Sensitivity study on the equivalent mechanism model for conceptual design of vehicle body crashworthiness

Sensitivity study on the equivalent mechanism model for conceptual design of vehicle body crashworthiness

S. Candreva[1], D. Straface[1], C. Garre[1], D. Mundo[1], L. Farkas[2], S. Donders[2], P. Mas[3]
[1] Department of Mechanical, Energy and Management Engineering, University of Calabria
Ponte Pietro Bucci, 87036 Rende, Italy

[2] Simulation Division – LMS, A Siemens Business
Interleuvenlaan 68, 3001 Leuven, Belgium

[3] Engineering Services – LMS, A Siemens Business
84 Quai Charles de Gaulle, 69006 Lyon, France

Abstract
The present work focuses the attention on the sensitivity study of the *equivalent mechanism (EM)* conceptual modelling technique for vehicle body crashworthiness analyses. A library implementation of EM was presented by the authors in a previous work. This paper presents an extension by treating a more complex application case: a planar beam assembly that follows the approximate topology of the front side part of a vehicle. The analysis case consists of an impact against a rigid wall. A sensitivity analysis is performed w.r.t. variations in the geometry, using detailed FE simulations as a reference to validate the procedure in terms of accuracy. The sensitivity analysis covers two slight modifications in the geometry of two joints. The results show that the deformation modes of the EM model vary in agreement with the modes predicted through detailed FE simulations and that an acceptable correlation between the two simulation models is achieved in terms of rigid wall displacement and deceleration curves.

1 Introduction

The term *crashworthiness* refers to a measure of the capacity of the vehicle structure and each of its components to protect passengers' lives in case of crash. The vehicle structure is crucial for guaranteeing the safety of the passengers. It must absorb the kinetic energy of the vehicle in a controlled manner (keeping crash deceleration pulse below the threshold of human resistance), and limit the deformation of the cabin for ensuring a minimal survival space for the passengers.

The analysis of the crashworthiness of a vehicle is a highly complex task and requires the development of specific tools to understand and improve the crash behaviour of the structure. In the last decades, the number of tools available to help engineers make the correct design choices toward crashworthiness have been increasing, thanks mainly to the development of non-linear finite element (FE) techniques and to the evolution of computers. Running computerized simulations of vehicle crashes using FE methods is a very useful approach, but these simulations are still too expensive in terms of (CPU and elapsed) time. For this reason, FE simulation is not the most appropriate tool in the initial phases of the vehicle development process, when only limited data is available and a lot of design alternatives are to be evaluated. In this concept phase, the availability of tools to generate design variants in less time is crucial as well as the need to have models that can be simulated with affordable computational cost, even with some compromise regarding the accuracy. The main goal in this phase is not to reproduce real behaviour of the structure with a high level of accuracy, but rather to establish a domain of admissible solutions that could be explored later with more accurate simulations, and to identify the right 'trends': identify design modifications that improve the performance.

In literature, there exist different approaches for crashworthiness analysis, some of them oriented to the first stages of the design (concept models) and other for a more advanced phase when all the geometric details must be defined (detailed models).

Regarding detailed models, topology optimization [1] is a technique which can be used to optimize the topology of beam components. In this way, a mass reduction can be achieved while keeping the same energy absorption in the case of impact with a mass with a specific direction and velocity. In [2] the authors presented the mode match adjustment method for a front impact, consisting in modifying the design variables of the structure for obtaining the desired deformation history. An expert engineer can estimate the proper deformation history and introduce the necessary modifications to the structure to achieve it. In this way, an acceptable solution can be obtained running only a few FE simulations. Surrogate models are general-purpose meta-models (model of models) used in a lot of engineering applications [3]. In applications implying models with an elevated computational cost, a meta-model can represent a convenient solution. The problem of these models is the high level of abstraction and the lack of physical sense. Once the model is built, its performance (such as the deformation energy) varies depending on the input variables (such as the dimensions of the structure), but it gives no insight on which element of the structure is more or less deformed.

Concerning concept models, some authors propose to use simplified FE models [4]. These models are characterized by the simplification of mesh and elements formulation, allowing a reduction of computational cost. The problem of these models is that, if a very coarse mesh is used, the results can be too inaccurate, while using a too big time step may cause the loss of information about the transient phase of the solution and thus the estimated deformation history could be completely different from reality. The simplified formulation of the elements may also result in deformations without associated energy (the so-called 'hourglassing'). A compromise between simulation time and accuracy is then needed and very often the simulation time must be very largeto keep a reasonable level of accuracy.

Kamal [5] introduced lumped-parameter models for simulating the response of a vehicle under frontal impact. This model approximates the vehicle with a 1D mass-spring system. Due to the high simplification of the representation, this model requires engineers with a lot of experience in the field of structures to parameterize springs and masses and to translate the output into design data. With such a model it is very difficult not only to setup the mass-spring system but also to understand how to modify the structure once the results from the simulation are obtained.

To overcome such difficulties, new simplification methods were developed with the aim to obtain the behaviour of the vehicle starting from the behaviour of the components of the structure (i.e. shells and beams). The approximation of these elements is the core of the most recent techniques for physics-based concept modelling of vehicle structures for crashworthiness. New lumped-parameters models were developed focusing not only on the global response of the structure but rather on the behaviour of each component. The global response can then be obtained by correctly assembling the components that form the structure. Among the many choices of concept models developed for this purpose, equivalent mechanisms (EM) models proposed by Karim et al. [6] present the advantage of being physics-based and thus allow an easy interpretation of the simulation results. The core idea of the method is that the main components of the vehicle collapse structure, which is typically modelled with shell finite elements, can be approximated with a network of rigid bodies connected through prismatic and revolute joints (characterized by springs with non-linear stiffness). These springs are set up to replicate the force-displacement (or torque-rotation) characteristic of the beam components of the whole structure. To parameterize the non-linear springs, some FE simulations must be run first.

A library implementation of EM concept modelling technique within LMS Imagine.Lab AMESim environment for conceptual design of vehicle body crashworthiness was presented by the authors in a previous work [7]. The library contains a set of already characterized components, such as beams or joints, which can be easily reused on many different assemblies. Modelling and testing a new design variant with the implemented library requires only placing the components with the mouse (thanks to the intuitive GUI of AMESim) and running the simulation of the concept model, which is achieved in a few minutes or hours depending on the complexity of the model, compared with the many hours or even days required for the corresponding detailed FE simulation.

The present work focuses the attention on the sensitivity study of the EM conceptual modelling technique, extending previous work by the authors through the analysis of a more complex application case. The latter consists in a planar collapse structure, approximating a vehicle front part that impacts against a rigid wall. For the sensitivity analysis, two simulations are run with slight modifications in the geometry of two joints. The results show that the deformation modes of the EM model vary in agreement with the modes predicted through detailed FE simulations and that an acceptable correlation between the two simulation models is achieved in terms of rigid wall displacement and deceleration curves.

The remainder of the paper is organized as follows. Section 2 presents a short summary of previous work related to the EM concept modelling technique for crashworthiness analysis implemented in AMESim. Section 3 illustrates an application of the mentioned library and a sensitivity study on a complex application case. Conclusions are given in Section 4.

2 A library for concept crash analysis

In previous authors' work, a library has been developed in the environment LMS Imagine.Lab AMESim, as a tool for modelling and simulation of vehicle structures for crashworthiness, using the EM concept model. The library consists of ten super-components, i.e. formed by elements already implemented in Planar Mechanical and Signal Control libraries of AMESim, allowing assembly and simulation of any type of structure. Basically, the library is divided into three groups: Beams, Joints and Boundary Conditions.

Beam components are used to approximate the behaviour of a real beam structure. Joints are components allowing to link two or more Beam components, so that the whole structure is built in a modular fashion. The Boundary Conditions allow to clamp a Beam component or to define the contact (impact) with a rigid wall.

2.1 Beam Component

Beam component allows to simulate the behaviour of a beam structure after an impact. Figure 1 shows a simplified model of the Beam component in AMESim sketch mode.

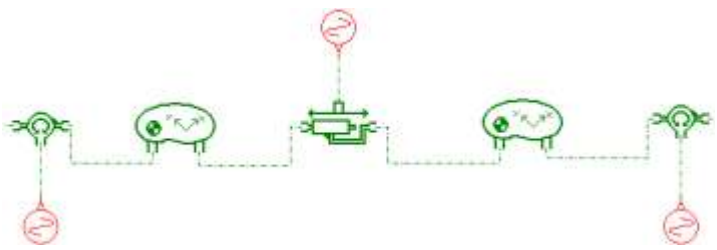


Figure 1: Schematic representation of a Beam component.

A prismatic joint is in the middle, coupling two masses with half of the value of the beam mass. At the other extremity of each mass a pivot joint is connected. The blocks in red contain the characteristic curves (force/displacement or torque/rotation) of the joints, obtained from detailed FE simulations, involving the beam structure under axial and flexional loads separately. The prismatic joint coupling the two masses is responsible for applying the reaction force to the axial displacement of the masses. The two pivot (revolute) joints simulate the formation of plastic folds and exert a reaction torque in consequence to relative rotation of other interconnected Beam components.

As an example, the result of a FE simulation for the characterization of a pivot joint of the Beam component is shown in Figure 2.

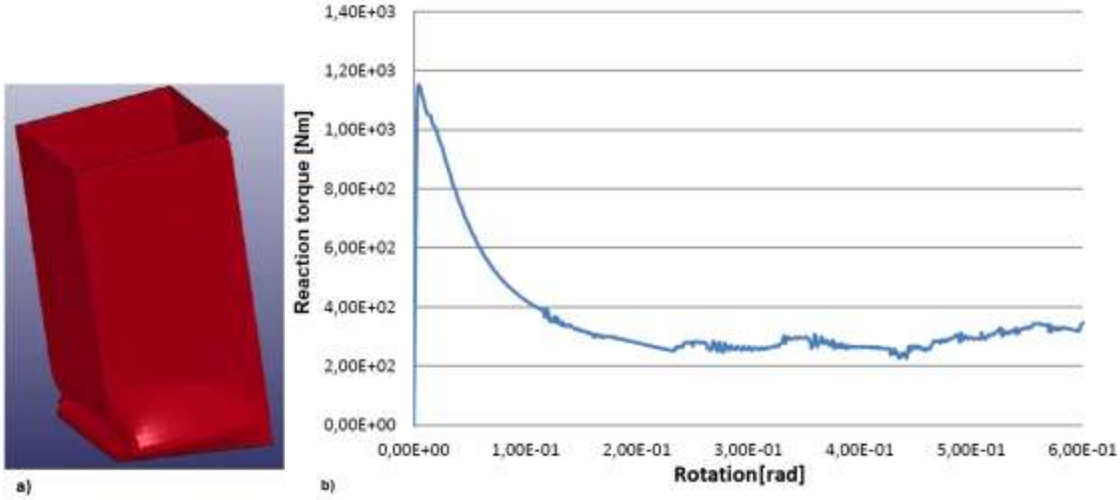


Figure 2: a) Deformation of the beam due to bending torque and b) curve of reaction torque with respect to rotation.

2.2 Joint Component

Joint components allow to interconnect several Beam components and to consider the contact of the structure with a rigid wall. They are used to model the critical parts of the structure with a behaviour that cannot be properly described using only Beam components, such as the intersection points of various beams with different angles. Various types of Joint components have been implemented. Figure 3 shows the schematic representation of one of the Joint components, which is a T-joint allowing coupling of three Beam components.

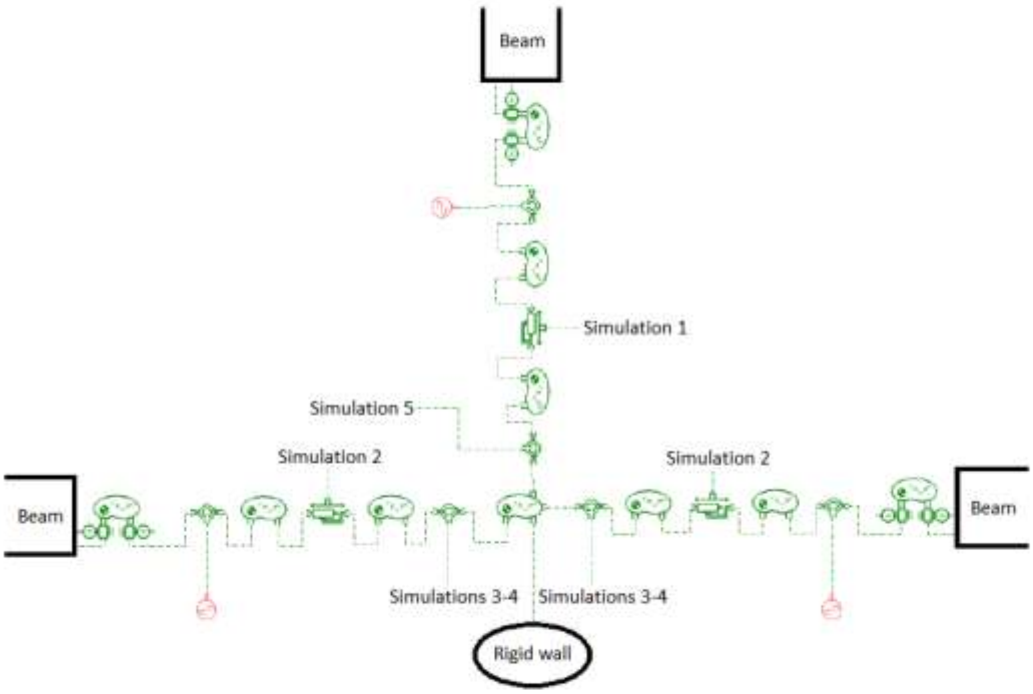


Figure 3: Schematic representation of a T-Joint component. The characteristic curves are assigned from different simulation cases at different points of the model.

This Joint component is composed of three Beam components coupled to a central rigid body with four ports: three of them allow to connect the Beam components while the fourth is available for a potential contact with a rigid wall. This component is composed of prismatic and pivot joints to be characterized. To obtain this characterization, FE simulations are conducted on the joint structure depicted in Figure 4.

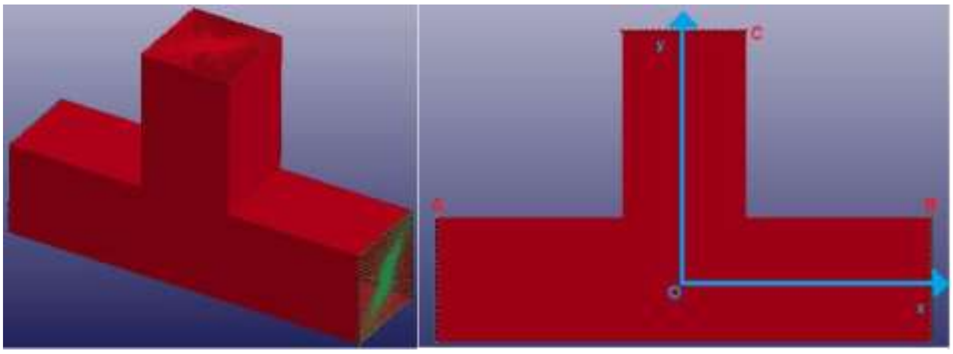


Figure 4: FE model of the three-beam joint.

Five simulations are run for this example, two with axial load and three with bending load:

- **Simulation 1**: axial load on the vertical arm (-Y direction).
- **Simulation 2**: axial load on the horizontal arm (-X direction).
- **Simulation 3**: bending load on the horizontal arm in upward direction (Y).
- **Simulation 4**: bending load on the horizontal arm in downward direction (-Y); the configuration is the same of the previous case, but since the structure is not symmetric with respect to X axis, the torque must be evaluated in both directions.
- **Simulation 5**: bending load on the vertical arm.

As an example, Figure 5 shows joint deformation and reaction torque curve obtained from Simulation 3.

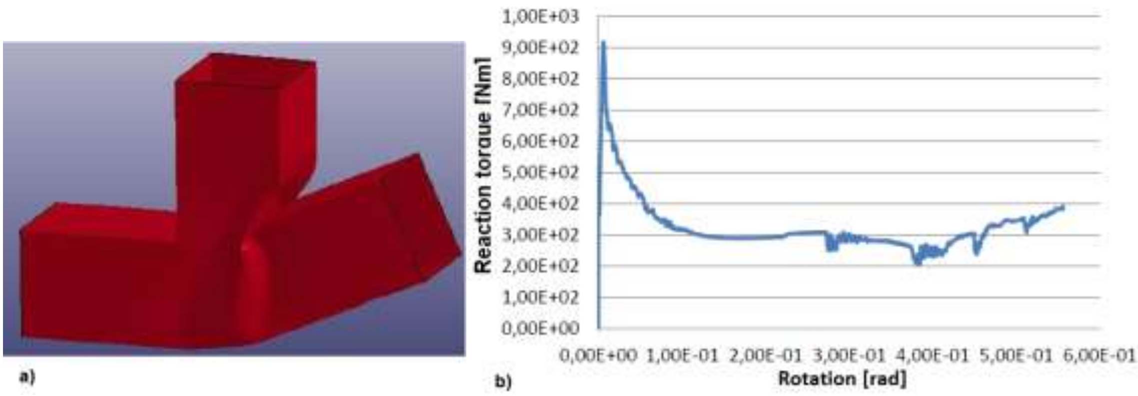


Figure 5: Simulation case of a rigid wall of 750 Kg impacting at one end of the T-joint with initial speed of 1 m/s. The other two ends are clamped to observe the generation of a plastic fold close to the point of intersection of the three beams. a) Final deformation. b) Corresponding reaction torque/rotation curve.

The obtained curve is clearly different from the curve obtained with the simulation of independent beam.

The same procedure is followed for the characterization of all the other Joint components. This characterization is needed to obtain accurate enough results, since the behaviour of these critical parts of the structure has a big influence on the global response of the structure itself. In these parts, the large plastic deformations are located and the resistance to collapse is lower with respect to a single beam because of geometric discontinuities.

2.3 Boundary Condition Components

Boundary Condition components allow to constrain the degrees of freedom of a Beam component (Clamp component) or to model the contact with a rigid wall (Rigid Wall component). A schematic representation of those two possibilities offered by the Boundary component is given in Figure 6.

For the Clamp component, a simple element from the Planar Mechanical library of AMESim is used, providing zero displacement and velocity to the connected beam.

The parameters used for contact modelling are: the elastic modulus of both bodies, the damping of the contact and the type of friction. This component requires also the definition in the plane of the contour of both objects in contact.

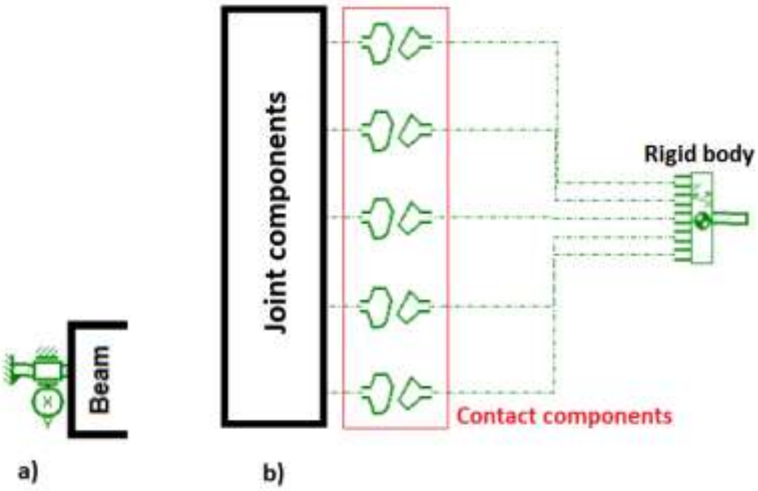


Figure 6: Schematic representation of Boundary Condition components.
a) Clamp component, b) Rigid Wall component.

3 Application case and sensitivity study

As application case, a complex structure with several critical parts, impacting with a rigid wall of 2000 kg with an initial speed of 1 m/s is considered. Figure 7 a) shows the structure with critical parts marked in green and the direction of impact with a blue arrow.

The structure is clamped at *A* and *B* ends. The section of the beams is square, with 50 mm of width and 1 mm of thickness. All the beams have a length of 200 mm. The material is elastic and perfectly plastic with an elastic modulus of 207 GPa, a yield point of 240 MPa, a Poisson coefficient of 0.3 and a density of 7800 kg/m^3. The modelling phase of the structure with the implemented library was very fast, requiring only to place few components in the sketch. Figure 7 b) shows the structure in AMESim Sketch.

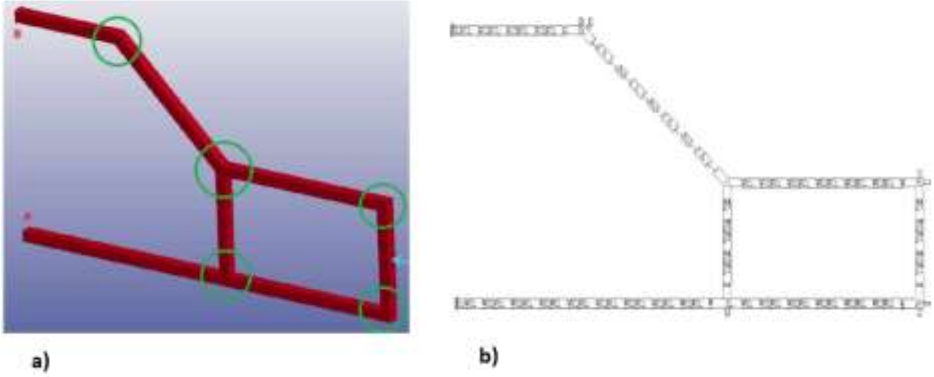


Figure 7: a) FE model of the complex structure and b) complex structure in AMESim.

The deceleration and the displacement of the rigid wall is compared with the corresponding detailed FE simulation, as shown in Figure 8.

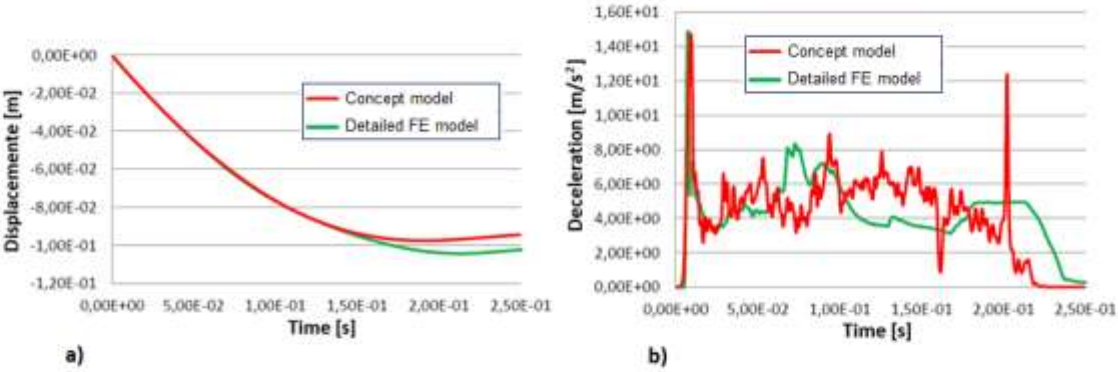


Figure 8: Comparison between the detailed FE model and the concept model. a) Curve of displacement of the rigid wall with respect to time. b) Curve of deceleration of the rigid wall with respect to time.

The displacement curves are almost coincident, having a maximum error of 6% at the end of the simulation. The deceleration curve shows a good coincidence in the initial phase, and after the collapse both curves oscillate around a value close to 5 m/s^2. The prediction of the deceleration peak presents an error around 1.5%. The saving in computation time is very remarkable: 3 days for the detailed FE model and 1.5 hours for the concept model, with a time reduction factor of 48.

Figure 9 shows the deformation of the structure obtained with the detailed FE simulation and with the concept model.

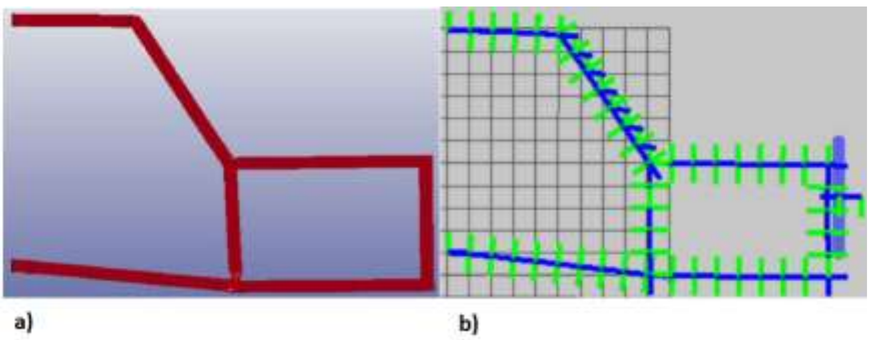


Figure 9: Visual comparison of the deformation on the a) detailed FE simulation and the b) concept model.

The concept model approximates very well the deformation mode, showing the structural failure on the lower part of the structure.

To measure the sensitivity of the EM concept model to variations in the geometry of the structure, slight modifications are introduced in some points. In particular, the structure is stiffened with the addition of material in some critical parts, at the intersection between some beams. The modified parts are marked in green in Figure 10.

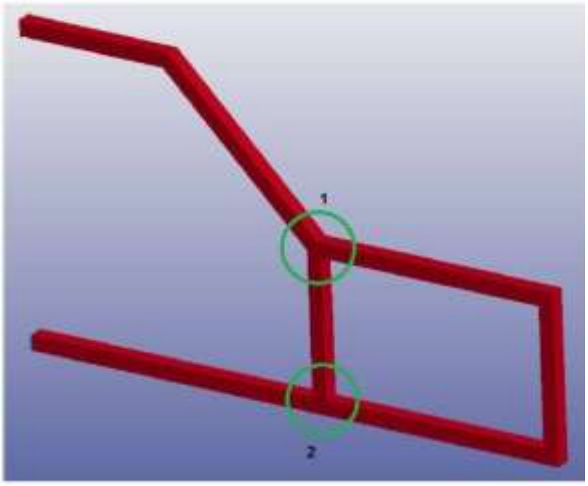
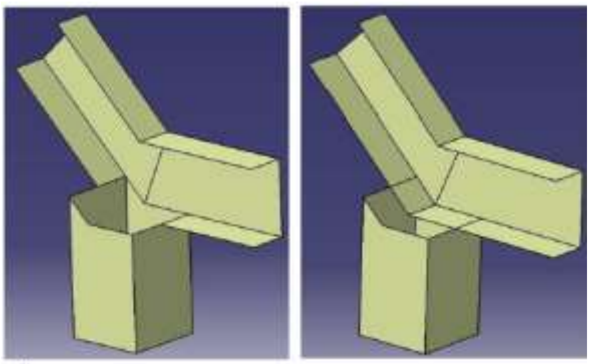


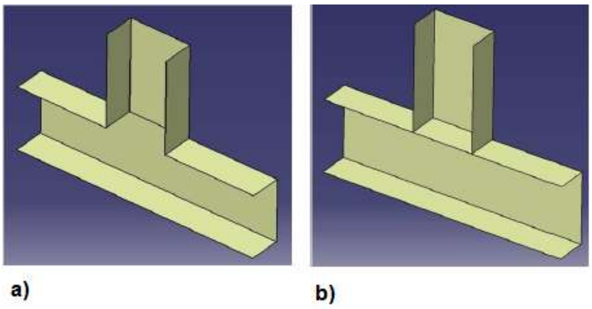
Figure 10: Modified critical parts.

Figures 11 and 12 show the modifications introduced in the two T-joints of the structure, named hereafter, respectively, critical part 1 and critical part 2.



a) b)

Figure 11: Modification of the critical part 1. a) Original part and b) stiffened part.



a) b)

Figure 12: Modification of the critical part 2. a) Original part and b) stiffened part.

To apply the above-described modifications to the EM concept model, new detailed FE simulations, regarding the modified parts, are needed to characterize the correspondent Joint components. Again, the results obtained with the modified concept model are compared with the detailed FE simulation of the modified structure. The deceleration and the displacement of the rigid wall are compared with the results from the corresponding detailed FE simulation, as shown in Figure 13.

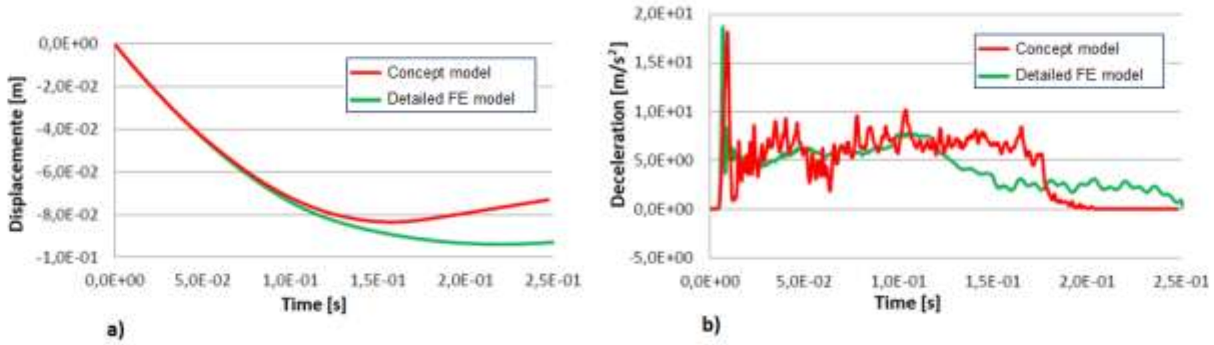


Figure 13: Comparison between the detailed FE model and the concept model. a) Curve of displacement of the rigid wall with respect to time. b) Curve of deceleration of the rigid wall with respect to time.

The curve of displacement obtained with the concept model shows again an initial part coincident with the detailed simulation, and a second part with a maximum error close to 10%. Regarding the deceleration curve, the concept model predicts the deceleration peak with an error around 3%. After the collapse of the structure, both curves oscillate around a value close to 5 m/s^2. Figure 14 shows the deformation of the structure obtained with both simulations.

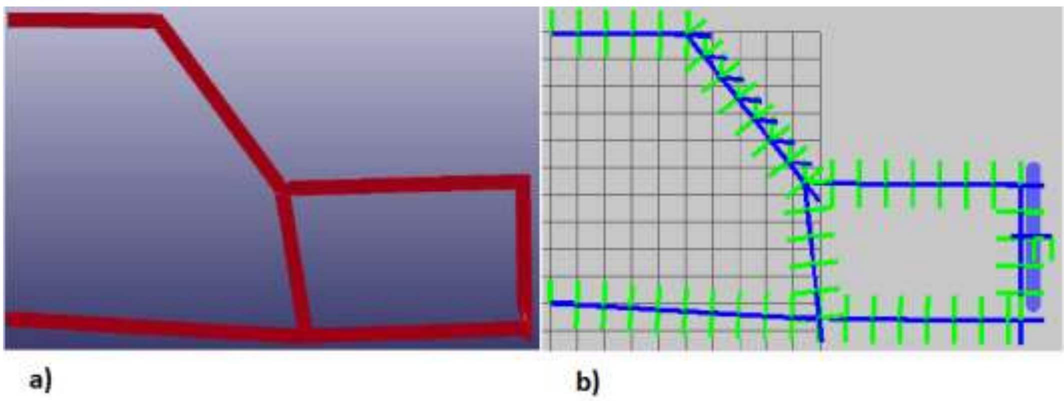


Figure 14: Visual comparison of the deformation on the a) detailed FE simulation and the b) concept model.

Figure 14 shows a different deformation mode compared with the case without the modifications. In the case without modifications, the failure is observed in the lower part (critical part 2) whereas after making the modifications there is a first failure in the front part and then in critical parts 1 and 2. This initial failure of the front part can be explained considering that critical part 2 is the most compliant part in the previous case. In the second case, critical part 2 is reinforced, causing limited deformations in this zone and new deformations in other parts such as the front one that, being subject to shear load on its section, has failed. It can be concluded that the EM concept model is sensitive to variations in the deformation mode caused by small and localized variations in the geometry of the structure and allows to predict the structure deformation with much lower demands of computational time but still with sufficient accuracy.

4 Conclusions

The presented work is an extension of a previous work by the authors and related to the implementation of the equivalent mechanism (EM) model in LMS Imagine.Lab AMESim environment, for conceptual analysis of vehicle body crashworthiness. The implemented library consists of ten components, allowing to assemble geometrically complex structures subject to boundary conditions. The modelling phase is extremely simple using the library, requiring only to place and connect the different components in the Sketch layout.

In this paper, a complex application case of the library was presented,. The analysis case consists in a planar collapse structure, approximating a vehicle front part that impacts against a rigid wall. Also with this complex application case, the results obtained in terms of rigid wall displacement and deceleration curves were accurate enough for conceptual design and with very reduced computation time with respect to the corresponding detailed FE simulation.

Moreover, the sensitivity of the EM concept model to variations in the geometry was assessed, by comparison with detailed FE simulations. For the sensitivity analysis, slight modifications in the geometry of two joints were considered. The results showed that the deformation modes of the EM model vary in agreement with the modes predicted through detailed FE simulations and that an acceptable correlation between the two simulation models is achieved in terms of rigid wall displacement and deceleration curves.

Acknowledgments

The research leading to these results has received funding from the People Programme (Marie Curie Actions) of the European Union's Seventh Framework Programme FP7/2007-2013/ under REA grant agreement n° [285808] – INTERACTIVE: Innovative Concept Modelling Techniques for Multi-Attribute Optimization of Active Vehicles (www.fp7interactive.eu).

References

[1] Neal M. Patel, Byung-Soo Kang, John E. Renaud, Andrès Tovar. *Crashworthiness design using topology optimization*. Journal of Mechanical Design, 2009, Vol. 131, No 6.

[2] Karim T. Hamza. *Design for vehicle structural crashworthiness via crash mode matching*. Ann Arbor, Michigan: University of Michigan, 2008.

[3] Karim Hamza and Kazuhiro Saitou. *Vehicle crashworthiness design via a surrogate model ensemble and a co-evolutionary genetic algorithm*. Long Beach, California: ASME IDETC/CIE Conference, 2005.

[4] Paul Du Bois, Clifford C. Chou, Bahig B. Fileta, Tawfik B. Khalil, Albert I. King, Hikmat F. Mahmood, Harold J. Mertz, Jac Wismans. *Vehicle crashworthiness and occupant protection*. Southfield, Michigan: American Iron and Steel Institute, 2004.

[5] M. M. Kamal. *Analysis and simulation of vehicle to barrier impact*. SAE Technical Paper, 1970.

[6] Karim Hamza, Kazuhiro Saitou. *Design optimization of vehicle structures for crashworthiness via equivalent mechanism approximations*. Journal of Mechanical Design, 2004, Vol. 127, No 3, pp. 485-492.

[7] C. Garre, D. Straface, S. Candreva, D. Mundo, L. Farkas, S. Donders, P. Mas. *A library for conceptual design of vehicle body crashworthiness using equivalent mechanism*. Proceedings of the 2014 FISITA World Automotive Congress, Maastricht, The Netherlands, 2014 June 2-6.