Structural Analysis of Brake System's Air Reservoir of Heavy Duty Vehicles

Mr C. Reis, Mr J. Tonioli, Mr F. Anjos and Mr M. Argentino, debis humaitá ITServices Latin America, Brazil

ABSTRACT

The present work describes the use of computer simulation supported by experimental data on the structural analysis process of a heavy duty commercial vehicle. There is a worldwide trend to use more computer simulation during the development phase of new vehicles, in order to improve vehicle structural behavior and decrease development time and costs even before the prototype has been constructed.

In this paper the occurrence of cracks in the brake system's air reservoir prototype of a heavy duty commercial vehicle during track tests has been investigated.

The FEA software MSC.Nastran was employed and specific simulations were carried out for the reservoir and its brackets which were inserted in the full vehicle model.

The results obtained for the initial configuration were in agreement with the failure that occurred in the test track. Geometric and welding alterations in accordance to design criteria were proposed. New failures have not been verified in experimental tests since these modifications have been implemented.

INTRODUCTION

Present trend in automotive industry design involves the reduction of time and cost development. In this sense, the use of computer simulation supported by experimental data allows precise evaluation of different configurations in a short period of time. Not only does it contribute to a better understanding of vehicle's dynamic behavior but also structural strength, permitting a more effective approach in design process.

In this work, it was investigated the crack occurrence on a brake system's air reservoir of a heavy duty vehicle, when submitted to a track test. The cracks appear in the welded region between the reservoir and its brackets, causing an air leak and consequently pressure loss and performance of the break system.

A numerical tool based on the Finite Element Method has been employed and specific simulations have been carried out for the reservoir and its brackets, which have been inserted in the full vehicle model.

PROBLEM DEFINITION

In the track test of a heavy duty truck, its brake system's air reservoir presented failure, forcing the test interruption. The aim of this work is identifying the possible causes of the failure and suggest proposals in order to solve this problem.

The main loads acting in the brake system's air reservoir are the track excitation and the reservoir's internal air pressure. Analyses simulating this loads have been carried out in order to investigate its effects in air reservoir. There have been carried out static analyses to determine the constant stresses due to internal pressure, and dynamic analyses in order to obtain the stresses due to track excitation.

SYSTEM MODELING

There have been generated finite element models for the air reservoir, batteries' subsystem (where the reservoir is attached) and for the complete vehicle in order to perform the analyses. Different air reservoirs' configurations have been analyzed until configuration that complies with the design criteria have been reached.

Air reservoir models

It has been developed one separated model for each air reservoir configuration analyzed. The models for the original and final configurations of the reservoir's fixation can be seen in figure 1, shown below:

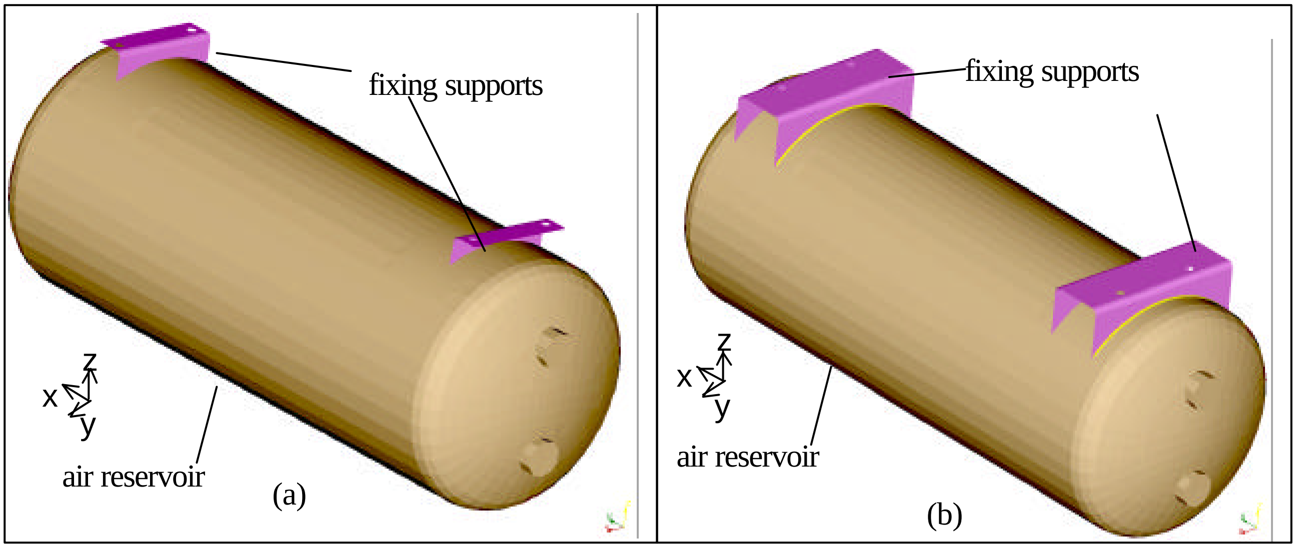

Figure 1 - Air reservoir models: a) original configuration; b) final configuration.

For the final configuration (figure 1.b), the geometry of the fixation supports was changed from a "L" shape to an "U" shape and its length (along the y axis) was increased and both sides of the connection region between the support and the air reservoir have been considered welded.

Batteries subsystem model

The batteries' subsystem has been modeled with shell elements including the reservoir model and considering part of vehicle's frame. The subsystem model can be viewed in figure 2.

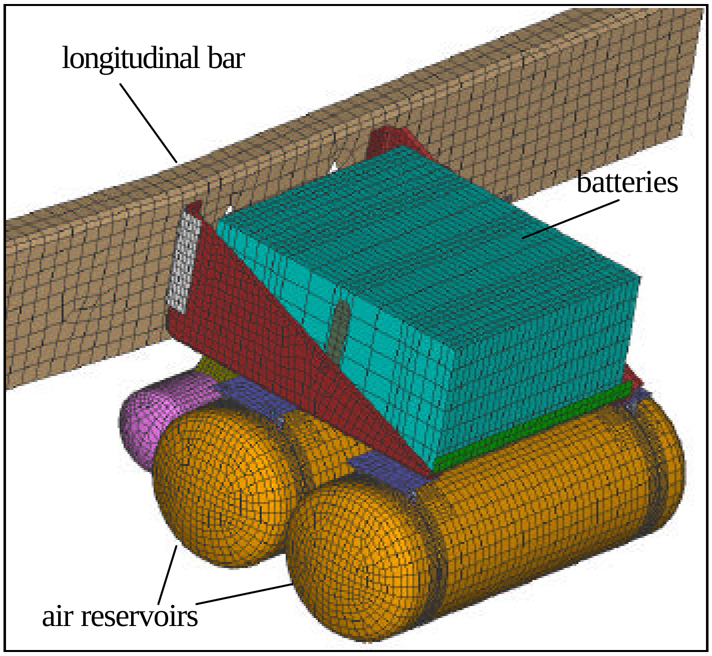

Figure 2 - Battery subsystem model.

Complete vehicle model

A full vehicle model has been generated considering the frame, suspension, cab, engine, load compartment, and other components. The air reservoir and the battery subsystem model have been included into it. Figure 3 illustrates the full vehicle model.

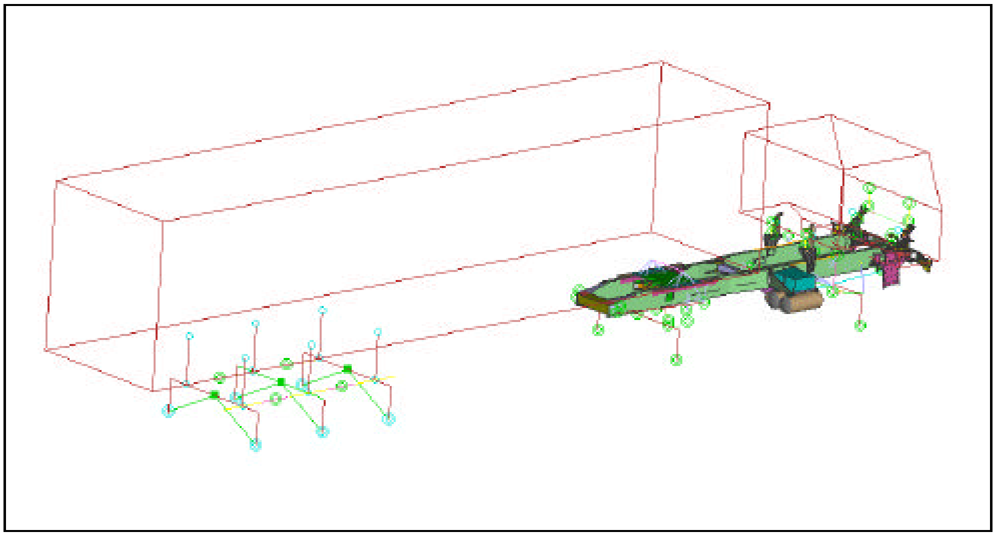

Figure 3 – Complete vehicle model.

MATERIALS

The material which has been used in the air reservoirs and its brackets is standard steel. Its mechanical properties are listed in table 1, shown below.

Table 1 – Material's mechanical properties.

Component	**Air reservoir**	**Fixing supports**
Material	**St 37-2**	**SAE 1008**
Elasticity Modulus (N/mm^2)	210000	210000
Poisson Coefficient	0.3	0.3
Yield strength (N/mm^2)	235	170
Tensile strength (N/mm^2)	360	303

ANALYSES

The analyses have been carried out by means of software MSC/Nastran v. 70.0 for linear static and modal analyses and v. 67.5 for transient dynamic analyses. The load acting in the air reservoir is a composition of a static load due to internal air pressure, and a dynamic load due to track excitation. To understand the dynamic behavior of the batteries and air reservoirs' subsystem, it's important to know its natural modes and frequencies of vibration.

Internal pressure

A load equivalent to an internal pressure of 10 bar has been applied to the air reservoir models (figure 1) for the linear static analyses (MSC.Nastran SOL 101) realized. The air reservoirs were supposed to be supported on a rigid base.

Modal

A modal analysis (MSC.Nastran SOL 103) for the batteries' subsystem (figure 2) has been carried out with a view to obtaining the natural frequencies and modes in the range of interest, that is the critical frequency excited by the track. This might lead to a better understanding of its dynamic behavior and the failure mechanism.

In the batteries' subsystem model the translation DOFs in both section edges of the modeled longitudinal bar had been restrained.

Transient dynamics

A transient analysis (MSC.Nastran SOL 112) with track excitation for the full model (figure 3) has been performed for each analyzed configuration of the air reservoir. To this transient analysis a signal data acquired from the standard test track, where the failures have been verified , have been imposed to the full vehicle model. The analyses have been carried out for maximum load condition to the original configuration and for maximum load and unload conditions for the final configuration.

DESIGN CRITERIA

The stresses obtained in the static analyses have been used to correct the materials' fatigue limit to the case where the mean stresses are not zero. This correction is made using the Goodman equation [1]. For welded regions, it has been considered additionally static and dynamic stress concentration factors, depending on welding configuration and loading direction at the region [2]. The maximum allowable stresses were determined for each analyzed configuration. The allowable stress limits for the original and final configuration for the dynamic transient analyses are shown in table 2:

Table 2 – Allowable stress limits for welded region.

Fixing support configuration	**Allowable stress (N/mm^2)**
Original	33
Final	43

The stress limits for the welded regions have taken into account SCF (Stress Concentration Factor) due to welds and the correction of the mean stress when it differs from zero.

RESULTS

The modes and frequencies found, for the air reservoir, in the modal analyses performed are listed in table 3:

Table 3 - Natural modes and frequencies.

Configuration	**Original**	**Final**	**Description**
Mode	**Frequency (Hz)**	**Frequency (Hz)**	
1	26.84	26.74	reservoirs' movement around X axis
2	28.38	28.03	reservoirs' movement along X axis

The static and dynamic analyses results for the original and final configuration are shown in table 4 and in figures 4 to 7.

Table 4 - Static and dynamic analyses results.

Loading	**Original configuration**	**Final configuration**
$\sigma_{pressure}$ (N/mm^2)	88	70
$\sigma_{transient}$ (N/mm^2)	82	39

DISCUSSION

The modal analyses results show low values to the reservoirs' natural frequencies. The first and second frequencies are in the range of the track excitation. Therefore the failures are most likely to be caused by resonance effect. The determination of the predominant direction of the natural modes helps to locate the critical points to be reinforced or changed to minimize the resonance effect.

A stress of 88 N/mm^2 is found in the welded region between the air reservoir and its bracket in the static analysis (figure 4) performed and a value of 82 N/mm^2 for the transient dynamic one (figure 5). This value is above the design criteria limit of 33 N/mm^2 for that region. This result justify the failure occurred in the track tests.

The alternative configuration proposed and analyzed has met the adopted design criteria. The maximum stress obtained, in the transient analysis, for the welded region between the air reservoir and its bracket was 39 N/mm^2 (figure 7), which is below the limit of 43 N/mm^2.

The improvement in the stress distribution and limits, for final configuration, is due the new weld configuration that introduce a smaller SCF, and the greater weld length that allow better stress distribution and the "U" shape of the new fixing support that causes a smaller bending load in the weld fillets.

CONCLUSIONS

The numerical results for the original version of the fixing supports really identify the failure occurred in the track test. The analyses have shown that the final configuration complies with the design criteria and, therefore, can be used in the production vehicle. New failures, in the analyzed component have not been verified in the track tests since the new configuration have been adopted.

The present work has described the successful use of numerical simulation in the design processes of a heavy duty commercial vehicle. The numerical simulation has allowed different concepts of the air reservoir to be tested before a new prototype has been constructed. As a consequence, development cost and time of such component can be reduced. From this, one can conclude that numerical simulation is a powerful tool to be used in the development phase of new vehicles, in conjunction with experimental data and validation procedures.

REFERENCES

[1] - Shigley, Joseph Edward, Elementos de Máquinas, Livros Técnicos e Científicos S.A., 3ª edição, 1984.

[2] - Niemann, G., "Elementos de Máquinas", vol. 1, Editora Edgard Blücher Ltda, 1971.

[3] - SAE J1397, Jul/82;

[4] - DIN 17100, Jan/98;

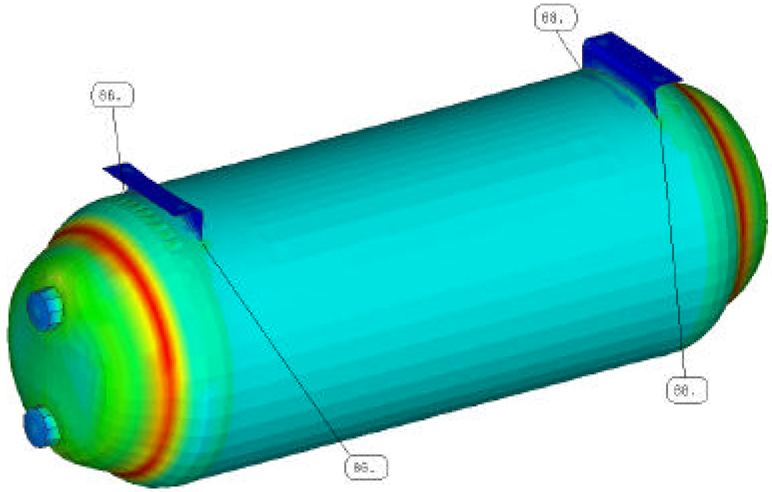

Figure 4 - Original configuration: internal pressure load.

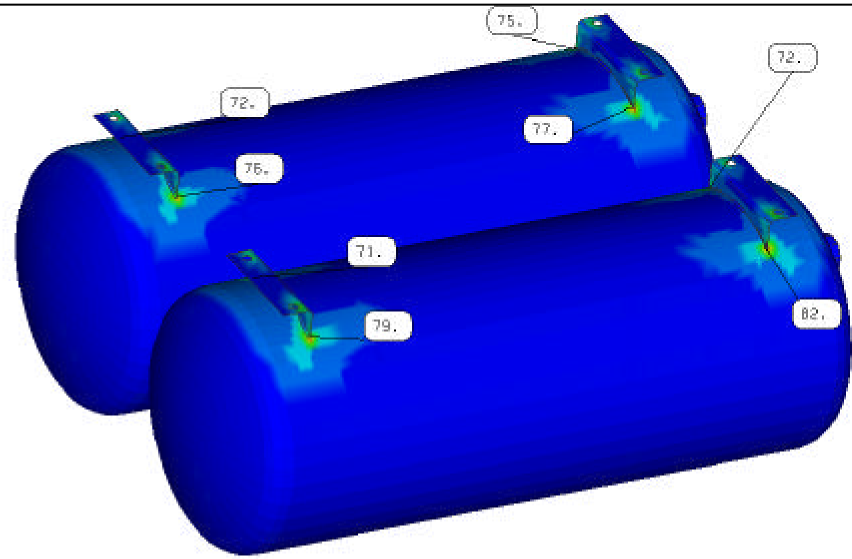

Figure 5 - Original configuration: transient dynamic analysis.

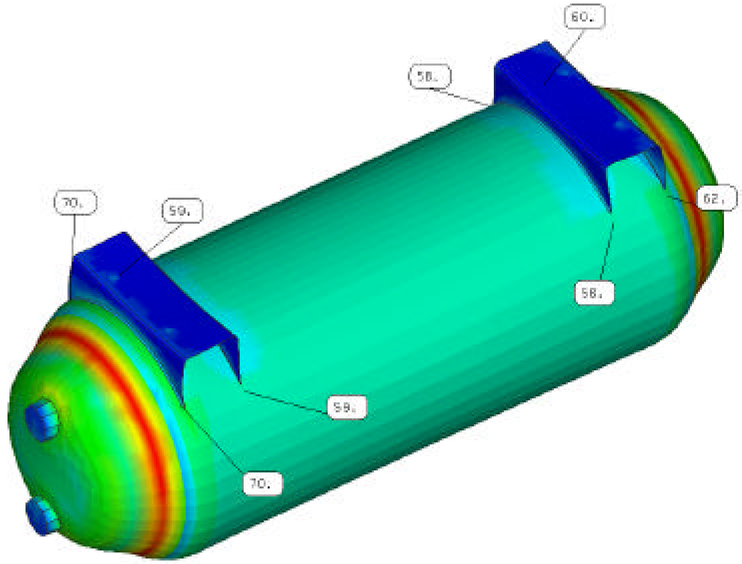

Figure 6 - Final configuration: internal pressure load.

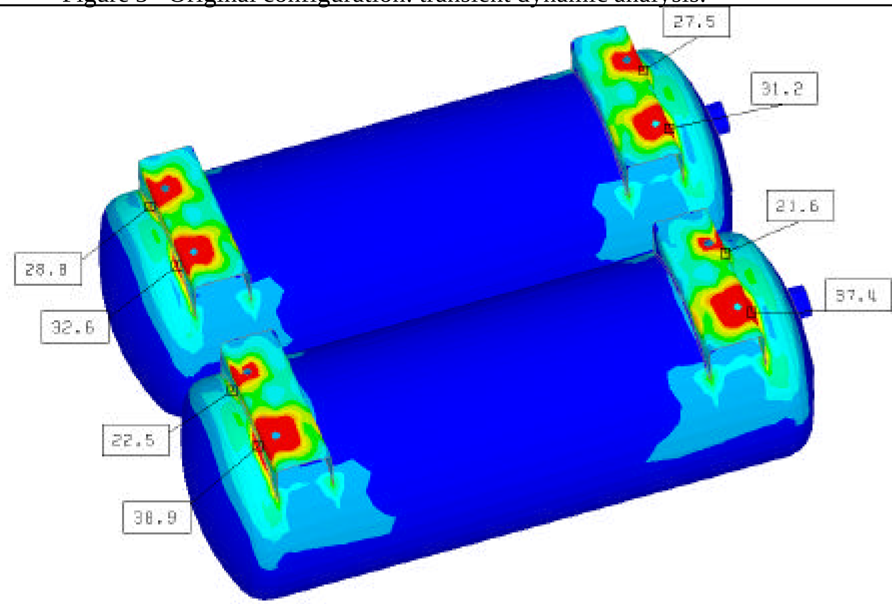

Figure 7 - Final configuration: transient dynamic analysis.