Finite element modelling of a plan polymeric composite product under impact and thermal conditions

S M Ion, D Nitoi, O Chivu and C G Opran
University Politehnica of Bucharest, Splaiul independentei no. 313, sect.6, Bucharest, Romania
E-mail: dan.nitoi@upb.ro

Abstract. Polymeric composites products are undeniable trends today, especially in the most innovative fields and with a permanent degree of novelty. The specialized literature presents a very wide range of their applications in the avant-garde fields such as the aerospace field and that of the automotive industry. Paper presents the results of research on modelling and impact simulation of these types of products besides a transient temperature analyse using the finite element method. Starting from the experimental part through which a flat composite product is made, the authors research its behaviour on impact and at the same time try to make a mathematical model by the finite element method. This comes to complete the experimental research by providing information on the mechanical stress that occur during experimental testing. The properties of the materials both those used in the realization of the experimental product and in the simulation are those offered by the manufacturers. The input data used in the finite element simulation tries to provide an image as close as possible to the probable behaviour of the product in case of car crash. Considering obtained results, large-scale industrial application is envisaged for automotive polymeric composite parts leading to the increase of the safety of the traffic participants in optimal vehicle manufacturing conditions.

1. Introduction
The manufacturing methods for fibre reinforced materials are largely dependent on the matrix. Such materials are generally heated, melted and injected or pressed into a mould. The final geometry of the component is therefore determined by the mould. Fibre reinforced materials are used in a variety of applications that require very low-weight materials with particularly good mechanical properties. This includes the energy sector, the aerospace industry, military and automotive industry. Reducing the weight of airplanes and cars/trucks offers the highest potential to lower fuel consumption and CO₂ emissions, thus forcing the manufacturers to explore materials alternatives with better specific mechanical properties [1].

This paper refers to the production, testing and modeling and simulation of the characteristics of a flat type composite product. The first part of the article presents the execution of the mould in which the production of the composite material will take place, presenting its main elements. In the following the composite product that will be used in the automotive industry is presented after which it will be tested similar to the various conditions that may be encountered automotive industry exploitation. The last part of the research presents the finite element analysis of the way in which the studied product responds to the applied conditions with answers close to the experimental results.

Considering polymeric composite product manufacturing, the Resin Transfer Molding process (RTM), is the most used technology to create large parts made by polymer matrix composites because:
the production time is relative small; the consumption of materials is reduced to minimum; the obtained parts are in a constant tolerance of ±0.5 mm after de-molding; both sides of the part are shiny and smooth; determinate resistance of the wall; it can be used with gelcoat on both sides or only on one; the reinforcement materials can be fiber glass, carbon fiber, sandwich materials, or metallic inserts; it does not need pressing during the curing of the resin; reduced cycle time, and low styrol emissions [3,4]. Polymeric composite constituents used to manufacture the plane composite structure used in experiments has the following properties: gelcoat is Enguard SD, grey 25620, produced by Ashland; finish fiber is Finishmat D77 0.3 mm thick, produced by Lantor; reinforcement fiber is Combidflow of 1150 g/m2 specific density and 2.6 mm thick, produced by Ahlstrom; polyester resin is Enydyne I 69277 A, HDT ISO 75-2 A, 92, produced by Ashland 0.3 mm thick. A plane-shaped research demonstration mould is designed, built and used to produce 4-mm thick fiber glass – polyester resin composite samples manufactured by vacuum-resin transfer molding (V-RTM) process (figure 1). A single layer of glass fiber fabric is placed between the outer and inner molds and then resin is transferred into the mold through nozzles shown in the picture. Working parameters are: temperature +25°C; vacuum pressure for mold closing -0.8 bar; vacuum pressure for resin flow -0.5 bar [2].

**Figure 1.** Composite product manufacturing: (a) mould elements; (b) closed mould with composite product; (c) mould geometry and structure.
A plane-shaped research demonstration mold is designed, built and used to produce 3.2 mm thick fiber glass – polyester resin composite samples manufactured by vacuum-resin transfer molding (V-RTM) process. A single layer of glass fiber fabric is placed between the outer and inner molds and then resin is transferred into the mold through nozzles shown in the picture. Working parameters are: temperature +25°C; vacuum pressure for mold closing -0.8 bar; vacuum pressure for resin flow -0.5 bar.

2. Impact testing of the plane composite product
The experimental tests performed on the flat composite product focused on the impact tests. These tests were performed on an Instron type machine. The impact tests were performed from several heights, namely H = 300, 400, 500, 600, 750, 800 mm. Figure 2 shows the force-energy-time graphs made by the software of the equipment used [3, 5-7].

![Testing graphs for the part in different testing conditions.](image)

Figure 2. Testing graphs for the part in different testing conditions.

Figure 3(a) shows the fixing of the composite specimen on the table of the impact test machine. Following the shock test, the specimens were destroyed as can be seen in figure 4(b). For all working heights, figure 4(c) shows the specimens resulting from the tests.

![Fixing of the composite specimen on the table of the impact test machine.](image)

(a)
Figure 3. Testing the product: (a) composite part fixing; (b) testing of composite parts in different conditions H = 300 mm, H = 400 mm, H = 500 mm, H = 600 mm, H = 750 mm H = 800 mm.

According to the impact tests performed, in table 1 are presented the main results for six types of tests performed, namely for the heights h = 400, 500, 600, 700, 750, 800 mm. The mass of the punch is m = 3.94 kg.

Table 1. Impact test results.

| Test no. | Specimen ID | Weight [Kg] | Drop Height [mm] | Peak Load [KN] | Deflection at Peak Load-1 [mm] | Energy to max load-1 [kg m] |
|----------|-------------|-------------|------------------|----------------|-------------------------------|--------------------------|
| 1        | 1.1         | 3.9400      | 500.0000         | 3.2951         | 6.4428                        | 0.8266                   |
| 2        | 1.2         | 3.9400      | 600.0000         | 3.5116         | 6.1800                        | 0.9590                   |
| 3        | 1.3         | 3.9400      | 700.0000         | 3.1915         | 7.0515                        | 1.1347                   |
| 4        | 1.4         | 3.9400      | 800.0000         | 3.3776         | 7.2806                        | 1.2493                   |
| 5        | 1.5         | 3.9400      | 750.0000         | 3.6079         | 7.2573                        | 1.2932                   |
| 6        | 1.6         | 3.9400      | 400.0000         | 3.6864         | 6.7343                        | 1.0541                   |
| Average  |             | 3.9400      | 625.0000         | 3.4450         | 6.8244                        | 1.0862                   |
| Median   |             | 3.9400      | 650.0000         | 3.4446         | 6.8929                        | 1.0944                   |
| Minimum  |             | 3.9400      | 400.0000         | 3.1915         | 6.1800                        | 0.8266                   |
| Maximum  |             | 3.9400      | 800.0000         | 3.6864         | 7.2806                        | 1.2932                   |

3. Finite element modeling of the impact test and thermal behavior

The prediction of the behaviour in certain conditions of mechanical stress represents a very important stage in the design of any product used in the automotive industry, fact for which, in addition, the input data for a structural analysis are presented. Establishing the material properties to characterize as well as possible the behaviour of the composite product is the most important element in the proposed finite element analysis [4, 5, 8-10]. Depending on the properties of the materials provided by the manufacturers and presented previously, in table 2 are presented the material characteristics used as input data.

Table 2. Mechanical properties of the composite structure layers.

| Material        | Ex [GPa] | Ey [GPa] | Ez [GPa] | v   | Gx [GPa] | Gy [GPa] | Gz [GPa] | ρ [Kg/m³] |
|-----------------|----------|----------|----------|-----|----------|----------|----------|----------|
| Finishmat 30%   | 62       | 62       | 15       | 0.22| 2.1      | 3.2      | 2.1      | 913      |
| Resin           | 4.5      | 4.5      | 4.5      | 0.35| 3.2      | 3.2      | 3.3      | 1120     |
| Combyflow 30%   | 45       | 45       | 11       | 0.22| 4.5      | 2.2      | 4.5      | 1194     |
Using this material structure, a composite product with LxL = 100 x 100 mm was manufactured and modelled by FEM. This simulation includes the analysis of impact test behaviour in two situations. In the first the free fall height is h = 400 mm and in the second h = 800 mm. Figure 4(a) presents the test result in the first case (h = 400 mm) on both sides of the composite plate and in figure 4(b) the FEM analysis. From the FEM analysis of deformations on the OZ axis, with blue colour a maximum deformation UZt = 6.8 mm is observed. The experimental tests presented in table 1 gives for the same drop height the deflection on the peak load (line 6, table 1) the value UZexp = 6.73 mm, very close to theoretical value. Table 3 presents the output data for the impact test from drop height = 400 mm.

Table 3. Testing values for impact test from drop height = 400 mm.

| Specimen ID | Weight [Kg] | Drop Height [mm] | Maximum load-1 [kN] | Time to max load-1 [ms] | Impact velocity-1 [m/s] | Impact energy-1 [kg m] | Total energy-1 [kg m] | Total time-1 [ms] |
|-------------|-------------|------------------|---------------------|-------------------------|------------------------|------------------------|----------------------|-------------------|
| 1.6         | 3.9400      | 400.00           | 3.6864              | 2.7112                  | 2.7793                 | 1.5517                 | 1.5696               | 4.4263            |
The second analysis refers to the case of the test from the height \( h = 800 \) mm. Figure 6(a) shows the two sides of the composite plate in the case of testing and in figure 6b the resulting deformation in the case of FEM. The FEM calculates the displacement \( U_{Z} = 7.83 \) mm. Testing machine measured (line 4, table 1) \( U_{Z_{exp}} = 7.28 \) mm, very close to theoretical value.

Table 4. Testing values for impact test from drop height = 800 mm

| Specimen ID | Weight [Kg] | Drop Height [mm] | Maximum load-1 [kN] | Time to max load-1 [ms] | Impact velocity-1 [m/s] | Impact energy-1 [kg m] | Total energy-1 [kg m] | Total time-1 [ms] |
|-------------|-------------|-------------------|----------------------|------------------------|------------------------|------------------------|----------------------|------------------|
| 1.4         | 3.9400      | 800.0000          | 3.3776               | 1.9666                 | 3.9371                 | 3.1138                 | 2.9503               | 6.0120           |
The study of the stresses that appear in the composite structure offers a very useful image in the analysis of the appearance of cracks or destruction of the studied piece. Without the use of tensometric gauges the FEM offers results very close to reality and an extremely useful overview of the map of the different types of stresses used in the strength of materials. Figure 6 shows the analysis of the state of stresses of type S2 (the one that represents in a balanced way the state of tensile and compression stress in the material). From the analysis of the two images can be seen how in the first test case (h=400 mm) the maximum value of the stresses is $S_2 = 0.76 \times 10^9$ N/m$^2$ (figure 6(a)). For the second analysis (h = 800 mm) the value maximum stresses is $S_2 = 0.87 \times 10^9$ N/m (figure 6(b)). Both values of the stresses are located right in the zone of the values of the elasticity modules of the materials that constitute the composite structure. The fact that both parts were destroyed can be explained by these high values of stresses and by the possible occurrence of defects or non-uniformities in the material [11].
Figure 6. S2 stress analysis: a – $S_{2\text{max}} = 0.76 \times 10^9$ N/m$^2$ for $h = 400$ mm testing case; b – $S_{2\text{max}} = 0.87 \times 10^9$ N/m$^2$ for $h = 800$ mm testing case; c – S2 stress section.

Figure 8 shows the calculation of the shear stresses in the XOY plane. As can be seen, in this plan, there is an alternation of tension and compression stresses inside the structure which may explain its destruction during the tests. In the first case of test the maximum value of the stresses is $S_{xy} = 0.8 \times 10^6$ N/m$^2$ and for the second case $S_{xy} = 0.1 \times 10^7$ N/m$^2$ (figure 7). Even if the stress values are not high, their arrangement and alternation can cause the material to crack and break because the position and adhesion of glass fiber. Stress distribution is the same on both situations, but the values are different.

Figure 7. SXY shear stress analysis. a – $S_{\text{2max}} = 0.76 \times 10^9$ N/m$^2$ for $h = 400$ mm testing case; b – $S_{\text{2max}} = 0.87 \times 10^9$ N/m$^2$ for $h = 800$ mm testing case.
From the point of view of the exploitation of such a composite product, for example in the automotive industry, where the temperatures do not generally exceed 90\(^0\) C, a thermal analysis was performed. Material properties used as input data are presented in table 5. Figure 8 presents the equilibrium thermal analysis from where it can be seen how on the opposite side of the surface heated to T = 90\(^0\) C the temperature remains at that of the environment, about T = 20\(^0\) C. It is thus found that the composite part can function as a good thermal insulator for automotive industry.

| Material      | Thermal conductivity [W/mK] | Specific heat [J/Kg K] |
|---------------|-----------------------------|------------------------|
| Glass fibre   | 1.2                         | 7.35                   |
| Epoxy resin   | 0.35                        | 1000                   |

Figure 8. Simulation of the thermal field when applying a temperature of 90\(^0\) C on a surface of the composite product.

4. Conclusions
The paper seeks to make a new contribution in the field of products obtained from composite materials that are used in the automotive industry. It is presented here the way in which the product is made by using an injection mold whose components are showed. Further, research presents several impact tests to which the composite product has been subjected, tests that are similar to what can happen in the event of a car accident. In the last part of the article are presented several results obtained by modeling and simulation starting from the input data such as geometric shape, properties of materials that are part of the product and the impact force acting on it are analyzed displacements and stress appears in the structure of the material. The results of the calculations offered by the program regarding the displacements are very close to those obtained experimentally in the two analyzed situations, namely the impact from the height of h = 400 mm respectively h = 800 mm. This proves that the working assumptions initially considered are valid. Furthermore, from the point of view of the analysis of the stress state that appears as a result of the application of the impact force, it can be seen that their values are at the limit of the elastic modulus for both fiberglass and resin used to produce composite product. The values of the calculated stresses do not exceed the modulus of elasticity being even at their lower limit. In the case of a homogeneous and isotropic structural material, the appearance of stress that do not exceed its modulus of elasticity would not have led to its rupture or destruction. The physical destruction of composite products, however, taking place in conditions where the stresses do
not exceed the modulus of elasticity shows that there may still be defects inside the structures that can lead to their cracking. This, in practical applications regarding composite materials, a safety factor must be taken regarding the occurrence of defects that can lead to their possible cracking.

In the second part of the simulation was considered an equilibrium thermal analysis from which it can be seen that this type of material can be considered a good thermal insulator because applying a constant temperature $T = 90^\circ C$ on one of its surfaces leads only to an insignificant increase in temperature on the opposite surface at ambient temperature.

5. References

[1] Lengsfeld H., Wolf-Fabris F., Kramer J., Lacalle J., Altstadt V., 2016 Composite technology-Prepregs and Monolithic Part Fabrication Technologies (Hanser Publisher).

[2] Ion S. M., Opran C., Lamanna G. 2020 Mathematical Model of Vacuum–Resin Transfer Molding for Polymeric Composites Products Macromolecular symposia, 389(1) doi: 10.1002/masy.201900108.

[3] Melro A. R., Camanho. P. P., Andrade Pires F. M., Pinho S. T. 2013 International Journal of Solids and Structures 50 1906-1915.

[4] Doitrand A., Fagiano C., Irisarri F., Hirsekorn M. 2015 Composites Part A: Applied Science and Manufacturing, 73, 143-154.

[5] Chawla N., Chawla K. 2006 Journal of Material Science 41 395-40.

[6] Tsai S. W., Melo J. D. 2015 Composite Materials Design and Testing: Unlocking mystery with invariants (Stanford Aeronautics & Astronautics JEC Group).

[7] Tsai S. W. 2018 Strength & Life of Composites 2008 (JEC Group).

[8] Lamanna G., Opran C., Perfetto D. 2018 Development of an Established FE Model for the Simulation of LVI Tests on GFRP Plates, 9th International conference on “Times of polymers and composites” doi.org/10.1063/1.5045998.

[9] Herath M. T., Bandyopadhyay K., Logan J. D. 2010 Modelling of delamination damage in composite beams; 6th Australasian Congress on Applied Mechanics, ACAM 6, 12-15.

[10] Michel J. C., Suquet P. M. 1993 On the Strength of Composite Materials: Variational Bounds and Computational Aspects; Topology Design of Structures 355-374.

[11] Feng W. 1991 A Failure Criterion for Composite Materials; J. Composite Materials, 25 88-100.