30th International Conference
ENGINEERING MECHANICS 2024
Milovy, Czech Republic, May 14 – 16, 2024

INFLUENCE OF SIMPLIFYING THE GEOMETRY OF THE FE MODEL OF THE VEHICLE FRONTAL FULL BARRIER CRASH TEST

Jančář, R.*, Dlugoš, J., Fridrichová, K.*****

Abstract: *The deformation of the vehicle as a result of the accident serves as the input for determining the impact speed for the purposes of the expert report. One possibility of calculating the impact speed is by means of a verified finite element analysis (FEA) of the vehicle impact. However, to develop such a sufficiently accurate finite element (FE) model requires a significant financial and time investment. Such a model contains a number of key simplifications that are difficult to detect without prior experience. Fortunately, there are at least a few freely available detailed FE vehicle models that have been experimentally verified. On these it is possible to investigate the effect of different modelling methods on observed parameters such as overall deformation. This paper is specifically concerned with the sensitivity of FEA results to different levels of vehicle geometry modelling. A Chevrolet Silverado 1500, model year 2014, is used as a detailed FE vehicle model.*

Keywords: Finite element analysis, crash simulation, crashworthiness, accident reconstruction.

1. Introduction

In accident analysis, a number of methods are used to determine crash conditions. The Energy Equivalent Speed (EES) is used to quantify the deformation energy of the vehicle that caused the plastic deformation. The EES value can be determined in various ways such as by expert estimation, comparison, correlation diagram, energy grid, from crash test values and the CRASH3 method. Each of these methods is described in detail by Bucsuházy et al. (2023).

When a vehicle hits a barrier or another vehicle, kinetic energy is converted into deformation energy. This is a system of bodies that can be modelled using the finite element method (FEM). Among the first, this was attempted by Winter et al. (1981) on a DeLorean DMC-12 vehicle. Damage analysis of a 2010 model year Toyota Yaris vehicle using FEM was performed by Numata (2018). Numata investigated the distribution of the deformation energy as a function of the overlap ratio in a frontal impact. Burg (2013) used the results of the finite element analysis (FEA), which he visually compared with the real deformed shape of the vehicle. Based on this, he determined the impact speed. A similar approach was used by Görtz (2016), who compared the deformation depths in a frontal impact on a column determined by FEA and measured on a real vehicle.

Creating a finite element (FE) model of a vehicle for the purpose of assessing its crash deformation energy can be a costly and time-consuming process. Therefore, the objective of this paper is to determine which geometric features have the greatest influence on the results.

---

[*] Bc. Radek Jančář: Institute of Automotive Engineering, Brno University of Technology, Technická 2896/2; 616 69, Brno; CZ, radek.jancar@vut.cz

[**] Ing. Jozef Dlugoš, PhD.: Institute of Automotive Engineering, Brno University of Technology, Technická 2896/2; 616 69, Brno; CZ, jozef.dlugos@vut.cz

[***] Ing. Kateřina Fridrichová: Institute of Automotive Engineering, Brno University of Technology, Technická 2896/2; 616 69, Brno; CZ, katerina.fridrichova@vut.cz

doi: 10.21495/em2024-146

2. Methods

A detailed FE model of the Chevrolet Silverado 1500, model year 2014, was created as part of Project No. DTNH22-13-C-00329 (Singh, 2012) between NHTSA and EDAG, INC. (Fig. 1). The project focused on exploring full-size pickup truck lightweighting options with production processes available between 2020 and 2030. Investigation of the impact of vehicle lightweighting was observed on the verified FE model mentioned above. This model was created using reverse engineering tools such as 3D scan, CAD model reconstruction, material property measurements on real vehicle samples and others. Verification was performed using results from crash tests such as NHTSA test no. 8456 or test no. 8316, by comparing the natural frequencies of the cab or by using torsional and flexural stiffness of the body and frame. This resulted in a FE model containing over 1 400 components and nearly 3 million elements. Glued joints and spot welds were modeled by advanced methods with failure considerations based on e.g. Malcolm (2007).

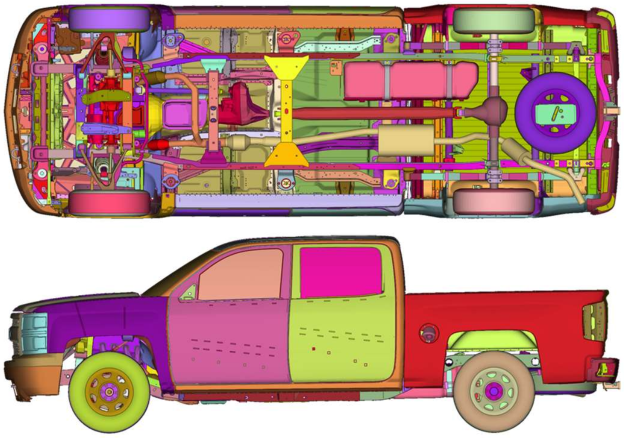

Fig. 1: Detailed FE model of Chevrolet Silverado 1500 (Singh, 2012).

The effect of geometry simplification (Drapal, 2016) on the results is investigated on this FE model. The simplification consists of reducing a discretized component or group of components to a mass point that is connected to adjacent parts by RBE3 (Fig. 2). The geometry modifications are divided into the following groups:

- The body group includes the windshield and rear glass, the cab doors and bucket doors, the front grille and the headlights.
- The accessories group includes exhaust, spare tire, tow bar, tank and tow hooks.
- The interior group includes the dashboard, front seats, steering wheel and pedals.
- The suspension group includes front suspension springs, airbag model tires and brakes.
- The engine bay group includes the battery, radiator and steering.
- All of the above groups at the same.

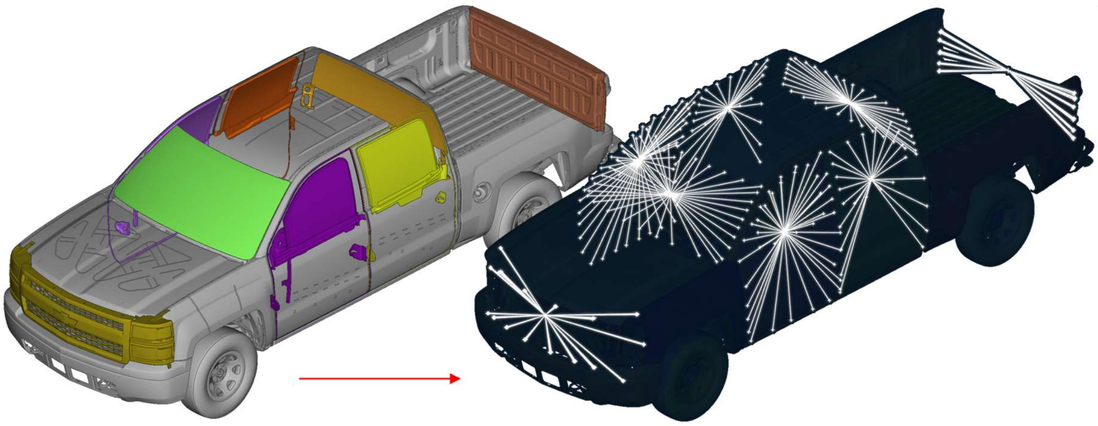

Fig. 2: The body group: replacement of the parts by mass points and RBE3.

The NCAP Frontal Full Barrier Test is simulated. The evaluated outcome parameter is the total longitudinal deformation of the vehicle at 56 km·h^{-1}. From the results of the FEA, it is evident (Fig. 3) that the longitudinal deformation is generally not linearly dependent on the impact speed. This is due to the nonlinearity of the model due to the nonlinear materials and the significant amount of contacts. This nonlinearity is more dominant for higher impact velocities (Bucsuházy et al., 2023; Görtz, 2018). Therefore, in addition to the total longitudinal deformation of the vehicle, the parameter representing speed change required to increase the longitudinal deformation by 1 mm is also investigated. This parameter is determined as a slope of the deformations for speeds of 42 km·h^{-1} and 56 km·h^{-1}.

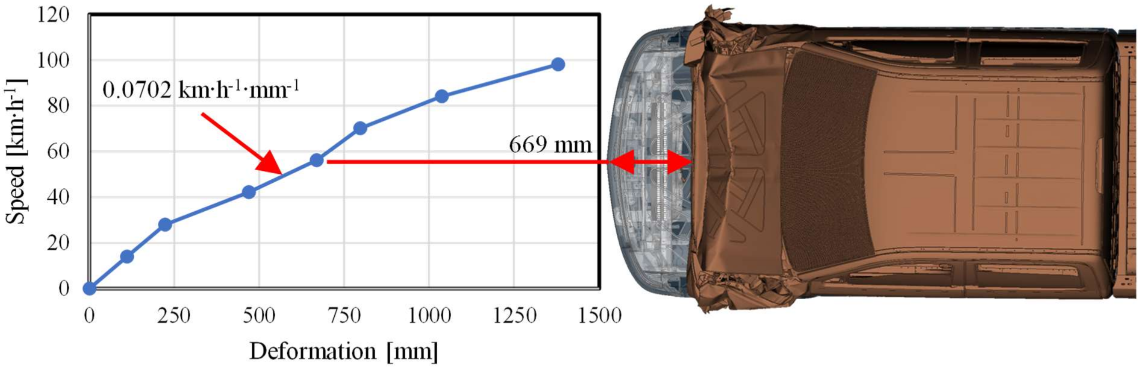

Fig. 3: Output parameters evaluated; the total longitudinal deformation 669 mm and the slope 0.0702 km·h^{-1}·mm^{-1} for the full FE model.

3. Results

The results of the simplified geometry models are shown in Tab. 1. The combination of all groups at once leads to the largest difference from the results of the full FE model. The absence of selected parts and their replacement with mass points and RBE3 reduces the stiffness of the vehicle, leading to an increase in the total longitudinal deflection from 669 mm to 700 mm (an increase of 4.62 %). The engine bay group contributes the most to this decrease in stiffness by reducing components in the front area of the vehicle (battery, radiator and steering). These components are subject to the largest deformations. The radiator, in particular, contributes to a significant change in the overall deformation. In the case of a full FE model, contact between the powertrain and the radiator occurs during the impact. In the absence of the radiator, the powertrain moves significantly further due to the new clearance. This phenomenon can be seen by comparing the deformed shapes of the front of the vehicle at the same time for the FE model with and without the radiator (Fig. 4).

The accessories group has the smallest effect on the deformation. The components such as exhaust, spare tyre, towing bracket, tank and towing hooks do not contribute too much to the vehicle's stiffness. Their influence is primarily through added mass.

FE model	**Deformation [mm]**	**Difference [%]**	**Slope [km·h^{-1}·mm^{-1}]**	**Difference [%]**
Full	669	-	0.0702	-
All groups bellow	700	4.62	0.0623	-11.33
The body group	672	0.37	0.0708	0.78
The accessories group	670	0.14	0.0708	0.82
The interior group	648	-3.13	0.0726	3.39
The suspension group	679	1.43	0.0758	7.99
The engine bay group	695	3.89	0.0641	-8.65

Tab. 1: Results of the impact of geometry simplification.

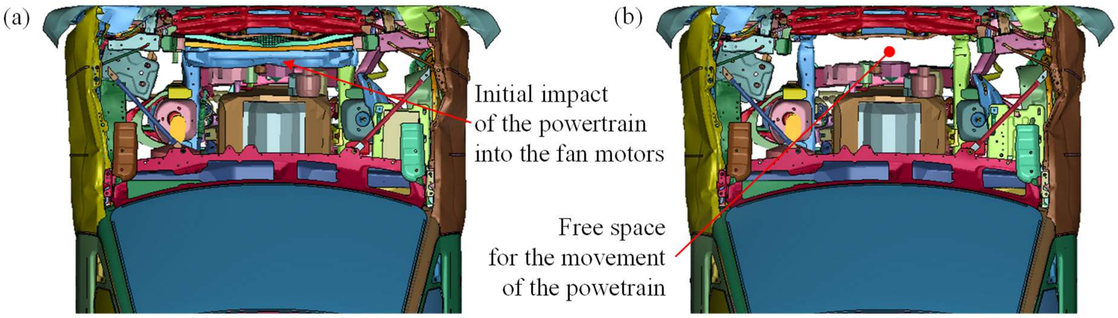

Fig. 4: Deformation of the front of the vehicle at 27.5 ms: a) model with radiator – the full model, b) model without radiator – the engine bay group.

4. Conclusions

In FEA of a vehicle frontal impact, the most important aspect from the geometry model point of view is to capture the area closest to the deformed zone as detailed as possible. This is confirmed by the results of the sensitivity analysis i.e., the engine bay group. The absence of the radiator causes an increase in the total longitudinal deformation of 3.89 %. However, if the parts involved are made of very compliant material such as the plastic front grille or the headlights (the body group), the reduction of these parts to mass points has a minimal effect on the results. The reduction of parts (even sufficiently rigid ones) in the areas far from the most deformed zone using mass points and RBE3 does not significantly degrade the results (the body group and the accessories group).

Interesting results arose when reducing the interior group. In this group, the dashboard (represented by a cross brace), front seats, steering wheel and pedals were replaced by mass points. The RBE3 connection should not result in a stiffer cabin, yet the results provide 3.13 % less deformation compared to the full model. The explanation for this phenomenon requires further investigation.

To better understand the sensitivity of the detailed FE model, it would be useful to investigate other influences such as the material model. The detailed FE model contains materials with multilinear stress diagrams for different strain rates. Further, it would be useful to investigate the influence of shell element formulation, mass scaling settings, and different methods of joining individual parts.

Acknowledgement

The authors gratefully acknowledge the funding from the specific research on BUT FSI-S-23-8235. The authors wish to thank NHTSA for providing Chevrolet Silverado FE model publicly available that enabled this research to be made.

References

Bucsuházy, K., Moravcová, P., Semela, M. et al. (2023) *Vehicle Crush Investigation: A Guidebook to Documentation and Analysis*. 1st edition. Brno.

Burg, H. (2013) Zur Einstufung von EES-Werten bei außergewöhnlichen Unfällen. In: *15 AREC Meeting*, Wildhaus.

Drapal, L. and Voparil, J. (2016) Shape Design of a Piston Ring. In: *Proc. of 20th Int. Sci. Conf. Transport Means*, Juodkrante, Lithuania. Kaunas Univ. of Technology, pp. 767–771.

Görtz, M. (2016) 3D-Vermessung von Schäden an Fahrzeugkarosserien, In: *16. Aachen Interdisziplinäres Verkehrssymposium*. Aachen.

Görtz, M. (2018) *Model for determining deformation energies on vehicles after traffic accidents*. Dissertation, Žilina: UZVV UNIZA, (in Slovak).

Malcolm, S. and Nutwell, E. (2007) Spotweld Failure Prediction using Solid Element Assemblies, In: *6th European LS-DYNA Users' Conference.* Gothenburg, pp. 47–56.

Numata, S. and Okumura, D. (2018) Validation of Crush Energy Calculation Methods for Use in Accident Reconstructions by Finite Element Analysis. *SAE Int. Journal of Transportation Safety*, 6, 2, pp. 133–146.

Singh, H. (2012) Mass Reduction for Light-Duty Vehicles for Model Years 2017-2025. Report No. DOT HS 811 666, Program Reference: DOT Contract DTNH22-11-C-00193. Contract Prime: Electricore.

Winter, R., Mantus, M. and Pifko, A. B. (1981) Finite Element Crash Analysis of a Rear-Engine Automobile (No. 811306). SAE Technical Paper.