Structural Analysis of an Exhaust System for Heavy Trucks

Mr N. Vasconcellos, Mr F. dos Anjos and Mr M. Argentino, debis humaitá IT Services Latin America, Brazil

ABSTRACT

The present work describes the use of computer simulation supported by experimental data on the design process of an exhaust system for heavy duty commercial trucks, with regard to structural behavior.

A finite element model was generated including the complete vehicle and the exhaust system. Static and dynamic analyses were performed in MSC.Nastran software simulating different loading conditions and system configurations. The results obtained assure the structural integrity of the exhaust system and also contribute to a better understanding of this system behavior and its structural strength.

INTRODUCTION

As a traditional procedure, track tests are made to assure the proper structural design of vehicles and life determination. There is a worldwide trend to use more computer simulation during the development phase of new vehicles, in order to improve structural behavior and decrease the time to market and costs. Accurate finite element model must be generated in order to predict the system structural behavior and to allow fast optimization processes. This work shows the CAE procedure applied to an exhaust system for a heavy truck.

PROBLEM DEFINITION

In the track test of a dumper truck, its exhaust system presented failure in the welded region within the primary tube and its bracket (figure 3), forcing the test interruption.

The aim of this work is to identify the possible causes of the failure and suggest proposals in order to solve this problem. An exhaust system of a vehicle is mostly excited by the track through the frame fixations, by the engine vibration and by the powertrain translations on its brackets. Therefore, all these potential alternatives of excitation must be investigated.

SYSTEM MODELING

A finite element model was generated for this study including the complete vehicle and the exhaust system fixed on the longitudinal member and on the powertrain. The vehicle – a dumper truck - was modeled considering the frame, suspension, cab, engine, load compartment, and other components such as fuel tank and batteries. The model can be seen in figure 1.

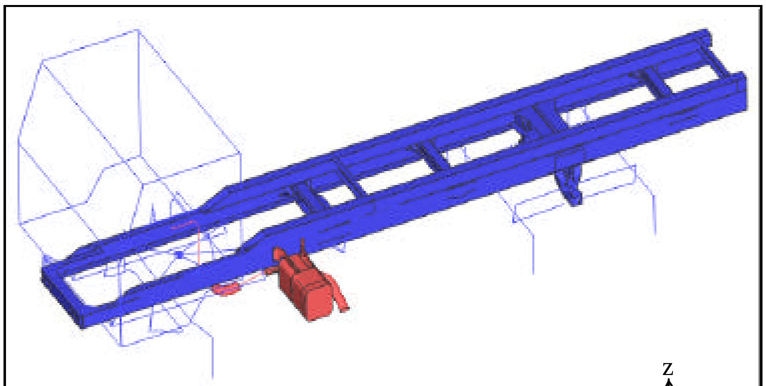


Figure 1 – Complete finite element model.

The exhaust system model (figure 2) was developed including the primary exhaust tube and its bracket with stripes, the flexible tube, the muffler and its bracket, and the secondary exhaust tube. The model of the primary tube bracket, which presented failure, is shown in figure 3.

LOADS

Static and dynamic analyses were performed, by means of MSC.Nastran software, simulating different loading conditions and considering linear elastic behavior of all components modeled.

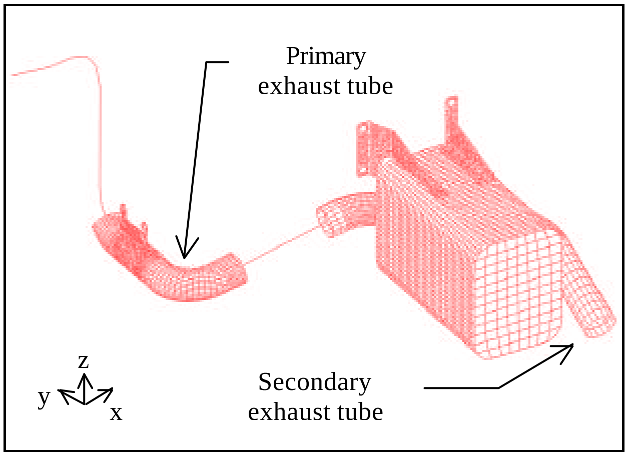


Figure 2 – The finite element model of the exhaust system.

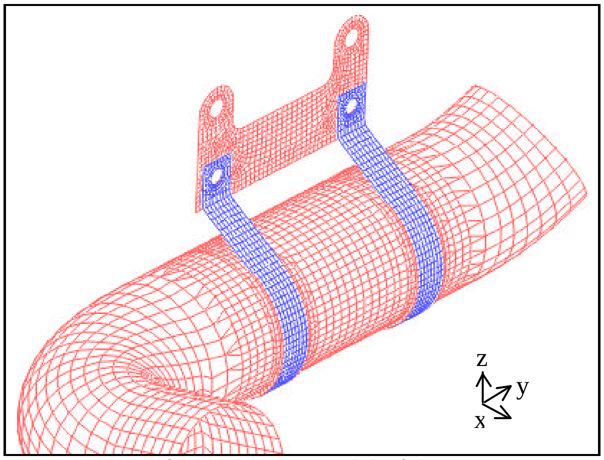


Figure 3 – The finite element model of the primary exhaust tube bracket with stripes.

Powertrain translation

As the powertrain translates, do to braking or curves for example, it pulls the exhaust system through the primary tube bracket. In this situation, the flexible tube must deform. However, experimental tests made for this component, showed that it can apply the maximum axial force of 600 N in the primary tube, before deforming as expected. A linear static analysis (MSC.Nastran SOL 101) was performed to simulate this condition, taking in account that the dynamic effects are not representative but considering a repetitive load that causes fatigue damage. The primary tube was restrained, as fixed in the engine, and a 600 N force was applied in the flexible tube.

Track test

For track test analysis, a signal data obtained from a real track test was used as input excitation applied to the tires of the numerical model, in a transient dynamic analysis (MSC.Nastran SOL 112), simulating the whole vehicle ride.

Engine vibration

According to modal analysis (MSC.Nastran SOL 103), the critical mode that can be excited by the engine vibration to cause the failure, is the longitudinal one, at a frequency of 62.3 Hz (table 4). The acceleration peak, due to engine vibration, measured at the primary tube bracket in the longitudinal mode direction was 1.4 g.

To represent this load case, harmonic analyses (MSC.Nastran SOL 108) were performed, considering a base sinusoidal 1.4 g acceleration applied to the primary exhaust tube bracket attachment points. The modal damping rates, estimated by means of the experimental results, were 3.1% for the longitudinal mode. This value was also used as a parameter for the harmonic analyses.

MATERIAL PROPERTIES

The exhaust system components are made of steel, and its mechanical properties are shown in table 1, considering isotropic material.

DESIGN CRITERIA

The maximum allowable stresses were determined based on the Ref. [1][2][3] for each of the three load cases and are shown in table 2. Since the powertrain translation and the engine vibration loading have a high number of cycles, the stress limits for both cases were determined for infinite life. Fatigue life reduction factors were already considered on this value, such as size and reliability.

Table 1 – Materials mechanical properties.

Component	Material	Yield strength [N/mm^2]	Tensile strength [N/mm^2]	Density [Mg/m^3]	Poisson Coefficient	Elasticity Modulus [N/mm^2]
Muffler bracket	St44-2	275	410	7.85	0.3	210000
Muffler, tubes and stripes	St1203	170	303	7.85	0.3	210000
Primary tube bracket	LNE28	280	390	7.85	0.3	210000

Table 2 – Allowable stress for each load condition.

Material	St1203	St44-2N	LNE28
Allowable stress in harmonic and static analysis [N/mm^2]	135	168	162
Allowable stress in track test analysis [N/mm^2]	151	190	220

For welded regions, it was considered an additional stress concentration factor depending on the geometry and loading direction of the regions. Thus, the stress limit was reduced by this factor as shown in table 3.

Table 3 – Allowable stress for welded parts.

Material	St1203
Allowable stress in harmonic and static analysis [N/mm^2]	30
Allowable stress in track test analysis [N/mm^2]	34

MODEL VALIDATION

The complete vehicle was previously calibrated to appropriately represent track tests. The exhaust subsystem models were validated by comparing the natural frequencies and modes of the numerical model to the experimentally measured values. To calculate the natural frequencies, it was used the computational Lanczos method. Table 4 shows theoretical and experimental modal analysis results for the exhaust system.

RESULTS

Track test

From track test analysis results, we can see that the von Mises equivalent stresses were above limits for the welded region between the primary tube and the fixing stripes, as shown in figure 4. The other parts, as the muffler bracket shown in figure 5, have not presented stresses above acceptable limits.

Table 4 – Natural frequencies and modes for the exhaust system.

Modes	Description	Theoretical result Frequency [Hz]	Experimental Frequency [Hz]
1st mode	vertical motion of the muffler	17.9	18.2
2nd mode	longitudinal motion of the secondary tube	22.3	23.6
3rd mode	roll motion of the muffler	32.2	not characterized
4th mode	vertical motion of the secondary tube	49.8	47.6
5th mode	longitudinal motion of primary tube	62.3	60.5

Engine vibration

The von Mises equivalent stresses obtained on the primary exhaust tube and its bracket as a result of the application of the base harmonic excitation, at the natural frequency of the longitudinal mode (62.3 Hz) are shown in figure 6. The critical welded region between the primary tube and the fixing stripes reached the maximum value of 69.9 N/mm^2, which is above the limit.

Powertrain translation

For the maximum force in the flexible tube, the static analysis also reveled high stresses level in the welded regions between the primary tube and the fixing stripes. The maximum value was 93.8 N/mm^2, which is above limit, as shown in figure 7. On the primary tube bracket, however, the stresses were below allowable limit in all three analyses and the same occurred for the other system components.

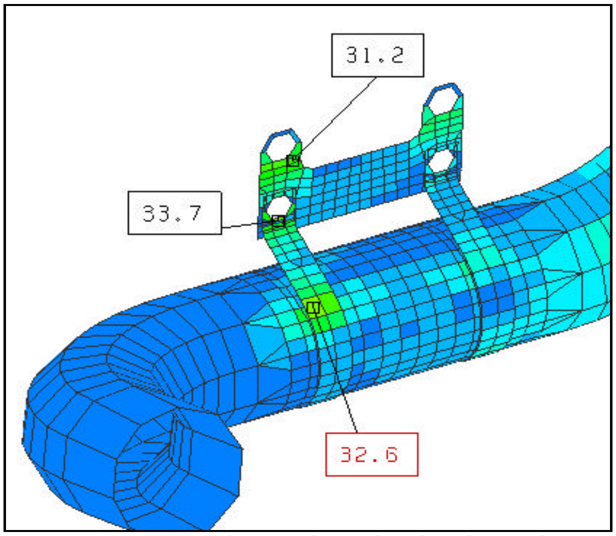


Figure 4 – Primary exhaust tube and its bracket with stripes. Stress distribution for track test analysis.

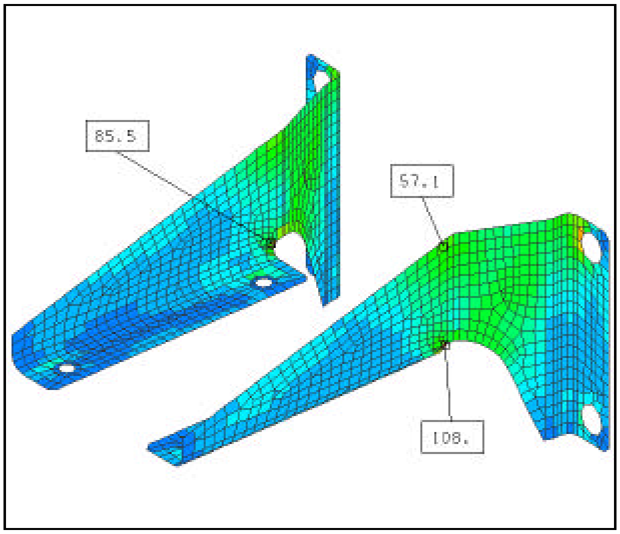


Figure 5 – Muffler bracket. Stress distribution for track test analysis.

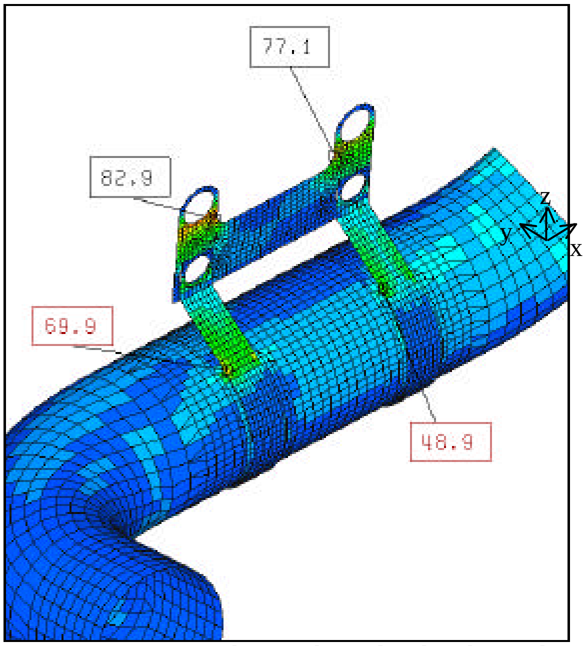


Figure 6 – Primary exhaust tube and its bracket with stripes. Stress distribution for base harmonic excitation analysis.

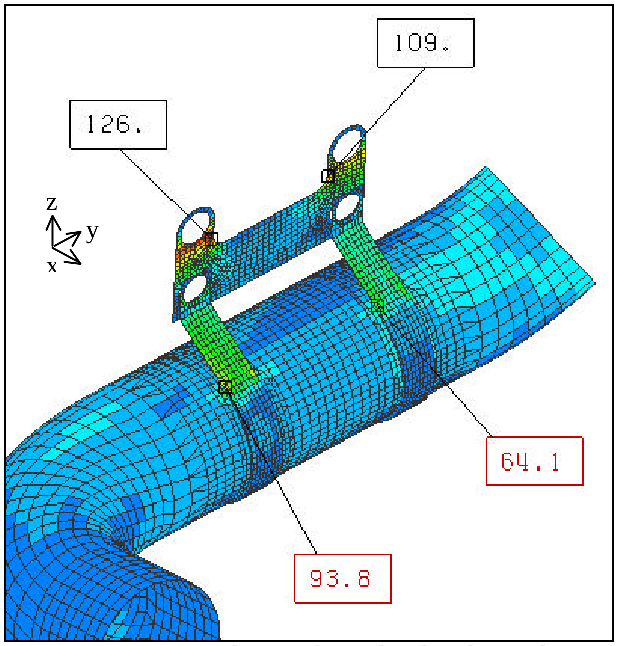
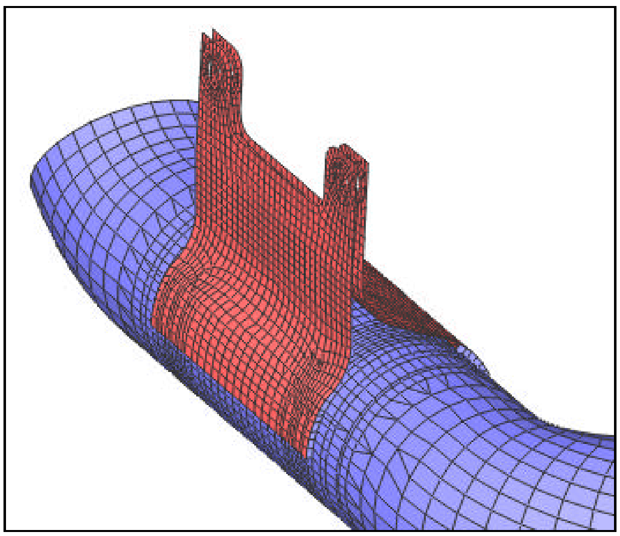
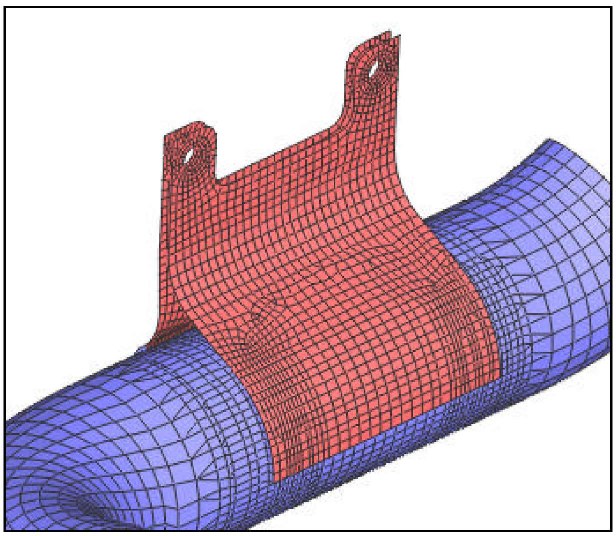


Figure 7 – Primary exhaust tube and its bracket with stripes. Stress distribution for maximum force in the flexible tube analysis.





bracket – front view.

bracket – rear view.

OPTIMIZATION

In order to decrease stresses level in the critical welded region between the primary tube and the fixing stripes, it was generated a modified version, joining the primary tube bracket and the fixing stripes into one unique piece, made of St1203 steel, as shown in figure 8 and 9.

Engine vibration

This new version presented the longitudinal natural mode for the exhaust system at a frequency of 69.9 Hz. Assuming the same damping rate for small frequency variation in a similar mode, the base harmonic excitation gives the results shown in figure 10. The maximum stress value reached in welded regions was 20.3 N/mm^2, which is below the limit.

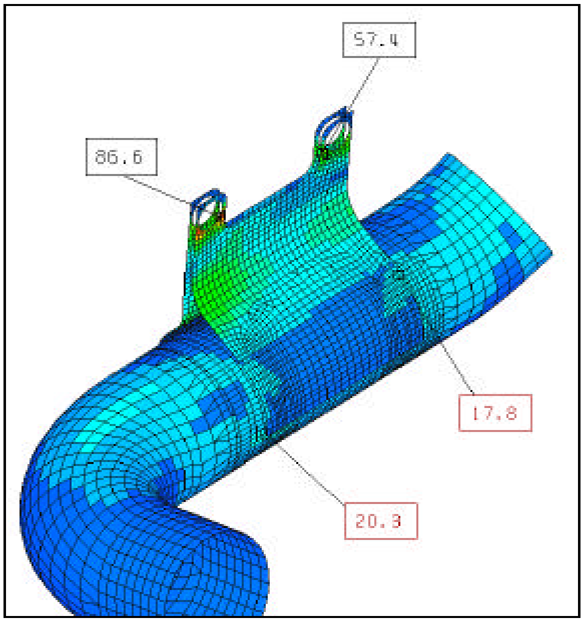


Figure 10 – Primary exhaust tube and its modified bracket. Stress distribution for base harmonic excitation analysis.

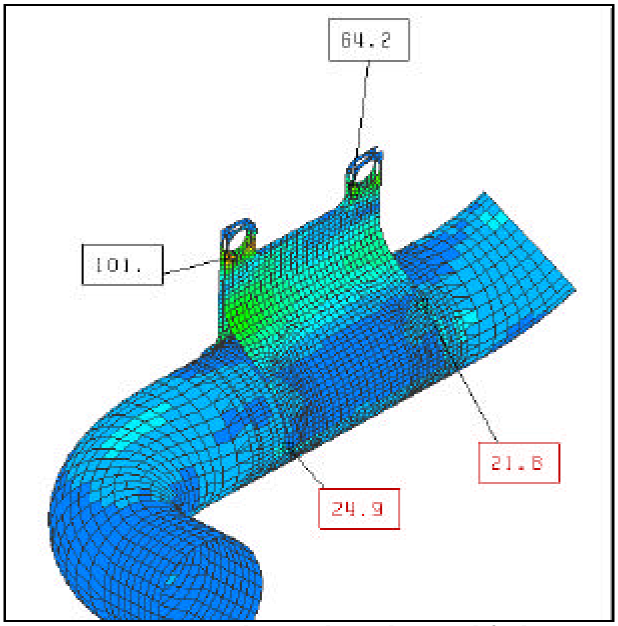


Figure 11 – Primary exhaust tube and its modified bracket. Stress distribution for maximum force in the flexible tube analysis.

Powertrain translation

Verifying the results for the maximum force (static analysis) in the flexible tube, the critical stress value was 24.9 N/mm^2 (figure 11), which is also situated below the material limit adopted for welded regions. The other exhaust system parts didn't present stresses above the material limits, as well as in the original version, in all analysis.

DISCUSSIONS

From the natural frequencies and modes analysis, it can be noted good agreement between experimental results and simulation, assuring satisfactory calibration of the model. Furthermore, the precision of the numerical model is helpful to guide experimental tests, regarding measurement instruments position. The tests were intended to get specially the 5^{th} natural mode, that excites the system in the longitudinal direction, making the flexible tube work axially. In this direction the flexible tube resists until the 600 N force limit. The other natural modes were forcing the flexible tube to act transversally, and then the movements were not transmitted to the rest of primary tube, where the failure occurred. Based on these behaviors, only the 5^{th} mode harmonic analysis was performed. The 3^{rd} mode verified in the numerical model was not characterized in the experimental test because the measurement instruments were disposed in unfavorable position to get a roll motion of the muffler.

By analyzing the results for the original version, we can see that the stresses are above limits for all three analyses in the welded region within the primary tube and the fixing stripes, justifying the failure occurred in track test. It was also verified that, in the original version, the stripes were generating a bending moment at the welded joints, while in the modified version, this undesired solicitation was eliminated in the double bracket configuration. This way, the stress level at the welded region could decrease to acceptable values, as shown in figures 10 and 11. Since the harmonic excitation and the static analyses have reveled more critical situation than the track test analysis, and the dynamic behavior of the two versions are very similar, it can be considered that the optimized version is properly designed for this application.

CONCLUSIONS

From the analyses performed, it was possible to identify the failure causes and to find a solution for the problem, reaching the aim of this work. The results obtained assure the structural integrity of the modified exhaust system when implemented on the vehicle. This study also contributes to a better understanding of this system behavior and its structural strength, for future projects application. Also, all the tests made to improve the primary tube bracket, using a traditional method, would take around 120 days per version to be designed, produced and tested, and around an year to find a satisfactory configuration. Using the Finite Element Method, it is necessary about 45 days per version to have the results, and around 115 days to find a solution, also decreasing the expenses associated to prototypes development.

In numerical simulation, experimental input parameters are simply applied and model properties can be easily modified giving accurate results and leading to a fast solution. All in all, it was verified that numerical models validated with experimental data are a powerful tool during the development phases of a new vehicle, reducing project time and costs.

REFERENCES

[1] SHIGLEY, J. E., “Mechanical Engineering Design”, McGraw-Hill, 1989.
[2] NIEMANN, G. “Elementos de Máquinas”, Volume 1, Editora Edgard Blücher Ltda, 1971.
[3] JUVINALL, R. C., MARSHEK, K. M. “Fundamentals of Machine Component Design”, John Wiley & Sons, 1991.
[4] CLOUGH, R. W., PENZIEN, J “Dynamics of Structures” McGraw-Hill, 1982.
[5] MICHAEL, R., MILLER, M. “Quick Reference Guide”, version 68, MacNeal-Schwendler Corporation, 1994.