MATERIALE PLASTICE
https://revmaterialeplastice.ro
https://doi.org/10.37358/Mat.Plast.1964

Experimental Research Regarding the Defects Occurring at the Injection-molding of Long Technical Parts, Made of Thermoplastic Material, Using CAE Systems. Case Study

ANDREI PRADA[1]*, FLORIN BLAGA[1], STEFAN MIHAILA[1], MIHAI AGUD[2]
[1]University of Oradea, Faculty of Managerial and Technologial Engineering, Industrial Engineering Department, 1 Universității Str., 410087, Oradea, Romania
[2] S.C. Plastor S.A., Engineering Department, 175 Calea Clujului Str., 410546, Oradea, Romania

Abstract: *In this paper, the authors propose to carry out a case study, on a visual part made of thermoplastic material from the automotive industry. Using the most up-to-date CAE systems, we will try to highlight the main appearance problems in long parts, namely the appearance of weld lines at the meeting of the flow fronts. The part will be processed using sequential, as well as non-sequential injection systems, respectively, in order to highlight the difference between the two, respectively to compare these two tests with the results of rheological simulation (CAE).*

Keywords*: CAE systems, rheology, sequential injection, weld lines, injection systems*

1. Introduction

In order to obtain a good final quality of the visual parts, it is very important to pay special attention to the details related to the design of an injection system as appropriate as possible during the phase of development and validation of the part. Research in the field aims at essential elements of process design, starting from elements such as: general principles for optimizing the parameters of the injection process [1], case studies regarding the choice of the optimal location of the injection points [2], the thermal study of of the process in order to understand the phenomena that lead to the appearance of welding lines [3], the influence of process parameters on the mechanical behavior of the part in areas with welding lines [4], or for example the influence of cooling on the phenomenon of the appearance of these defects [5]. The specialized literature supports technologists, using often theoretical approaches, based on mathematical models of the phenomena in the process, which is why in the present work the authors propose to carry out a practical, less sophisticated study.

The case study will consider a rear bumper ornamental part, made of PMMA. It is a visually appearance important part, from the automotive industry, with length = 1500 mm. The part with this profile was chosen in order to be able to highlight elements such as the optimal injection system, appearance problems and deformations. Before the phase of designing the mold in which the part will be injected, a series of finite element analyzes of the flow of the plastic material is performed.

The CADMould program will be used for the simulation. The first stage involves importing the three-dimensional CAD geometry of the part into the program and discretizing the surface of the model.

In order to obtain results as conclusive and as close to reality as possible, it is recommended that this discretization (mesh) be as fine as possible. However, there is also the disadvantage that fine discretization can cause processor performance problems. Therefore, a middle solution must always be chosen, a mesh as fine as possible for optimal results, but large enough so that the simulation results can be generated in a timely manner. In Figure 1, it can be seen that in this case the program recommended a triangulation with mesh = 2.54 mm, while in Figure 2 it can be seen how much this triangulation value means from a dimensional point of view, in relation to with a track section.

**email: andrei.prada.97@gmail.com*

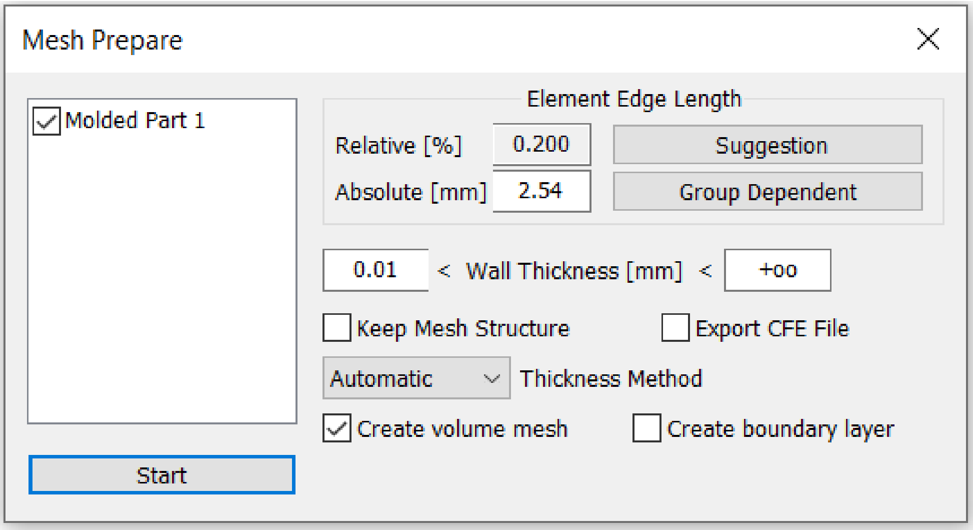

Figure 1. The recommended discretization

Figure 2. Discretized surface of the part

2. Materials and methods

2.1. Choosing the number of injection points

The next stage involves choosing the number of injection points. For each thermoplastic material there are recommendations regarding the ratio between the length of the flow path and the thickness of the part. In the case of PMMA, specialists recommend a ratio of 110:1 to 170:1 [6].

In the case of injection at a single, centrally located point, the value of this ratio is 229.831 (with a flow front length of 655.5 mm), which means that a single point is not enough. If 3 injection points are chosen, the ratio is 188.718 (shortest flow front length = 535.5 mm), a considerably better ratio, but not within the recommended range. Choosing 5 injection points, the ratio of 107.606 (with the length of the shortest flow front = 292.9 mm) represents the value closest to the recommendations, which can be seen in Figure 3.

In conclusion, for this part, 5 injection points is the optimal value. Regarding the location of these points, the specifics of the part must be taken into account (being an appearance part, they cannot be placed on visible areas), so the location of the injection points will be on the side, on the assembly elements provided on the part.

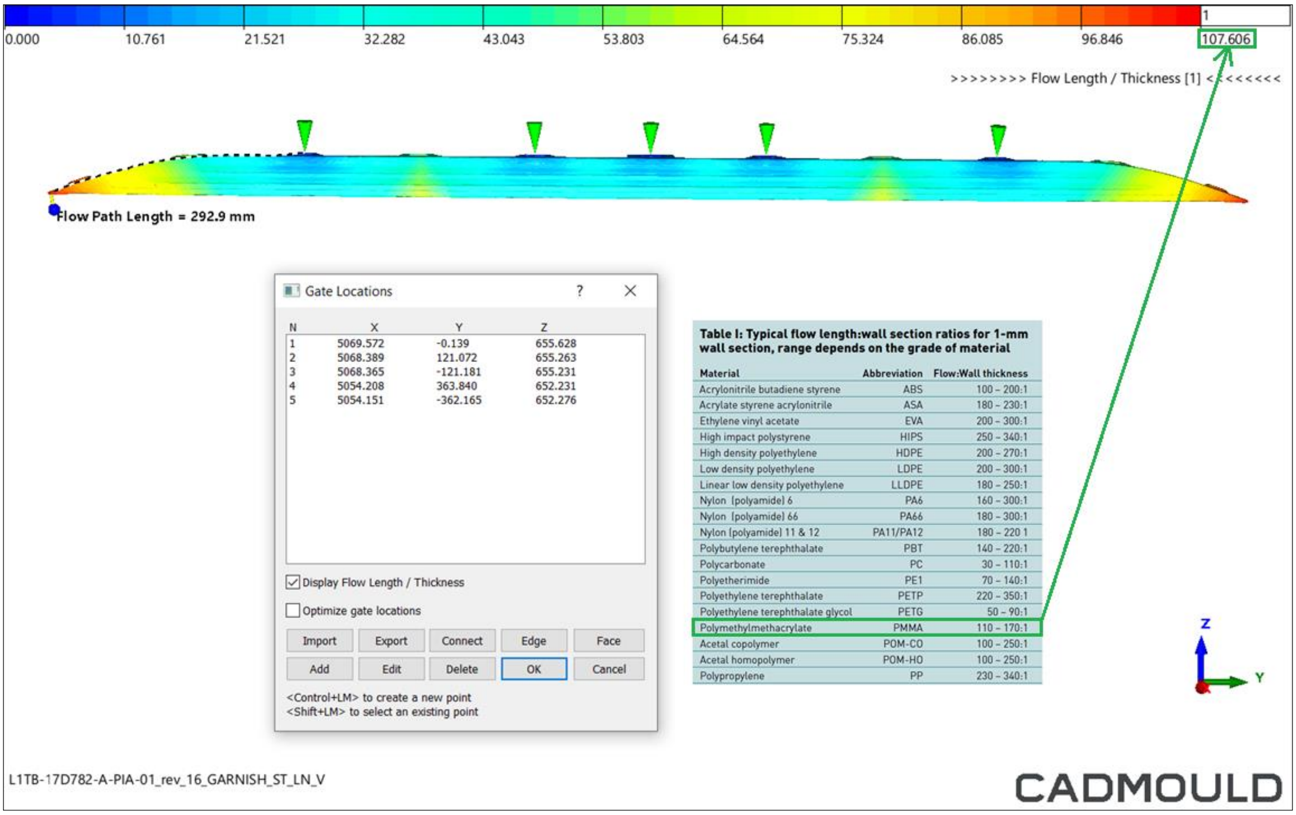

Figure 3. The flow path / wall thickness ratio in the case of the 5-point injection variant

2.2. Injection system design

After the number of injection points has been established, respectively their location on the part, the injection system will be designed. In order to have the best possible control of the entrance of the material fronts into the mold cavity, but also taking into account the specifics of the material (in this case PMMA), injection through a hot runner system mounted in the mold is recommended. The distribution network in the mold is composed of several elements (segments) that will be defined within the simulation program: the sprue represents the segment that connects the injection machine and the distribution channels that lead the material to the hot nozzles through which inject the part.

As can be seen in Figure 4, the injection system consists of a hot runner network, consisting of the nozzle entering the mold (according to the specialized literature [7], but also the experience with other similar parts, the chosen diameter = Φ8 mm), a series of distribution channels (runners) with diameters Φ18 mm, respectively Φ15 mm, some cold networks of the tunnel channel type, which connect the hot distribution block and the part itself.

The profile of the injection gate at the end of the tunnel channel was determined based on the nomograms recommended by the manufacturers of hot nozzles [8]. There is also the option of choosing a curved tunnel gate, but considering that the part is not injected under the visible surface, but on the side of the assembly elements, the ideal solution is given by the use of the simple tunnel channel gate.

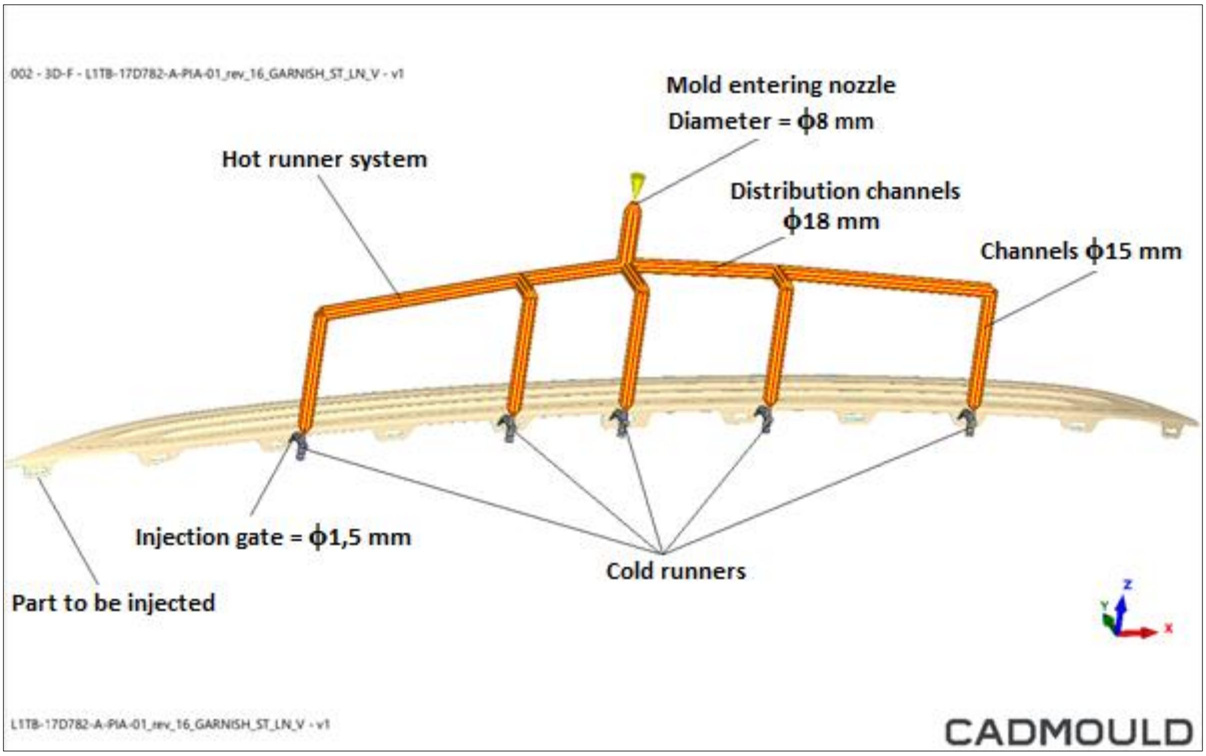

Figure 4. Injection system in the mold

2.3. Process parameters

The next step is to establish the process parameters. In the present case, the program has recommended certain default parameter values based on the input data entered up to this stage. The recommended filling time is = 1.828 s. It should be noted that all these parameters will be set to the same values, both in the case of sequential injection and in the case of direct injection.

The switch to holding phase will be made when 99% of the mold cavity is filled within the dynamic injection phase. Switching can be anywhere between 95% - 100% depending on several criteria such as e.g. the nature of the polymer (semi-crystalline / amorphous).

The melting temperature on the plasticizing cylinder (215°C), on the hot distribution network (230°C), respectively in the mold walls (80°C) are recommended by the material manufacturer. In the simulation, the cooling is purely theoretical, a constant tempering of the cavity walls is considered, which is impossible in practice, where a series of variations appear due to the thermal conductivity of the steels. The temperature when ejecting the final part is estimated at around 95°C. As a compaction time, the part will be pressed for 8 s, after which it goes to the cooling phase, for 30 s.

Considering the fact that the part will be filled sequentially, but also taking into account the location of the injection points, the recommendation is to use constant injection speeds, respectively constant holding pressures. In the case of direct injection, the same recommendations will be kept, in order to highlight certain differences. The centralization of all default process parameters re-commended by the program can be viewed in Figure 5.

MATERIALE PLASTICE
https://revmaterialeplastice.ro
https://doi.org/10.37358/Mat.Plast.1964

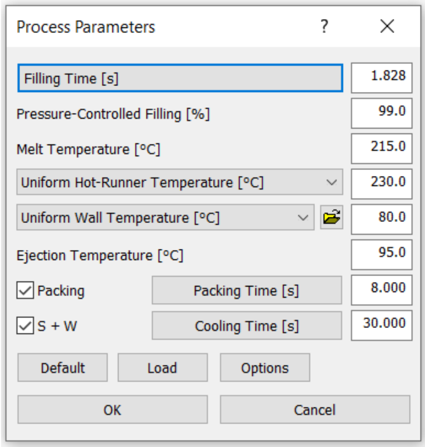

Figure 5. Parameters of the process

2.4. Applying pressure sensors

In order to simulate the sequential filling process as well as possible, a piezoelectric pressure sensor will be placed in the mold, next to the injection points 2 and 4. The location of these sensors can be seen in Figure 6. Thus, when the sensor next to point 2 detects the passage of the front, the opening of the nozzle 3, arranged symmetrically with respect to point 2, will be commanded. The same work procedure for point 4, when the sensor detects the passage of the front, the opening of the valve 5 will be commanded.

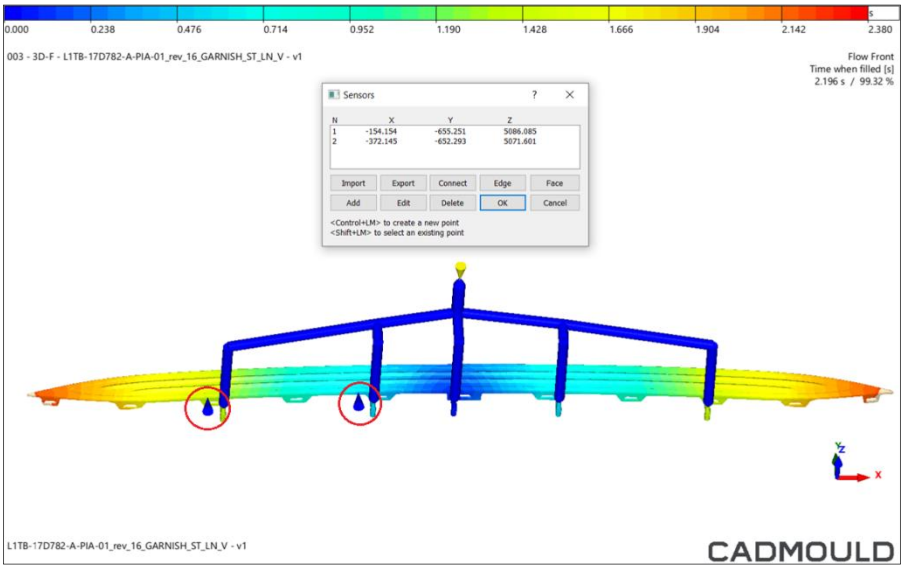

Figure 6. Application of pressure sensors to sequential injection

3. Results and discussions

3.1. Part filling analysis

The way in which the material melt flows in the mold cavity has a particularly great importance on the quality of the injected part. The melt of plasticized material flows into the mold due to the pressure applied by the screw of the injection machine, which in this phase works like a piston on which the piston of the hydraulic injection cylinder acts, inside which the injection hydraulic pressure (Hp) acts.

The moment the plastic melt comes into contact with the side walls of the mold cavity, it starts to cool immediately, resulting in a layer of cooled plastic material that forms a kind of heat-insulating layer of material, which coats the melt inside the mold. This is why the temperature in the center of the wall thickness of the melt is much higher than at the edges, so the fluidity of the melt is much higher in the center (relative to the edges).

Thus, different flow velocities appear between the layers in the center and those on the outside, which determines a flow front similar to a balloon wrapped in a well-stretched membrane, also known as the Fontaine effect (or "fountain" effect). The portion between the melt advancing inside the mold and that still unfilled is called the melt flow front. Behind the flow front, the melt pushes forward into the mold, causing the front to tense up and expand like a balloon [9,10].

In Figures 7 and 8 it can be observed how this flow front advances in the wall thickness of the part, both in the case of the injection variant without activating the sequential system, where we practically have 5 melt advance fronts, fronts that will meet and will generate welding lines; respectively in the case of the sequential injection variant, where we have a single melt flow front, fed by the progressive opening of the valves of the hot runner injection system.

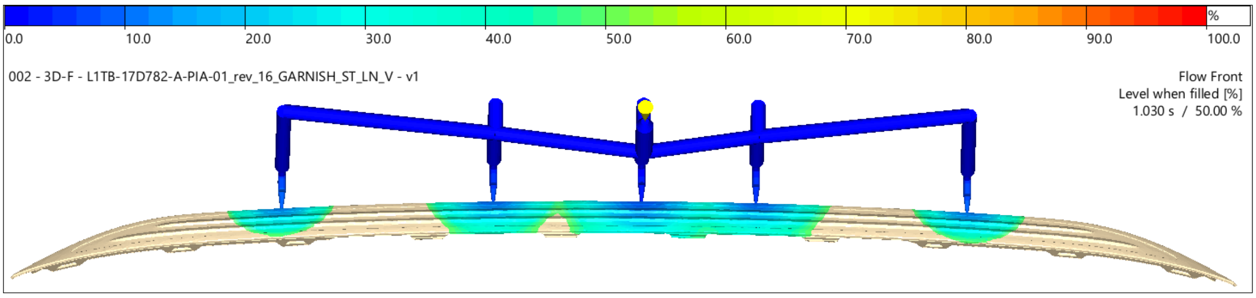

Figure 7. Part fill on non-sequential injection - level 50%

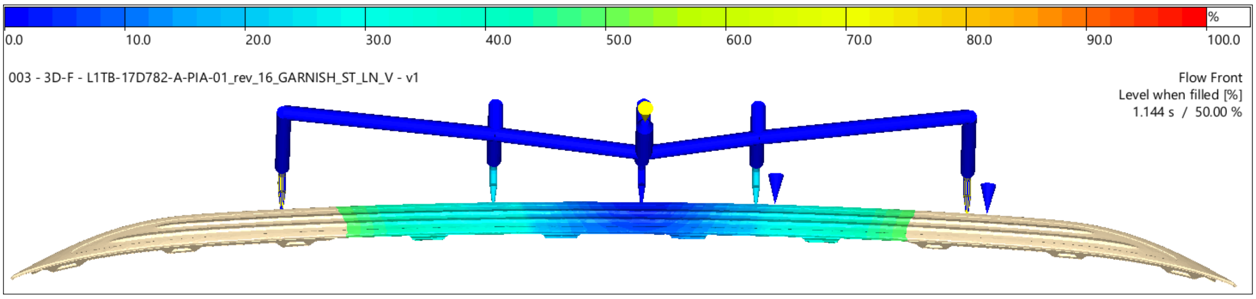

Figure 8. Part filling on sequential injection - level 50%

When the melt of plasticized material flows in the mold cavity, a series of phenomena and effects specific to the flow can be encountered, such as:

- the phenomenon of freezing: if the advance speed of the flow front is too low, respectively the contact time of the plastic material melt with the mold walls is reduced, a pronounced cooling of the melt will take place, which leads to increasing the thickness of the solidified superficial layer, or even up to the solidification of the entire wall thickness of the injected part (freezing effect); as a result the process will generate incomplete parts (or material missing parts).

For example, in some cases, due to the shrinking of the flow section at the same injection rate there will be a sharp decrease in the advance speed of the flow front in the thinner sections. As a result, the front freezes, leading to incomplete parts [9,10].

In the case of this part, the wall thickness is large enough not to lead to freezing of the flow fronts, which is why we will have full parts at the end of the dynamic injection phase in the mold (100% level), both in the non-sequential version (Figure 9), as well as in the sequential version (Figure 10).

MATERIALE PLASTICE
https://revmaterialeplastice.ro
https://doi.org/10.37358/Mat.Plast.1964

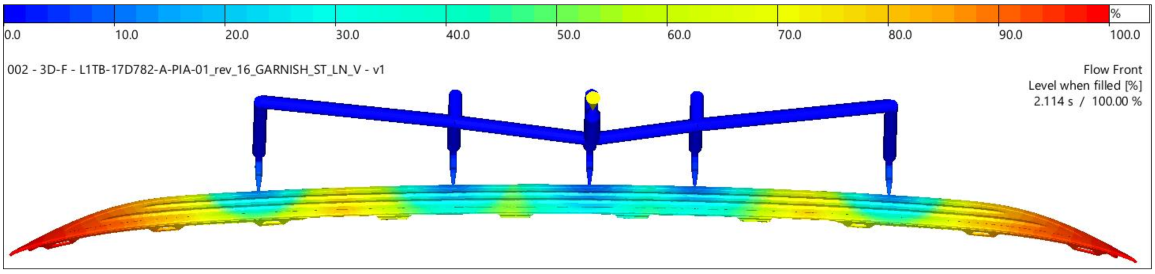

Figure 9. Part fill on non-sequential injection - level 100%

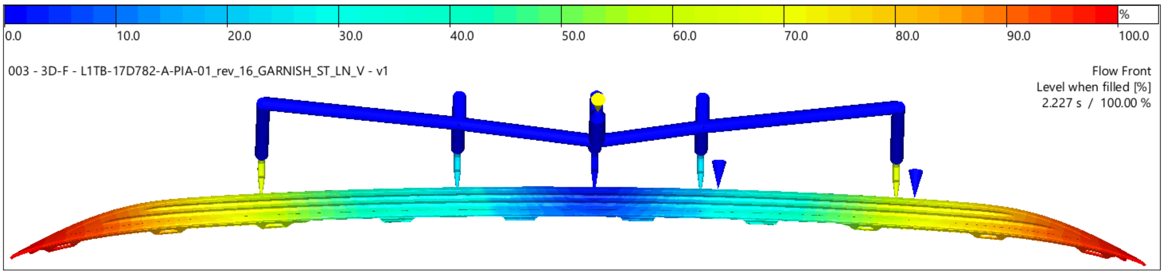

Figure 10. Part filling on sequential injection - level 100%

3.2. Temperatures profile during part filling

In the part fill simulation, the flow front temperatures indicate the temperature of the polymer when the advancing front reaches a specific point in the center of the part section.

The temperature of the flow front should not vary more than 2-5°Cduring the filling phase. Larger variations often indicate that the injection time is too short or that there are areas where the hesitation phenomenon may occur. If the front temperature is too low in a thin area on the part, the hesitation phenomenon can lead to an incomplete or missing part [11].

If one opts to move the gates out of areas with hesitation problems, this only moves the problem to other areas on the part. Increasing the temperature of the mold is not an optimal solution either, as it increases the cooling time [12].

Figure 11 indicates that by simultaneously opening all 5 valves, this will result in somewhat narrower areas of reduced temperature, which is why weld lines will appear in the marginal areas of these intervals. This is not true for sequential injection, since, as can be seen in Figure 12, these low-temperature ranges represent more continuous temperature dispersion environments.

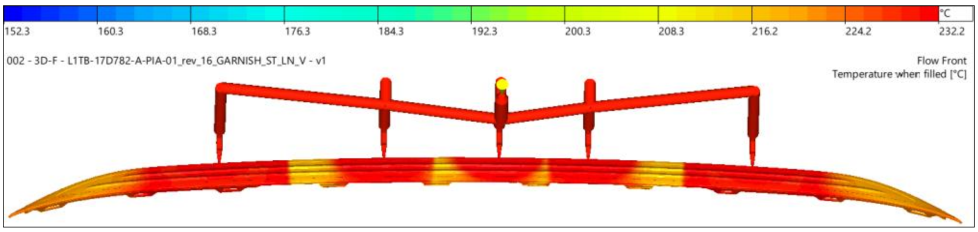

Figure 11. Temperature profile when filling the part – non-sequential version

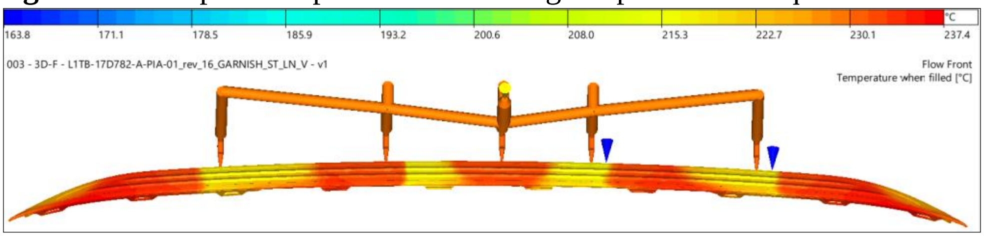

Figure 12. Temperature profile when filling the part – sequential version

3.3. Pressure loses

As the plastic flows into the mold, a series of phenomena occur that disrupt the flow phenomenon.

When the melt enters the distribution channels in the mold, it has a temperature T, respectively a pressure P. The mold is much colder than the plastic material, the temperature being usually set around the material's crystallization point (for crystalline materials).

As the plastic enters the channel, certain front pressure drops occur due to friction. The new pressure becomes P-ΔP. In addition to this, as the melt hits the walls of the cavity, it solidifies. Thus, the new temperature becomes T-ΔT. These phenomena lead to an increase in the viscosity of the material, which requires an increase in the applied pressure. The surface layers that form on the mold walls thin the section of the flow front. All of these factors indicate that by increasing flow distance and fill time, the plastic requires increased pressure to maintain constant injection rates. This causes a reduction in pressure in front of the flow front, which we actually call pressure loss.

The injection machine has some default maximum values of pressure and injection speeds that can be set. It is quite obvious that the pressure required to push the screw should never be greater than what the injection machine can provide [13].

In Figure 14, all 5 pressure drops specific to each progressive opening of a valve are highlighted very well, while in Figure 13, in the case of non-sequential injection, the pressure drops could not be highlighted as visibly.

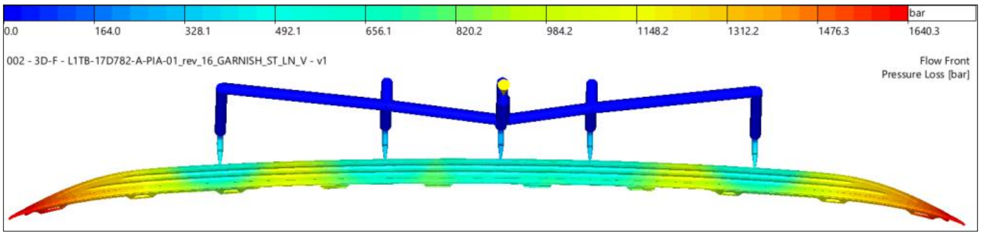

Figure 13. Pressure drops in non-sequential injection

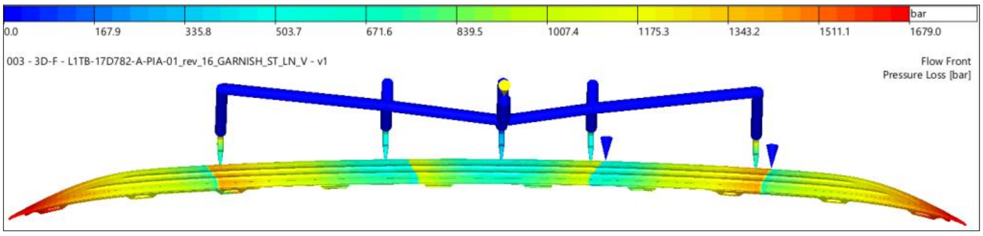

Figure 14. Pressure drops in sequential injection

3.4. Speeds profile at part filling

During the filling phase of the injection process, the speed of the melt flow fronts has a significant influence on the quality of the final part. The flow front is defined as the section of the advancing front: either the length of the melt multiplied by the wall thickness of the part, or the section of the distribution channels, or their summation if the melt must travel through both areas [14].

At any instant, the product of the velocity and the area of the flow front results in the volume of the flow front. For any mold with a complex cavity geometry, a constant injection rate does not necessarily cause the flow front to advance at a constant rate. At any variation of the wall thickness section, the velocities of the fronts also vary, with some areas being filled faster.

During the filling phase, the polymer molecules, respectively the reinforcing fibers (glass, talc, etc.) in the material structure, will orient in a direction influenced by the shear rate of the flow front. As the melt enters a much lower temperature cavity, it freezes almost instantly upon contact with the mold walls [19]. The orientation of the molecules and fibers will be influenced by the flow dynamics, which is why it is recommended to obtain constant velocities of the flow front from the process, in order to generate homogeneous orientations of the molecules.

This inconstancy of the speeds of the flow fronts can be observed in Figure 15, there are very small portions that flow a little faster, even in the areas where we will highlight on the physical part the appearance of the welding lines, while in Figure 16 we can see that the speed of filling is as constant as possible.

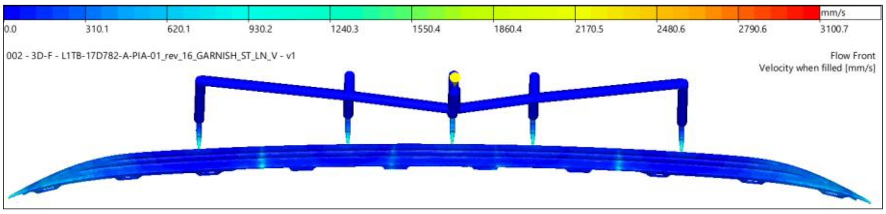

Figure 15. Profile of filling velocities for non-sequential injection

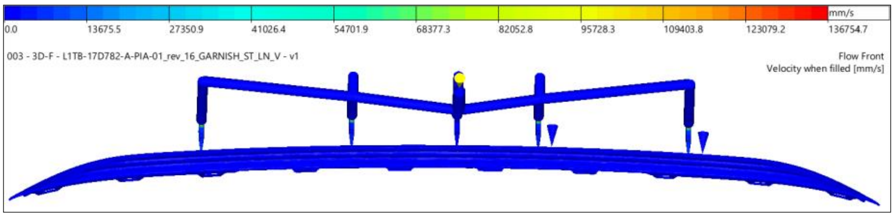

Figure 16. Profile of filling velocities for sequential injection

3.5. Sequential injection

Below, it can be seen the valve control panel for sequential injection (Figure 17), respectively the graph of injection pressures and speeds (Figure 18). The pressure drops on the green line are due to the decrease in flow distance and fill time with each valve opening. Valves 2 and 4 open at the same stroke (32 mm), while there is a slight difference between valve 1 (14mm) and valve 5 (12mm) due to the defalcated positioning of the central injection point on the assembly elements of the part.

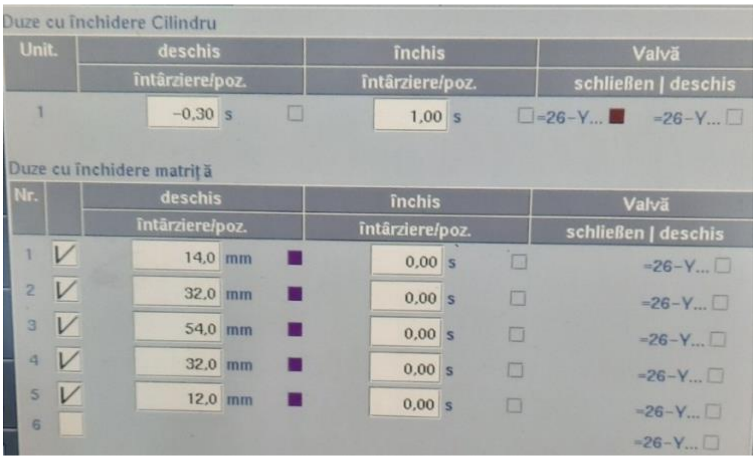

Figure 17. Sequential injection valve control panel

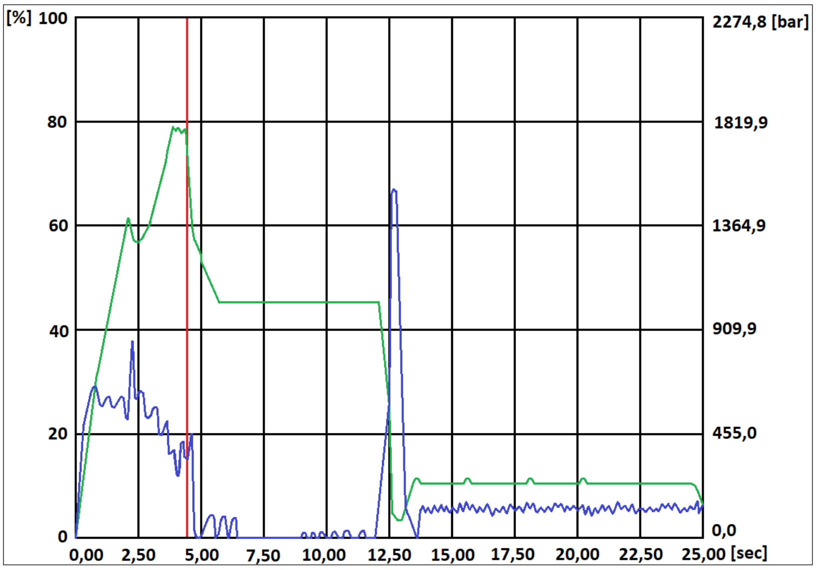

Figure 18. Plot of pressures (green) and injection rates (blue) for sequential injection

3.6. Non-sequential injection

Figure 19 shows the valve control panel for non-sequential injection. Virtually all valves will be open at the same piston stroke (54 mm). In the case study, 3 batches of parts with 2 parts per batch will be injected:

- first batch: 2 parts injected sequentially;
- second batch: 2 parts injected non-sequentially;
- the third batch: 3 parts injected non-sequentially, but with reduced injection speeds (Figure 20);

After the actual injection, each part will be inspected to be able to highlight how the chosen injection system influences the quality of the final part. Attention will be focused on two elements: defects in appearance (welding lines), respectively the dimensional variation of the parts, as well as weights. In the case of the injection of batch 3 of parts, it was found that by reducing the injection speeds, the holding pressure tended to decrease towards the minimum ("zero" pressure). This is explained by the fact that the PMMA from which the studied part is injected is a material that freezes quickly. As a result, the part will fill in terms of volume, but due to the inhomogeneity caused by the presence of prematurely frozen layers, the parts will not be as dense, which was noticed by comparing the weights of the parts in this batch with the weights of the parts in the first two batches. In this work we will only focus on the welding lines, as well as the impact they have on the final part.

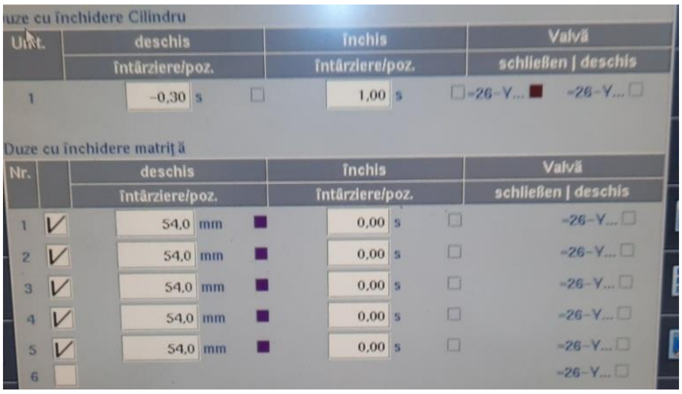

Figure 19. Valve control panel for non-sequential injection

Figure 20. Dropping speeds when injecting the third batch of parts

3.7. Aspect / appearance defects

MATERIALE PLASTICE
https://revmaterialeplastice.ro
https://doi.org/10.37358/Mat.Plast.1964

If no welding lines were detected on the sequentially injected part, they could be seen with the naked eye on the parts from batch 2 and 3 (Figure 21), being somewhat more pronounced on the landmarks from batch 3 (Figure 23). In order to better highlight these appearance defects, optical instruments were used to enlarge the area where they appear. It was also found that in the area of the ends of the tunnel channels on the injection network, the Diesel effect is manifested (Figure 22), due to inadequate ventilation.

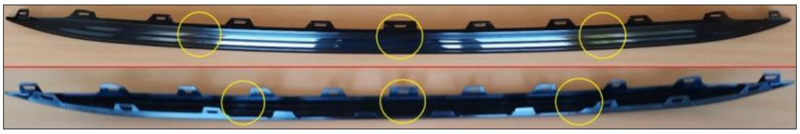

Figure 21. Highlighting areas with weld lines on the part (front and back view)

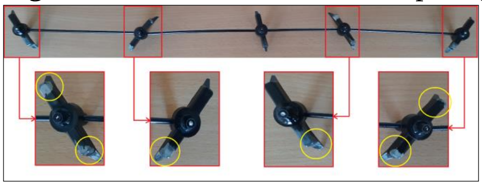

Figure 22. Highlighting burnt areas on the injection network

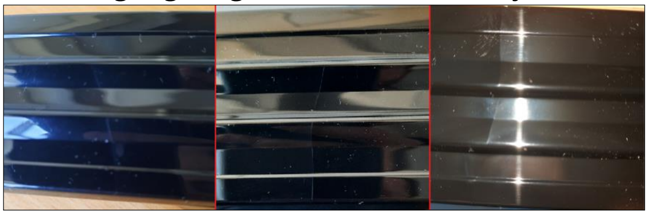

Figure 23. Weld lines visible to the naked eye on the surface of the injected part

3.8. Elongation testing of parts

In order to highlight the mechanical properties, 2 elongation tests were performed, up to 0.15 mm (Figure 26 - non-sequential, Figure 27 - sequential), respectively up to 0.30 mm (Figure 28 - non-sequential, Figure 29 - sequentially), to see how the loads graph varies by injection type. It was found that in order to elongate the sequentially injected parts, higher loads are required, which means that they are mechanically stronger. This is due to the presence of weld lines on non-sequentially injected parts, areas that weaken the mechanical strength in the areas where they appear. These tests were performed on 4 samples of length l = approx. 100 mm (an example of a specimen can be seen in Figure 25), obtained by sectioning the parts: 2 specimens each with welding lines in the middle, respectively 2 specimens without welding lines. Figure 24 shows the apparatus on which these elongation tests were performed.

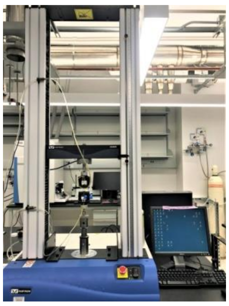

Figure 24. INSTRON 3365 Universal Tester [20,21]

Figure 25. The specimen of length l = 100 mm to be tested at tension

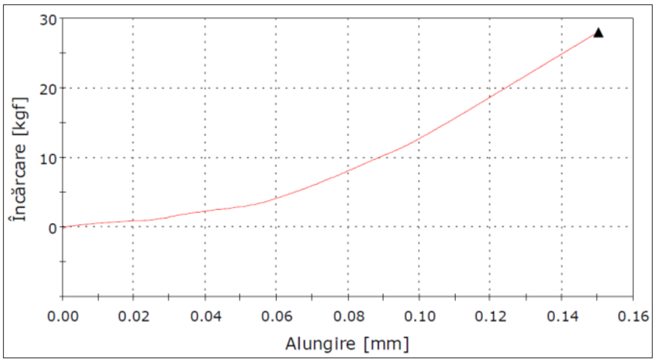

Figure 26. Load required for 0.15 mm elongation of specimen A1 (injected non-sequentially)

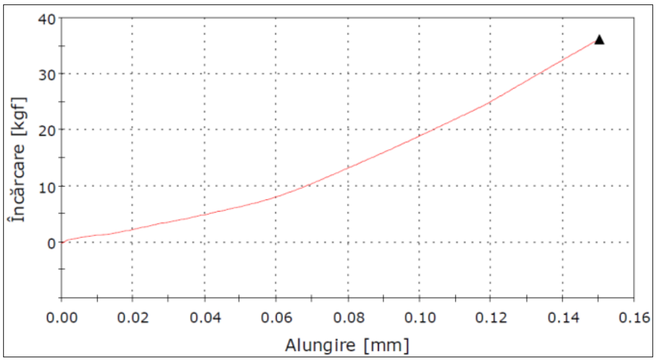

Figure 27. Load required for 0.15 mm elongation of specimen A2 (injected sequentially)

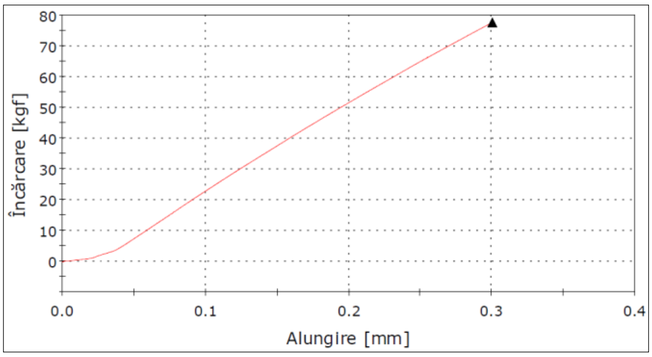

Figure 28. Load required for 0.30 mm 0.30 mm elongation of specimen B1 (injected non-sequentially)

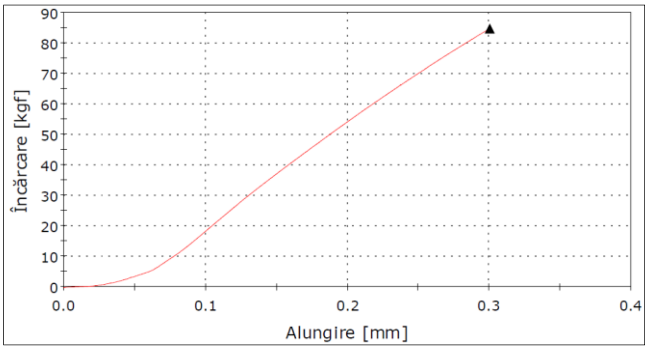

Figure 29. Load required for 0.30 mm 0.30 mm elongation of specimen B2 (injected sequentially)

4. Conclusions

It can be concluded that the simulation correctly indicated the possibility of weld lines in the case of the non-sequential injection variant, also accurately suggesting their location. These defects are inconvenient for two reasons: first of all from the point of view of their appearance on the visible and glossy areas, but also from the perspective of the fact that these areas present weaker mechanical characteristics, a fact demonstrated by the elongation tests performed. Due to the correct location of the injection points, respectively the good design of the wall thickness of the landmark, other defects, such as incomplete parts due to the freezing of flow fronts in thin sections, or deformations due to pressure loses, could not be observed during the tests.

References

1.DANG, X.P., *General frameworks for optimization of plastic injection molding process parameters*, published in *Simulation Modelling Practice and Theory*, Vol. 41, 2014, p. 15-27

2 ZHAI, M., XIE, Y., *A study of gate location optimization of plastic injection molding using sequential linear programming*, published in *Int. J. Adv. Manuf. Technol.*, Vol. 49, 2010, p. 97-103

3.XIE, L., ZIEGMANN, G., *A visual mold with variotherm system for weld line study in micro injection molding*, published in *Microsystem Technologies*, Vol. 14, 2008, p. 809-814

4.XIE, L., ZIEGMANN, G., *Influence of processing parameters on micro injection molded weld line mechanical properties of polypropylene (PP)*, published in *Microsystem Technologies*, Vol. 15, 2009, p. 1427-1435

5.KITAYAMA, S., TAMADA, K., TAKANO, M., AIBA, S., *Numerical optimization of process parameters in plastic injection molding for minimizing weldlines and clamping force using conformal*

cooling channel, published in *Journal of Manufacturing Processes*, Vol. 32, 2018, p. 782-790
6.****Injection World Magazine* (February, 2011, p. 39-41). Available online: http://www.injectionworld.com/Archive.aspx
7.****HASCO – Hot Runner Technology Brochure*. Available online: https://media.hasco.com/marketing/Content/Mediathek/Broschueren/Bedienungsanleitung_HR/HK_Technik_BDA_EN.pdf
8.****Meusburger – Gate Types (Anschnittvarianten) Brochure*. Available online: https://ecom.meusburger.com/files/pdf/e/e4000_info.pdf
9.ILIE, S., Injectarea materialelor termoplastice, Plastor, Oradea, 2010
10. ȘEREȘ, I., Injectarea materialelor termoplastice, Editura Imprimeriei de Vest, Oradea, 1996
11. PANTANI, R., DE SANTIS, F., BRUCATO, V., TITOMANLIO, G., *Analysis of Gate Freeze-Off Time in Injection Molding*, Polymer Engineering and Science, 2004
12.***Autodesk Knowledge Network. Available online: https://knowledge.autodesk.com/support/moldflow-insight/learn-explore/caas/CloudHelp/cloudhelp/2019/ENU/MoldflowInsight-Results/files/GUID-2BC51F30-583F-4E5C-8824-1BF7BAF0A144-htm.html
13.***Distinctive Plastics – Technical Injection Molding Services. Available online: https://www.dpi-tech.com/Whitepapers/details.asp?Title=Introduction-to-Pressure-Drop-in-Injection-Molding&id=5
14. ***Wiley Online Library. Available online: https://onlinelibrary.wiley.com/doi/abs/10.1002/pen.25092
15. NEDELCU, D., FETECĂU, C., CIOFU, C., MÎNDRU, D., Aspects Regarding the Use of FEM for Calculus at the Injection Moulding of a High Accuracy Part., *Mater. Plast*., **46**(3), 2009, 269-273
16. ISOPESCU, R., POSTELNICESCU, P., Effect of Cooling Agent on Temperature Profile During Molding Injection: Case Study for Polylactic Composites., *Mater. Plast*., **58**(1), 2021, 228-236 https://doi.org/10.37358/MP.21.1.5462
17. POP, A.P., ILEA, S., *Optimization of the injection moulding process for plastics parts by using FEA.*, Annals of the Oradea University, Fascicle of Management and Technological Engineering, Vol. XXXI**,** Nr. 2, 2022
18. FETECĂU, C., STAN, F., Computational Prediction of Defects During Injection Molding in a Complex Part., *Mater. Plast.*, **44**(3), 2007, 180-184
19.*** Autodesk Knowledge Network. Available online: https://knowledge.autodesk.com/search-result/caas/CloudHelp/cloudhelp/2015/ENU/MoldflowInsight360/files/GUID-099634AE-DB7A-41BA-B70C-5A23FB013B06-htm.html
20.***UC Irvine Materials Research Institute. Available online: https://imri.uci.edu/facilities/tempr/tempr-instruments/instron3365/
21.***Instron: Materials Testing Machines. Available online: https://www.instron.com/-/media/literature-library/products/2011/06/3300-series-table-model.pdf

Manuscript received: 31.01.2023