compression: low and uniform shrinkage, thus ensuring the solidification of gate location.

The analyzes carried out warping are the same as packing (same values for time and % Pressure Filling). The first was done with the pressure profile and default time Moldflow. The second and third analyzes were performed based on the results of previous analyzes.

After three analyzes of the results, it was observed that the total deflection part has a value of 1.06 mm, and the deflection by the cooling effect corresponds to 0.72 mm (0.34 mm corresponds to volumetric deflection and deflection for orientation effects).

As cited above, an improved cooling system (with waterfalls) will generate a smaller temperature gradient and directly influence the effects of warpage of the part, since the cooling effects of deflection corresponds to more than 60% of the total deflection piece.

7. CONCLUSION

The performance analysis through software Moldflow presented results for the main parameters that should be evaluated in the injection process. At the end of the analysis of each step - filling, cooling, packing and warping - were decided which parameters are the best performers, and consequent justification.

Regarding the fill analysis is concluded that the process was optimized by increasing the temperature of the mold surface, lowering the temperature of the melt, increasing the diameter of the sprue and setting the injection time to 0.502 sec.

To cool analysis, it is still necessary to better evaluate the parameters to improve the results of this step. Despite having conducted three different analyzes, final results showed that the heat extraction part is not uniform. Since the section most critical part was the cavity (the outside part presented a good cooling), created a thermal gradient between these two extreme areas. A possible solution to this problem is to apply a cascaded at this location.

The packing and warpage analysis was performed together and used as input parameters further analysis of the data cooling. The best results were achieved for 100% of a pressure applied to the 18s.

Table 7 shows the final characteristics of the process and part.

Table 7. Characteristics of the process and part

Technical features	
Weight	97,6 g
Volume	135 cm³
Area	279,5 cm²
Cycle time	45 s
Material	Polyethylene
Melt temperature	230 ºC
Instant V/P	0,52 s

8. ACKNOWLEDGEMENTS

This work was supported by CNPq, National Council for Scientific and Technological Development - Brazil, through a stock exchange of the Science Without Borders Program. This article was the result of a research project developed during the course Processing polymers and casting at Superior Technical Institute, Technical University of Lisbon.

9. REFERENCES

CAMPUS - A material information system for the plastics industry. 10 Nov 2012. <http://www.campusplastics.com>
CES Edupack 2012. GRANTA Design
Coutinho, Fernanda M. B., Mello, Ivana L., Maria, Luiz C. de Santa, 2003. Polímeros: Ciência e Tecnologia, vol. 13, nº 1, p. 1-13, 2003. <http://www.scielo.br/pdf/po/v13n1/15064.pdf>
Moldflow Insight Tutorials. 14 Nov 2012. <http://www.molding-simulation.com/browse-moldflow-insight-tutorials-videos-1-date.html>
Pires, Ines, 2012. Notas de aula: Transformação de polímeros e fundição.
Resin Material Prices: Secondary Markets. 10 Nov 2012 <http://www.ides.com/resinpricing/Secondary.aspx>
[S.I.:s.n.]. "Tipos de polímeros". 13 Nov 2012. <http://polimeros.no.sapo.pt/tipos.htm>

10. RESPONSIBILITY NOTICE

The author is the only responsible for the printed material included in this paper.