LOAD DETERMINATION IN SMALL URBAN BUSES, USING A COMBINED FINITE ELEMENT AND MULTIBODY SYSTEM APPROACH

José Pasquini Junior
Sérgio Gasparinetti
Odilon Terreri Perseguim
Marcelo Alves Bezerra
Rodivaldo Henrique da Cunha
debis humaita IT Services Latin America

Álvaro Costa Neto
University of São Paulo

Summary

The present work describes a methodology being adopted at debis in order to determine realistically joint loads in suspension parts of small urban buses.

Finite element models available at the moment do not allow for many non-linearities and rigid body large motions, typical of suspension parts when they cross large road irregularities, such as bumps and potholes.

Multibody system models can easily represent these effects. However, ideal kinematic joints do not consider local stiffness of connecting points of suspension parts and therefore tend to result in larger reaction forces than reality. Also, many real systems tend to be overconstrained from a kinematic point of view, resulting in automatic elimination of constraints in the multibody code and unrealistic loads at the remaining constraints. Finally, many parts may suffer elastic deformation, which also alter the joint load.

The approach adopted by this work takes local stiffness of connecting points and the flexibility of same parts using finite element models previously developed. Local stiffness is obtained through static application of a unit displacement in all degrees of restraints of the joint. This stiffness is represented using a bushing element in ADAMS. For parts whose flexibility is important, a MNF (Modal Neutral File) representing the flexible body is imported into ADAMS.

A special tyre and contact model is implemented to represent model excitation, using a GFORCE and prescribed motion in ADAMS, with the help of the impact function.

Realistic suspension motions and joint reaction forces have been obtained, validating the adopted methodology.

1 Introduction

Present practice in Vehicle Design is taking Simultaneous Engineering a step further in Simulation Technology. Finite Element (FE) analysis is a well-established method for determining stress-strain states in mechanical system components. On the other hand, Multibody System (MBS) simulation has been widely used to study rigid body (full vehicle and subsystems) behaviour.

It can be observed a strong need to include large motions and many non-linearities in finite element models as well as to represent boundary conditions adequately. Multibody system models can easily represent these effects. However, ideal kinematic joints do not consider local stiffness of connecting points of mechanical systems. Also, many real systems are overconstrained from a kinematic point of view, resulting in automatic elimination of redundant constraints in the multibody code and unrealistic load at the remaining constraints. Finally, many parts may suffer elastic deformation, which also alters joint loads.

The engineering community is adopting different approaches. People from a FE background tend to implement their shortcomings within their own simulation environment, whereas MBS engineers are trying to increment the multibody tools. The approach adopted by FE people results in very large and very time consuming models, especially for dynamic analysis. Typical programs for these studies are LS-DYNA, ABAQUS, DYTRAN, PAM-CRASH, MADYMO, etc.

The approach adopted by MBS people includes importing inertia, damping and stiffness characteristics of flexible bodies into MBS models and exporting loads to most common FE codes. Shortcomings of such approach are only linear stiffness may be considered and the fact that it requires the swapping of simulation environments to get the desired results.

In our company, we are presently undergoing a process of evaluating the pros and cons of both approaches. Linear FE analysis has been a long tradition within the company. MBS simulation has been implemented in recent years and non-linear FE has been in use even more recently. The present work describes the work carried out with the MBS and linear FE approach, as presented in figure 1.

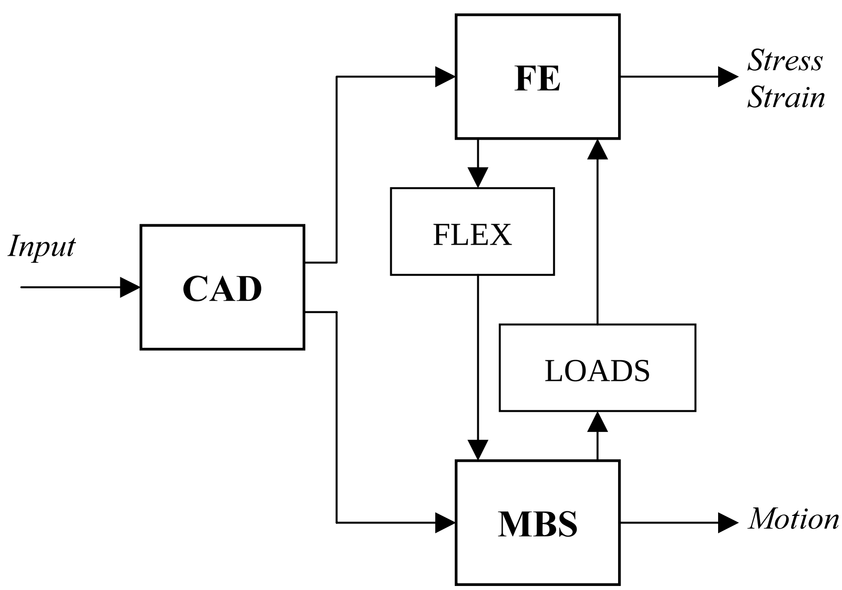

Figure 1 – MBS and FE approaches.

The vehicle studied in this work is a small urban bus (9 ton) with separate frame and car body assembled together. Front suspension is comprised of leaf springs, dampers and stabilizer bar and the rear suspension is made of leaf spring, drag links, air springs, supporting tray, stab-bar, dampers and Panhard rod. An overview of this bus can be seen in figure 2.

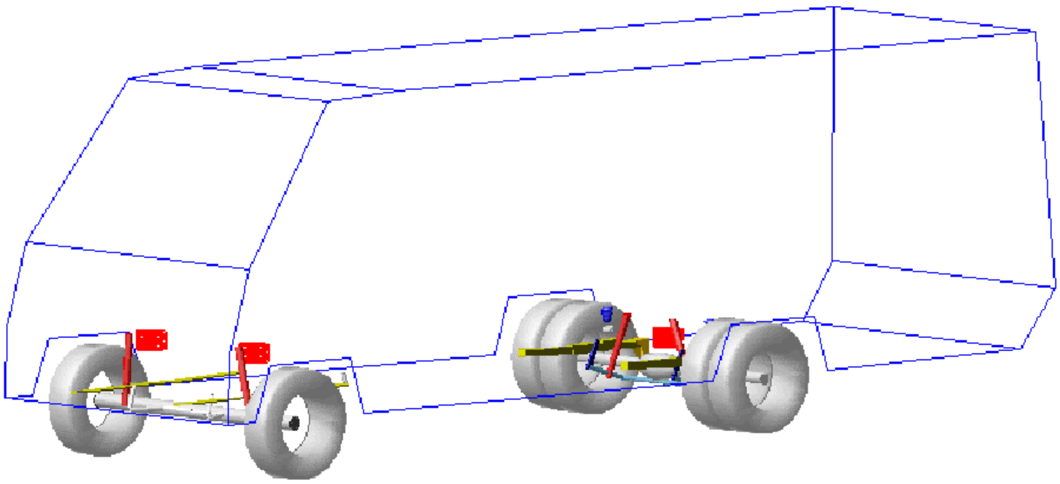

Figure 2 – Multibody model of the bus.

The objective of this analysis is to determine impact loading into elements and supports of the rear suspension.

2 Finite Element Model of Full Bus

A linear FE model of the bus has been developed for durability analysis. The model has 30.000 elements and 33.000 nodes. The model includes linear representation of front and rear suspensions and the tires. Figure 3 given below illustrates such model.

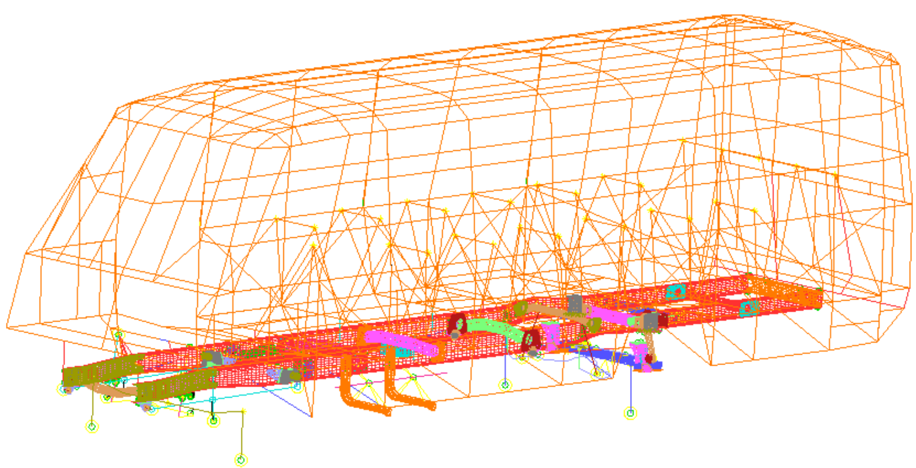

Figure 3 – Finite element model of the bus.

The rear suspension model includes linear air and leaf springs, the supporting tray for the air springs, linear dampers and a Panhard rod. This model is illustrated in figure 4.

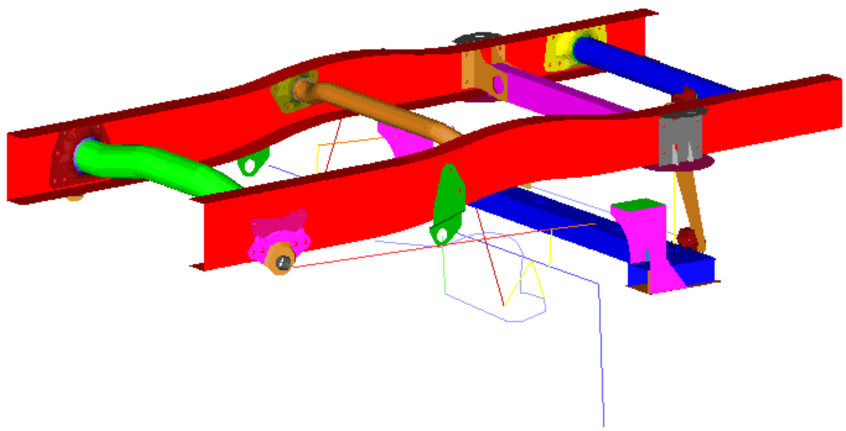

Figure 4 – Finite element model of the rear suspension.

Front suspension is comprised of linear leaf springs and linear dampers. It can be seen in figure 5. In both cases the axle is considered a rigid body.

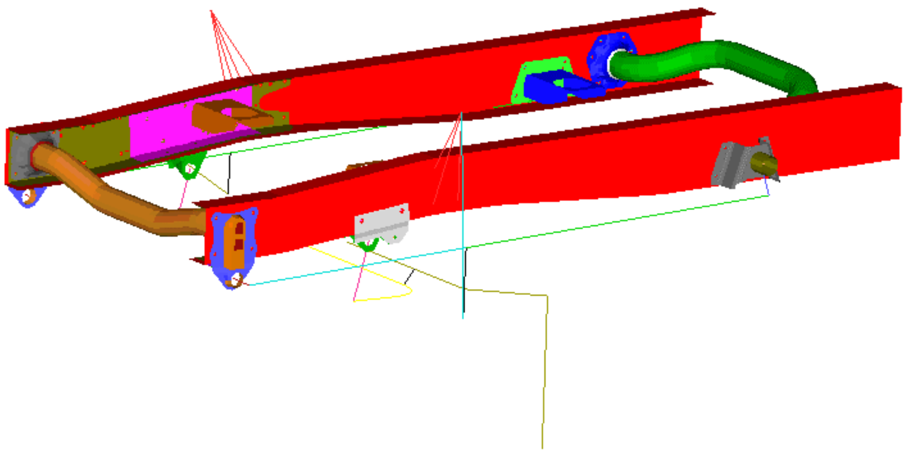

Figure 5 – Finite element model of the front suspension.

One of the main focuses of a durability analysis is the frame. In this case, it is modelled in greater details. The model includes the longarines, crossbars and supports. It is illustrated in figure 6 below.

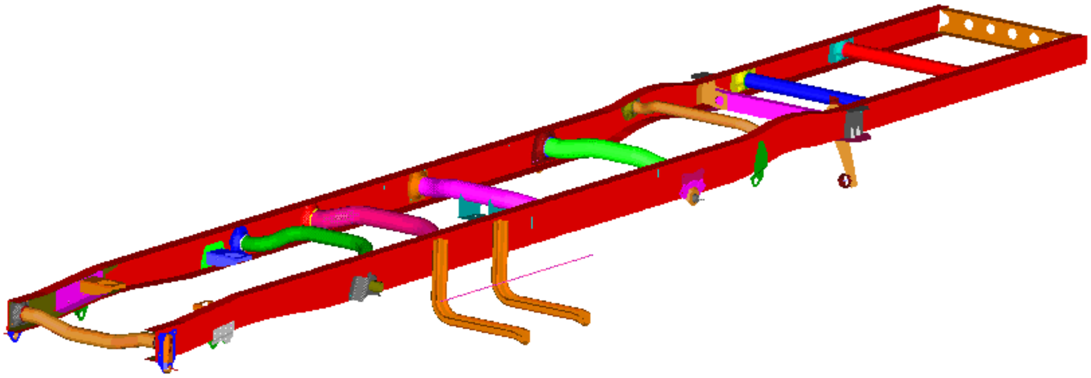

Figure 6 – Finite element model of the frame.

Superposed onto the frame is an approximate model of the car body, which is built by a third party. The representation of this model is given in figure 7.

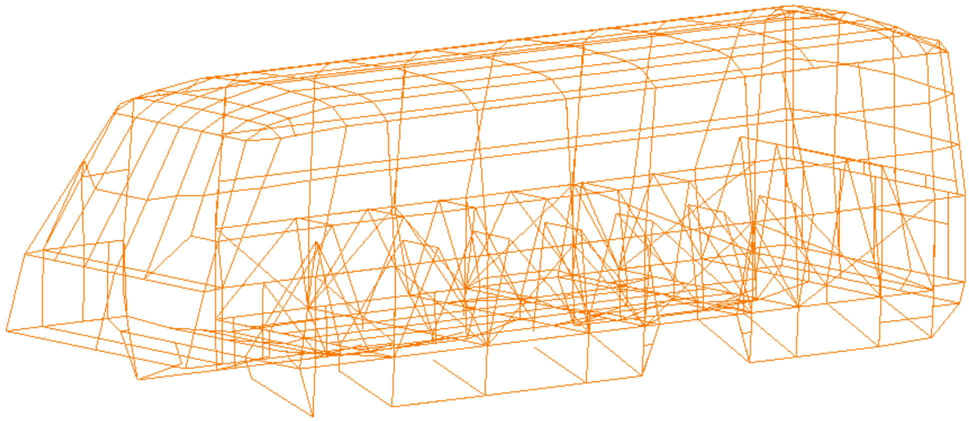

Figure 7 – Finite element model of the car body.

The FE model will serve many purposes. One of them is the determination of local stiffness at attachment points of front and rear suspensions. Also, it will be used to provide a flexible representation of the rear air springs tray (leaf springs are modelled using the ´beam´ element in ADAMS).

Later, separate portions of the frame are used as load input points from the suspension attachment points and the car body connecting points. The objective of such approach is to simulate a simpler model, which includes correct boundary conditions for the suspension supports.

3 Multibody System Model of the Bus

The multibody model of the bus is comprised of the following elements: front suspension, rear suspension, bus body, frame (considered as a rigid body) and air-spring tray imported as a flexible body.

3.1 Subsystem Models

Front suspension layout can be seen in figure 8 below and it is comprised of wheels and axle, air springs, leaf springs and stab-bar.

Rear suspension is comprised of air spring tray, axle and wheels, leaf springs/drag link and Panhard rod. Its model is illustrated in figure 9.

For the purpose of the MBS model, the bus body and the frame are considered as rigid bodies. The reasons for adopting this approach are as follows: The FE models of the bus body and frame for stress analysis are very detailed and the resulting MNF is too large. And also, to perform a remeshing of these models would take too long.

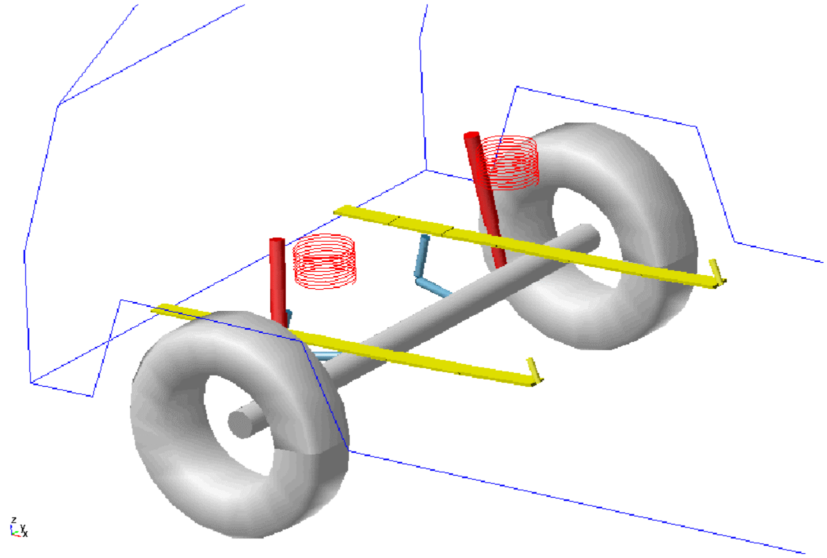

Figure 8 – Multibody model of the front suspension.

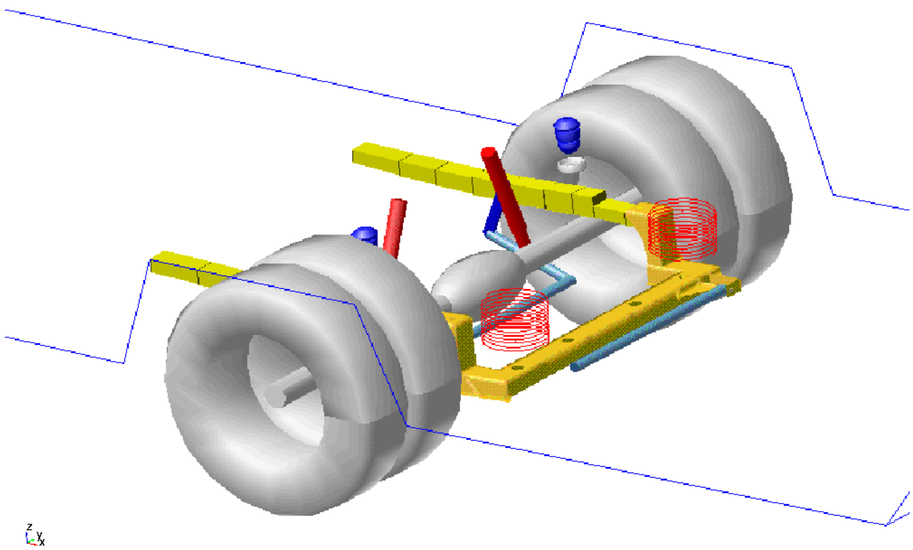

Figure 9 – Multibody model of the rear suspension.

The option adopted was to represent this flexibility locally using the ´bushing´ element in ADAMS. Static simulation was carried out at each connecting point of the frame and suspension and local stiffness were obtained.

The air springs tray was imported into ADAMS as a flexible body as illustrated in figure 10.

3.2 Force Elements

The air spring was modelled as a G-force with force-deflection characteristics dependent on air pressure. The air pressure is dependent on vehicle load in a way of maintaining a constant ride height. Typical curves are shown in figure 11.

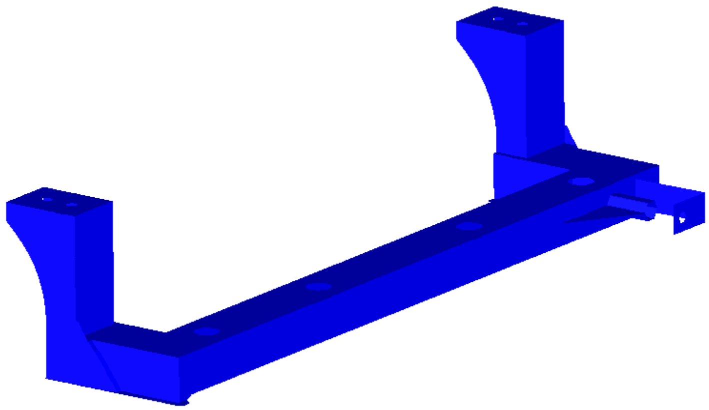

Figure 10 – Finite element model of the air spring tray.

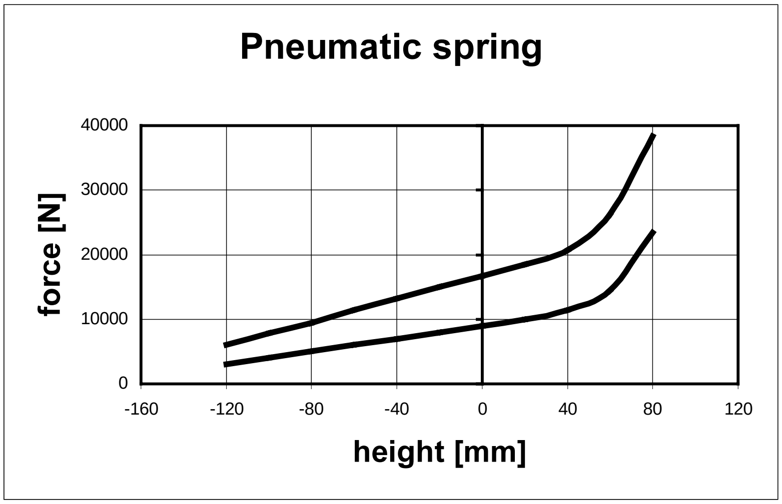

Figure 11 – Typical curves of air springs.

The leaf springs were modelled as ´beam´ elements in ADAMS, using the Vehicle module.

Damper characteristics obtained with the manufacturer were represented by splines in ADAMS.

The tire-road contact was modelled in two different ways. For both of them, terminated ramp road profiles were considered, with different ramp length.

For one of them, a non-rolling radial spoke model for the contact patch sector represented the tire. In this situation, the ramp length was short in order to generate larger longitudinal load components. The other one considered a single contact point model. In this case, the ramp was made long enough in order to hold the single point hypothesis. Varying level of road input severity was obtained transversing the ramp at increasing speeds.

3.3 Vehicle Simulation

Full vehicle simulations carried out include in-phase and out-of-phase obstacles crossing in a straight line and in-phase ramp and single drop at a lateral acceleration of 0.4 g. In all cases, the vehicle was at a velocity of 20 km/h, and the obstacle height was 75 mm.

4 Simulation Results

Two substructures were analysed: the rear portion of the frame and the air-spring tray. In both cases, loads passed into NASTRAN were the inertia forces of the parts and the reaction forces at the connections.

The resulting connection forces are obtained at the bushing elements used to represent local stiffness of the structure, with the exception of a fixed joint connecting the leaf spring and the air spring tray.

The MBS simulation results of the loaded bus with the non-rolling radial spoke tire model can be seen in figures from 12 to 14. The vertical acceleration of the rear axle can be seen in figure 12 for three simulation conditions. The vertical force generated by the air springs can be seen in figure 13. Figure 14 illustrates the forces acting on the left leaf spring support. These results were used to preliminary validate the model based on road tests carried out with similar vehicles and FE durability simulation experience.

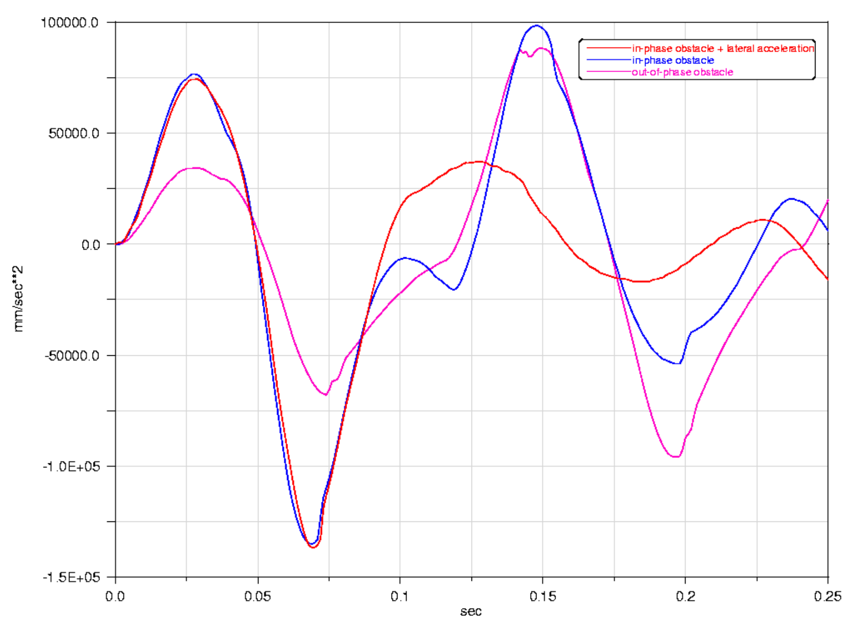

Figure 12 – Vertical acceleration of the rear axle.

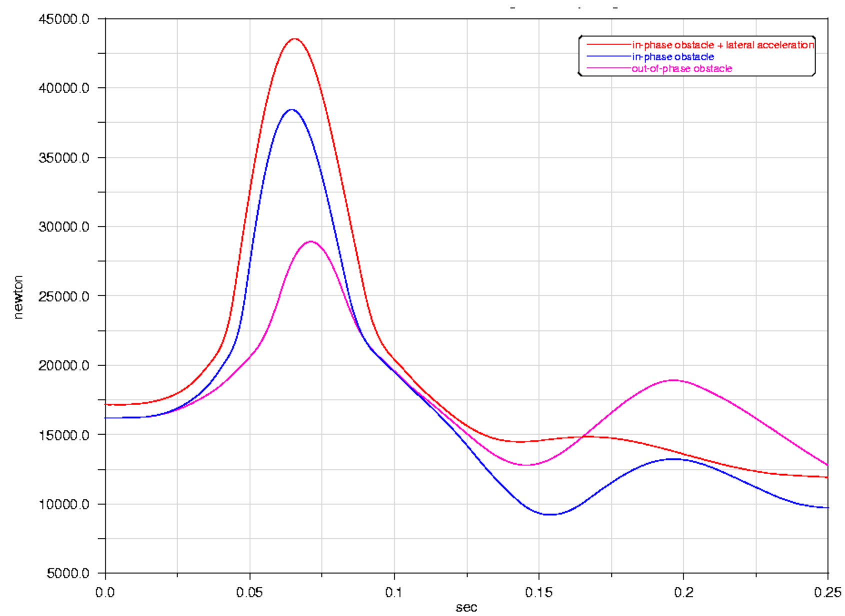

Figure 13 – Vertical force on the right air spring.

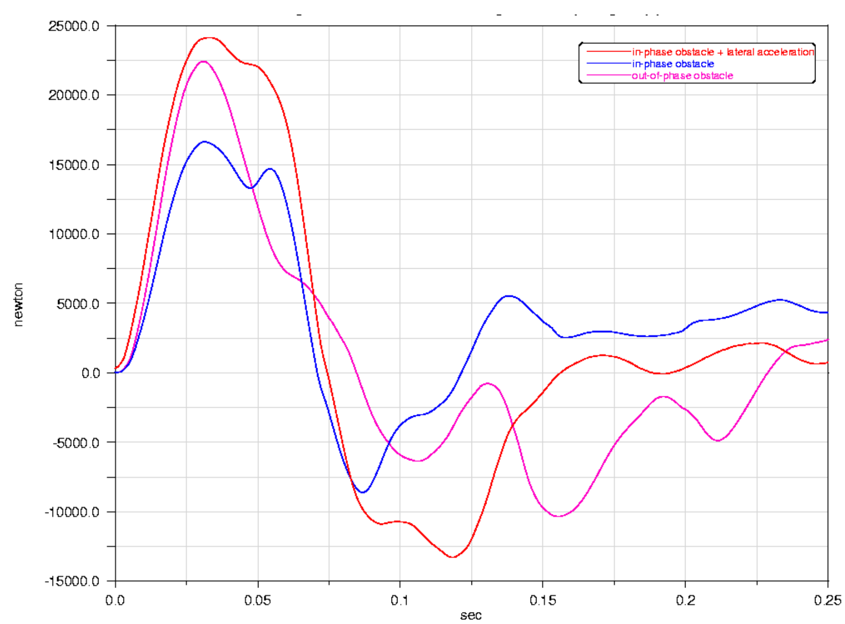

Figure 14 – Longitudinal force on the right leaf spring support.

An equivalent static analysis was carried out in NASTRAN in which inertia plus reaction forces are imported from ADAMS.

The stress distribution on the air spring tray and on some components of the rear suspension can be seen in figures 15 and 16, respectively.

The results obtained are in close agreement with known results from linear FE analysis for portions of the frame and axle accelerations measured in field tests. However, for the rear suspension parts, MBS simulation resulted in larger stress values, possibly due to the nonlinear effects. Nevertheless, confirmation of such results can only be obtained from experiments, which we have not been able to perform yet.

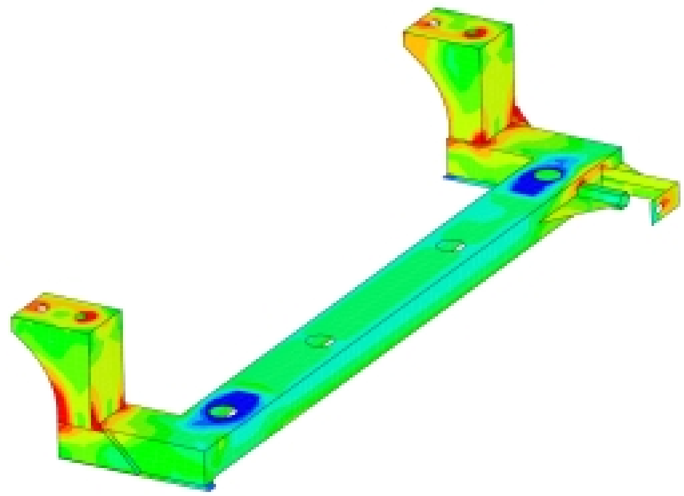

Figure 15 – Stress distribuition on the air spring tray.

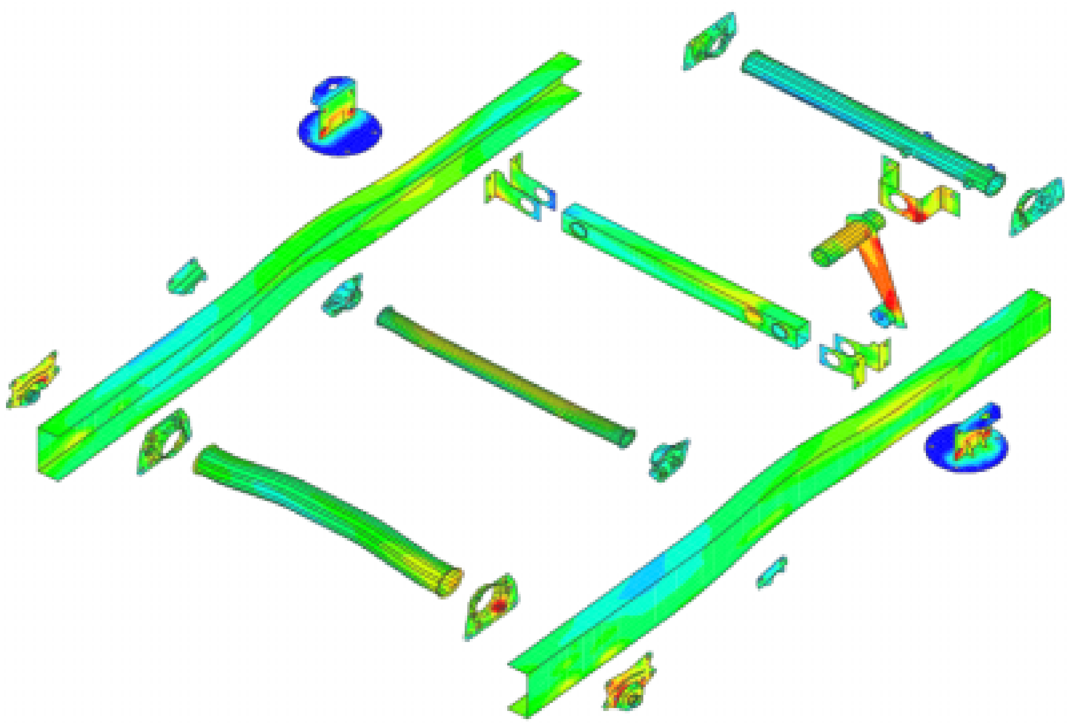

Figure 16 – Stress distribuition on some components of the rear suspension.

5 Conclusions

The simulation of a small urban bus has been carried out using a MBS/FE integrated approach. The objective of the analysis was to evaluate the stress on the elements of the rear suspension under some critical operating conditions. The MBS simulation were performed and the results exported to the FE model. This approach was used in order to simplify the analysis, with the following advantages:

- Low computational costs;
- Simpler MBS and FE analysis using the main features of each technology;
- Use of an already refined FE model;
- Possibility to study the dynamic behaviour of the vehicle, with the MBS model.

Future work includes the development of a more representative tire model; a study to validate the static analysis with low natural frequency structures; finally, and mainly, validate these MBS/FEA simulations with experimental results.

Acknowledgements

The authors wish to thank Mercedes-Benz do Brasil for the permission to publish this work and the support and data provided by their TAS and TCO divisions.

References

[1] Using ADAMS/Solver Version 9. Mechanical Dynamics, Inc., 1998.

[2] MSC/NASTRAN Version 70 Reference Manual. The MacNeal-Schwendler Corporation, 1997.