# Numerical and Experimental Correlation of a Survival Cell Designed for a Bus Body Structure

Alexsandro Sordi[1], Felipe Biondo[1], Guilherme Magnabosco[1]

[1]Marcopolo S.A.

## 1 Abstract

The behavior of mechanical structures, when subjected to impact load, is a matter of great relevance and its applications in terms of vehicle collision. When we analyze the superstructure of a bus, those vehicles must be tested according to prerequisites established in standards such as UNECE ECE 29 (European standard) or CONTRAN 629/2016 (Brazilian standard). The standards prescribe to use a pendular system to evaluate the frontal structure of the vehicle. In this regulation is defined the height and the mass that will collide with the structural modulus. However, the procedures described in these standards do not represent the real collisions involving these types of vehicles. This can be seen when comparing the energy imposed on the test module, detailed in CONTRAN 629/2016, where the energy imposed on the vehicle is approximately 20 kJ on each side of the test module, this corresponds to a collision of a 5 tons vehicle at 10 km/h or a 20 tons bus at 5 km/h.

In this context, this study aimed to develop a survival cell for the bus driver called FIA - (Frontal Impact Absorber), aiming to increase vehicle safety in situations where a frontal collision occurs. To reach the final concept of impact absorber, several nonlinear explicit structural analyzes were performed using the LS-Dyna software. In order to prove the efficiency of the FIA, a physical test was performed. The test consisted of a semi-frontal collision between two buses, where one vehicle remained stationary and the other collided with it at a speed of 40 km/h using an offset of 50% between the vehicles. A device was developed in order to keep the vehicle in its trajectory during the motion.

When evaluating the results, a correlation was noted between the experimental assay and the FEA analysis. The biggest differences in displacements found are in the range of 14%. When comparing the vehicles tested, the difference between the models with and without the FIA package becomes evident. The bus without this component had a deformation value measured at the point of greatest displacement of 648 mm, whereas the vehicle with FIA device, 273 mm, resulting in 58% less deformation in this vehicle. Therefore, it is evident that this device can help to minimize the damage caused to the occupants of the vehicle that is equipped with this device during the event of a collision.

Keywords: crash test; numerical and experimental correlation; crashworthiness; coach bus; frontal crash; Frontal impact absorber; survival cell

## 2 Introduction

Brazil is a country with continental dimensions and collective road passenger transport is vital for its social and economic development. According to the National Union of the Industry of Components for Motor Vehicles [1], between the years 2012 and 2018 the fleet of coaches grew 8,2% reaching the number of 386.417 vehicles, however the quality of the roads did not follow this evolution.

According to data published by the Brazilian National Transport Confederation [2], Brazil has 1,720,756 km of highways, of which only 211,480 km are paved, that is, 12.3% of the total. In a survey carried out in 2017, the same study shows that, of the 105,814 km evaluated, only 14,283 km had double lane, equivalent to 13.5% and that 91,031 km had a simple two-way lane, corresponding to 86% of the total analyzed by the survey. When this data is analyzed, is possible to correlate this road condition to the number of accidents involving this kind of transportation.

With regard to the safety of bus passengers, this attribute must be achieved in such a way as to comply with current safety standards. For Sánchez [3], the safety of the occupants, when an impact occurs, is

© 2021 Copyright by DYNAmore GmbH

an extremely important item. He also mentions that for a vehicle to be considered safe, it must not only a prevent collision involvement but minimize the degree of injury when involved in an accident as well.

The structure of bus-type vehicles is basically composed of thin-walled profiles and steel plates, those are the components that absorb the greatest amount of energy during an impact situation. The parameter that measures the capacity of the structure or part of the vehicle to absorb kinetic energy resulting from an impact and maintaining the integrity of the occupants' space is called crashworthiness.

In this context, the present work aims to present a device called frontal impact absorber (FIA), was developed specially for road buses. In this study will be presented the numerical simulation of a crash test simulating the frontal crash between a regular coach bus, that meets the CONTRAN criteria, versus a coach bus with the FIA, as well as the real crash test between these vehicles. In the end of the paper will be presented the numerical and experimental results of the two buses, showing the performance of the FIA and the correlation between the numerical and the experimental model.

## 2.1 Current Scenario

Currently bus manufacturing companies must comply with standards and requirements regarding the safety of public transport passengers. In Brazil, CONTRAN [4] establishes procedures and test methods with the minimum parameters for safety requirements in these vehicles.

In annex II of CONTRAN resolution number 629 [4], parameters are established to develop the frontal impact resistance test. This consists of raising a mass in its normal pendulum trajectory to a height of 2,000 mm above the point of impact, letting it fall freely and impacting perpendicularly to the frontal region of the vehicle. This procedure consists of a pendulum of dimensions 700 mm x 700 mm, with a mass of 1,000 kg and its body must be firmly connected to two rigid bars with length, from the point of articulation to its center of mass, between 4,500 and 5,000 mm.

The pendulum must impact on two regions of the frontal structure of the vehicle, one centralized with the driver's seat, at a height of 200 mm from the geometric center of the pendulum to the floor line and, similarly, the pendulum must impact on opposite side of the structure. Figure 1 shows a schematic drawing of the pendulum test, where (a) represents the trajectory of the pendulum, (b) a frontal image of the test before the pendulum falls, (c) measurement of the height before the pendulum falls and (d) a side view of the moment before the impact.

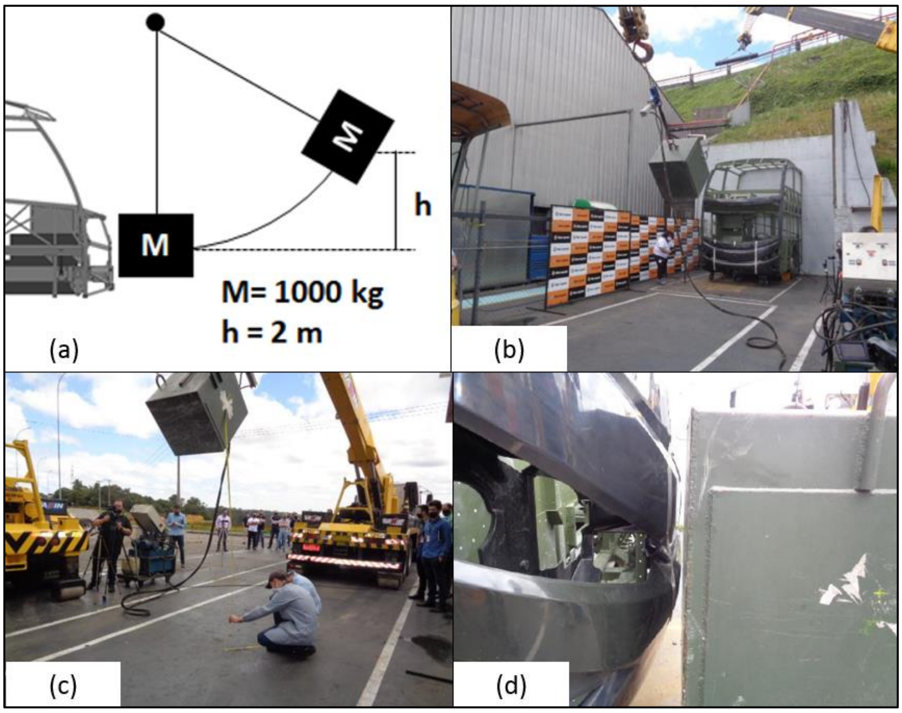



*Figure 1 - Pendulum test according to CONTRAN Regulation No. 629*

In order to be considered approved, after the two impacts are carried out, no point of the vehicle structure may suffer permanent longitudinal deformation greater than 200 mm. The velocity of the pendulum at the moment of impact is determined by means of an energy balance according to Equation 1,

$$mgh_i = mV_f^2/2 \qquad (1)$$

Where:
m = mass of the pendulum;
g = gravity;
$h_i$ = initial height of pendulum fall;
$V_f$ = impact velocity of the pendulum;

The energy imposed on the structure in the pendulum test is 19.6 kJ in each impact, regardless of whether the vehicle is 5 ton or 20 ton. This test corresponds to collide a 20 tons vehicle at 5 km/h or a 5 tons vehicle at 10 km/h. In other words, the pendulum test does not represent a frontal collision situation. This is corroborated by Figure 2, in which some accidents involving buses are presented.

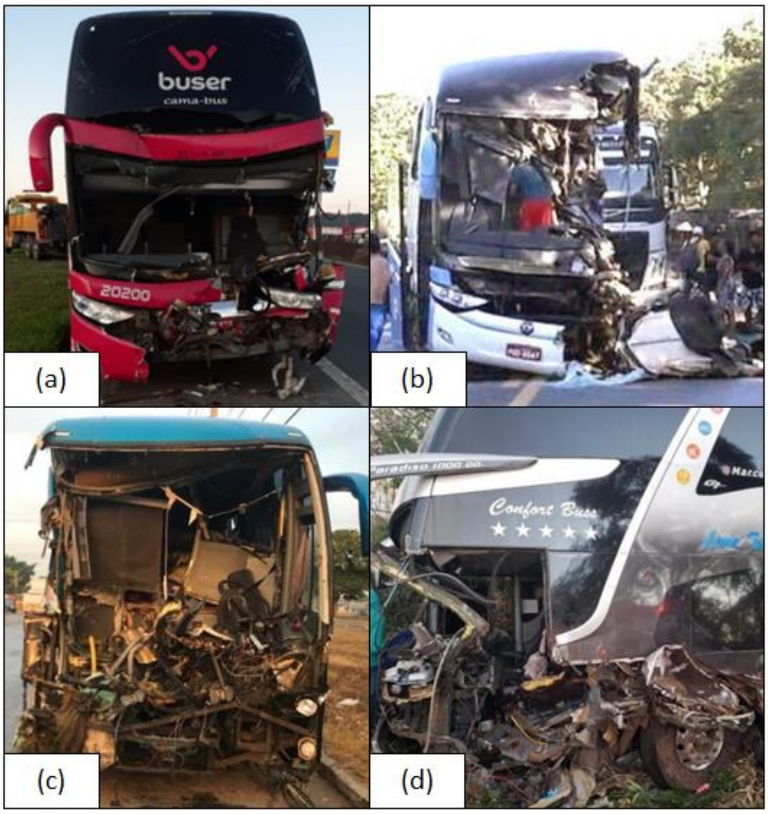



*Figure 2 - Colisions involving buses: ((a) Diário do Transporte [7] ; (b) Folha de São Paulo [8]; (c) A Gazeta [9]; (d) Gaucha ZH [10]*

Generally, in collisions involving buses, large deformations occur on the structures, resulting in major tragedies. According to data from the Volvo Traffic Safety Program [5], between 2008 and 2017 there were 84.731 traffic accidents involving buses in Brazil, only on federal highways, resulting in 6.427 deaths, 16.596 serious injuries and a total of 269.080 people involved.

According to data from the Brazilian Federal Highway Police [6], between the years 2017 and 2018 the most dangerous traffic accident was the roll over, where the vehicle rests on its side, front or rear, but without rolling on yourself. Table 1 shows the numbers of accidents according to their nature. In this table, the most dangerous accident is the rollover, in which the vehicle rests on its side, front or rear, but without turning on itself. The rollover causes 1 death for every 12 accidents. Another dangerous accident is the frontal collision of this vehicles, which leaves at least 1 seriously injured after every 12 occurrences.

© 2021 Copyright by DYNAmore GmbH

*Table 1 - Accident numbers according to their motivation [6]*

| | Accident | % | Death | Seriously Injured Graves |
|---|---|---|---|---|
| Overturning | 139 | 1.4 | 2 | 6 |
| Frontal collision | 2143 | 22.3 | 49 | 176 |
| Lateral collision | 1754 | 18.2 | 8 | 36 |
| Transverse collision | 1002 | 10.4 | 3 | 28 |
| Rear collision | 3879 | 40.3 | 27 | 125 |
| Rollover | 709 | 7.4 | 60 | 102 |

## 3 Experimental test

To perform the test, two coaches Paradiso 1200 were used, they are called PV1 and PV3. Both had a total length of 14,000 mm. The front impact absorber (FIA) was built on the PV1 vehicle, the mass of this vehicle was 15,770 kg. While the PV3 vehicle did not have the FIA and its mass was 18,500 kg.

The test was set to be performed at a speed of 40 km/h with a displacement of 50% off set (driver to driver). To obtain the impact velocity a slope with a 20% inclination was used. Vehicle PV3 remained static, while vehicle PV1 collided with it at that speed.

To keep the vehicle PV1 in motion and on the impact trajectory to PV3, an electronic control system was developed and installed in the steering system of the vehicle in motion, consisting of: ultrasonic sensors (front and rear axle) that checked the position of the vehicle during the test. The speed sensor was installed to measure the velocity and determine the starting point of the movement; LVDT-type displacement sensor located on the steer-bar damper was used to control the angle of the wheels; A stepper motor linked to the steering column through a system of chains and gears perform the movement of the steering system and correct the vehicle's route during the test (Figure 3).

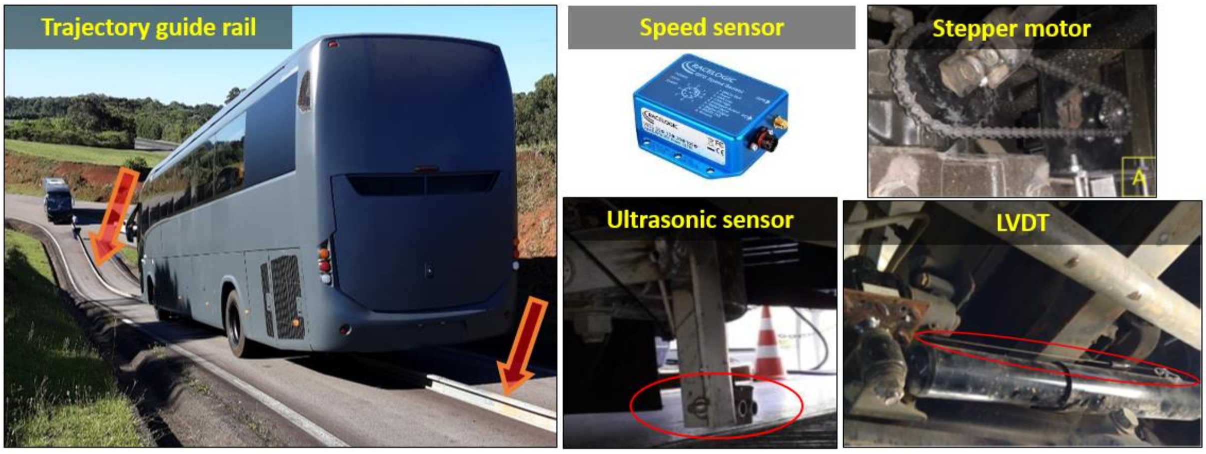



Figure 3 - Experimental test

Both vehicles were loaded with dummies to represent the mass of passenger’s area, in addition, mannequins with human characteristics (weight, joints, etc.), where positioned in the driver's seat (Figure 4).


Figure 4 - Dummies

## 4 Numerical Model Description

In this chapter, will be presented the methodology for the FEA model that represents the superstructure of the analyzed vehicle, describing in details the parameters and criteria for the analysis, as well as the boundary conditions established to obtain the results.

The finite element model was built considering all the important items that contribute to vehicle rigidity regarding a frontal collision situation. Some components have been simplified in order to guarantee a minimum element size of 5 mm. Figure 5 (a) and (b) presents an external view of vehicles built in the FEA software, in Figure 5 (c) an internal view of the drivers area and in (d) an internal view of the passengers area.

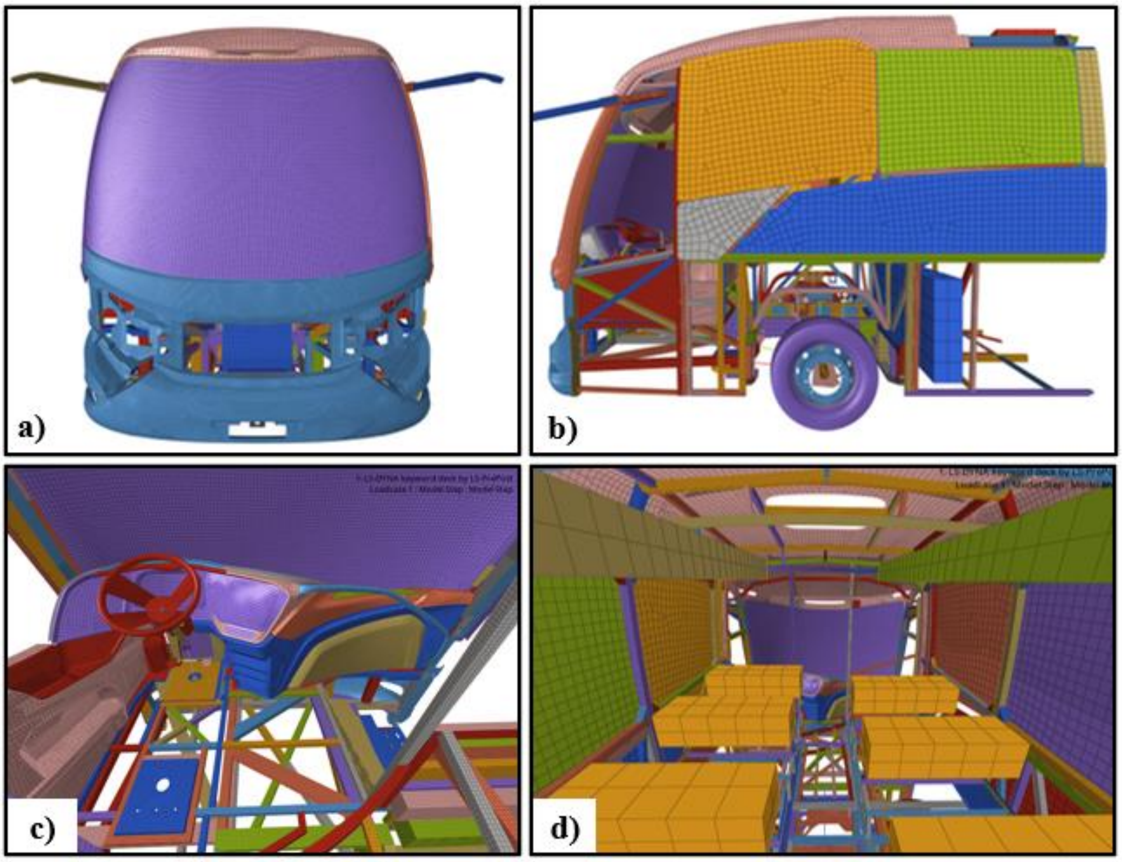



Figure 5 - Discretization of the numerical model

For the construction of the mesh, the software *Hypermesh* version 2020 was used. Some parameters were observed, such as: components with thin thickness (tubes, profiles and sheets) were made with shell-like elements; the uniform mesh was used in the energy absorption region of the structure; components should have a minimum of 3 elements per face of any section in the entire vehicle or use a fully integrated element (EQ.16: Fully integrated shell element); the total number of triangular elements should be limited to 10% of the total; the maximum number of aspect ratio must be 10, where the model must have at least 98% within this criterion; the maximum warpage angle number should be 15, where the model must have 98% within this criterion. The models used in the simulation had a total of 1,933,019 nodes and 1,979,471 elements for each vehicle.

After the finite element model was completed, a simplification was performed. In this simplification, the vehicle was divided into two regions: region 1 is composed by the components of the frontal structure of the vehicle and region 2 with the rear part of the coach, as shown in Figure 6.

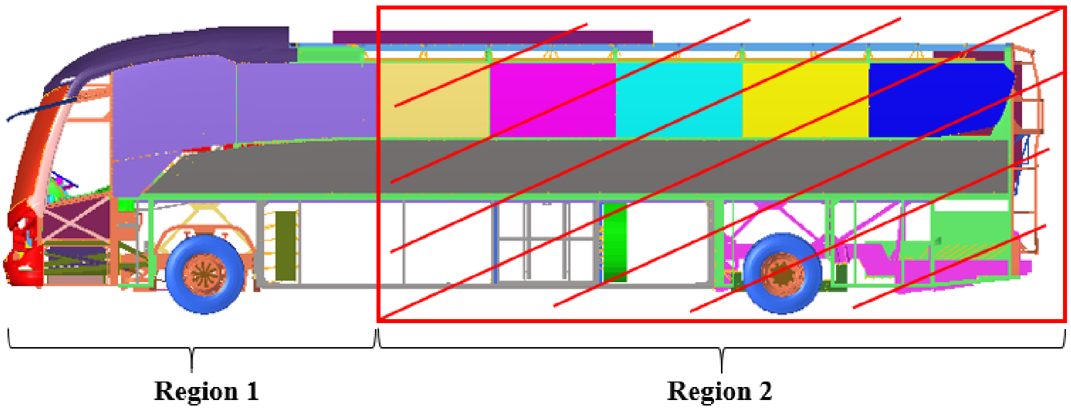



Figure 6 – Division of the finite element model

In a frontal collision situation, the largest plastic deformations are present in the components of region 1, so region 2 was excluded from the numerical model, leaving only the wheels and tires. For that, an inertia element, with the mass and inertia of region 2, was added to its CoG, as shown in Figure 7.

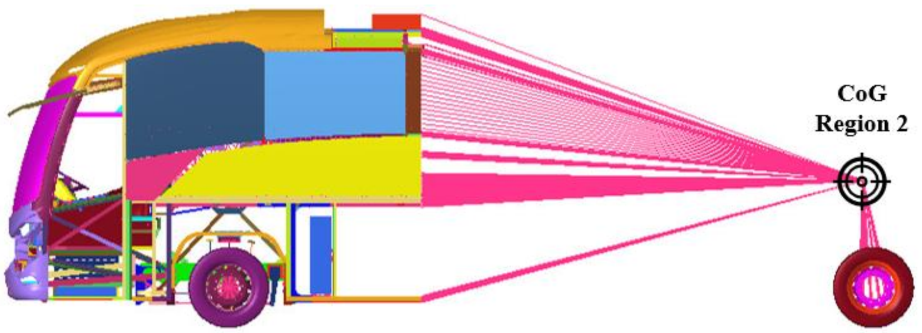



Figure 7 - Model Simplified

The connections between the components of the simulated model were made in 3 ways: the first type of connection used was the "matched mesh", where in this case the edges of the components are superimposed in order to leave the connected components in the geometry. When the mesh is generated, it will be done continuously, and the interface nodes of the components will be the same. The second type of connection used is the CNRB (Constrained nodal rigid body). CNRB are elements used to model a rigid structure. It can connect as many nodes as needed, being just one of them independent. The element imposes that the result of the dependent nodes is the same as that of the independent node for the defined degrees of freedom. The third type of connection was the use of 1D beam elements (Beam) to represent the region of interest screws (regions that are considered important in the collision). This type of connection allows to assign a material and section to this beam, to represent the properties of the screw, including the failure in this type of element.

© 2021 Copyright by DYNAmore GmbH

In this simulation, different types of element formulation were used. For solid elements, the formulation of element 1 was used (EQ.1: constant stress solid element), default of the Ls-PrePost software for solid elements. In the Beam elements, the formulation of element 9 (EQ.9: spotweld beam) was applied, which allows assigning a constitutive material model with a failure criterion when necessary. For the shell elements, the formulation of element 2 and the 16 (respectively EQ.2: Belytschko-Tsay and EQ.16: Fully integrated shell element) were used, both with 5 integration points in the thickness (NIP = 5) and 1 as “shear factor” (SHRF).

The materials used to manufacture the vehicle were characterized through standardized stress-strain tests according to the ASTME 8 standard. The traction test is performed on standardized specimens as described in the standard. This aims to obtain the stress-strain curves of the materials.

In the LS-Dyna software, the vehicle structure materials are represented by the material that represents the elastoplastic behavior by the option “Mat_Piecewise_Linear_Plasticity” (MAT123). The coordinates of the stress-strain curve in the plastic region are considered according to the correction of the stress-strain curves. Table 2 lists the properties for the steels used in the model.

*Table 2 - Material Properties*

| Material | Density (kg/m³) | Poisson's Coefficient | Young Modulus | Yield Strength (MPa) | Ultimate Strength (MPa) | Elongation (%) |
|---|---|---|---|---|---|---|
| ZAR230 ZF | 7,850 | 0.3 | $21 \times 10^4$ | 360 | 497 | 14 |
| ZAR230 | 7,850 | 0.3 | $21 \times 10^4$ | 245 | 380 | 19 |
| ZSTE380 | 7,850 | 0.3 | $21 \times 10^4$ | 450 | 586 | 16 |
| LN280 | 7,850 | 0.3 | $21 \times 10^4$ | 334 | 551 | 14 |

To compute the strain rate effect, the constitutive law proposed by Cowper-Symonds was used, where the constants C = 40.4 and p = 5 are commonly used values for steels [10]. The main deformation in the plane (EPSMAJ) was used as a failure criterion. This deformation state is the most critical mode for collision situations. LS-DYNA calculates the “major deformation in plane” on all elements at each time interval. When plastic deformation exceeds the failure criterion in an element, that element is removed from the finite element model.

The screws were modeled using the material MAT100 (MAT_SPOTWELD) in the LS-DYNA software. The properties were obtained from the class of these screws, where the class defines the tensile strength limit and the yield strength. Considering the first number as "i" and the second as "j", we have:

- Ultimate strength limit: $\sigma_u = i \cdot 100$
- Yield strength: $\sigma_y = j \cdot i \cdot 100$

Table 3 list the properties for the screws used in the model.

*Table 3 - Properties of the screws*

| Screw Class | Density (kg/m³) | Poisson's Coefficient | Young Modulus | Yield Strength (MPa) | Ultimate Strength (MPa) | Elongation (%) |
|---|---|---|---|---|---|---|
| 8.8 | 7,850 | 0.3 | $21 \times 10^4$ | 640 | 2392.5 | 11 |
| 12.9 | 7,850 | 0.3 | $21 \times 10^4$ | 1080 | 2825.28 | 8.6 |

The failure criterion for screws is defined by the EFAIL option, where the maximum equivalent plastic deformation is considered. This failure criterion has a simple operation, using the effective plastic deformation state of the element to deactivate it when reaching the limit value.

The components that are part of the front suspension of the vehicles used in the numerical analysis were represented in a simplified way. The tension and stabilizer bars were simplified and represented using beam elements, these components are represented in Figure 8. Other items represented in the suspension of the virtual vehicle were the shock absorbers and pneumatic suspension. For these, discrete elements of the damper type were used to represent the dampers and discrete elements of the

© 2021 Copyright by DYNAmore GmbH

nonlinear spring type to represent the pneumatic suspension. These components are highlighted in Figure 8.

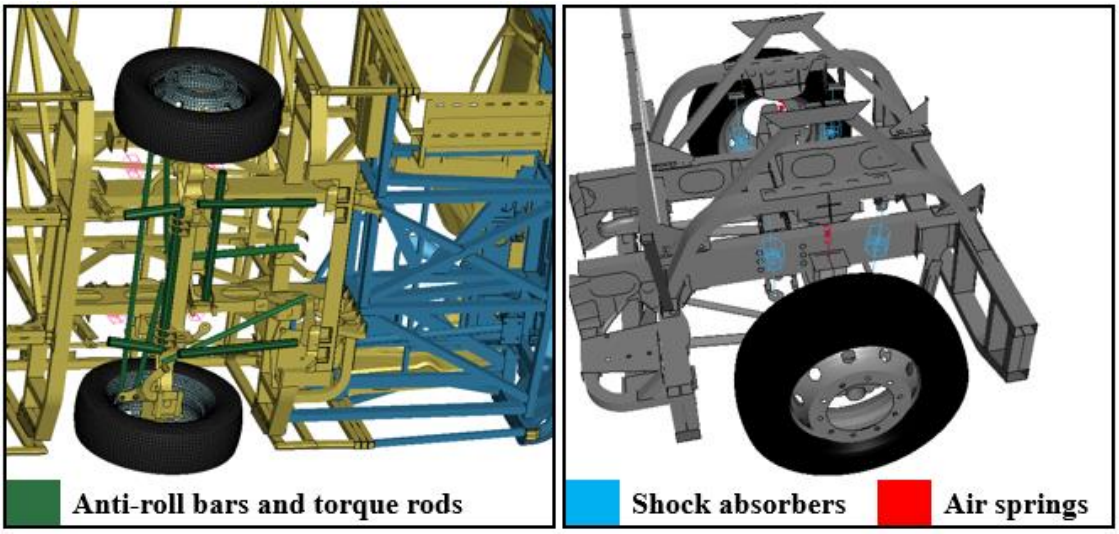



Figure 8 - Tension and stabilizer bars simplifications

The components that are characterized with the elastic material are those whose function is not structural, that is, they are only in the virtual model to add mass at specific points. In the case of this numerical simulation, this material was used to represent the mass of the following components: air conditioner, package holder, passenger mass and fuel tank. These components and their respective locations on the vehicle are shown in Figure 9.

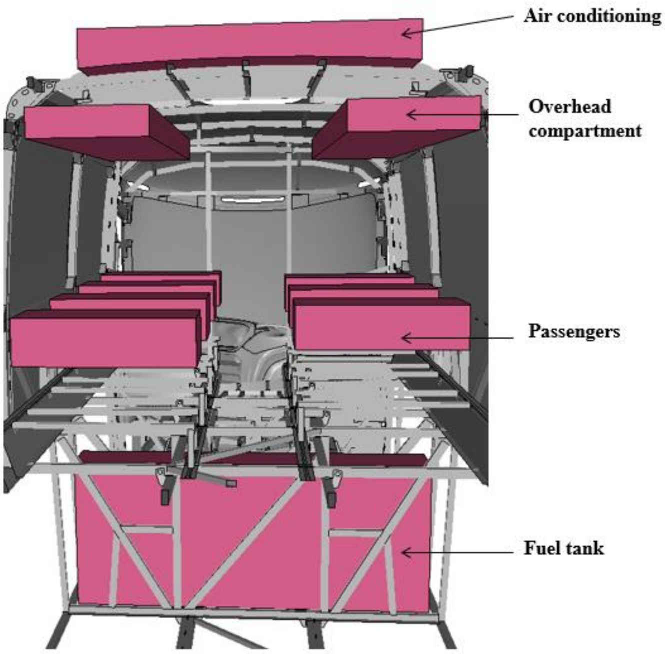



Figure 9 – Masses of non structural parts

The masses of the virtual vehicles were calibrated according to the experimental weight. In this case, the PV3 vehicle, which was static before the collision, had a mass equivalent to its GVW, that is, close to 18,500 kg. Figure 10 shows the mass of this vehicle in the virtual model, where it was 18,574 kg. This same figure also shows the location of the CG in terms of its height in relation to the ground and longitudinally in relation to the front axle.

Vehicle PV1, which collided with vehicle PV3, had a mass equivalent to 15,770 kg, that is, equivalent to its mass plus the addition of approximately 10 people. Figure 10 shows the mass of this vehicle in the

© 2021 Copyright by DYNAmore GmbH

virtual model, where it was 15,718 kg. This same figure also shows the location of the CG in terms of its height in relation to the ground and longitudinally in relation to the front axle.

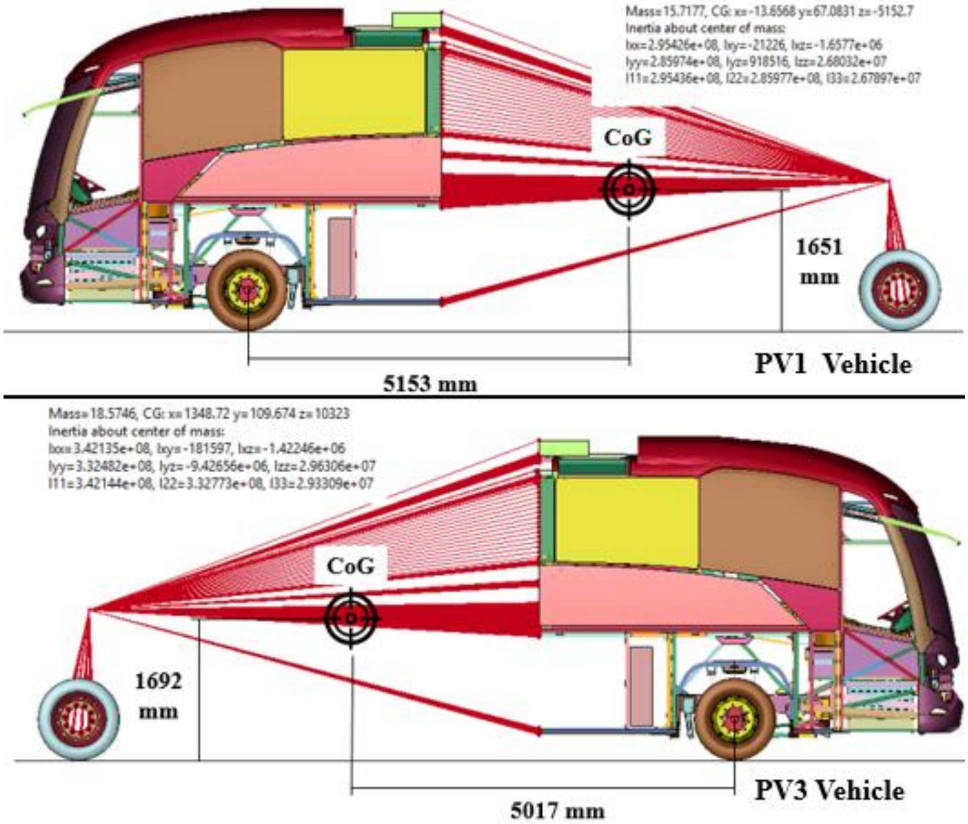


Figure 10 - Mass and CoG

The contact used in the numerical model was the "contact automatic single surface". This contact is used when the exact position of the contact is unknown, and in a collision situation there are several contacts between the parts of the vehicles involved. The use of this contact prevents penetration between vehicle components. As static friction factor (FS) and dynamic friction factor (FD), 0.30 and 0.25 were used respectively.

To represent the collision between the two vehicles, they were first placed in the condition immediately before the first contact. Vehicle PV3 remained static with the proper mass, while vehicle PV1 added a speed of 40 km/h corresponding to that of the experimental measurement. An offset of 50% between the two vehicles was also maintained, representing a driver-to-driver collision. In Figure 11 the collision condition in the virtual analysis is shown.

© 2021 Copyright by DYNAmore GmbH

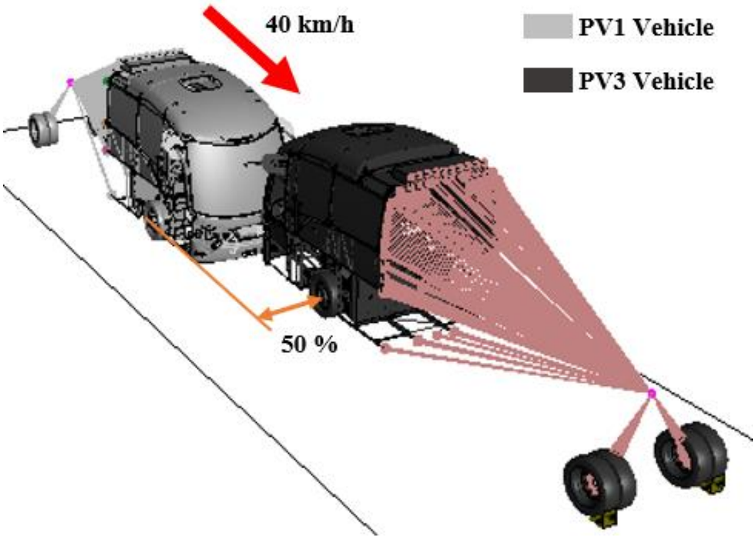



Figure 11 – Initial condition of the virtual analysis

The velocity in vehicle PV1 was entered through the Initial velocity card in the LS PrePost, where the velocity VZ = 1.1111e+04 mm/s² was entered. In the PV3 vehicle, wheel chocks were added in order to reduce its movement at the moment of collision, these chocks were included in the finite element model as shown in Figure 12.

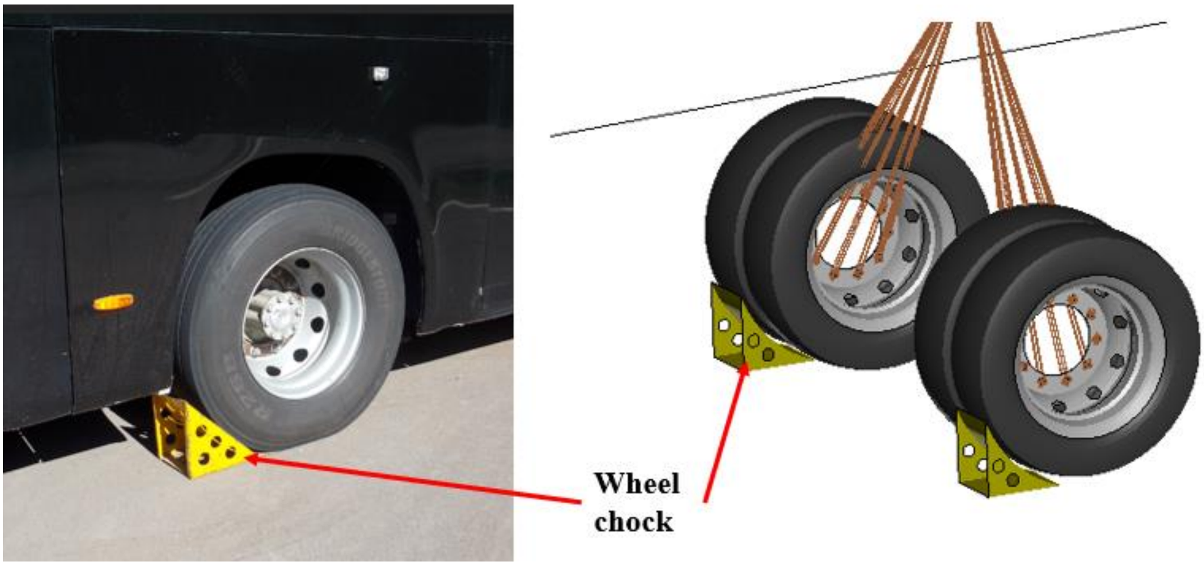



Figure 12 - Wheel chocks

## 5 Results

### 5.1 Experimental Results

Through the displacement sensors arranged in the PV1 vehicle, it was possible to verify that it followed the correct trajectory for the impact, reaching a speed of 40 km/h right before it and with a deviation of 11.28 mm to the right side, meeting the test criteria (Figure 13).

© 2021 Copyright by DYNAmore GmbH

Figure 13 - Crash test

Figure 14 shows the conditions of the driver (dummies) after the collision, it is observed that in the case of PV3 vehicle (without FIA) the dummy was pressed into the structure and in the vehicle with FIA this does not happen, showing the efficiency of the FIA compared to the vehicle without.


Figure 14 - Results of the experimental test - internal view

After inspection, all finishes, and non-structural components were disassembled to perform a digitalization of the structure of both vehicles. With this, it was possible to overlay the initial condition of the structure with the final condition. For both vehicles, the greatest displacement was with the upper fixation of the steering column, and for the vehicle without the package (PV3) it was about 2.4 times greater (Figure 15).

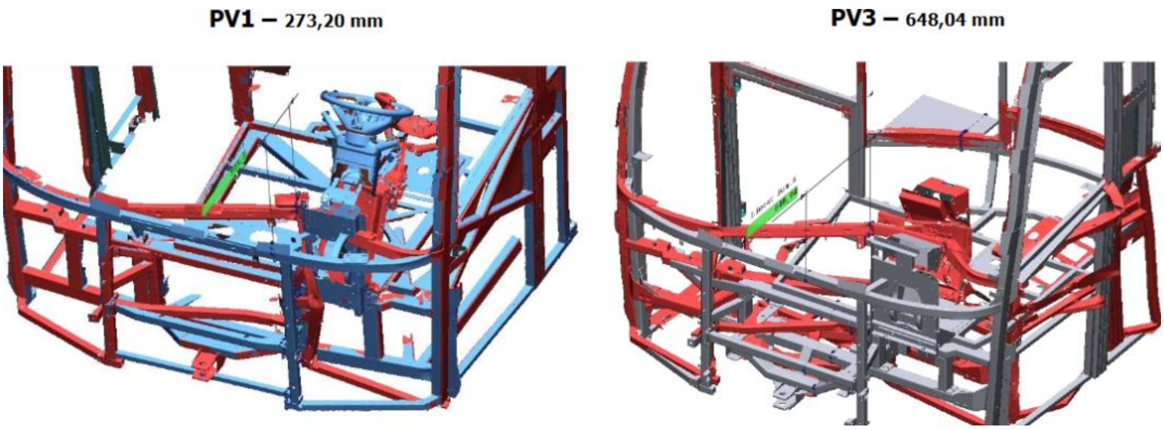


Figure 15 - Digitalization of the models

### 5.2 FEA Results

Figure 16 shows the FEA simulation result of the analyzed load case. Vehicle PV1 is highlighted in gray and vehicle PV3 is highlighted in blue.

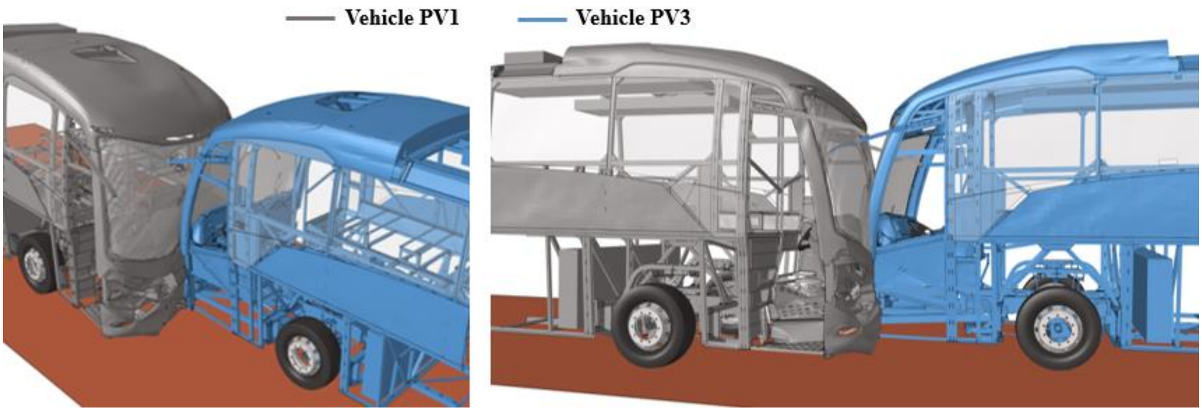


Figure 16 - Virtual Crash test

The numerical simulation was calculated until the moment when the speed of the two vehicles were close to zero and with a constant speed trend. To obtain exactly the speed of zero km/h in both vehicles, the computational cost is extremely high, considering that what is acting to stop them from that moment on is just the friction between the tires and the ground. Figure 17 shows the speed versus time graphs for the two test vehicles.

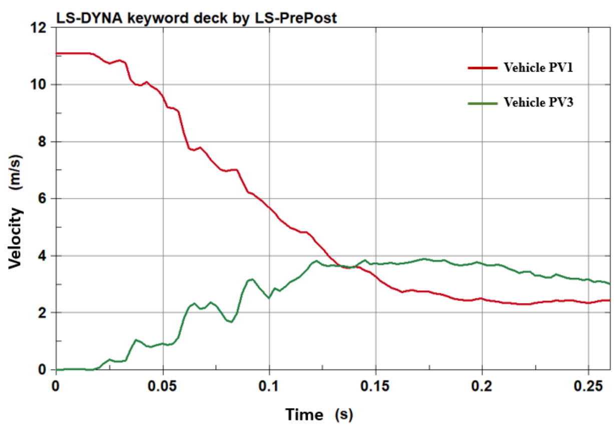


Figure 17 - Speed as a function of time

After the collision, the resulting residual space for the driver in both vehicles was measured. Figure 18 shows the measurement performed between the partition wall behind the driver and the steering wheel.

This Figure shows that the vehicle equipped with the FIA (PV1) results in a larger residual space than the standard vehicle (PV3).

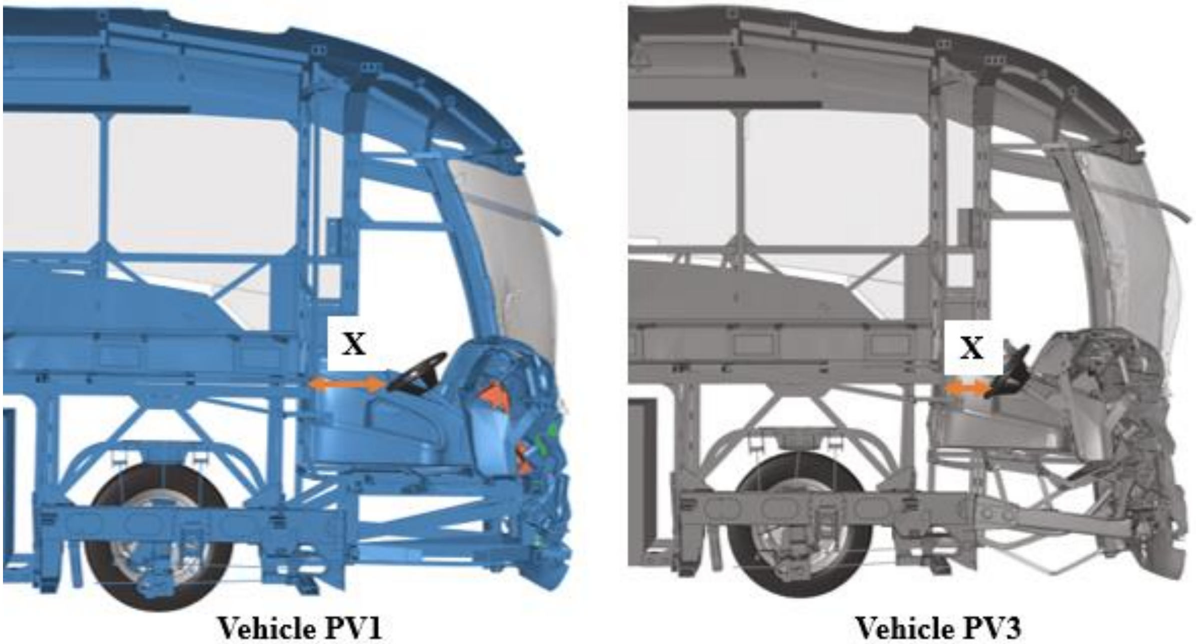



Figure 18 - Comparison of residual spaces

To measure the difference between the residual space of the two vehicles a chart was plotted as a function of time. In Figure 1 the difference between the models is shown, resulting in 499 mm for the vehicle with FIA and 325 mm for the vehicle without FIA.

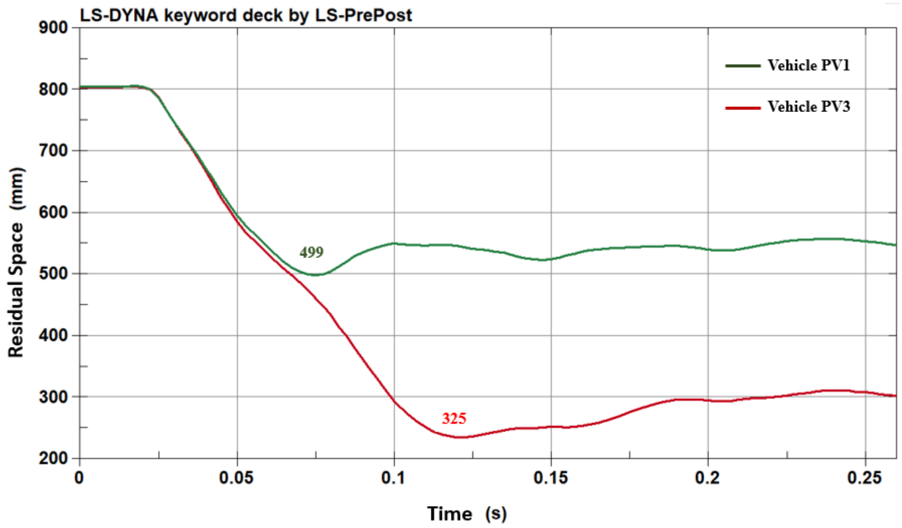



Figure 1 – Change in residual space after the collision for vehicle PV1 and PV3

### 5.3 Comparison between FEA and Experimental Test

Figure 19 shows, respectively, a lateral, isometric, and superior view of the comparison between the performed experiment and the simulation. Through them it is possible to observe a good correlation between the numerical and the experimental vehicles.

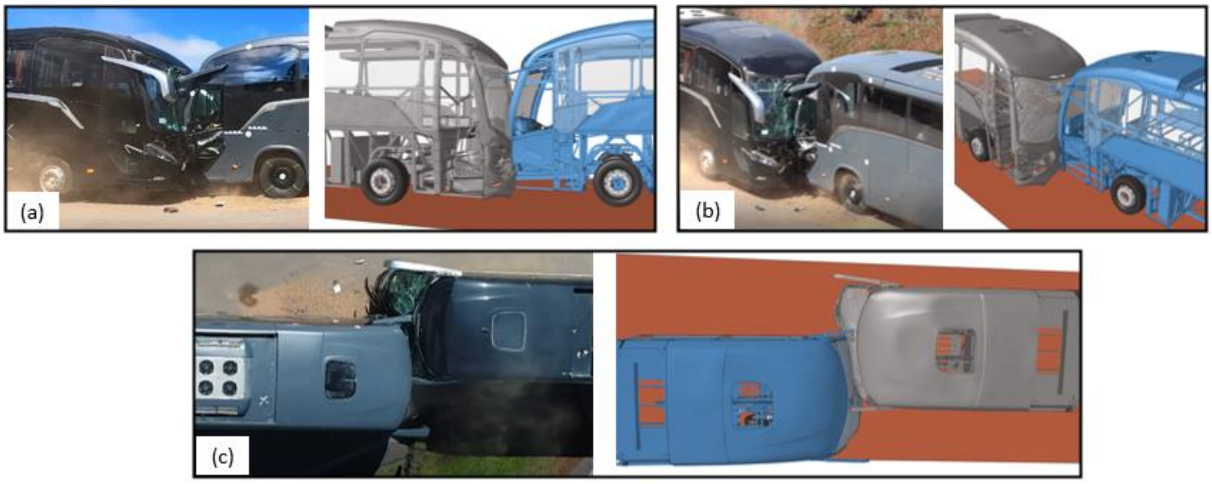


Figure 19 - Comparison between experimental and simulation tests

Figure 20 shows, respectively, the comparison between the experiment test performed and the simulation of the vehicle without FIA and the vehicle with FIA. Once again it is possible to observe a correlation between them.

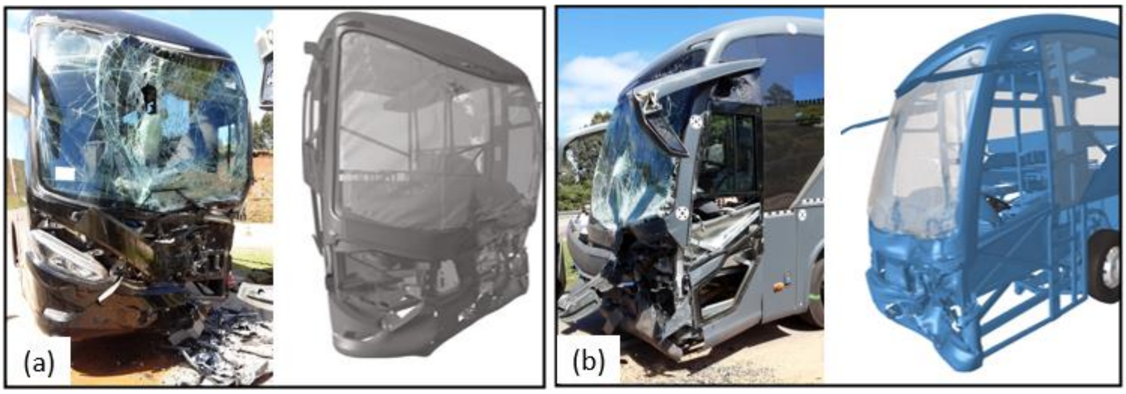


Figure 20 – Crash test (a) without FIA – (b) with FIA

Figure 21 shows a comparison between the experiment and the simulation, of the vehicle with FIA, specifying the region of the driver's station, as this region is the object of study in this work. These Figures show the points with the greatest deformations resulting from the collision performed. It is possible to observe a correlation between the experiment and the simulation.


Figure 21 - Comparison between experiment and simulation

© 2021 Copyright by DYNAmore GmbH

As previously mentioned, the structures were digitized, before and after the collision, and the digitization results were superimposed with the simulation deformation, in order to identify and measure the differences between them. Figure 22 (a) shows the overlap between the digital file resulting from the digitization of the experiment and the FEA deformed file. The Figure 22 (b) shows all the sections where a comparison was made between the experimental and virtual vehicles.

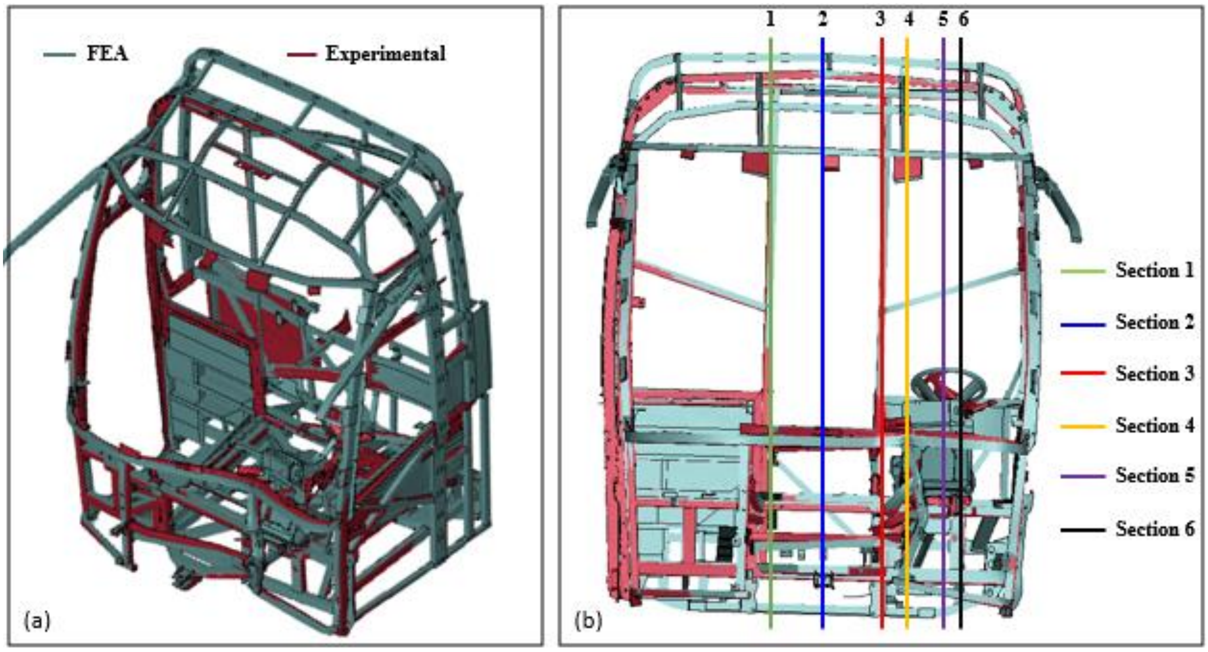
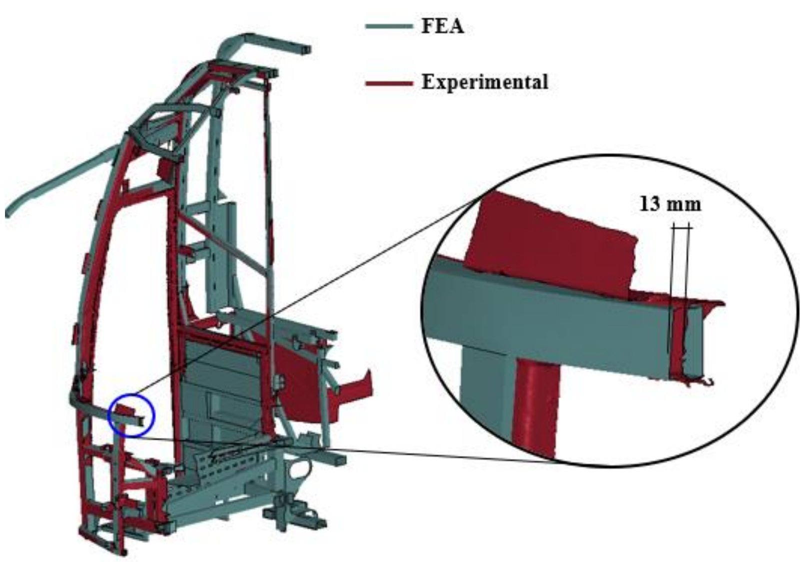
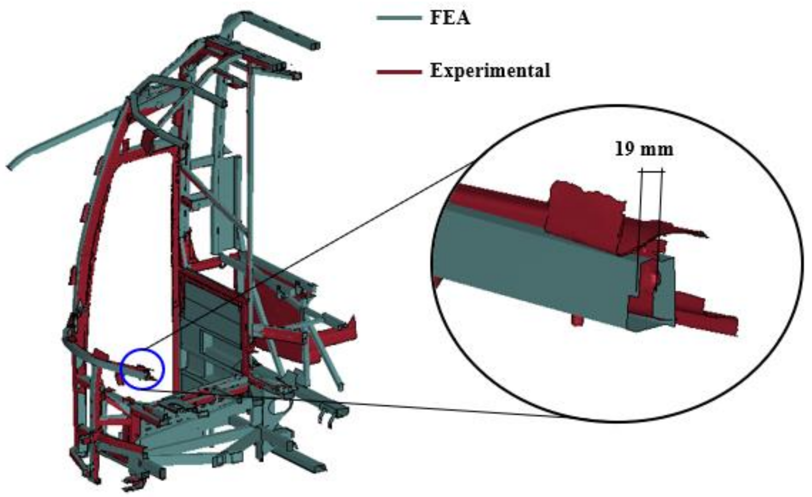
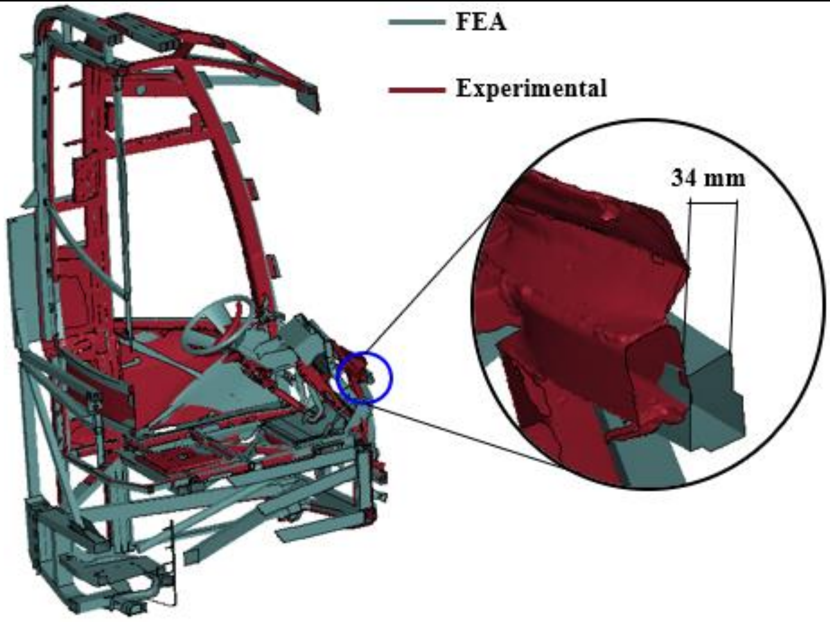
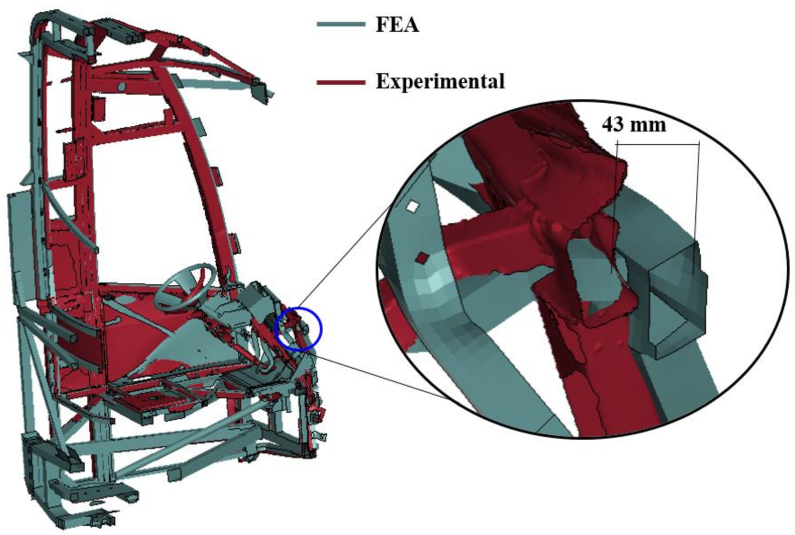



Figure 22 - Scanned model overlay and simulation

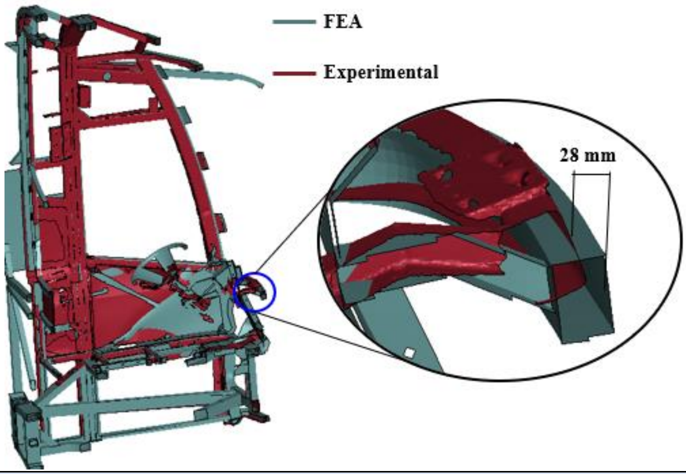
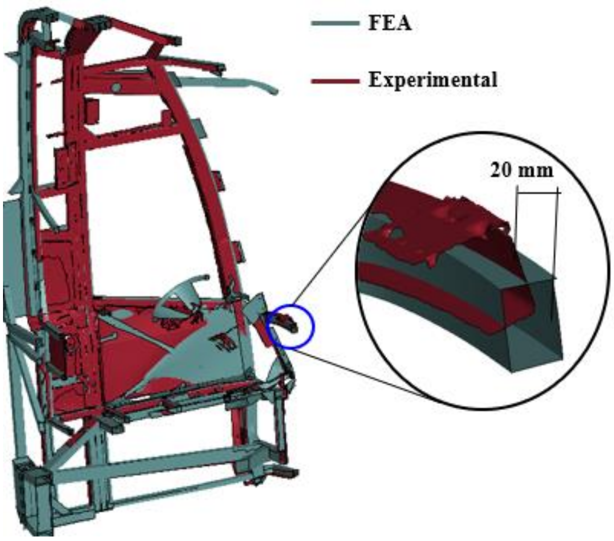
Table 4 presents the percentage difference between the FEA and the experiment, where it is possible to observe that the maximum difference occurs in section 1 with 19% and the smallest difference occurs in sections 2 and 6 with 12%. Averaging the differences in the evaluated sections, 14% was found. It is important to note that the virtual model deforms less than the experimental one. This difference may be related mainly to the constitutive model of strain rate used.

*Table 4 - Difference between FEA and Experimental test*

| Section | Difference between FEA and Experimental test |
|---|---|
| Section 1 | |
|  | 19 % |

| Section 2 | |
|---|---|
|  | 12 % |
| **Section 3** | |
|  | 13 % |
| **Section 4** | |
|  | 16 % |

© 2021 Copyright by DYNAmore GmbH

| Section 5 | |
|---|---|
|  | 13 % |
| **Section 6** | |
|  | 12 % |

# 6 Summary

This study aimed to develop a methodology to validate a survival cell for the bus driver called FIA (Frontal Impact Absorber), aiming to increase vehicle safety in situations where a frontal collision occurs. To reach the final concept of impact absorber, several nonlinear explicit structural analysis were performed using the LS-Dyna software. In order to prove the efficiency of the FIA, a physical test was performed. This test consisted of a semi-frontal collision between two buses, where one vehicle remained stationary and the other collided with it at a speed of 40 km/h using an offset of 50% between the vehicles. A guide-type device was developed to keep the vehicle's trajectory in motion.

When evaluating the results, it was noticed a good correlation between the experimental test and the FEA analysis, where the average displacement difference found is in the range of 14%. When comparing the vehicles tested, the difference between the models with and without the FIA package is evident. The bus without this component had a deformation value measured at the point of greatest displacement of 648 mm, whereas the vehicle with FIA device, 273 mm, resulting in 58% less deformation in this vehicle. Therefore, it is evident that this device can help to minimize the damage caused to the occupants of the vehicle that is equipped with this device during the event of a collision.

It is noteworthy that the vehicle analyzed as the current product complies with all current legislation. However, an additional structure was proposed because it is considered that the tests proposed in the current regulations do not represent a real accident condition for coach bus vehicles.

© 2021 Copyright by DYNAmore GmbH

## 7 Literature

1. Sindicato Nacional da Indústria de Componentes para Veículos Automotores. “Relatório da Frota Circulante,” 2017.
2. Confederação Nacional de Transporte. “Pesquisa CNT de Rodovias 2017”, 2017
3. Sánchez, C. A. A. “Estudo de Impacto Usando Elementos Finitos e Análise não Linear”. Tese de doutorado, Universidade de São Paulo, 2001.
4. CONTRAN. “Resolução No 629 - Estabelece os Requisitos de Segurança Para Veículos de Transporte Público Coletivo de Passageiros”. 2016.
5. Programa Volvo de Segurança no Trânsito. “Atlas da Acidentabilidade no Transporte Brasileiro”, 2018.
6. Polícia Rodoviária Federal. “Dados Abertos Acidentes de Trânsito”, 2019.
7. Diário do Transporte. **Acidente com ônibus a serviço da Buser deixa motorista morto em Minas Gerais**. Disponível em: https://diariodotransporte.com.br/2019/11/13/acidente-com-onibus-a-servico-da-buser-deixa-motorista-morto-em-minas-gerais/ Acesso em: 10 de dezembro de 2019
8. Folha de São Paulo. **Acidente de ônibus em GO deixa ao menos sete**. Disponível em: https://www1.folha.uol.com.br/cotidiano/2018/02/acidente-de-onibus-em-go-deixa-ao-menos-sete-mortos.shtml Acesso em: 10 de dezembro de 2019
9. A Gazeta. **Acidente entre ônibus e caminhão deixa 1 morto e 8 feridos em São José dos Campos.** Disponível em: http://agazetarm.com.br/?p=29886Acesso em: 10 de dezembro de 2019
10. Gaúcha ZH. **Feriadão de Independência é o mais violento nas estradas em 2018.** Disponível em: https://gauchazh.clicrbs.com.br/geral/noticia/2018/09/feriadao-de-independencia-e-o-mais-violento-nas-estradas-em-2018-cjlvgxecm01xu01mn1w3z6usc.htmlAcesso em: 10 de dezembro de 2019
11. Junior, A. D. M. “Avaliação do comportamento da estrutura de ônibus rodoviário solicitado a impacto frontal”. Tese de doutorado, Universidade Federal do Rio Grande do Sul, 2010.
12. Jones, N. “Structural Impact”. Cambridge University Press, 2rd edition, 2012.
13. FMVSS. “Standard No 208 - Occupant crash protection”. 2007.
14. UNECE. “Resolution 66 - Uniform provisions concerning the approval of large passenger vehicles with regard to the strength of their superstructure”. 2010.

© 2021 Copyright by DYNAmore GmbH