TECHNICAL UNIVERSITY OF CLUJ-NAPOCA

ACTA TECHNICA NAPOCENSIS

Series: Applied Mathematics, Mechanics, and Engineering
Vol. 57, Issue III, September, 2014

FRONTAL CRASH SIMULATION OF A CHASSIS FRAME

Liviu SCURTU, Iulian LUPEA

Abstract: *Nowadays, finite element analysis is used widely in crashes simulations in order to determine the deformation, stress and dynamic behavior of the vehicle during the impact. A finite element model of a chassis frame and frontal bumper is developed. The frontal crash simulation between the chassis frame is fully simulated using Radioss explicit integration solver. The proposed model is simulated under frontal collision with a rigid wall in the first case and in the second case with a rigid pole. In both cases the resultant deformations and stresses are determined for an initial velocity of 15m/s. The results of this simulation can be used to assess the damage of the observed model and to investigate new ways to optimize the frame design.*

Key words: *crashworthiness, frontal impact, explicit solver, chassis frame.*

1. INTRODUCTION

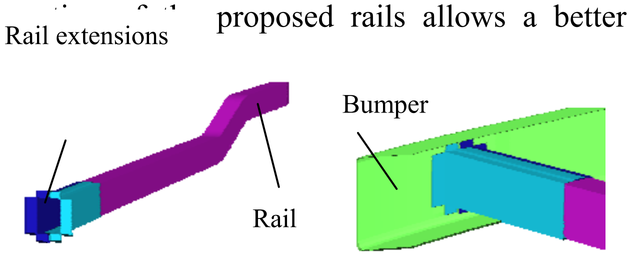
In recent years the safety requirements for autovehicles lead to an improvement of their structure. Recently, finite element analysis is widely used in order to improve the autovehicles structure. A great advantage of finite element analysis is the possibility to evaluate various situations almost similar to the reality. The automobile chassis is designed to include progressive crush zones to absorb the kinetic energy by plastic deformations. Several papers regarding the vehicle frame crash using finite element analysis have been published [1], [3]. In addition the finite element analysis is frequently recognized as a useful tool for roadside hardware design [4], [6].

In this paper the frontal crash simulation of a chassis frame using Radioss explicit integration solver is presented. The frame model is simulated under frontal collision with a rigid wall in the first case and in the second case with a rigid pole.

2. MODEL DESIGN

The geometry of the model was designed by using SolidWorks surface module. The frame of the rail extension consists of two parts that have a U-shape as can be seen in figure 1.

Both extensions of the frame rail are welded on the rail. Total length of the extension is 450 mm and the length without the weld assembly is 300 mm. This distance is designed to reduce the shock during the frontal impact. Tubular ... proposed rails allows a better



a) Rail model   b) Zoom in rail extensions

Fig. 1. Rail model

The geometry is meshed in shell elements using HyperMesh software. The size of shell elements is 10 mm. After the meshing geometry the finite elements model is exported for preprocessing in HyperCrash. The first crash simulation model is designed for determination of the maximum displacement and maximum von Mises stress when the frame is impacted into a rigid wall. In the second case the front frame is impacted into a rigid pole, keeping the same conditions as in the first case.

3. MATERIAL MODEL

An important point of the crash study is the material model of the chassis. The material type Johnson-Cook (Law2) is assigned to all components. Johnson-Cook material type models is an isotropic elastic-plastic material expressing the flow stress in a material as a function of strain ε, strain rate $\dot{\varepsilon}$ and temperature T. The stress σ during plastic deformation is given by the relation (1) [7]:

$$\sigma = \left(a + b\varepsilon_p^n\right)\left(1 + c\ln\frac{\dot{\varepsilon}}{\dot{\varepsilon}_0}\right)\left(1 - T^{*m}\right) \qquad (1)$$

where a is the plasticity yield stress, b the plasticity hardening parameter, c the strain rate coefficient, n the plasticity hardening exponent, $\dot{\varepsilon}_0$ the reference strain rate, $T*$ is a temperature coefficient and m is the temperature exponent. When the plastic strain ε_p in shell elements reaches the failure plastic strain ε_{max}, the shell elements are deleted. When the strain rates are important and known from measurement, the yielding true stress versus true plastic strain curves for various strain rates can be included in the material models [5]. Plastics experience stronger strain rate effects comparing with metals. In table 1 the mechanical properties of this material are presented.

Table 1

Initial density	$7.85e^{-6}$ Kg/mm3
Poisson ratio	0.3
Hardening exponent	0.5
Young modulus	210 GPa
Hardening parameter	0.5 GPa
Failure plastic strain	0.3

The thickness of the rail and the extensions parts is 2 mm thick sheet metal and the bumper thickness is 2.2 millimeters.

4. LOAD CASE CREATION

The gravitational acceleration of 9.81 m/s^2 has been applied for all nodes of the model. A general purpose interface type 7 has been chosen for the contact simulation during the crash and the buckling process. The stiffness is increasing with the penetration and the force variation of a node versus penetration is increasing in a non-linear manner, preventing the node from going through the shell mid-surface. A Coulomb friction coefficient of 0.2 has been chosen. All components of the frame are interconnected by welding line.

a) Rigid wall impact b) Rigid pole impact

Fig. 2. Top view of proposed crash set up

The initial velocity of the frame is set to 15 m/s. In first case the frame is crushed into a perpendicular rigid wall and in second case the frame is crushed into a rigid pole. Diameter of rigid pole is 300 mm. Figure 2 shows the top view of the front frame impact for both configurations.

The complete time history of the frame behavior during the crash simulation is recorded by the accelerometer placed on the frame extension. To simulate the weight, on the chassis frame are placed two concentrated masses. The first one of 300 kg situated in the rear of the frame determining the mass of the engine and gearbox and the second mass of 100 kg represent the weight of the driver with the safety equipments. The dynamic loading condition applied in this simulation is almost similar to the real crash situation. In figure 3, the general view and the positioning of the concentrated masses and the accelerometer, can be seen. The concentrated masses are linked to the chassis with rigid link elements. The complete model and linkages contains 62001 elements and 68670 nodes.

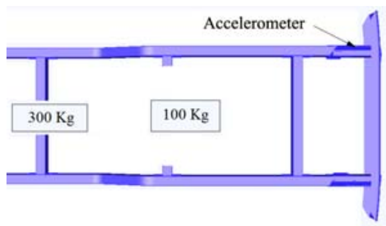


Fig. 3. Mass distribution and accelerometer

5. SIMULATIONS AND RESULTS

In figure 4 the von Mises stress distribution on the frame, can be seen. The maximum deformations of the frame assembly are located on the rails extensions of both rails and in rear of the chassis in the bended area of the rail. The maximum value of the von Mises stress is 7.099E-1 GPa. The largest deformations are located at the rails extensions of the chassis.

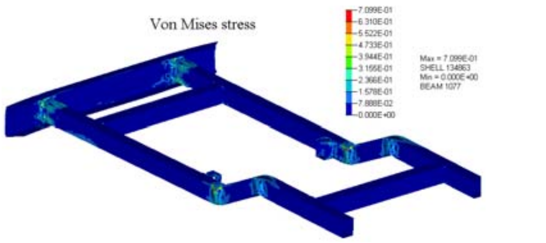
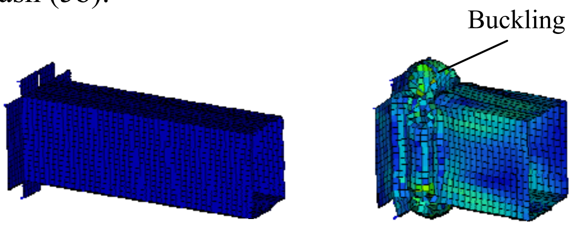


Fig. 4. Von Mises stress distribution on the chassis (GPa)

Most of the impact energy is absorbed by the rails extensions located between the bumper and rails. Figure 5 shows the shape of the rail extension before the crash (5a) and after the crash (5b).



a) Before impact b) After impact

Fig. 5. Rail extension behavior comparison

In figure 6 the enlarged view of displacement distribution of the frame crash into a rigid wall is presented.

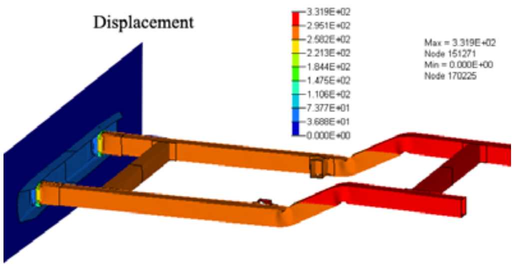
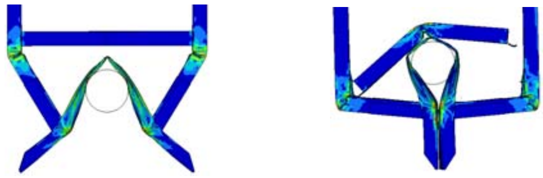


Fig. 6. Overview of displacement distribution on the chassis (GPa)

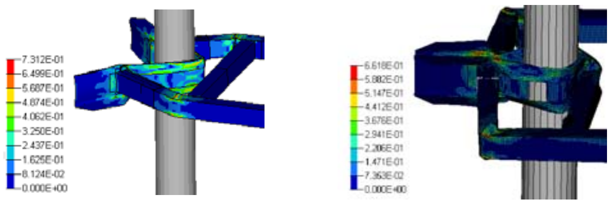
Figure 7a shows the top view of von Mises stress concentration of the front frame after 40 milliseconds and in figure 7b is presented the front part of the frame at final configuration after 100 milliseconds. The maximum value of the von Mises stress is 6.618E-1 GPa.

The largest deformations are located in the front of the chassis frame. For the pole crash simulation the enlarged view and values of the von Mises stress concentration of the frame is presented in picture 8 at time 40 milliseconds and 100 milliseconds.



a) after 40 ms b) after 100 ms

Fig. 7. Top view of von Mises stress



a) 40 ms b) 100 ms

Fig. 8. Zoom in, front frame behavior after impact (GPa)

After the rigid wall impact analysis is performed the energy curves are plotted as shown in figure 9. The kinetic energy curve starts from the initial energy value of 56427 joules remains constant to the first 2.7 milliseconds, time when the bumper get in contact with the rigid wall. After that the kinetic energy drops continuously until the 40 milliseconds, the level being close to zero. The deformation stops when the kinetic energy in the model is completely absorbed in plastic deformation and gets converted into internal energy.

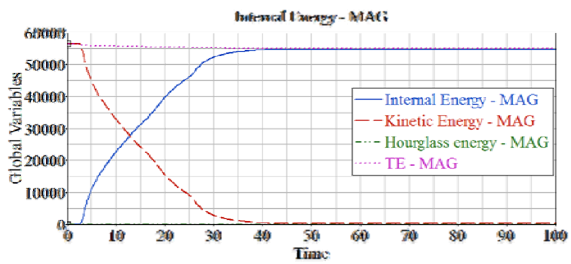


Fig. 9. Energy balance during rigid wall impact

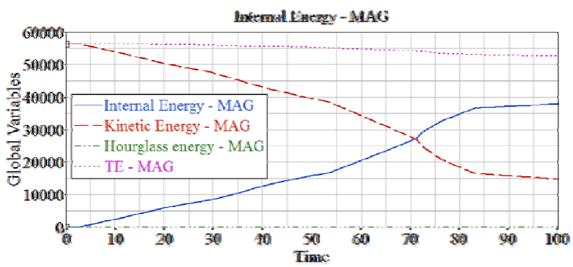


Fig. 10. Energy balance during rigid pole impact

In the second case of this study shown in figure 10 the kinetic energy starts from the same initial kinetic energy of 56427 joules decreasing continuously until the first 85 milliseconds. After this time the kinetic energy is decreasing slowly.

One of the indicators for correct analysis is that the total energy of the system remains constant. The hourglass energy is negligible.

6. CONCLUSION

In this study is presented a methodology to analyze the frame structure during the fontal impact into a rigid wall and into a rigid pole.

In the first crash simulation can be observed that the largest deformations are located in rails extensions. Most of kinetic energy is absorbed by the rails extensions, heaving in this case the status of shock absorber.

In the second crash simulation the displacement are much larger than in the first simulation. In this case a way to optimize the front side of the frame is of interest.

This methodology can be used to evaluate comparatively the effect of any automotive structural modification on the frontal crash before finalizing the design. The energy balance is a method to evaluate the correctness of the numerical analysis.

The main advantage of the simulation is the possibility to study the crash in more details, to perform parametric studies and the optimization of the structure.

7. REFERENCES

[1] Ananda Babu, T., Vijay Praveen, D., Venkateswarao, M., *Crash analysis of car chassis frame using finite element method*, International Journal of Engineering Research & Technology, ISSN-2278-0181, Vol. 1, Issue 8, 2012.

[2] Belsare, V., Pathak, C., Kulkarni, Milind, *Rollover analysis of passenger bus as per AIS-031*, International Journal of Engineering Research and Development, e-ISSN 2278-067X, Volume 4, Issue 5, pp. 49-59, 2012.

[3] Benhizia, A., Outtas, T., *Numerical simulation of frontal offset crash test for the vehicle frame using Ls-Dyna*, World academy of science, Engineering & Technology, Issue 56, p5, Aug. 2011.

[4] Lupea, I., *Assessing the efficiency of a guardrail system by simulation*, International Conference on Motor Vehicle and Transportation, MVT 2006, pp. 85-92, Timişoara, November 15-17, 2006, FISITA & SIAR, Proceedings ISBN 10 973-638-284-2.

[5] Lupea, I., Cormier, J., Shah, S., *FEM for Impact Energy Absorption with Safety Plastic*, 8-th International LS-DYNA Users Conference, Proceedings pp. 5.57-5.68, May 2-4, 2004, Dearborn, Michigan, USA.

[6] Sean M., O., Wekezer, W., *Crash impact analysis of the G2 guardrail: A validation study*, FAMU-FSU, TRB National Research Council, 1998.

[7] *** Radioss Theory Manual 10.0 version, *Large Displacement Finite Element Analysis*. Part 2, 2009.

SIMULAREA UNUI IMPACT FRONTAL AL UNUI ŞASIU UTILIZÂND METODA ELEMENTELOR FINITE

În zilele noastre, analiza cu elemente finite este utilizată în scară largă în simularea accidentelor cu scopul determinării deformaţiilor, tensiuniilor şi a comportării dinamice a autovehiculelor pe durata impactului. Este realizat un model discretizat în elemente finite al unui şasiu şi a unei bare de protecţie pentru un cart. Simularea impactului frontal al şasiului este realizată folosind solverul explicit Radioss. Modelul este supus în primul caz la coliziunea frontală cu un perete rigid şi în cel de al doilea caz la coloziunea frontală cu un stâlp rigid. În ambele cazuri deformaţiile şi tensiunile von Mises sunt determinate la o viteză iniţială de 15m/s. Rezultatele acestei simulări pot fi utilizate în evaluarea eficienţei la impact a modelul propus şi căutarea unor metode pentru optimizarea structurii.

Liviu-Iacob SCURTU, Assist. Prof. Ph.D. Technical University of Cluj-Napoca, Dept. of Automotive and Transport, 103-105 Muncii Blvd, 400641 Cluj-Napoca, +40-264-401610, e-mail: liviu.scurtu@auto.utcluj.ro

Iulian LUPEA, Prof. Ph.D. Technical University of Cluj-Napoca, Dept. of Mechanical Systems Eng., 103-105 Muncii Blvd, 400641 Cluj-Napoca, +40-264-401691, e-mail: iulian.lupea@mep.utcluj.ro