DESIGN AND FEA CRASH SIMULATION FOR A COMPOSITE CAR BUMPER

A. CALIENCIUG[1]   Gh.N. RADU[2]

Abstract: To create a quality structure with low stress concentration we need to make improvements to that structure starts with the design, structure, quality of the materials and we will obtain a mixture of the materials named composite materials with better properties and a new structure shape optimization. The present papers analyze the impact behavior of a composite car bumper made from new materials. The study is performed using Solidworks software for the design of the new car bumper made from new composite materials and the stress concentration distribution is evaluated by use of finite element analyze with Abaqus software for the impact cases.

Key words: structure, car bumper, impact.

1. Introduction

Nowadays the automotive industry is concerned based on development of primary car safety equipments. This study focuses in the application of new composite materials in a front car bumper.

New composite materials made for the automotive industry every year because they are lighter, high strength and better than the present ones. EuroNCAP [4] company was founded for the first time in 1997 and has two important objectives.

The first objective it provides important information about each car safety rating as a score. The second objective it provides important data to producers for safety improvements. Some examples of their tests are: frontal, side, pole impact and children, pedestrian protection. RCAR the Research Council for Automobile Repairs, is an international organization works towards reducing costs by improving cars

damage ability, repair, safety by low-speed car crash test [5].

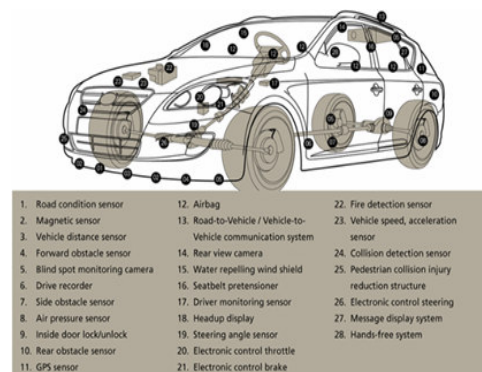There are two ways for automotive industry safety equipments: active and passive, Figure 1 [6]:

1. Road condition sensor	12. Airbag	22. Fire detection sensor
2. Magnetic sensor	13. Road-to-Vehicle / Vehicle-to-	23. Vehicle speed, acceleration
3. Vehicle distance sensor	Vehicle communication system	sensor
4. Forward obstacle sensor	14. Rear view camera	24. Collision detection sensor
5. Blind spot monitoring camera	15. Water repelling wind shield	25. Pedestrian collision injury
6. Drive recorder	16. Seatbelt pretensioner	reduction structure
7. Side obstacle sensor	17. Driver monitoring sensor	26. Electronic control steering
8. Air pressure sensor	18. Headup display	27. Message display system
9. Inside door lock/unlock	19. Steering angle sensor	28. Hands-free system
10. Rear obstacle sensor	20. Electronic control throttle	
11. GPS sensor	21. Electronic control brake	

Fig. 1. *Active and passive safety systems* [6]

The active safety car systems means that all the equipments integrated in the car that helps the driver to avoid the collision with

---

[1] Dept. of High-Tech Products for Automotive Industry, *Transilvania* University of Braşov.
[2] Dept. of Strength of Materials and Vibrations, *Transilvania* University of Braşov.

another car or with every object or person on the road and here we have some examples of this systems: Abs (anti-lock braking system), Esp. (Electronic stability program), Impact sensors, traction control, Bas (brake assist) or pre-safe brake system.

The pasive safety car system means that all the physical systems that the passengers of the car are protected in case of the collision and some examples are here: front and rear car bumpers capable to absorb impact energy, pretensioner seatbelts, airbags, roll cage.

By using composite materials, bumper structure will lead to a weight loss of vehicle weight, lower fuel consumption and pollutant emissions of greenhouse gases, and high resistance to impact and corrosion.

Any kind of car is based on active systems but when this systems or human errors will failure, the passive systems will take places.

2. Objectives

The most important objective has been the impact analysis with the wind and an auto-towing hook with a steel ball at the end, using the finite element analysis using Abaqus software after the modeling with Solidworks software of the new shapes car composite bumper was made from new composite materials.

3. Solidworks Design and Abaqus Fea

The designer that creates the shapes of the car bumpers must taking into consideration to reduce the stress concentration in that structure and for this reason; the best think is to use good software for a best shape optimization. Some couple of year ago someone has suggested a simple geometric algorithm to ''clone'' the shape of trees but practically is not so simple [1].

In computer simulation, a CAD car bumper model is first created using Solidworks. All the geometric dimensions of the car bumper are measured and some modeling techniques such as sweeping, reflecting and shell are applied to create the structure with a long curve surface. Figure 2 shows the car bumper's CAD model [2].

After finishing the CAD model, for FEA study, the Solidworks models were exported to the Abaqus software. FEA study considered linear-elastic case for the new composite car bumpers. In creating the finite element model, the 2D shell element, shell 181, is used to mesh the car bumper surface. The material of the bumper surface is ABS, and the materials of the impact ball-towing hook are steel.

The mounting point was supported using elastic boundaries, with an elastic stiffness values that have not influenced the stress gradient in the stress concentration area.

The new composite car bumpers were meshed and in the structural analysis the elements with shared nodes states stresses independently at the node with independent stress calculations provides the accuracy based on von Misses stress.

The accuracy value at a given node is calculated with the formula [3]:

$$Accurancy \frac{0.5(\max val - \min val)}{Global \cdot vonmisses \max}. \quad (1)$$

According to these equations (1) the accuracy will be between zeros the best case and 0.5 deviations. Nodes that are shared between the elements have only one stress and deviation is zero. With this equation and condition, we can validate the FEA for the new composite car bumper.

The complete period of time for Solidworks modeling, FEA study (preprocessing-processing-post processing) for each of the studied cases was in average 360 minutes.
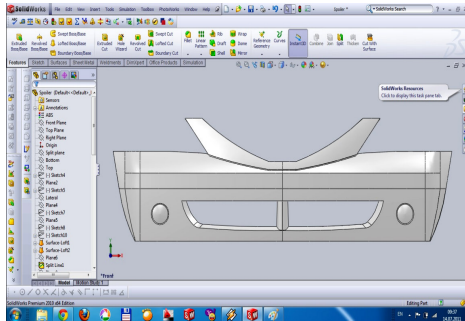
Fig. 2. *Front car bumper made from new composite materials*

In Figures 3 and 4 are presented the model from Solidworks exported in finite element analysis Abaqus software.
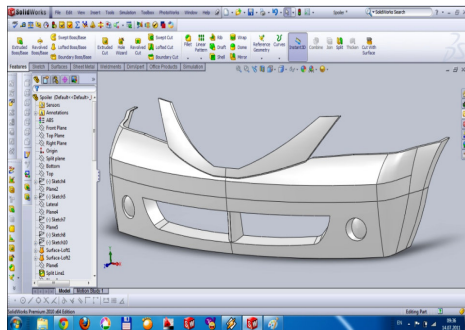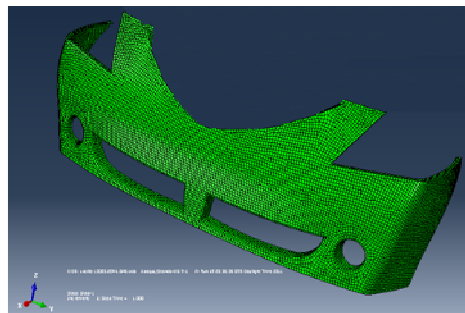

Fig. 3. *Exported all database for FEA simulation*

Fig. 4. *Meshing the new composite car bumper*

Bellow are presented the properties of the following composite materials: ABS, epoxy resin reinforced with fiber glass fabric made

with a weight 280 g/m^2 and the last one epoxy resin reinforced with overlapping layers of nylon 6/10 in two directions.

In Figures 5 and 6 are presented the ABS properties:
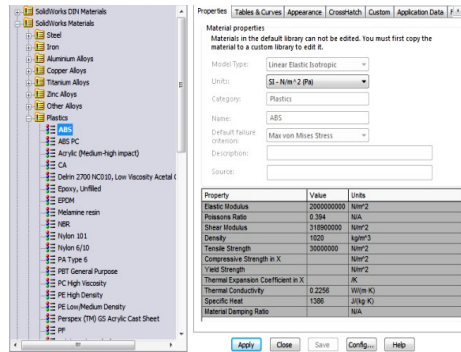
Fig. 5. *Properties of ABS*
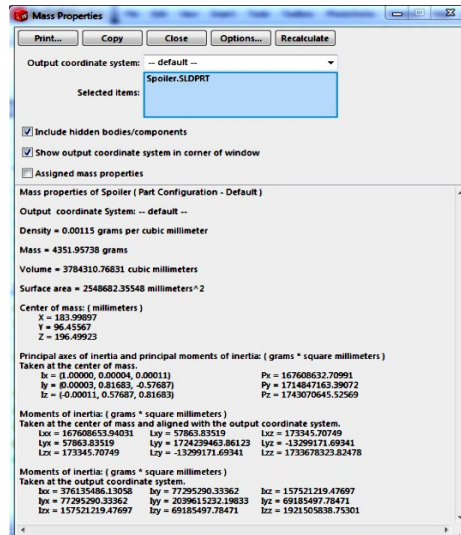
Fig. 6. *Properties of ABS*

In present days, the most of all materials that are used on bumpers is ABS because of its properties like rigidity, elasticity, corrosion resistance.

In Figures 7 and 8 are presented the Epoxy resin reinforced with fiberglass fabric made properties like high tensile, resistance to heat, are resistant to chemical agents and good resistance to radiation.
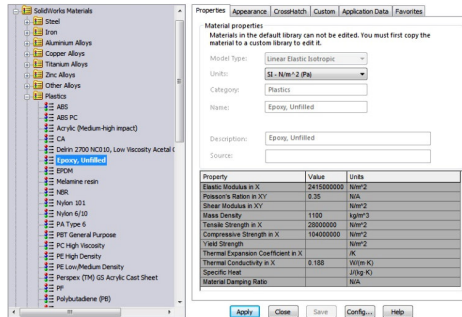
Fig. 7. *Epoxy resin reinforced with fibber glass fabric made with a weight 280 g/m²*
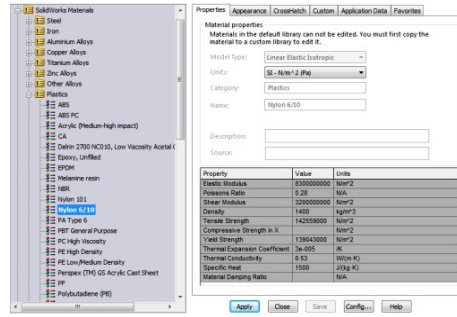*properties*

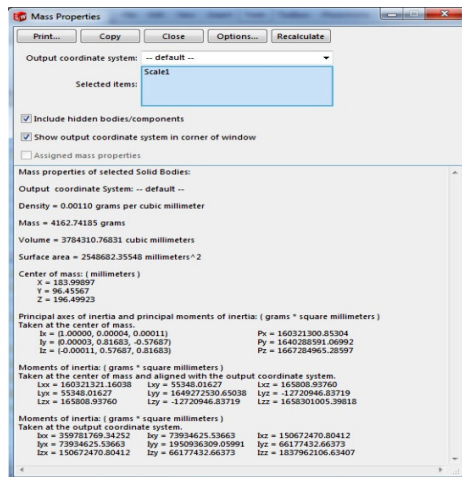Fig. 9. *Epoxy resin reinforced with overlapping layers of nylon 6/10*

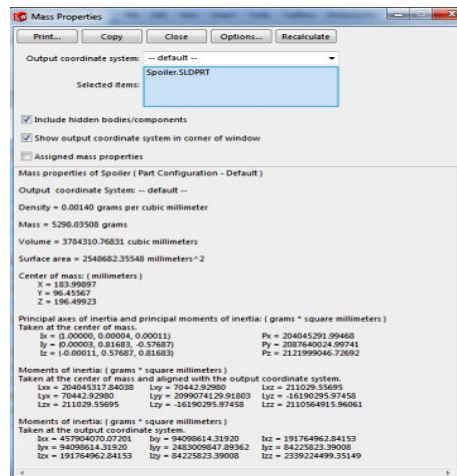Fig. 8. *Epoxy resin with fibber glass*

Fig. 10. *Epoxy resin reinforced with overlapping layers of nylon 6/10 in two directions properties*

In Figures 9 and 10 are presented the epoxy resin reinforced with overlapping layers of nylon 6/10, fibers exhibit high Young's modulus, high strength and high toughness. The improved toughness of the composite fibers arises from the ability of adopting different conformations under stress by the flexible spacers at the nylon 6/10 interface.

After they were presented their new material properties on the directions past the finite element analysis of the front car bumper tests for two cases: with the wind speed of 100 km/h and with a steel ball part of a structure of a towing car hook with the speed of 45 km/h.

In Figure 11 is presented the towing car hook with the steel bal in the top.

Fig. 11. *Steel Structure of a towing car hook with a steel ball on the top* [7]

In Table 1 are presented the elements which composes the car: the new front

composite car bumper made from two material types and a composite car structure. The number of the elements from FEA and dimensions which are the weight, diameter of the steel ball and the thickness of each element are writed.

The whole impact process will be simulated on a computer by running a transient analysis. Finite element analysis using Abaqus software for a fixed front car bumper is subjected to wind pressure at a speed of wind equal to 100 km/h like in Figure 12 with a maximum tension and Figure 13 with the maximum displacements.

As a contact problem, the front surfaces of the bumper is defined as the target surface, and also the contact surface.

Finite element analysis using Abaqus software of a front car bumper, moving at a speed of 45 km/h and its impact with a steel ball diameter 60 mm and weight 0.5 kg, wich is a part of a structure of a car towing hook like in Figure 14 with a maximum tension and Figure 15 with maximum displacements.

Composite car properties   Table 1

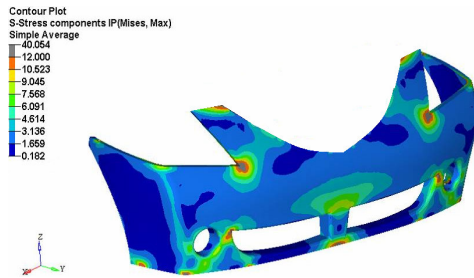| Element	Material	FEM Elm	Elm Num	Dimensions
Steel ball	Rigid steel	Solid 164	123	Weight/diameter 0.5 kg/60 mm
Composite car bumper 1	Epoxy resin & fibber glass fabric	Shell 181	3066	Thickness 3.5 mm
Composite car bumper 2	Epoxy resin & Nylon 6/10	Shell 181	1148	Thickness 3.5 mm
Composite car	Steel (20%) & composite material (80%)	Mass 166	4	Weight 1000 kg

Fig. 12. *Following the analysis of maximum tension of 40.054 MPa*
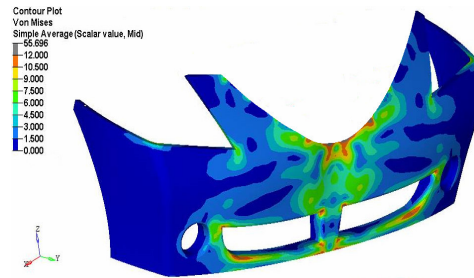

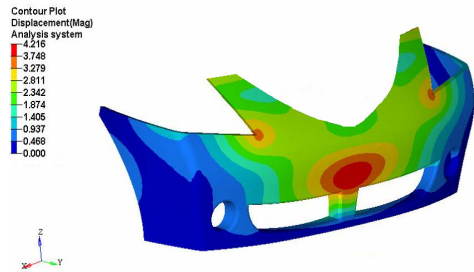Fig. 14. *Following the analysis of maximum tension of 55.096 MPa*

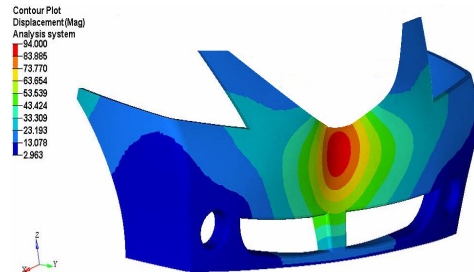Fig. 13. *Maximum displacements in the analysis of 4.216 mm*

Fig. 15. *Maximum displacements in the analysis of 94 mm*

4. Conclusions

Analyzing the reports made by Stress Analysis software on the maximum stress occurring in studied new composite car bumpers is found that the greatest tensions arise at these movements:

1. In the first step of the impact with a steel ball for a movement of 94 mm was obtained a maximum stress of 55.696 MPa.

2. The second step of the impact with the wind for a movement of 4.216 mm was obtained a maximum stress of 40.054 MPa.

Through the experiments with the computer simulation, the behavior of the bumper under velocity impact can be determined. During the impact, the transferred force reaches the highest value and shortly after the impact then reduces to its static value. In the future work we will have a comparison between this theoretical test simulation and the real test simulation.

Acknowledgements

References

1. Aignatoaie, M.: *Fea Study on the Influence of Using Hyperbolic Shapes in Cad Design of Rotating Machine Elements*. In: Proceedings of the 15th International Conference Modern Technologies, Quality and Innovation, Vol. I, Vadul lui Vodă, Chişinău, 25-27 May, 2010, 1-4.

2. Calienciug, A., Radu, N.,Gh.: *Fea and Design of an Automotive Bumper Made from Composite Materials*. In: Proceedings of the 4th International Conference Computational Mechanics and Virtual Engineering, COMEC'2011, Vol. 4, Braşov, 20-22 October, 2011, p. 605-609.

3. Kleisner, V., Zemčik, R.: *Analysis of Composite Car Bumper Reinforcement.* In: Applied and Computational Mechanics **3** (2009) No. 2, p. 287-296.

4. http://www.euroncap.com/tests/front impact.aspx. Accessed: 20-05-2008.

5. http://www.rcar.org/index.htm. Accessed: 20-05-2008.

6. http://www.4tuning.ro/tehnica-auto/ sisteme-de-siguranta-pasive-si-active-care-este-diferenta-10759.html. Accessed: 07-10-2011.

7. http://www.4tuning.ro/tehnica-auto/.

8. http://www.vreaureducere.ro/wp-content/ uploads/2010/06/logan1.jpg. Accessed: 12-09-2011.