Small electric car front cross-member assembly low speed impact simulation

Ioannis Diamantakos[1], Iratxe Lopez Benito [2], Konstantinos Fotopoulos[1], George Lampeas[1]

[1]Laboratory of Technology and Strength of Materials (LTSM), Department of Mechanical Engineering and Aeronautics, University of Patras, Patras 26500, Greece
[2]Iratxe Lopez Benito, Batz S. Coop., Torrea, 2, 48140 Igorre, (Bizkaia) - Spain

1 Summary

In the frame of EVOLUTION project a small electric car has been developed. One of the demonstrators of car structure design is the front cross-member assembly. The basic features of the front cross-member assembly structure comprise the application of foam type materials and crash boxes design, which are assembled behind a transversal beam; both features aim at the maximum possible impact energy absorption during a crash event.

In the present work the numerical simulation of the front cross-member assembly low speed impact is presented. The crash is considered according to Allianz test protocol. FE models of all assembly parts are built and the FE model of the whole front cross-member assembly is constructed applying proper contact definitions and initial conditions. The developed FE model is solved using LS-DYNA explicit FE code. Strain-rate depended material properties are utilized for the foam materials.

For the validation of the numerical model a physical test according to Allianz test protocol has been used. Numerical simulation results compare well with experimental test measurements, leading to successful validation of the developed FE model.

2 Introduction

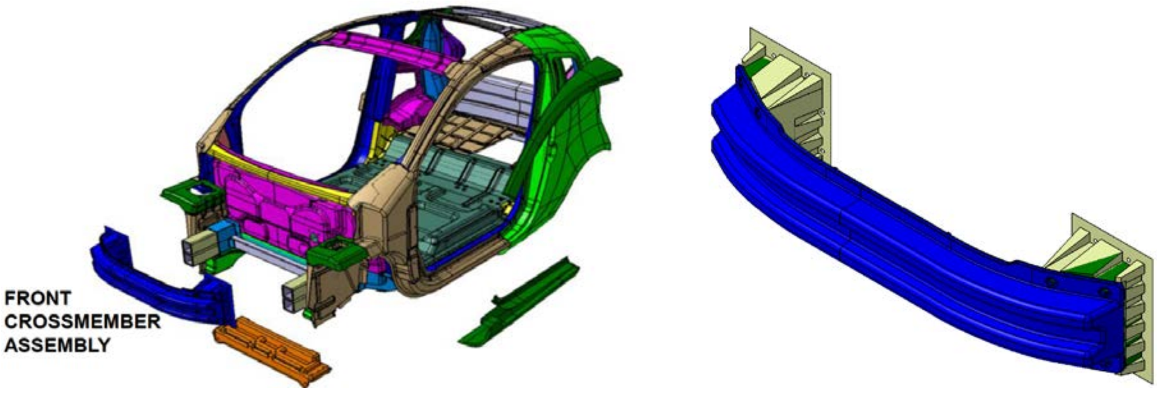
EVOLUTION [1] has been a research project funded by the European Commission dealing with the development of a small, modular electric car. In the frame of this project a number of car sub-structures (called demonstrators) have been developed. One of these demonstrators was the front cross-member assembly, positioned in the front of the car structure, aiming in absorbing impact energy during crash. In order to increase the impact energy absorption capabilities of the structure, crash boxes have been designed to be assembled behind the transversal beam of the car bumper. Additionally, crash boxes and part of the transversal beam structure have been filled with foam type materials (more specifically low density poly-urethane foams).

Numerical simulation is widely used for the assessment of cars structural behaviour due to crash incidents (e.g. [2 - 6]), as it offers a convenient and cost effective mean to test different design concepts and configurations and evaluate their effect on the safety and integrity of car structures. In the frame of EVOLUTION project, numerical simulations of the front cross-member assembly low speed impact have been performed and validated, in order to provide a validated FE model of the front cross-member assembly to be used in the whole car global FE model development. Strain rate depended material properties have been derived from respective experimental tests of the foam materials used. Consequently, the FE model of each structural part of the demonstrator part has been built. Combining the individual parts FE models and applying proper contact definitions, the FE model of the whole front cross-member assembly has been constructed.

For the validation impact according to Alliance Center for Technology (AZT) low-speed front crash test protocol (AZT test protocol) [2] has been considered. The developed FE model has been solved using LS-DYNA explicit FE code. Numerical simulation results concerning structural deformations, absorbed impact energy, displacements, speed and acceleration of the vehicle during impact evolution have been drawn and compared to respective experimental measurements obtained during a physical test under the same conditions.

3 Front cross-member assembly geometry and materials

The front cross-member assembly is placed in the front of the car structure, as presented in Fig 1(a). Its main objective is to absorb impact energy during a low or high speed crash.



(a) (b)

Fig.1: (a) Front cross-member assembly position in car design [7], (b) front cross-member assembly

The front cross-member assembly consists of a transversal beam and two crash boxes positioned behind the transversal beam (shown in Fig. 1b). The transversal beam consists of a front cover, a rear cover, while the transversal beam front cover stiffness is improved by internal ribs. Part of its interior is filled with low density Poly-Urethane (PU) foam, with density of 100g/l. The transversal beam front and rear cover material is PPG Fabric, while the adjacent ribs are made of PPNNCFG50 material. The raw material used for the crash boxes is XENOY, while crash boxes interior has been filled with low density (80g/l) PU foam.

4 Foam materials simulation

Foams are materials manufactured by some expansion process. Foam material numerical simulation is quite tricky, due to the particular mechanical behaviour of the specific materials. Foams can be classified as elastic or crushable foams [8] and several material models exist in LSDYNA code for these two types of material behaviour. As poly-urethane foams behaviour (elastic or crushable) could not be defined a-priori, both types of material models have been studied.

Experimental testing results from quasi-static and dynamic uniaxial compression tests have been used for the definition of foam material model parameters. Strain rates from $0.0133s^{-1}$ to $200s^{-1}$ have been used in the tests. PU foams with densities of 80g/l and 100g/l have been tested. Based the above mentioned data, the PU foams material behaviour data have been extrapolated, which have then been used in the front cross-member impact FE simulation. The respective curves according to LS-DYNA user's manual [9] should contain no more than 100 points, preferably equidistant along the abscissa. Therefore the experimental data have been properly processed, in order to extract the necessary tables.

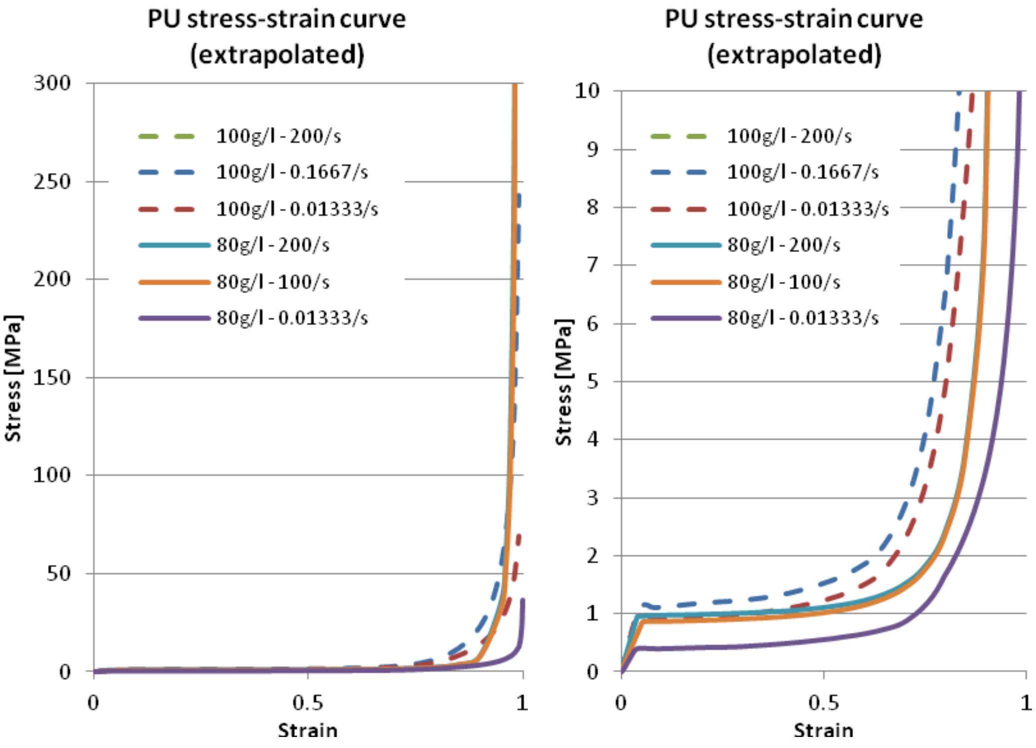
The first step is to obtain a single stress-strain curve for each combination of material / strain rate. Quasi-static and high strain-rate curves for each material are then plotted together and any intersections are eliminated. These intersections are more likely in the densification phase. Each of the curves in the densification phase is then differentiated (the tangent modulus is calculated) to determine where their slope stops increasing. As the extrapolated stress-strain curves from the testing data in the present cases terminated much lower than the stresses likely to occur locally in numerical simulations, extrapolation of the smoothed stress strain curves for higher stress values is necessary. A hyperbolic function of order n is used for this purpose [8],

$$\sigma_{n+1} = \sigma_n + \left.\frac{\partial\sigma}{\partial\varepsilon}\right|_{\varepsilon_1} \left(\frac{1-\varepsilon_1}{1-\varepsilon_n}\right)^n \Delta\varepsilon;\ \varepsilon_n > \varepsilon_1 \tag{1}$$

where *n* is defined as,

$$n = \frac{\ln\left(\dfrac{\sigma_2 - \sigma_1}{\left.\dfrac{\partial\sigma}{\partial\varepsilon}\right|_{\varepsilon_1}\Delta\varepsilon}\right)}{\ln\left(\dfrac{1-\varepsilon_1}{1-\varepsilon_n}\right)};\ \varepsilon_2 > \varepsilon_1 \tag{2}$$

The resulting (extrapolated) stress-strain curves for the studied cases, that will be used for the numerical simulation of cross-member assembly are presented in Fig. 2(a). In Fig. 2(b) the same stress-strain curves for stress values up to 10MPa are presented, where the curves "plateau" is shown. The extrapolated material properties in tabular form have been used for the definition of material model tables in LSDYNA code.



(a) (b)

Fig.2: Extrapolated PU foam stress-strain curves to be used in FE analysis (a) full curves, (b) part of curves for low stress values

From this study on the simulation of compressed PU foams the following conclusions have been drawn:

- Either hexahedral or tetrahedral elements can be used for the foam material simulation.
- `*MAT_CRUSHABLE_FOAM` (MAT 63) and `*MAT_MODIFIED_CRUSHABLE_FOAM` (MAT 163) seem to be the most appropriate LS-DYNA material models for the simulation of low density PU foam material behaviour.
- If hexahedral elements are used, hourglass control defined by parameter `IHQ 4, 5` or `6` lead to more accurate prediction of the material behaviour as compared to `IHQ 1, 2` or `3`. On the other hand tetrahedral elements are insusceptible to hourglass phenomena.
- Element size of 4-6 mm is sufficient for the accurate simulation of foam material behaviour

5 Front cross-member assembly impact simulation

The FE model of the front cross-member assembly has been developed and solved using LS-DYNA explicit FE code. For material modelling, different material cards have been used taking into account the strain-stress curves of each raw material. Furthermore, strain-rate dependent material properties have been used (where available) in order to enhance simulation accuracy. The FE model has been updated according to the latest material curves and information provided by EVOLUTION project partners. In Fig. 3 the FE models of the several parts comprising the front cross member assembly are presented in an exploded, isometric view.

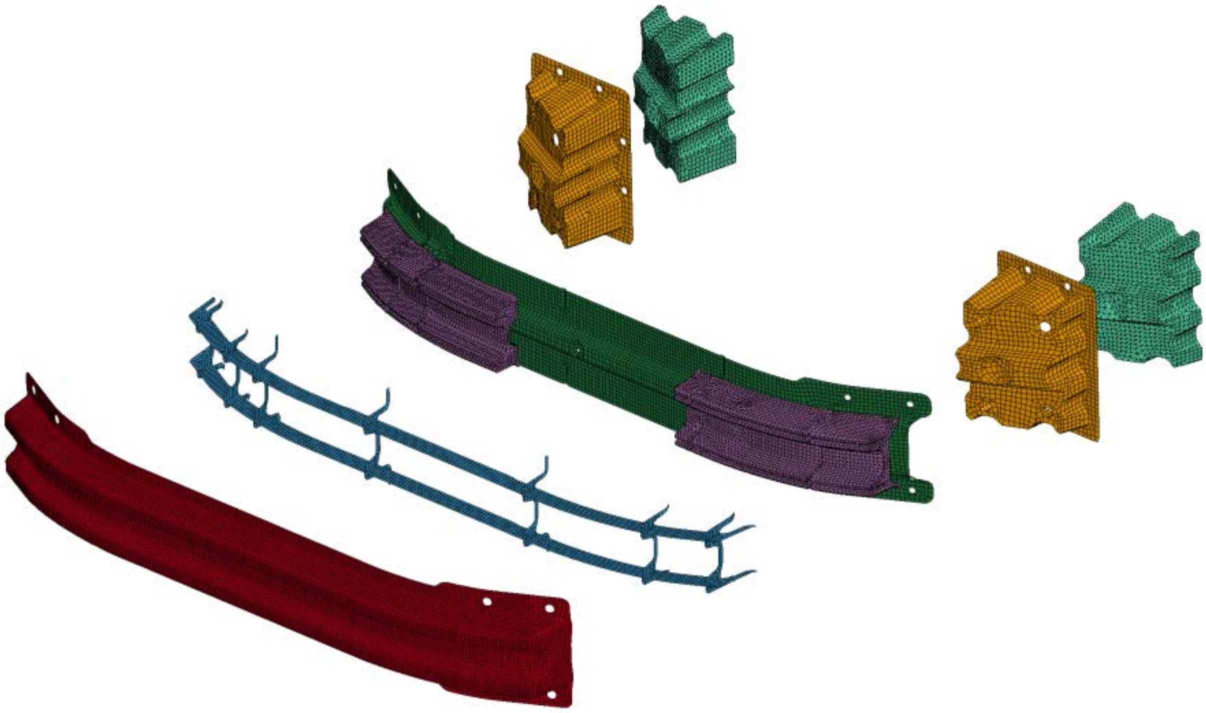


Fig.3: Exploded view of the FE model parts

The mule behind the front cross-member assembly has not been explicitly modelled, due to lack of information concerning its geometry. Therefore, apart from the front cross-member assembly, two plates, designated as mass plates, positioned at the back of crash-boxes (see Fig. 4), are used to simulate the mass of the vehicle (mule) behind the front cross-member assembly. The total vehicle mass is considered to be 1120kg, while the front cross-member assembly is about 6.69kg. Thus, each mass plate has a total mass of 556.65kg, which is achieved by considering an almost rigid material model with appropriate density value.

The physical impact test has been conducted according to AZT test standard. The parameters related to the barrier positioning and dimensions (related to Fig. 5) are D=172mm, A=10° and R=150mm. An initial speed of 2.75 m/s has been defined for the cross-member assembly. Nodes of the mass plates are constrained in all directions, except the impact direction. The displacement of all the nodes of both mass plates parts are common in the initial velocity direction. This boundary condition has been selected, as the mule holding the front cross-member assembly has been considered to move on tracks. Finally, the barrier is considered rigid and fixed in space.

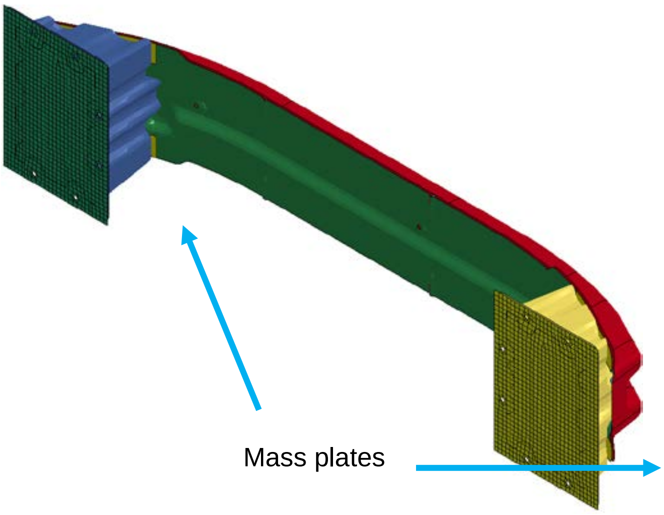


Fig.4: Mass plates positioned behind crash-boxes

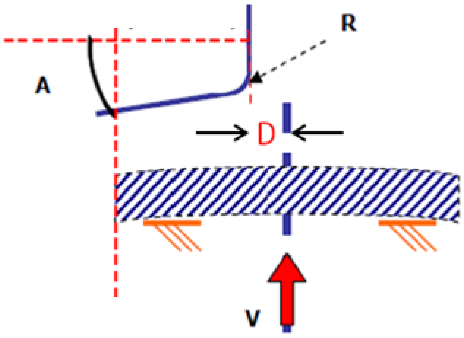


Fig.5: Simulated AZT test configuration

However, in the physical AZT test performed by FPK, after the front cross-member assembly impacted the barrier, the mule has been rotated around the vertical axis. This is illustrated in Fig. 6, where the rotation of the plates holding the front cross-member assembly is depicted by the dotted line of the right picture. Such behaviour could not be simulated by the FE model, because it would require modelling of the whole mule structure, whereas in the present case the whole mule has been simulated by the two mass plates as described above. This difference between the actual and simulated phenomenon is expected to add some discrepancies between the test measurements and the numerical simulation results.

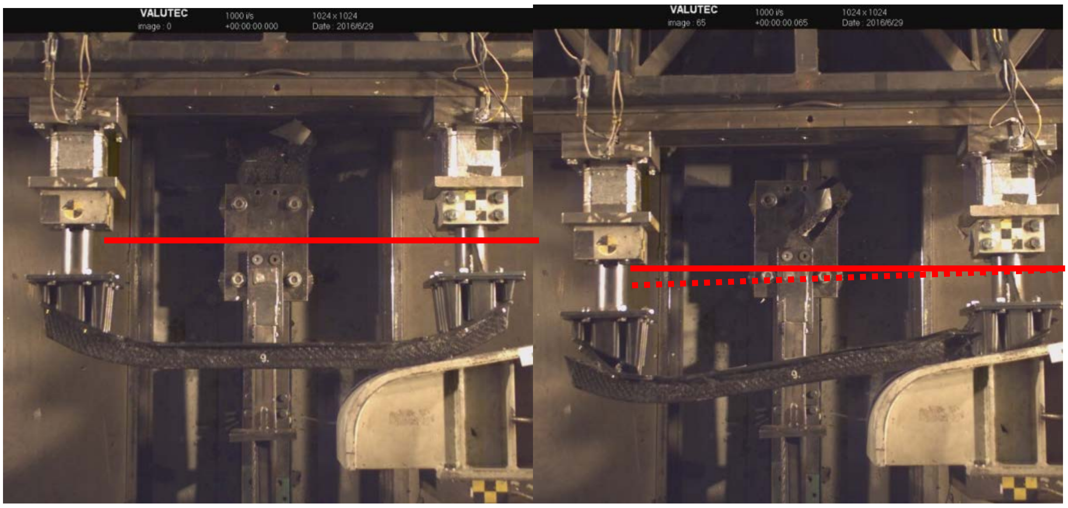


Fig.6: Illustration of mule rotation during the AZT impact test

The FE model has been solved for 100ms. This time period is considered enough for the AZT crash phenomenon completion, as it has been revealed by the experimental investigation of the crash. The total solution time has been 26.5 hours in a personal computer with i7 CPU and 32 GB of RAM.
In Fig 7 the evolution of internal energy (total and of individual parts) during impact is presented, which corresponds to the absorbed impact energy. It is observed that the structural parts absorbing the larger amounts of impact energy are the crash-boxes covers and the foams inside the transversal beam and the crash boxes.

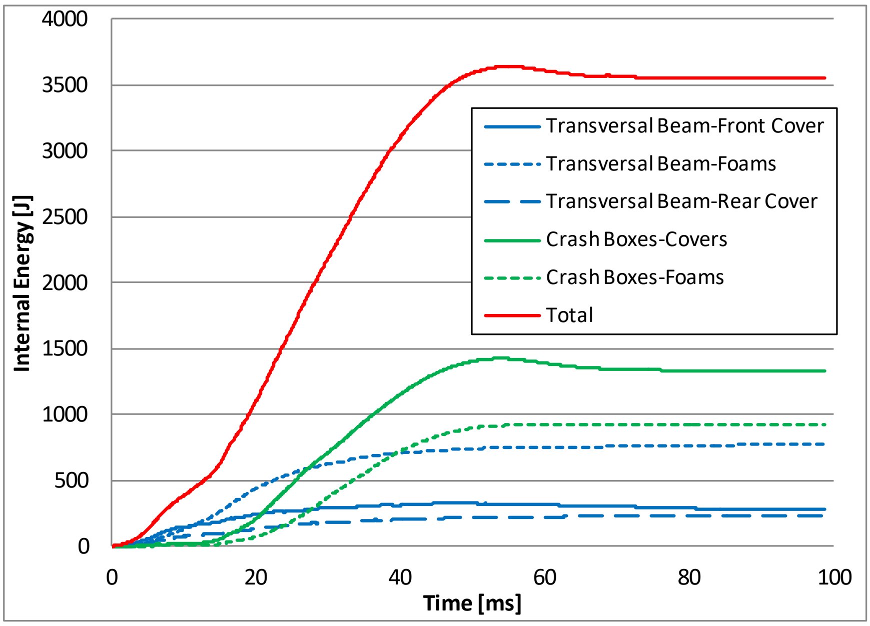


Fig.7: Simulation results - Internal energy vs. time diagram for the several parts

6 Comparison of numerical results and experimental measurements

The FE simulation results have been compared with the physical impact test measurements. Images recorded by a high-speed camera, as well as force and displacement measurements obtained during the AZT crash test, have been used for the correlation. The basic phenomena observed during the physical test that are required to be reproduced by the numerical simulation include:

- Deformation of the transversal beam in front of the crash box at the impact area, until it is fully compacted.
- Relatively stiff behaviour of the crash boxes.
- Failure of bonding between the transversal beam’s front and rear cover
- Separation between the right crash box and the transversal beam’s rear cover during final stages of the impact event.

In Fig. 8 the evolution of impact as recorded during the AZT test and as calculated by the numerical simulation is presented. It may be observed that deformation of the transversal beam in front of the crash box at impact area, presents the same behaviour in both the physical test and simulation, until it is fully compacted.
Furthermore, de-bonding between the transversal beam front and rear cover can be observed in both the physical test recording and simulation results. Moreover, in both cases deformation of the crash box at the impact side follows the initial deformation of the transversal beam, presenting a relatively stiff behaviour. It has to be noted that during the impact test, a separation between the transversal beam’s rear cover and the crash box cover, located away from the impact area (at the right side), was detected. However, the specific failure has not been predicted by the FE simulation, that may be due to the assumed fixed moving direction of the mass plates

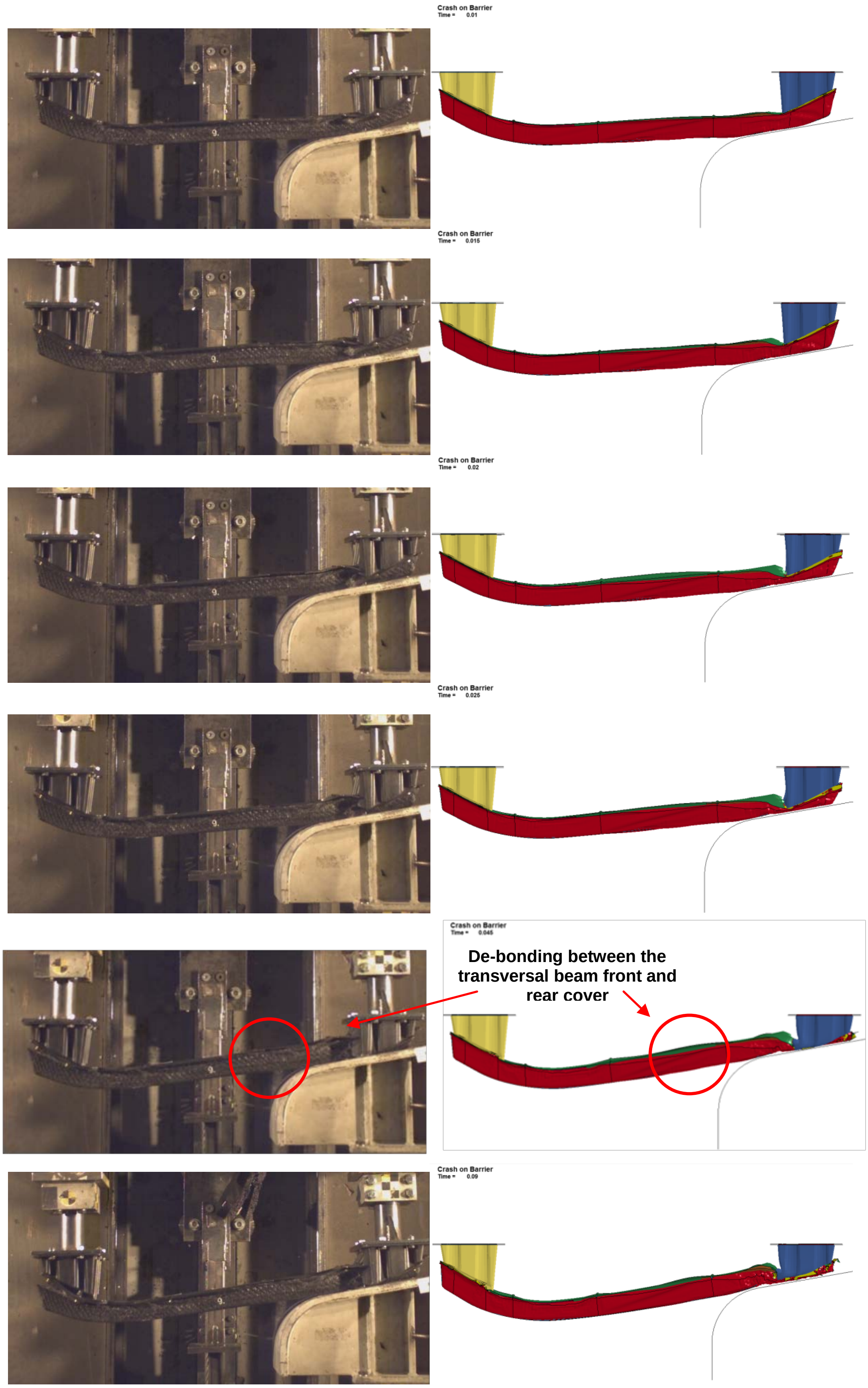


Fig.8: Front cross-member assembly deformations as recorded during the AZT test (left pictures) and as calculated by the numerical simulation (right pictures)

In the following Fig. 9 to Fig. 12 variation of velocity, acceleration, displacement, absorbed energy as a function of time are compared, as obtained by the numerical simulation and as acquired during the AZT experimental test. More specifically, in Fig. 9 the velocity vs. time diagram is presented. It may be observed that the numerically calculated velocity curve presents the same general behaviour compared to the experimentally derived curve, while only during the initial stages of the impact event, a small deviation can be recorded. In Fig.10 the acceleration vs. time curves derived from the experimental and numerical results can be compared. Numerical results correlate well with experimental results, regarding maximum acceleration values and slope of the curves.

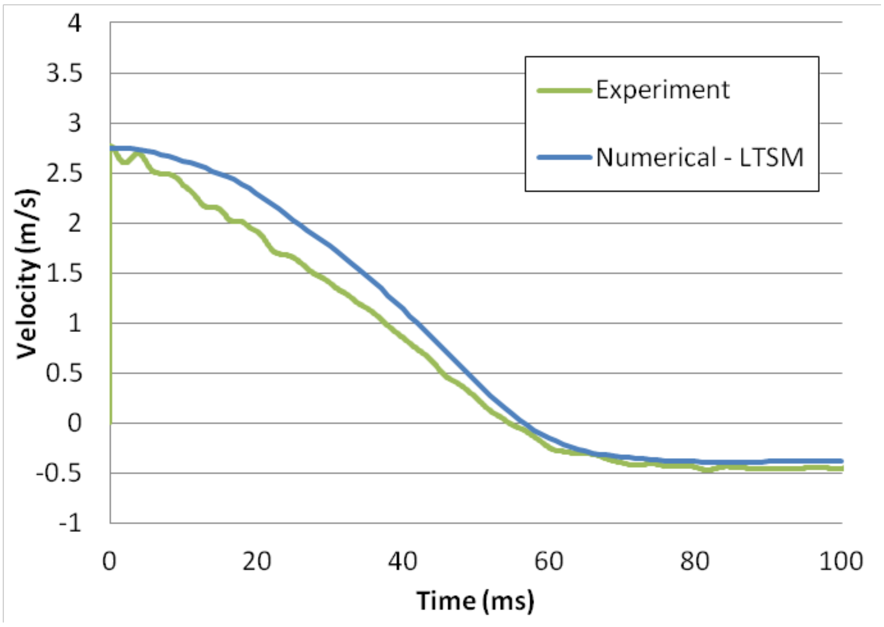


Fig.9: Comparison of experimentally measured and numerically calculated velocity vs. time diagram

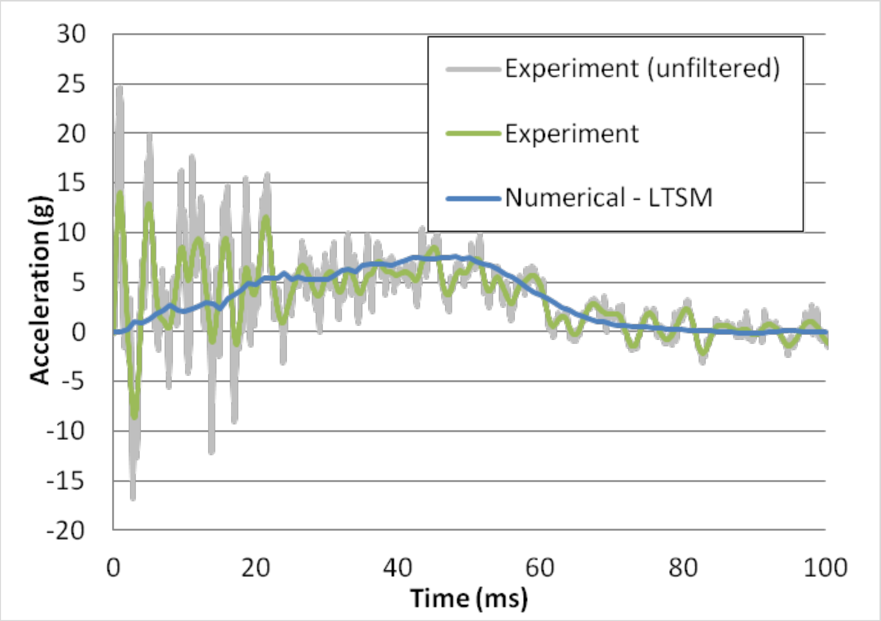


Fig.10: Comparison of experimentally measured and numerically calculated acceleration vs. time diagram

Subsequently, in Fig 11 the displacement vs. time diagram is presented. In this diagram except from the numerically calculated curve, two experimental curves are presented corresponding to the left and right mass plate. The difference between these latter curves is related to the twisting of the mule during the physical impact test. However, as mentioned above, this twisting has not been taken into account during the numerical simulation, as holding plates are assumed to “move together” (i.e. having the same x-displacement). Regarding the energy absorbance comparison between experimental and numerical results is presented in Fig. 12. During the initial stages of the impact event, there is a slight difference in the slopes of the experimental and numerical curves. However, subsequently, the general behaviour is identical, while the finally absorbed total energy is identical for the physical test and the numerical simulation.

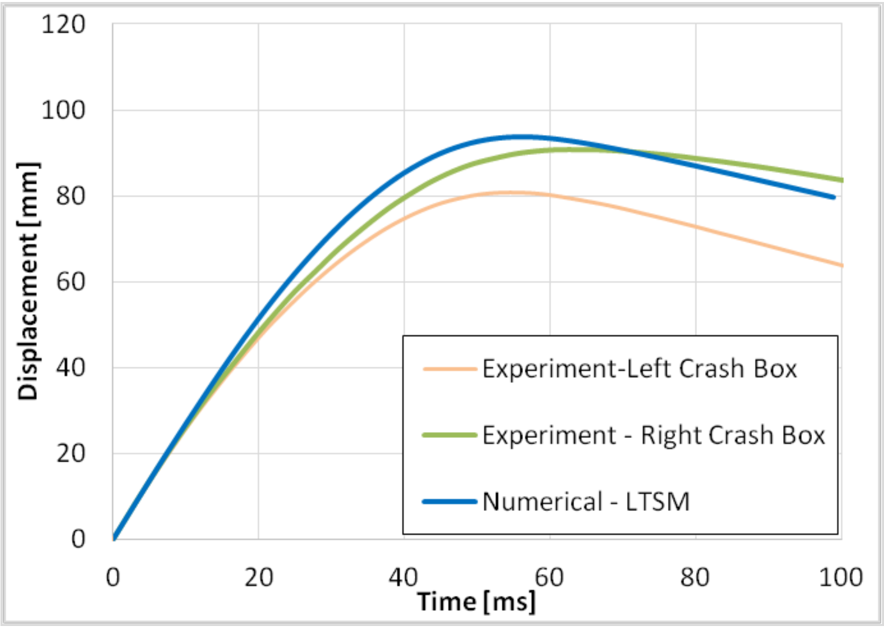


Fig.11: Comparison of experimentally measured and numerically calculated displacement vs. time diagram

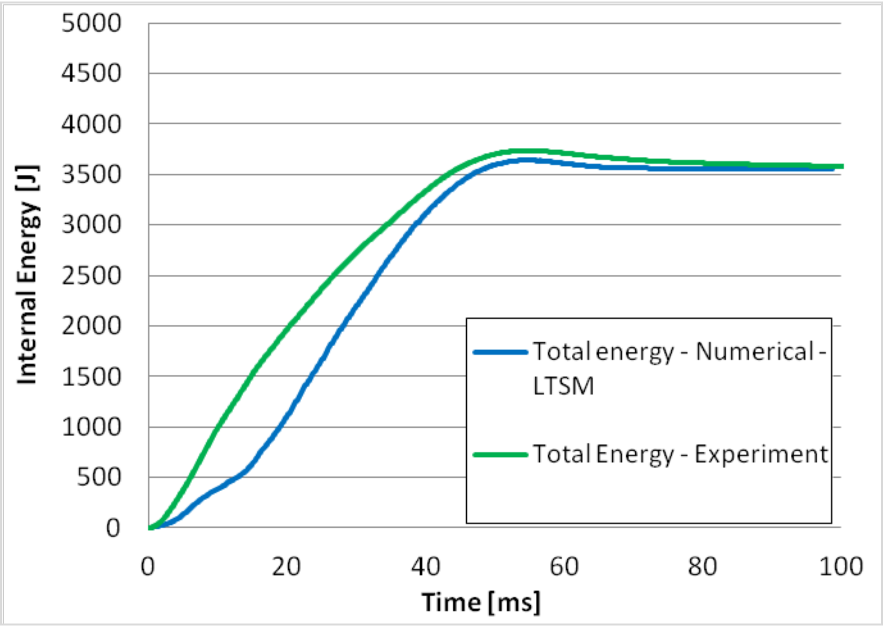


Fig.12: Comparison of experimentally measured and numerically calculated absorbed energy vs. time diagram

7 Conclusions

In the frame of EVOLUTION project a new fully plastic front cross beam subsystem demonstrator has been designed, developed and prototyped by FPK. Different materials have been used in the structure, including low density poly-urethane foams.

Numerical simulations of the front cross-member assembly low speed impact according to AZT crash test protocol have been performed. The following main conclusions may be drawn:

- Foam materials simulation is quite tricky due to their particular mechanical behaviour. Especially, when strain rate material properties are used, raw experimental data have to be processed, so that the required material model parameters are properly calculated.
- Numerical simulation results compare well with experimental test measurements, leading to successful validation of the developed FE model.
- It have been calculated that during impact crash boxes attached to the transversal beam and low-density polyurethane foams inserted in the transversal beam and the crash boxes absorb most of the impact energy.

8 Acknowledgment

The research leading to these results has received funding from the European Union Seventh Framework Programme (FP7/2007-2013) under grant agreement n° 314744.

9 Literature

[1] The Electric Vehicle revOLUTION enabled by advanced materials highly hybridized into lightweight components for easy integration and dismantling providing a reduced life cycle cost logic (EVOLUTION), Grant agreement no: 314744,

[2] Jovan Obradovic, Simonetta Boria, Giovanni Belingardi, Lightweight design and crash analysis of composite frontal impact energy absorbing structures, Composite Structures, Volume 94, Issue 2, January 2012, Pages 423-430.

[3] Jones, D., Gaewsky, J., Weaver, A., and Stitzel, J., "A Semi-Automated Approach to Real World Motor Vehicle Crash Reconstruction Using a Generic Simplified Vehicle Buck Model," SAE Int. J. Trans. Safety 4(2), 2016, Pages: 267-277.

[4] E. Nassiopoulos and J. Njuguna, Finite element dynamic simulation of whole rallying car structure: Towards better understanding of structural dynamics during side impacts, 8th European LS-DYNA Users Conference, Strasbourg - May 2011.

[5] Shokri El Houssini, Front Bumper Crashworthiness Optimization, 9th International LS-DYNA Users Conference, 2006

[6] S. Heimbs, F. Strobl, P. Middendorf, S. Gardner, B. Eddington, J. Key, Crash Simulation of an F1 Racing Car Front Impact Structure, 7th European LS-DYNA Conference, 2009

[7] EVOLUTION Technical Meeting for concept sign-off - OVERVIEW PRESENTATION - Assembly, interfaces & joinings, pdf presentation 22/07/2015

[8] LS-DYNA Aerospace Working Group, Modeling Guidelines Document, Version 12-1, June 2012.

[9] LS-DYNA, KEYWORD USER'S MANUAL, VOLUME II - Material Models, February 2013, Version R7.0, LIVERMORE SOFTWARE TECHNOLOGY CORPORATION (LSTC)