THE MOLDFLOW ANALYSE OF THE INJECTION PARTS, AND THE IMPROVEMENT OF THE INJECTION PROCESS.

Togan V.C.[1,2], Ioniță Gh.[1]

[1]Valahia University, Faculty of Material Engineering, Mechatronics and Robotics, 18-24, Unirii Blvd, Targoviste, Romania; [2]S.C. Arctic S.A., Product Development Department, Str. 13 December, No 210, Gaesti; Romania;
E-mail: valentin.togan@yahoo.com; ionitateacher@yahoo.com

Abstract: Processing by injection is the largest industrial way to obtain plastic parts. An analyzing accuracy of the injected part execution, improving the quality of the injection process and of course the quality of geometrical execution parts and injection process, has been performed with the new 3D simulation program Autodesk Moldflow Insight.

Keywords: Part contractions, injection process, material flow, Moldflow analyze.

1. INTRODUCTION.

The 3D drawing occurrence, led to the development of the manufacturing parts made from polymer (using directly the 3D drawing on molds execution). With the modernization of molds execution methods, major improvements have been made on the execution simulation of various polymers based parts. The new and innovative virtual simulation software brought a variety of tools used to verify any design error.

This study case is performed on an injected polyethylene (a cap) with a complex geometric shape design (fig 1.).

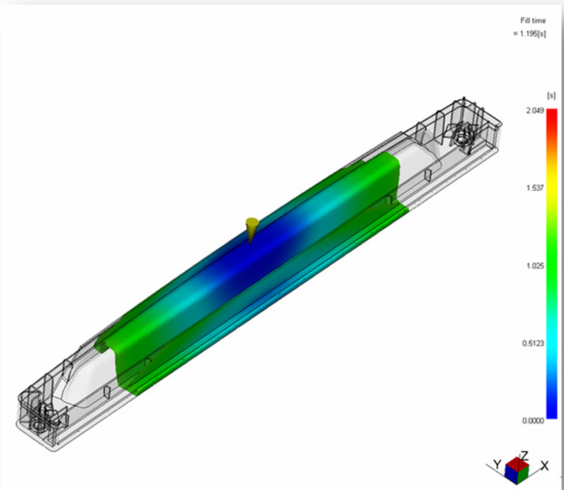


Figure 1: Injected part for analysis.

During the injection process it may appear some problems, because the parts made of polymeric materials are prone to surface defects that can be avoided only if the material flow and the 3D drawing design are correctly executed. To perform this part analyze will be used "Autodesk Moldflow Insight" software.

Thus, the aim of the virtual simulation is to optimize the part or mold design, in order to eliminate the possibility of surface defect occurrence during the injection process.

On the injection thermoplastic after processing, to achieve a certain part dimension, several criteria that must be taken into account, influence the quality on injection process, including the contraction phenomena, injection time or the cooling mold time calculation.

2. ACCURACY ANALYSIS OF INJECTED PART EXECUTION.

2.1 Injected part contraction.

Molded part contraction depends in the first place on the nature of thermoplastic injected material. Processing contracts must be from 24 to 168 hours after removal of the injected part from the mold. After the specified period contraction phenomena or post contraction can occur.

In the below figure you can see a section in the mold cavity in which material is injected at injection process runtime.

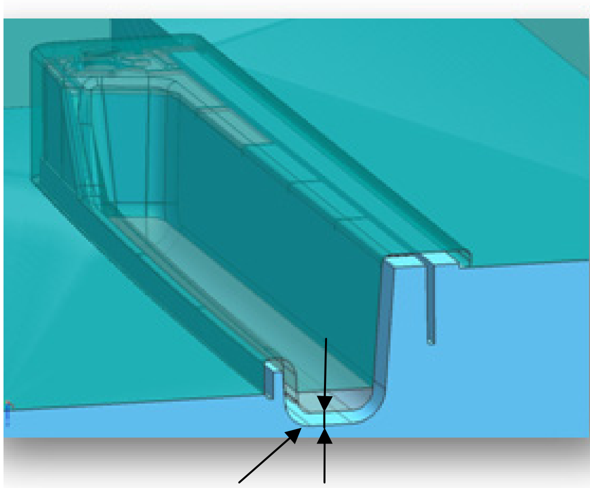


Figure 2: The mold cavity where the material is

2.2. Contraction calculating.

To understand better the contraction phenomena, below are presented some section images that show us the dimensional difference between the part (L) and the cavity in which the material is injected (Lm).

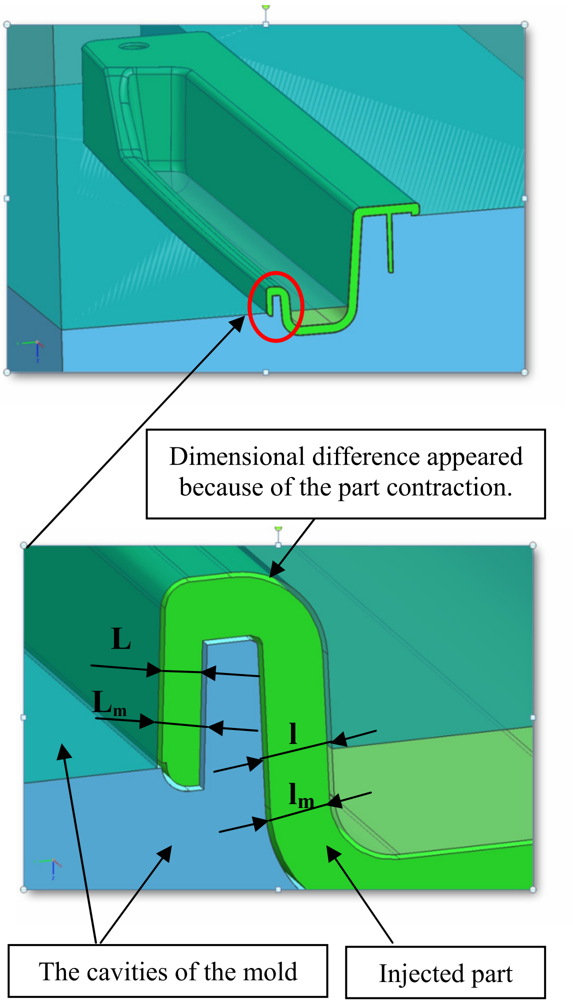


Figure 3: Mold section– injected part.

Where: l_m, L_m – cavities mold dimensions;
l, L – injected parts dimensions.
Regarding the studied part:
L = 2 mm
Lm = 2.01 mm

$$C_l = \frac{l_m - l_p}{l_m} = 1 - \frac{l_p}{l_m} \qquad (1)$$

Where: C_l – linear contraction;
l_m – mold cavity size to the average temperature of the mold at injection time;
l_p – length of the injected part at room temperature (20^0C)
Because in the injection runtime it is processed a melt material volume, after the cooling will take place a volume contraction:

$$C_v = \frac{V_m - V_p}{V_m} = 1 - \frac{V_p}{V_m} \qquad (2)$$

Where: V_m – cavity volume of work part inside the mold at average temperature mold in the injection runtime, (30…60ºC);
V_p – piece volume injected at ambient temperature, (20ºC)
On the mold construction for a wall thickness of ≈ 2mm is taken into account:

$$C_l \approx 0.6\%$$

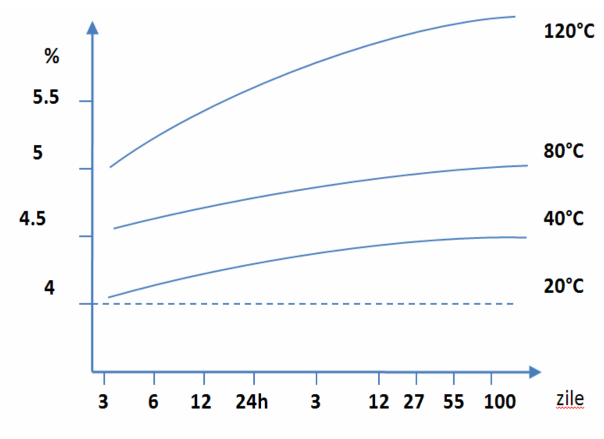
From equation (1) result:

$$\frac{L_p}{L_m} = 1 - Cl \qquad (3)$$

For an isotropic contraction of the three volume directions and replaceing the linear dimentions of the part and mold, volume contraction equation can be develop:

$$C_l = \frac{C_v}{3} \qquad (4)$$

Amorphous materials (polystyrene, polyacetal vinyl, ETC.) are showing a large isotropic properties with uniform contraction of processing by injection. In Figure 4 can be seen that post contraction is even greater when the storage temperature is higher. Even if the injection process is performed in optimal parameters, is very important both storage temperature and storage location (some injected parts, such as plastic handles, requires after the injection process a support to be placed in order to keep the initial shape unchanged).



------------ Polyethylene contraction
———— Polyethylene post-contraction

Figure 4: Contraction and post contraction of polyethylene injected part.

The injected part analyze has been performed with Autodesk Moldflow şi Unigraphics NX5. The analyze purpose is to avoid surface defects arising from the injection process (deformations, shrinkage cavity, burned area, exfoliations, flow material shape, defects related with initial material humidity.

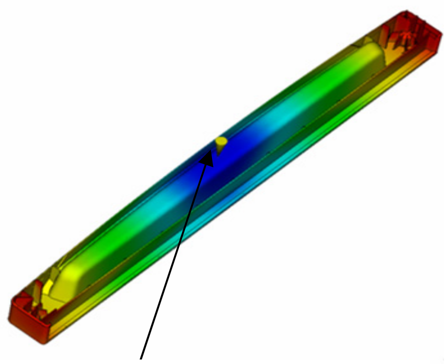


Figure 5: Injection point position.

As shown in Fig. 5 the injection point position is in the middle of the part, but not on the commercial surface. The reason for choosing this point is to fill the injection mold cavities in all areas and in the same time to reduce the part distortion risk. In Table 1 are mentioned the specific parameters of the process (mold temperature surface, minimum-maximum melting temperature, mold temperature at removing time the part from it, etc.).

Shrinkage Properties	Filler Properties	MuCell ® Material Properties
Description	Recommended Processing	Rheological Properties
Mold surface temperature	60	C
Melt temperature	240	C
Mold temperature range (recommended)		
Minimum	40	C
Maximum	80	C
Melt temperature range (recommended)		
Minimum	230	C
Maximum	250	C
Absolute maximum melt temperature	290	C
Ejection temperature	85	C

Tabel 1. Injection process parameters.

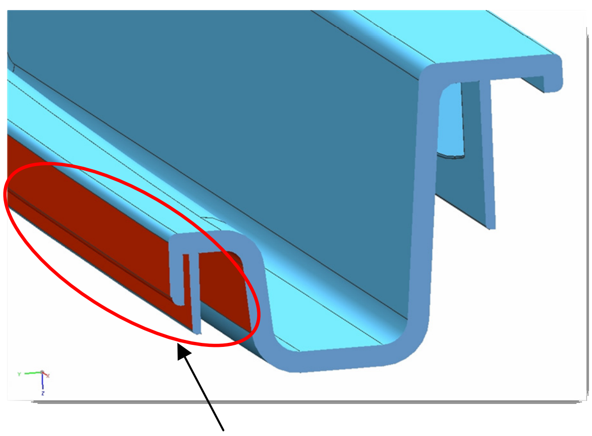


Figure 6: Cooling problem area.

Cooling the injected part from maximum value of plastic material temperature (during filling the mold cavities) at room temperature requires a relatively long time.
The formation of "hot area" with clumps of material, in case of a thick-wall molded part such as this, prolong the injection cycle due to increasing cooling time in the mold. The mass concentration of material and the forming of "hot area" are made especially in the corners or the injected part, where the side walls are joining the intersection of several walls.

3. COOLING PROBLEM AREA SOLUTIONS.

3.1 Improved design.

As a solution to this problem, the design will be improved by eliminating the continuous rib and by applying some local ribs on injected part as shown in the Fig 7.

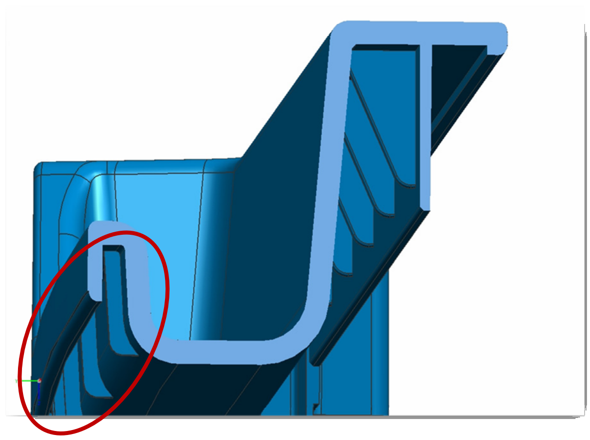


Figure 7: Improved design.

3.2 Injection process influenced by the mold temperature.

The mold temperature is a critical measure for cooling speed value and injection part properties. This critical value can be established according to heat exchange that takes place into the mold:
- Between the injected thermoplastic and the mold material Q;
- Heat exchange between the mold and the environment Q_T;
- Between the mold components and environments Q_E.

Are considered to be positive the thermal flows that are coming inside the mold and to be negatives those that are leaving it:

$$Q + Q_E + Q_T = 0$$

Heat balance expressed in this equation can be considered as a basic equation of the cooling process.

$T_M = 20...70^0C$ $\quad Q > 0$

$Q_E < 0$

$Q_T < 0$

Where: T_M – Output value or the mold temperature
Thus, the fluid temperature T_T will be adjusted between 30^0C şi 80^0C. Flow rate V_T of fluid temperature is a

adjustment control value and its disadvantage is that the flow at low speeds, the water in the mold is strongly heated, so cooling it at the entrance into the mold is more intense then the output from the mold. Possibility that the cooling intensity can be adjusted separately for each moderation can be considered the strongly advantage.

Material temperature T_m is one of the main factors that influence the dimensional deviations of an injected part. Thus, it seeks to achieve the desired material temperature through a theoretical adjustment of the cylinder and nozzle heating.

3.3 Injection cycle time calculation.

The total cycle time t_t influence the level of cooling (the cooling process takes places in the mold).

Total cycle time can be calculated with the relation:

$t_t = t_u + t_r + t_d$ (1)

Where: t_u – filling time;

t_r – cooling time; t_d – stripping time.

Total time determination t_t :

$t_t = t_u + t_{ul} + t_{cm} + t_{rm} + t_{rest} + t_{dm} + t_p + t_{im}$ (2)

Where: t_u – filling time;

t_{ul} – further pressure time;

t_{cm} – delay for the screw coupling ;

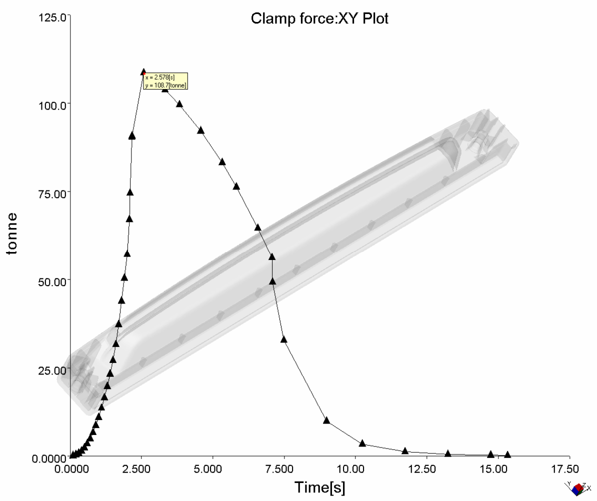


Figure 8: Injection total time.

t_{rm} – screw rotation time;

t_{rest} – left time – The time between the end of the screw rotation and beginning of the mold opening ;

t_{dm} – opening mold time between the end and start of the mold opening;

t_p – break time – time between the beginning of the opening and the end of mold opening ;

t_{im} – mold closing time – time between ending and starting mold closing.

To determinate the fill time calculation, we can use the relation: $t_u = t_{u\,min} + 0.3\,(t_{u\,max} - t_{u\,min}) \approx 2$ s (4)

$t_{u\,min}$ – minimum filling time

$t_{u\,max}$ – maximum filling time

Cooling time is calculated as:

$t_r = t_{pul} + t_{cm} + t_{rm} + t_R \approx 8$ s (5)

t_{ul} – further pressure time (compaction pressure)

t_{cm} – screw coupling delay time ;

t_{rm} - screw rotation time;

t_R – left time – time between ending screw rotation and mold opening start;

Stripping time can be calculated as:

$t_d = t_{dm} + t_p + t_{im} \approx 7.5$ s (6)

t_{dm} – mold opening time;

t_p – break time;

t_{im} – mold closing time.

$t_u << t_r + t_d$,

Process total time:

$\mathbf{t_t = 2 + 8 + 7.5 = 17.5}$ **s** (7)

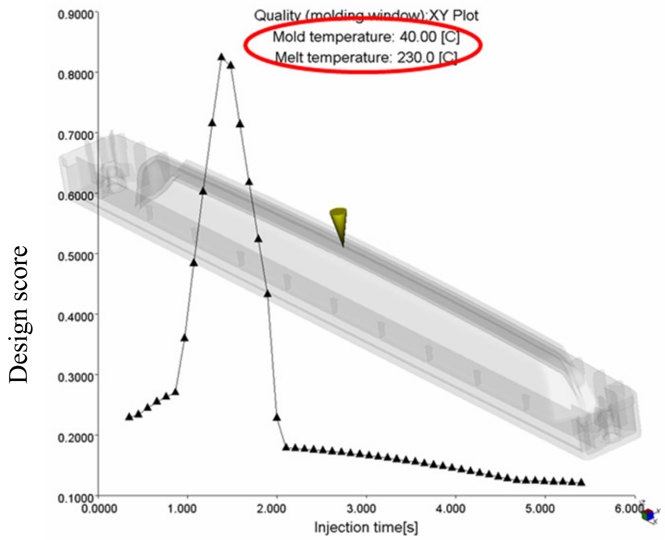


Figure 9: Qualitative analysis designs before solution apply.

Mold temperature = 40^0 C; Melt temperature 230^0 C
Score result = 0.82 (Maximum score = 1)

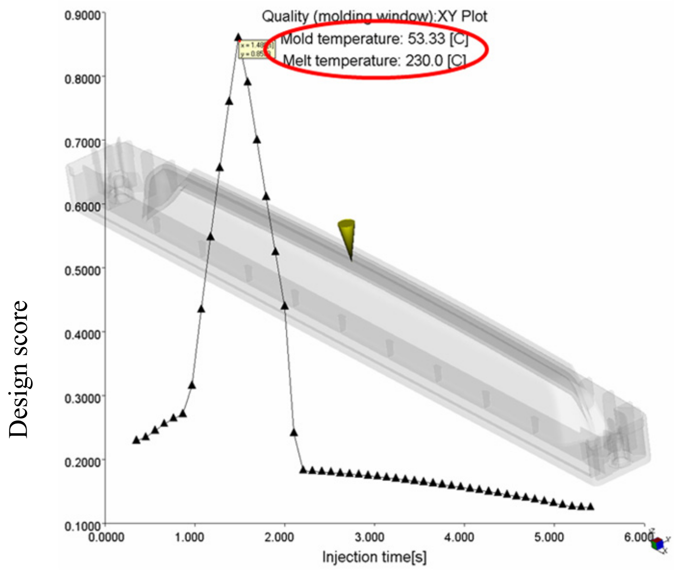


Figure 10: Qualitative analysis designs after solution apply.

Mold temperature = 53.33^0 C; Melt temperature = 230^0 C
Score result = 0.89 (Maximum score = 1)

4. CONCLUSIONS.

By means of the injected plastic fluidity analysis software Moldflow Autodesk Insight, can be highlighted both the injection process quality and injected quality part.

In the diagrams shown in Fig. 8 and Fig. 9 is illustrated qualitative analysis of the two injection processes – before and after applying the solution to improve the process execution. The charts are generated by the software according with the input data. Thus, diagrams can analyze the ratio of flow injection (correct part design) and injection total cycle time.

It has been shown that controlling the mold temperature will increase the part quality. If the temperature correction can be controlled, then will be avoided the surface defect (that can appear on the commercial surface of the part) from the injection process (contractions, deformations, shrinkage cavity, combustion zone, exfoliations, flow material shape, defects caused by material humidity)

Also, the importance of using a specialized program such as Moldflow, it follows from the fact that the analysis made in this software, avoids the design mistakes. In case that design errors occur and verifications with such special software are not performed, mold modification for correcting this errors is very expensive or sometimes impossible.

5. REFERENCES:

[1] Ion Sereş - *"Thermoplastic material injection"* Published Western Printing 1996.

[2] Mircea Ionescu Muscel Ianculescu – "*Mold designing for plastic injection parts*" Bucharest Technical Published , 1987.

[3] Tudose, Z., R., "*Processes and equipment in manufacturing macromolecular compounds*" Bucharest Technical Published, 1976.

[4] Mihail R., *"Polymer processing simulation.*" Bucharest Technical Published, 1989.

[5] Internet: http://www.biblioteca-digitala.ase.ro/biblioteca/carte2.asp?id=45&idb=