[1.]Ancuța N. JURCO, [2.]Liviu I. SCURTU

GEOMETRICAL DESIGN AND CRASH SIMULATION OF A SHOCK ABSORBER

[1,2.]Technical University of Cluj-Napoca, Faculty of Automotive, Mechatronics & Mechanical Engineering, ROMANIA

Abstract: The safety of the car passengers is an important issue that must be achieved in the structural design of the vehicles. To increase the safety of the passengers, it is intended to design and build the bodywork structure that have a geometry who can reduce the car damage and the passenger's injury in the crash impact. The shock absorber used for automotive structure are intermediate body elements placed between the front bumper and the chassis structure. The study presented on this paper shows the behavior in the case of a frontal impact of a shock absorber structure. In the first part of the paper are presented the latest research and methods used for virtual testing, using finite element methods. Second part of the paper presents the geometrical design and finite element analysis of the proposed shock absorber geometry. Each model is simulated in concordance with the real work condition, with same loading case. Obtained result are analyzed, determining the von Mises stress and displacement.
Keywords: bumper, shock absorber, FEA, solver

INTRODUCTION

In recent years, the rapid development of the automotive industry and required standards regarding the safety of the traffic participants involves the development of new structural parts that can increase the safety in the event of a crash. Current design, simulation and validation techniques improve the comfort and safety of the car passengers. Some of the studies from this research field are presented in the following paragraphs. An extensive study described in eight chapter present the base steps that determine the design, modelling, finite element crash simulation and results interpretation. With the purpose of shock absorber structure optimization for increase the passenger safety are presented various frontal bumper crash scenarios. After simulation, the resulting deformation and the absorbed energy it is observed [7]. The behavior of a chassis structure is crashed from a rigid wall and from a rigid pole at a velocity of 15 m/s. The chassis structure is solved in Radioss solver from both cases. The obtained results show the chassis structure have a better impact behavior of the rigid wall than the case of the rigid pole impact [4]. Another crash analysis that investigates the ways to improve the design of the frontal bumper is presented in [1]. The simulation of the crash is performed at a velocity of 30 m/s. The effect of the thin-walled structure in automotive crashworthiness are studies in [3]. The objective of this study is to optimize a thin-walled structure to attenuate the frontal impact. Are tested five structure models using the explicit finite element method of Ls-Dyna. Praveenkumar present the study of the frontal impact for three different bumpers. The simulation is done at the 17.778 m/s initial speed of the structure and the distance between the bumper and the barrier is 100 mm. The bumper material is an aluminum alloy [2].

FINITE ELEMENT METHOD

Currently the finite element analysis is a numerical simulation method implemented in engineering. In the next paragraphs is presented the base steps that must be follow in order to perform a crash finite element analysis. In figure 1 is presented the flowchart of the crash analysis process [6].

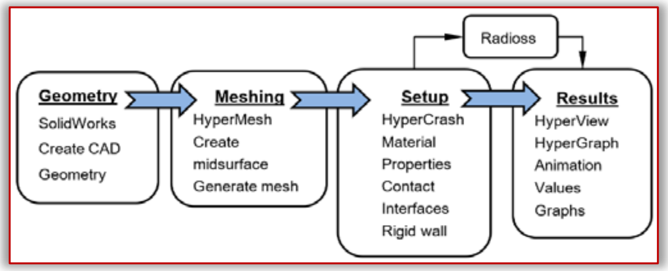

Figure 1 - Flowchart of the crash analysis process

CAD GEOMETRY

The study of the geometry is performed in order to obtain a shock absorber design that have a better performance in case of the frontal crash. The first step of this study is given by the conception and create the CAD model of the shock absorber. In figure 2 is presented the orthogonal and isometric projection of the shock absorber model.

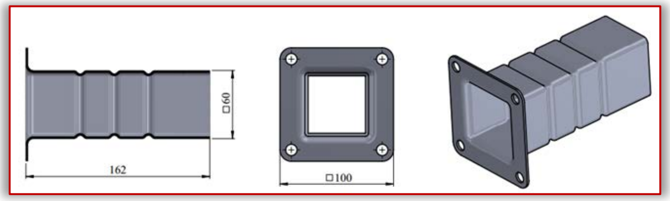

Figure 2 - CAD model of the shock absorber

Using the advanced modelling techniques in SolidWorks [9] software the proposed geometry of the shock absorber is generated. The model is created from a rectangular shape from 60 mm and a 4 mm thickness. The shock absorber geometry is exported from SolidWorks in IGES format.

GEOMETRY DISCRETIZATION

The structure geometry is imported into Hypermesh environment, where is prepared for meshing. After the cleanup surface process, the midsurface of the solid geometry is extracted by using the software commands. Geometry discretization is the fundamental steps required by finite element method and consist in transition from the continue structure to a discrete model with a finite number of points. The surface geometry is meshed automatically according to the pre-determined the quality criterion of the elements.

The property definition, material assigning, contact between surface, imposing the simulation condition and creating rigid elements are done in HyperCrash™. In order to set an imposed velocity of the meshed structure on the rear side of the absorber are created a rigid body element. The mass is attached from the master node in the center of mass of a slave node group. The resulted model is run in Radioss solver and the obtained results are performed in HyperView and HyperGraph [10].

MATERIAL MODEL

The assigned materials from both crash study from the shock absorber are steel and aluminum. The chosen material model from HyperCrash is an elastoplastic material Johnson Cook (Law 2). This material type includes strain rates and the temperature effect.

The mechanical properties of the assigned materials are shown in table 1. In law 2, the material behaves as linear elastic when the equivalent effort is less than yield stress. For high values of the strain, the material's behavior becomes plastic.

Table 1. Mechanical properties of the materials [8]

Steel		Aluminum	
Initial density	7.85e-6 kg/m3	Initial density	2.7e-6kg/m³
Poisson ratio	0.3	Poisson ratio	0.33
Hardening exponent	0.5	Hardening exponent	0.374618
Young modulus	210 GPa	Young modulus	60.04 GPa
Hardening parameter	0.5 GPa	Hardening parameter	0.22313
Failure plastic strain	0.3	Failure plastic strain	0.75

PROCESSING SETUP

To perform a higher accuracy finite element simulation, the shock absorber geometry is fine meshed and it is chosen QEPH shell formulation, which is recommended for crash analysis, and for plastic behavior of the material, five integration points are used.

The finite elements used in this crash simulation study are shell type that have a target length of 2 mm and a thickness of 4 mm. In figure 3 is presented the technique to meshing the symmetric geometry part. In this case is necessary to mesh just only one quarter of the part.

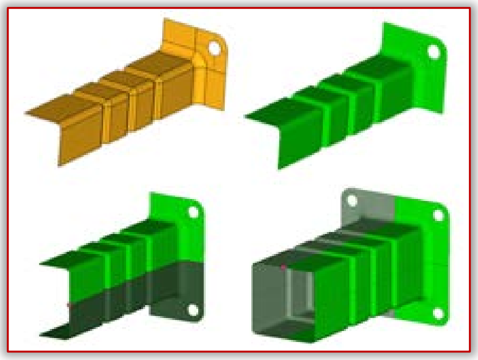

Figure 3 - Steps to create the symmetric mesh

After the quarter FEM model is mirrored from vertical and horizontal plane in order to create the completed model of the shock absorber. The surface contact between the shock absorber and rigid wall is "Type 7" with a Coulomb friction of 0.2. This contact interface is used for the most general types of crash analysis. For imitates the real work condition a mass of 300 kg is attached of shock absorber model. The imposed velocity of the model is 15m/s.

RESULTS

The crash result is presented in figures below for three thickness cases: 4 mm, 1 mm and 0.5 mm for both materials used.

This crash study is done on a graphic workstation, with an Intel Xeon CPU E5-2609v2, 2.5 GHz, and a 32 GB RAM and NVidia Quadro K620 graphical card.

In the figure 4 are presented the deformed model of the absorber, vonMises stress and displacement values from a thickness of 4 mm of the steel shock absorber.

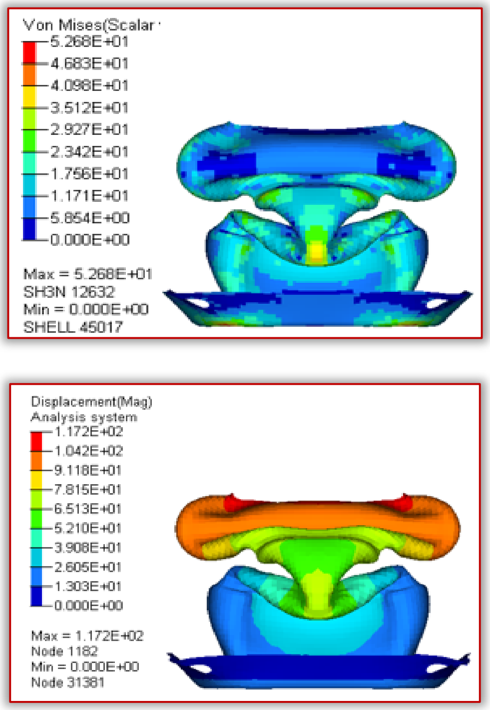

Figure 4 - Stress and displacement for 4 mm thickness of steel absorber

In figure 5 is presented the vonMises stress and displacement values of the aluminium shock absorber whit a 4 mm thickness.

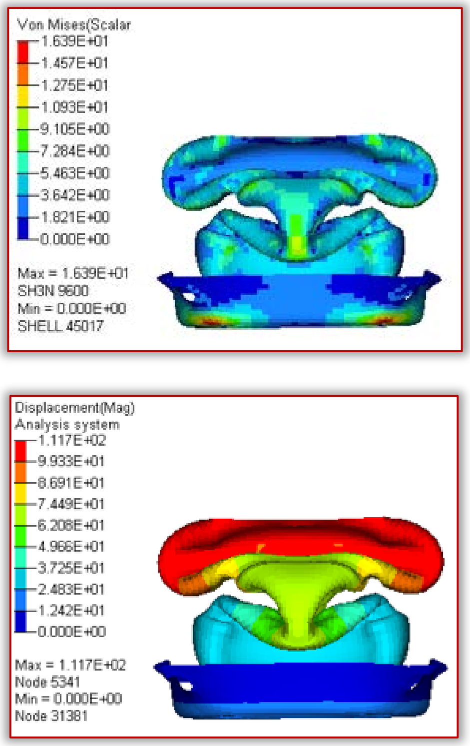

Figure 5 - Stress and displacement for 4 mm thickness of aluminum absorber

The stress and the displacement of the steel shock absorber are presented in figure 6.

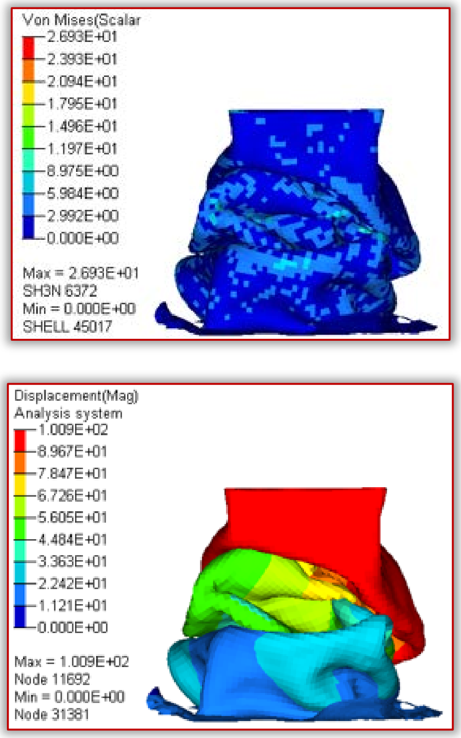

Figure 6 - Stress and displacement for 1 mm thickness of steel absorber

The stress and displacement resulted for an aluminum absorber with a 1 mm thickness are presented in figure 7.

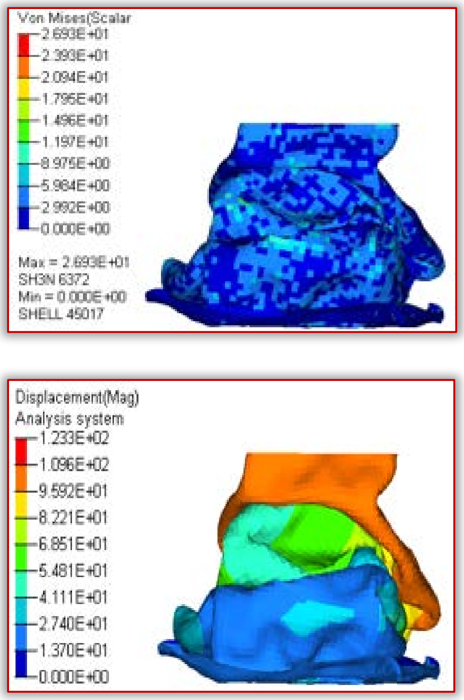

Figure 7 - Stress and displacement for 1 mm thickness of aluminum absorber

In figure 8 the von Mises stress and the displacement resulted for steel absorber are presented. It can be observed that the shock absorber model is totally destroyed.

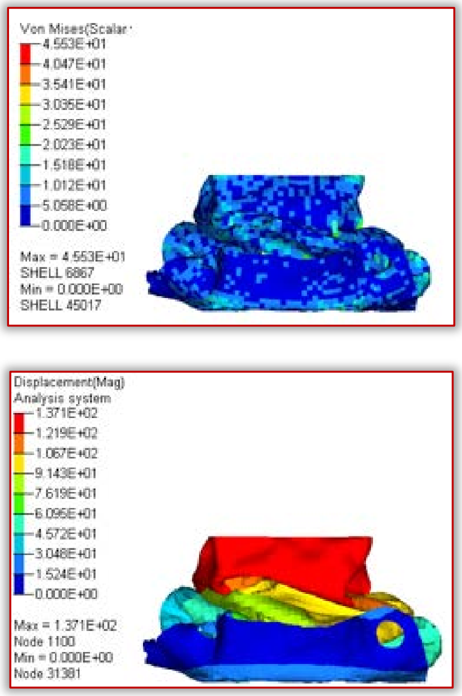

Figure 8 - Stress and displacement for 0.5 mm thickness of steel absorber

In figure 9 are presented the von Mises stress and the displacement of the aluminum shock absorber. The model is totally destroyed.

To a better view a buckling effect of the shock absorber in figure 10 are presented the phase during the crash simulations for a 4 mm from steel.

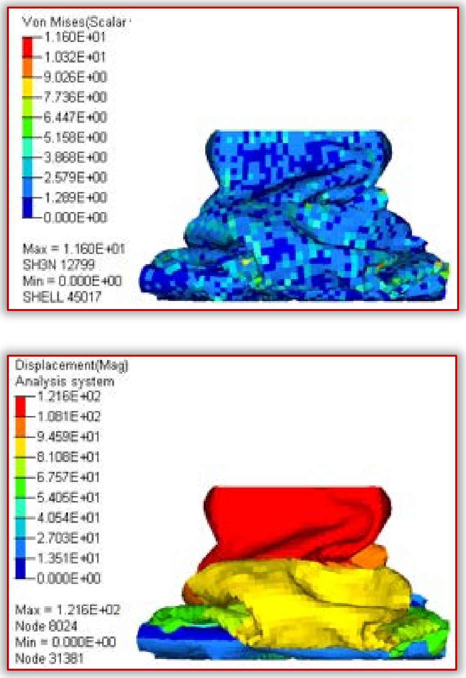

Figure 9 - Stress and displacement for 0.5 mm thickness of aluminum absorber

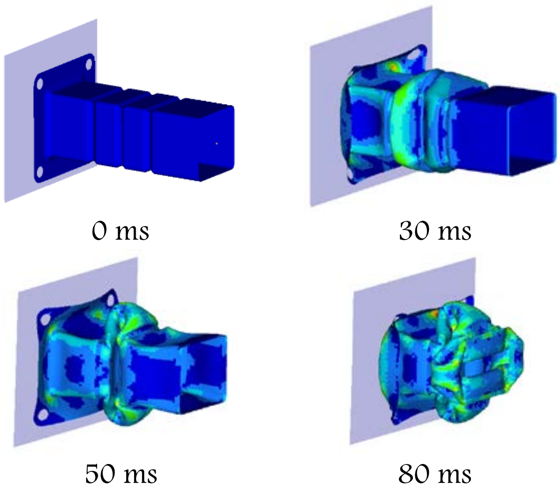

Figure 10 - Buckling effect of the shock absorber during the simulation

CONCLUSIONS

In this paper is present the crash behavior of the shock absorber used in construction of the vehicle's chassis. Are studied two cases of crash collision of a shock absorber from a rigid wall: in the first case, the attenuator is made from steel and the second case, the attenuator is made from aluminum.

During this study are presented all steps in order to create a crash simulation, from the geometrical design until the simulation results. An advantage of using the finite simulation analysis is that depending on the obtained simulation results it is possible to propose the final shape of the analyzed part. The protrusions from the model geometry allow a good axial compression of the model during the crash simulation process. It can be observed that the first simulation case with 4 mm thickness of the absorber takes over the internal energy better than in the other cases.

Note: This paper is based on the paper presented at ISB–INMA TEH' 2019 International Symposium (Agricultural and Mechanical Engineering), organized by Politehnica University of Bucharest – Faculty of Biotechnical Systems Engineering (ISB), National Institute of Research–Development for Machines and Installations Designed to Agriculture and Food Industry (INMA Bucharest), Romanian Agricultural Mechanical Engineers Society (SIMAR), National Research & Development Institute for Food Bioresources (IBA Bucharest), National Institute for Research and Development in Environmental Protection (INCDPM), Research-Development Institute for Plant Protection (ICDPP), Research and Development Institute for Processing and Marketing of the Horticultural Products (HORTING), Hydraulics and Pneumatics Research Institute (INOE 2000 IHP) and "Food for Life Technological Platform", in Bucharest, ROMANIA, between 31 October – 1 November, 2019.

References

[1] Yedukondalu, G., Srinath, A., Suresh Kumar, J., (2015), Crash Analysis of Car Cross Member Bumper Beam, Altair Tehnology Conference, Altair Technology Conference, pp. 1-9, India;
[2] Praveenkumar, A., Saran, V., Kavi Pradhap, M., (2016). Analysis of Typical Cross-Sectional Bumper Beam, International Journal of Modern Engineering Research, ISSN: 2249-6645, vol. 6(11), pp. 46-50;
[3] Sahzabi, Balaei, A., Esfahanian, Mohsen. (2017); The Effects of Thin-Walled Structure on Vehicle Occupants' Safety and Vehicle Crashworthiness, International Journal of Automotive Engineering, vol.7(2), pp. 2393-2403;
[4] Scurtu, I-L., Lupea, I. (2014), Frontal Crash Simulation of a Chassis Frame, Acta Technica Napocensis Series: Applied Mathematics, Mechanics, and Engineering, ISSN: 1221 – 5872, vol. 57(3), pp. 411-414;
[5] Scurtu I-L., Balcau M-C., Jurco A-N. (2019), Topography Optimization of a Bracket Used for Car Bodywork Reinforcement, Proceedings of the 4th International Congress of Automotive and Transport Engineering (AMMA 2018), Proceedings in Automotive Engineering. Springer, Cham;
[6] Tiago Miguel Encarnação Nunes, (2017), Multi-objective Design Optimization of a Frontal Crash Energy Absorption System for a Road-safe Vehicle, Thesis to obtain the Master of Science Degree in Aerospace Engineering;
[7] Vandanapu, Manideep, Kumar., Pandu, Ratnakar, (2015), Design Optimization of Passenger Car Front Bumper, International Journal of Science and Research (IJSR), ISSN (Online): 2319-7064, vol. 4(11), pp. 1411-1417;
[8] Altair Hyperworks user guide, (2017).
[9] SolidWorks user guide, (2018).
[10] Radioss tutorials, (2018).

ACTA TECHNICA CORVINIENSIS – Bulletin of Engineering
ISSN: 2067-3809

Faculty of Engineering Hunedoara,
5, Revolutiei, 331128, Hunedoara, ROMANIA
http://acta.fih.upt.ro