BULETINUL INSTITUTULUI POLITEHNIC DIN IAŞI
Publicat de
Universitatea Tehnică „Gheorghe Asachi” din Iaşi
Volumul 64 (68), Numărul 4, 2018
Secţia
CONSTRUCŢII DE MAŞINI

FINITE ELEMENT ANALYSIS IN OPTIMIZING COMPOSITE STRUCTURE INJECTION USED IN AUTOMOTIVE INDUSTRY

BY

IONUȚ CRÎȘMARU[1,*], MARIAN BOROȘ[2] and
CĂTĂLIN GABRIEL DUMITRAŞ[1]

[1]“Gheorghe Asachi” Technical University of Iaşi, Romania,
Department of Machine Tools
[2]Continental Automotive Iași

Received: November 15, 2018
Accepted for publication: December 19, 2018

Abstract. The objective of the study is to analyze and redesign the geometry of a composite material casing. The model considered is a multi-output module designed for power distribution and control, favoring better rationalization of the load. These types of modules come equipped with a range of workload protection (OCP, OVP, UVP, etc.). The reference support model used in the analysis is intended to equip a vehicle. The model under analysis is an adaptation of the reference model. Finite Element Analysis will allow you to determine the injection point and casing structure.

Keywords: composite; finite element; analyze; casing.

1. Introduction

In order to design plastic parts, the uniform thickness of the wall is critical. The variation of this thickness causes great problems in the deformation of the workpiece and the increase in dimensional imprecision.

To increase mechanical strength and rigidity the injected parts are often designed with stiffening ribs. The nerves are predominantly designed for thin

---

[*]Corresponding author; *e-mail*: crismaru.ionut@yahoo.com

and large wall parts. They are used both to strengthen the side walls and the bottom (Șereș, 1996). In Fig. 1 one present several types of nerves.

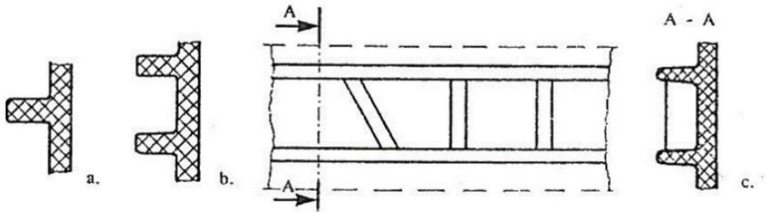
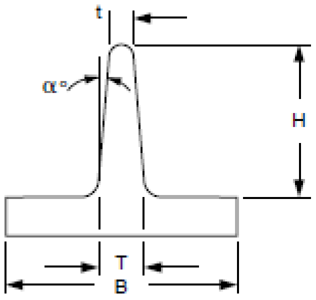


Fig. 1 – Longitudinal and transversal nerves
a) simple nerve b) double nerve c) Transversal and longitudinal nerve.

The calculation of the minimum thickness of stiffening ribs is shown in Fig. 2.



$t = T - 2H * tan\alpha$ (DuPont, 2008)

Fig. 2 – Minimum width of the stiffening ribs (DuPont, 2008).

Currently, the thickness of the ribs varies between 0.5 and 0.8 of the piece thickness. The reason for using a thicker layer than the wall is to prevent backslides. In the case of keeping the same thickness as the wall and applying at the base of the rib two connection rays with a value of 0.5 T, a circle encircled in the material of 1.5 T results in the base of the rib, is strictly in this area the thickness of the material is 50% greater than that of the piece, which favours the appearance of retards, Fig. 3 (DuPont, 2008).

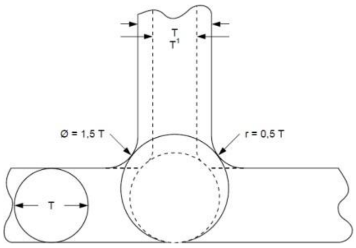


Fig. 3 – Dimensioning of stiffening ribs (DuPont, 2008).

If the piece has a good surface appearance, the use of stiffening ribs will cause local contraction of the material, resulting in the appearance of retrievals. The same type of local contractions also arises if the corners of the parts are left sharp. The variation in the thickness of the material will cause uneven contraction in the direction of the maximum material or the occurrence of the deformations. For the injected parts, which require special aesthetic conditions, these retouches can be masked with smaller ribs or decorative grooves Fig. 4 (Econology Ltd, 2004).

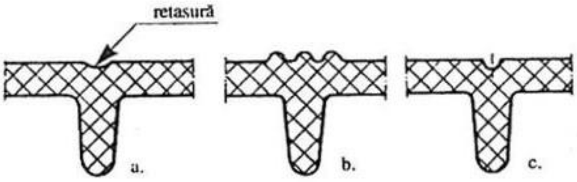


Fig. 4 – Construction solutions for hiding retches:
a) Retaining piece; b), c) correct constructive solutions (Şereş, 1996).

To prevent this problem, it is recommended to use the connections at the intersection of the walls both inside the workpiece and on the outside. The general formula for the value of the connection rays is as follows:

$$r = R - g; \tag{1}$$

inner radius; R – radius; g – material thickness of the wall of the piece.

It is recommended that the ratio of the connection radius to the wall thickness is not less than 0.6 ($r / s \geqslant 0.6$) (Șereș, 1996).

A similar situation is also presented in the design guide of DuPont, Fig. 5.

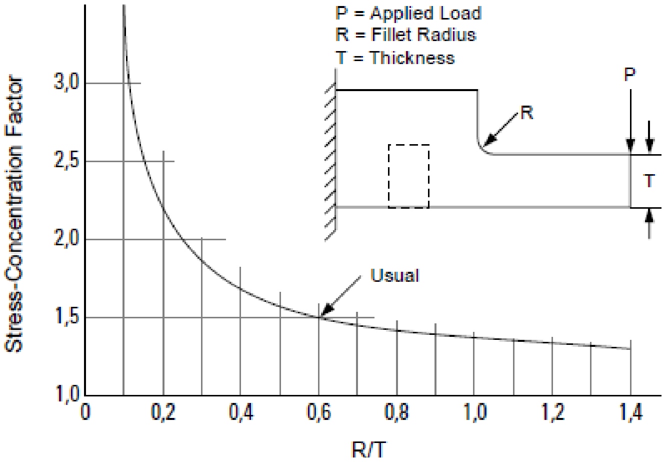


Fig. 5 – The variation of the voltage concentrators depending on the ratio of the applied connecting rays and the thickness of the piece (DuPont, 2008).

In the case of the joining of two walls of different thickness, the relation of calculation of the connection rays, according to Fig. 6, becomes:

$$r = (S_1 + S_2)/2 \tag{2}$$

$$R = S_1 + S_2 \tag{3}$$

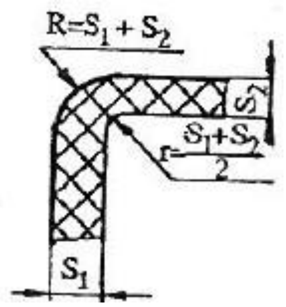


Fig. 6 – Calculation of the connecting rays for variable thickness of the injected piece wall (Şereş, 1996).

If the thickness of the wall cannot be kept constant for functional reasons, the transition from one thickness to the other is recommended to be achieved progressively, either by sloping surfaces or by applying rays. The sudden section change causes inside the workpiece internal stresses due to voltage concentrators or turbulence in the injection process, Fig. 7 (Şereş, 1996).

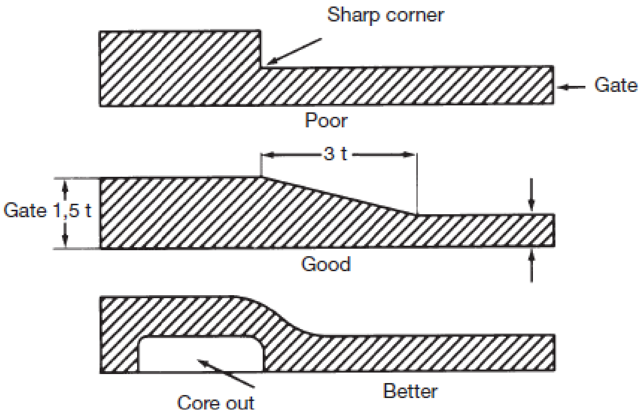


Fig. 7 – Wall thickness variation of an injected piece (DuPont, 2008).

For parallelepiped injected parts are proposed constructive solutions for the shape of the upper board, Fig. 8, respectively Fig. 9 for the bottom of the fish (Şereş, 1996).

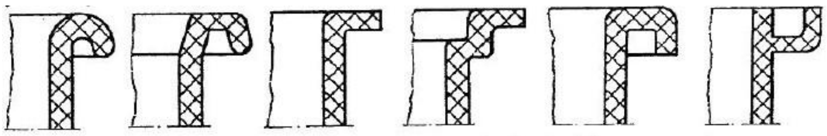


Fig. 8 – The shape of the upper board of the injected parts (Şereş, 1996).

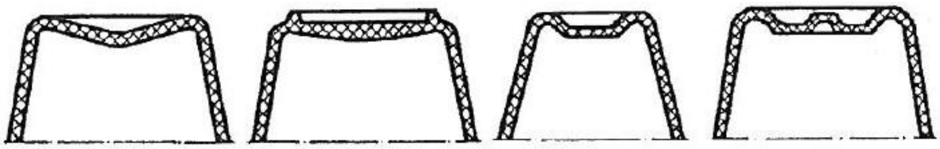


Fig. 9 – Constructive solutions for the bottom of injected pieces of parallelepiped shape (Şereş, 1996).

In the case of crystalline thermoplastic materials, there are large differences between longitudinal and transverse contraction, especially in the case of composite materials containing fibre reinforcement materials. For an injection moulded parallelepiped shape with a central injection point, the wider ribbing placed on the medial line of the lateral walls allows the plastic to reach first in this area, thus reducing the difference between the two types of contractions, Fig. 10 (Şereş, 1996).

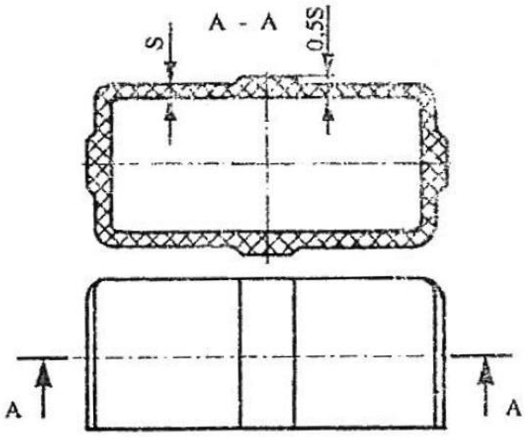


Fig. 10 – Side walls with central nerve (Şereş, 1996).

2. Considered Solution

To run an analysis in Moldflow, one consider a 3D model that must be converted to a finite number of elements. Each finite element is defined by a number of nodes. The purpose of this geometry is to be able to calculate the value of relative displacements in each network node (ECS Tuning, 2016).

These displacements can then be converted to different units of measure. This stage is called meshing.

The front model is composed of about 41,000 elements. Features of the Fusion Solver:

– Consider the laminar flow, after the generalized Newtonian fluid;

– The inertia and weight of the injected liquid is not taken into account;

– Conduction in the direction of injection is neglected in comparison with the direction of material thickness;

– The thermal convection in the sense of the thickness of the material is neglected;

– The loss of heat at the edge of the triangular elements is neglected (Schoemaker, 2006).

In Fig. 11 one present the meshed model.

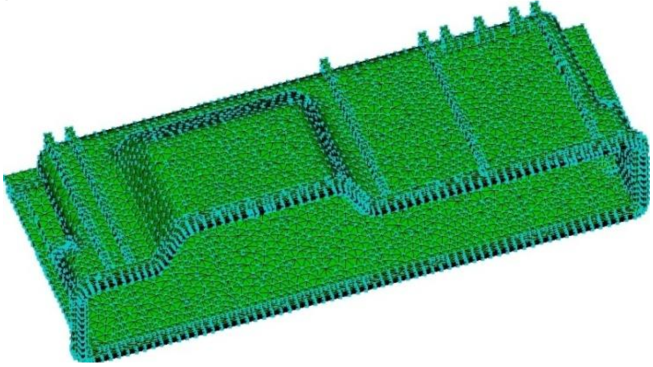
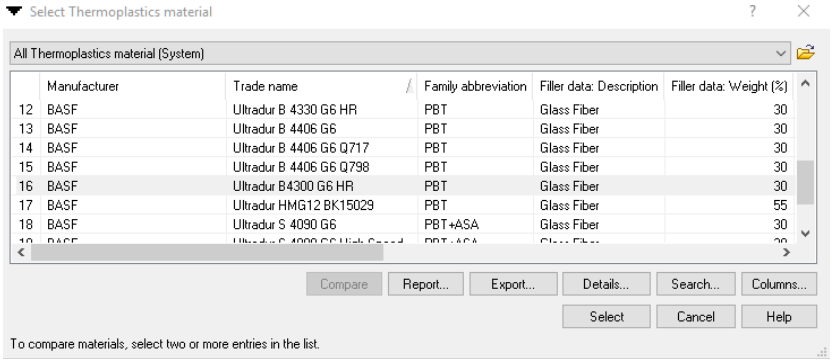


Fig. 11 – Meshed model.

3. Defining the Injection Point

Next, the thermoplastic material to be injected from the library of materials available in the program - Ultradur B 4300 G6 HR - PBT GF 30, produced by BASF (BASF, 2013), Fig. 12.



	Manufacturer	Trade name	Family abbreviation	Filler data: Description	Filler data: Weight (%)
12	BASF	Ultradur B 4330 G6 HR	PBT	Glass Fiber	30
13	BASF	Ultradur B 4406 G6	PBT	Glass Fiber	30
14	BASF	Ultradur B 4406 G6 Q717	PBT	Glass Fiber	30
15	BASF	Ultradur B 4406 G6 Q798	PBT	Glass Fiber	30
16	BASF	Ultradur B4300 G6 HR	PBT	Glass Fiber	30
17	BASF	Ultradur HMG12 BK15029	PBT	Glass Fiber	55
18	BASF	Ultradur S 4090 G6	PBT+ASA	Glass Fiber	30

To compare materials, select two or more entries in the list.

Fig. 12 – Selection of thermoplastic material.

The position of the injection point was determined following the analysis of the flow resistivity of the material due to the geometry of the piece, Fig. 13. The injection point was chosen on the vertical wall of the piece at the centre of the dark-blue area of Fig. 13. This area is defined by minimal flow resistivity, which will favour flow of material into the mould cavity. The operating principle of the main cylinder is described in the Fig. 14.

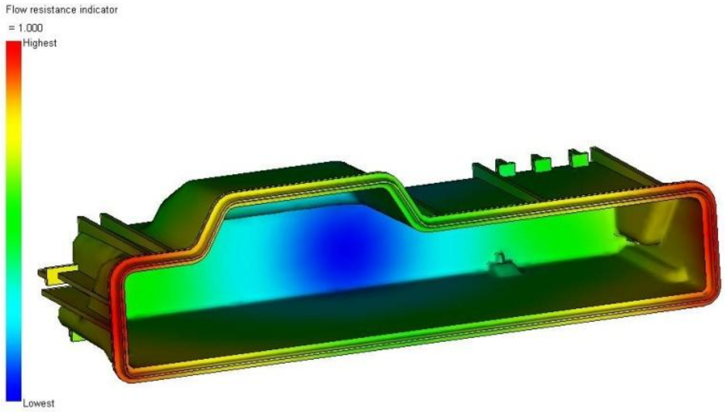


Fig. 13 – Main cylinder operating principle.

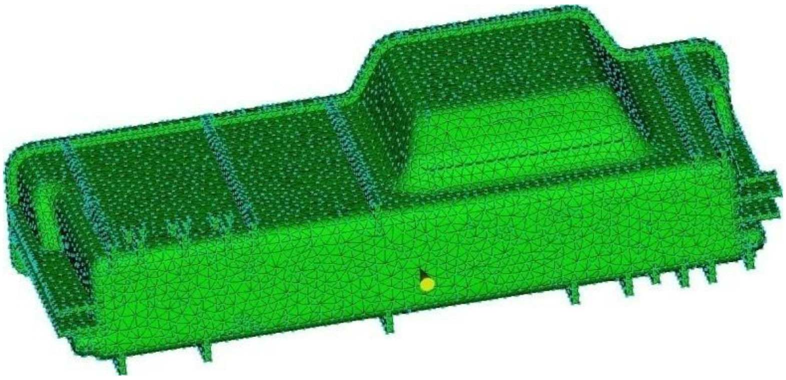
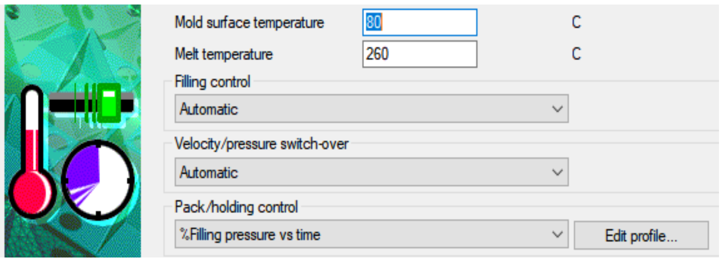


Fig. 14 – Yellow Con - position of the injection point at the workpiece level according to previous results.

4. Setting the Injection Mode Parameters and Simulating the Injection Process

To simplify this step, the injection parameters have been set to predetermined values, *i.e.* the mould temperature is kept constant at 80°C, the temperature of the injected thermoplastic material is 260°C, and the changeover from the speed control to the pressure is automatically, Fig. 15.



Mold surface temperature 80 C
Melt temperature 260 C
Filling control: Automatic
Velocity/pressure switch-over: Automatic
Pack/holding control: %Filling pressure vs time

Fig.15 – Defining injection parameters.

Throughout the study, the injection regimen parameters were kept constant to observe the effect of geometry modification on the induced deformations in the piece. To determine the deformations of the injection moulding piece, the Fill + Pack + Warp assay type was chosen (Fill + Compaction + Deformation). This type of analysis considers the pattern of the song to be uniform. From the perspective of the injection process, two 3D models for the same housing will be compared, the first model having non-optimized geometry, and the second model will be the result of series of iterations to reduce deflections.

The main element of interest for this analysis will be the shape of the outer board, geometry required for the laser welding process. In nominal geometry there is an interference between the two shells. This will represent the volume of displaced material that will ensure the unassembled assembly of the module, Fig. 16.

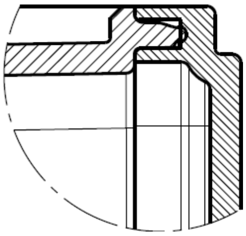


Fig. 16 – Detail. The shape of the outer board.

In addition to measuring the deformation value at the assembly area, the following will be followed:

– Actual cavity filling time;

– Temperature variation at the stage after completion of the injection process;

– Orientation of glass fibers at the end of the injection process;

– Training of retrievers;

– Intersection of faces of molten material - weld lines.

The fill time for the initial condition is 1.31 sec. The gradient indicates the time required for the material to come into contact with the surface, Fig. 17.

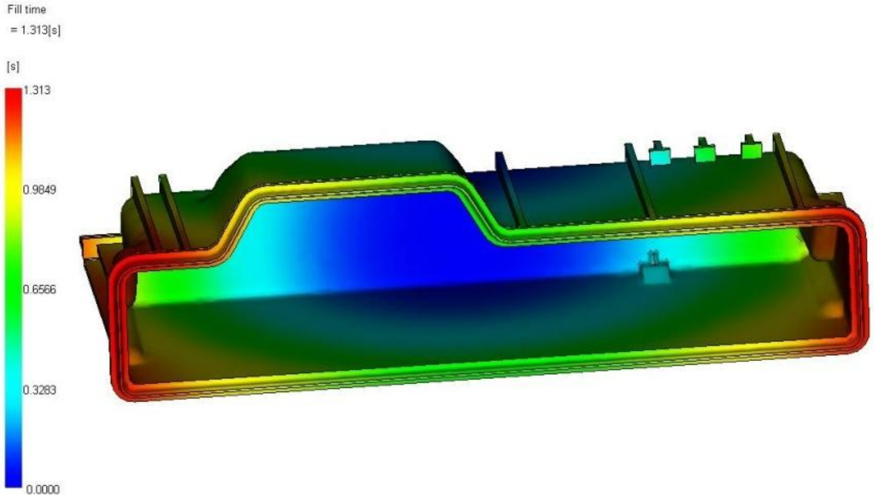


Fig. 17 – Actual cavity filling time.

The temperature of the front of the material is about 7.6°C. The front of the molten material is considered as an average surface in the section of the piece, equidistant to the exterior of the piece. It decreases from the injection point, maintained at 260°C to 252.4°C in the vicinity of the outer dash, Fig. 18.

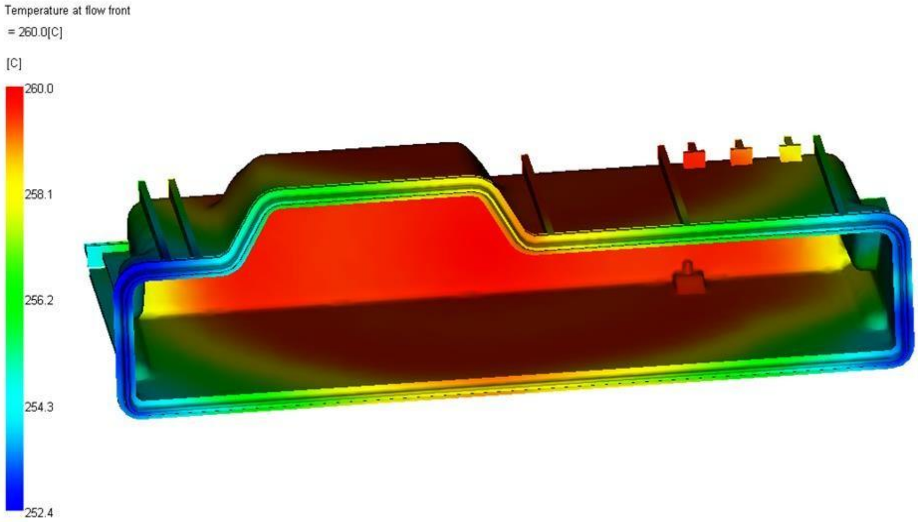


Fig. 18 – Temperature gradient of the molten material front.

In Fig. 19 the depth of the retrieval formed by the presence of stiffening ribs is estimated. The retrieval depth can reach 0.05 mm in the case of the girl.

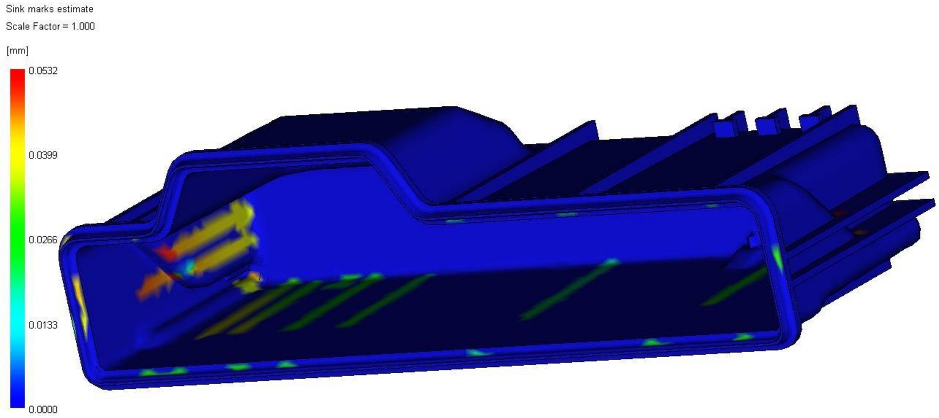


Fig. 19 – Impact of the use of stiffening ribs on the state of the inner surface, multiplicative factor 1x.

The presence of retouching in this application does not adversely affect the functionality of the module, the aesthetic aspect of the piece being second.

Fig. 20 shows the same result, in three different views, a) - axonometric view, b) top view, c) a rear view of the workpiece in order to emphasize the alignment of the glass fibers at the workpiece.

The scale of values represents the space orientation vector in relation to the axes of the absolute coordinate system of the piece. It can vary between 0 and 1. In the case of perfect alignment of the vector vector with the axle system, its value is 0.

The deformations induced in the piece will be measured in the three directions and finally the cumulated value will be presented.

The maximum total deformation criterion at the dash level is 1 mm.

For better visualization, the multiplicative factor will be individually defined for each axis to obtain a more suggestive representation of deformations. In each case, the transparent contour represents the nominal geometry of the part, the gradient being understood the direction of the deformation (red - deformation in the positive direction of the axis, blue - deformation in the opposite direction of the axis).

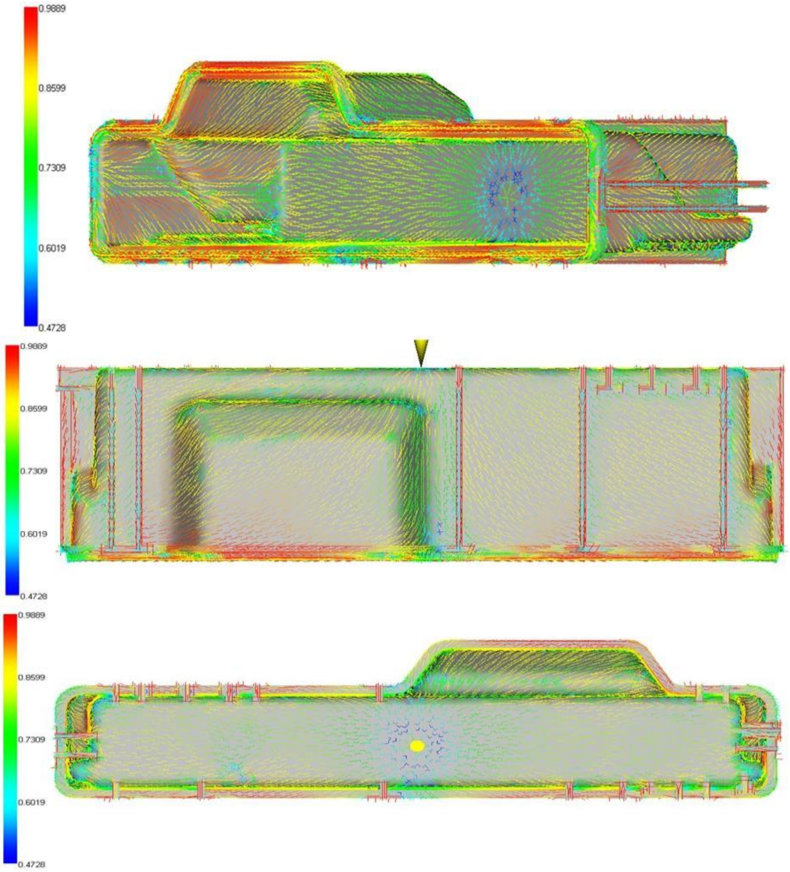


Fig. 20 – Layout of fiberglass fibers after completion of the injection process.

5. Conclusions

With Autodesk Moldflow, a series of simulations of the injection process of a fiberglass reinforced plastic material have been linked.

During the simulations, the injection parameters remained constant, the safety changes were made strictly to the geometry of the piece to observe their impact on the final product.

As a result of these simulations, the geometry of the piece has been optimized, with the following improvements being made to the original model:

– Reduction of the injection time from 1.31s to 1.21s due to the improvement of the material flow in the mold cavity;

– Reduction of the temperature gradient of the molten material front from 7.4°C to 7.2°C;

– Obtaining a more even glass fiber orientation in the mass of the injected material;

– Elimination of the joining zones of the molten material fronts from the assembly area;

– Overall decrease of the deformations in the piece by about 23% and for the assembly surface, the outer board, the maximum deformation was reduced by 59%.

Finally, I believe that this paper has achieved its goal of optimizing the geometry of a carcass by aiming to reduce the deformation values induced by the injection process below the threshold set by the 1 mm theme.

REFERENCES

BASF, *Ultradur (PBT) – Brochure*, Available at: http://www8.basf.us//PLASTICSWEB/displayanyfile?id=0901a5e1800bc1a9 (Accessed 09.06.2018) (2013).

DuPont, *DuPont Engineering Polymers General Design Principles - Module I*, Switzerland: E.I. du Pont de Nemours and Company (2008).

Econology Ltd, *Design Guides for Plastics*, Pratts Bottom: Tangram Techlonogy (2004).

ECS Tuning, *Genuine BMW - 12637591534 - Power Distribution M (12-63-7-591-534)*, Available at: https://www.ecstuning.com/b-genuine-bmw-parts/power-distribution-m/12637591534/ (Accessed 08.06.2018) (2016).

Schoemaker J., *Moldflow Design Guide*: *A Resource for Plastics Engineers***,** First Edition, Ed. Framingham (2006).

Șereș I., *Injectarea materialelor termoplastice*, Oradea: Edit. Imprimeriei de Vest (1996).

ANALIZA CU ELEMENTE FINITE UTILIZATĂ ÎN OPTIMIZAREA INJECȚIEI MATERIALELOR COMPOZITE ÎN INDUSTRIA AUTOMOTIVE

(Rezumat)

Obiectivul studiului îl reprezintă analiza și reproiectarea geometriei unei carcase din material compozit. Modelul luat în considerare reprezintă un modul prevăzut cu ieșiri multiple destinate distribuirii și controlului energiei electrice, favorizând o raționalizare mai bună a sarcinii. Aceste tipuri de module vin echipate cu o serie de protecții pentru lucrul în sarcină (OCP, OVP, UVP etc). Modelul suport de referință utilizat în analiză este destinat echipării unui autovehicul. Modelul supus analizei reprezintă o adaptare a modelului de referință. Analiza cu elemente finite va permite stabilirea punctului de injecție și structura carcasei.