23rd ABCM International Congress of Mechanical Engineering
December 6-11, 2015, Rio de Janeiro, RJ, Brazil

# NUMERICAL AND EXPERIMENTAL MODAL ANALYSIS IN AN OFF-ROAD COMPETITION VEHICLE CHASSIS

**Petterson Willians Rocha Abel, petterson.abel@ufv.br**
**Geice Paula Villibor, geicelin@yahoo.com.br**
**Rodrigo Kotinda Tanigava, rodrigo.tanigava@ufv.br**
**Bruno César de Carvalho Tavares, bruno.tavares@ufv.br**
**Joseph Kalil Khoury Junior, joseph.kalil@ufv.br**
Universidade Federal de Viçosa – Campus Universitário – CEP 36570-900 – Viçosa – Minas Gerais

***Abstract.*** *Knowing the natural frequencies and the chassis vibration modes is critical to design a more comfortable and safe vehicle. Thus, we conducted an analytical modal analysis by the Finite Element Method (FEM) and experimental in the prototype chassis developed by UFVbaja team. The experimental analysis was aided by equipment such as impact hammer, uniaxial accelerometers and a data acquisition system. Considering the free-free boundary condition, the first five natural frequencies were obtained with differences ranging from 4.67 to 22.22% between the simulation and experiment. Nevertheless, the Finite Element Method allowed validates a simplified model and have satisfactory result for a complex structure such as the chassis UFVbaja prototype.*

***Keywords:*** *Finite Element Method, Natural frequencies, free-free condition, Baja*.

## 1. INTRODUCTION

During the competition organized by the Society of Automotive Engineers (SAE), the prototype designed and built by students is subjected to different tests of which we can cite the ones involving comfort and endurance strength. Among all the requirements for a comfortable vehicle, the transmission of vibrations to the pilot, from the obstacles, should be minimal. Furthermore, the endurance strength, lasting four hours, subjects the driver and the vehicle structure to different dynamic loadings. The determination of the intensity, duration and characteristics of these charges are of paramount importance for the design of the prototype components and to prevent possible failures in components. The hypothesis that precedes the work involves the determination of the natural frequencies of the vehicle cage is possible to design a more comfortable and safe vehicle. In the analysis it will be possible to determine the natural frequencies and vibration levels incident on the structure, and comparing the studied methods, arrive at a possible solution modeling. Thus, the present work proposes a modal analysis on the off-road vehicle structure using numerical finite element method and experimental method to improve of the prototype design.

## 2. LITERATURE REVIEW

Modal analysis is a very effective procedure to determine the dynamic behavior of a structure and this technique has been used in structural and mechanical area, both from the point of view of theoretical modal analysis and experimental modal analysis.

### 2.1 Modal Analysis

The modal analysis is a powerful tool for understand the vibrational structure, thus allow better design components minimizing vibration problems and failures, improve comfort vehicle (Borges, 2006).

The modal analysis is based on three basic assumptions:

- Linearity of the dynamic behavior - the answer of a combination of forces applied simultaneously in the structure must be equal to the sum of the responses of each individual force;
- Invariable in time - constant physical parameters;
- Observable - the dynamic behavior is determined by the input / output relationship.

According to Ewins (2000), modal analysis can be performed by two processes. The first is known as a theoretical modal analysis which consists of mathematical modeling of a study by discretization technique called Finite Element Method. The second process is the experimental modal analysis that seeks to validate the theoretical model.

### 2.2 Modal Analysis Theory for Finite Element

The Finite Element Method has been used in the field of structural analysis. It is a numerical method that seeks an approximate solution of complex problems, which are difficult to obtain an analytical solution (Liu, 2003). The finite element formulation of the continuous elastic body is represented by a discrete system, formed by a set of structural elements attached to each other through a finite number of points called nodes of the model. This application of modal analysis will determine the natural frequencies and mode shapes of the model.

### 2.3 Experimental Modal Analysis

Experimental modal analysis serves to verify or validate the results obtained analytically or numerically. The dynamic characteristics of the model are obtained from the transfer function that defines a direct relationship of input / output system. This relationship is obtained by measuring the excitement and model response (Soeiro, 2013). The most common method of experimental modal analysis is to excite the structure and measure the excitation force signals and the response signals in the considered points. The excitement and response signals are sent to a signal analyzer that estimates Response Functions Frequency (FRFs) between measured points. Using the peak identification method, the modal parameters of the system are identified by the peaks present in the chart of FRFs. The main mechanisms used for experimental modal analysis are the following:

- Approach to the problem by the theory of modal analysis;
- Choice of experimental modal analysis method;
- Choose the way of acquiring modal data;
- Estimation of modal parameters;
- Validation of modal data.

## 3. OBJECTIVE

Realize two methods of modal analysis in Baja vehicle structure, finite element and experimental model to identify the natural frequencies and comparison the two methods.

## 4. MATERIAL AND METHODS

### 4.1 Used Equipment in the Vibration Test

The information about the vibrations are captured by the data acquisition system composed of an impact hammer, two uniaxial accelerometers, the data acquisition module from National Instruments®, next to proprietary program Labview®, a hack and a computer.

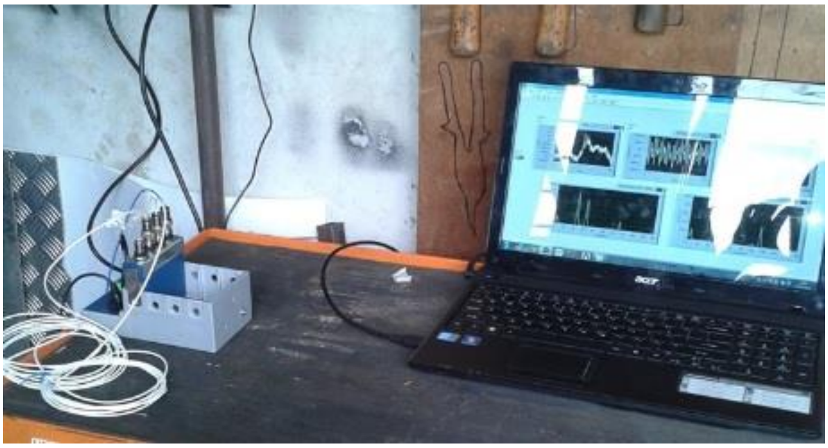

Figure1. Data acquisition system

### 4.2 Preliminary analysis

First it selected a simple geometrical model structure to understand the application of modal analysis. The structure is made of AISI 1020 steel and has dimensions of 0.54 m width, 1.05 m long and 1.34 m high. The cross section area of the square profile is 40x40 mm with 2.5 mm wall thickness and overall mass of the structure is 32.82 kg. The natural frequencies and mode shapes of the model were determined by ANSYS Workbench® software. The structure was imported Solidworks® as a single geometry with no partitions to prevent any part of the structure itself vibrates during analysis. The boundary condition was used to free-free condition where there is no restriction on translational and rotational movements. The five the first natural frequencies were determined along with their respective modes of vibration.

The excitation points, the impact hammer, and monitoring, by uniaxial accelerometers, previously established based on the vibration intensity of the computational model. Seeking no coincidence with us, sites were avoided next welded joints because of the disproportionate cushioning. To describe the dynamic behavior of the structure were obtained 60 data analysis in six monitoring points. The location of accelerometers prioritized the base of the structure, which showed higher levels of vibration. Figure 2 shows the simplified structure with the identified excitation points and vibration levels indicated in numerical analysis.

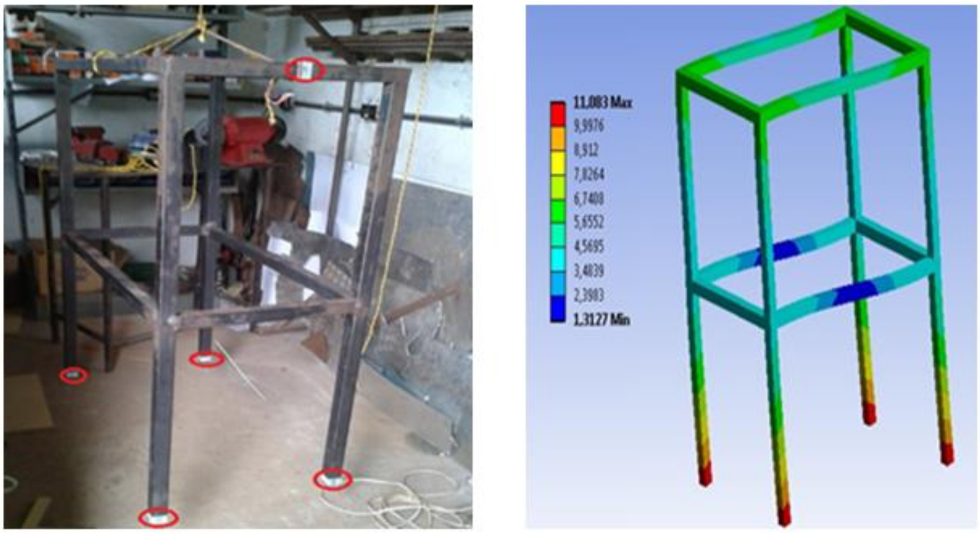


Figure 2. Location of points for excitation of impact hammer and vibration levels indicated in the simulation

The experimental modal analysis was defined by a direct relation of the system input and output. By exciting the impact hammer, the model's response was captured by two uniaxial accelerometers. To perform the experiment in the same free-free status of the simulation, the structure was suspended massless string. The assembly was made so that no external factors interfere with the test. Accelerometers were fixed in the structure rigidly and the impulsive excitement caused by the impact hammer, was made in the same direction for measuring accelerometers.

The model response was monitored by two accelerometers, featuring the method input / double-output. The sampling rates for data collection were 1000 Hz, defined based on the Nyquist theorem, which establishes a sampling rate of at least two times higher than the frequency being studied for elimination of problems with aliasing. The captured acceleration data were subjected to a Fast Fourier Transform (FFT) through LabView® computer program. This allowed transform the time domain data in the frequency domain data, thus obtaining the Frequency Response Function (FRF). This function was possible to determine the natural frequencies through the peaks found in the FRF chart. The Labview® program yielded data on acceleration and the frequency of the two accelerometers, in addition to the excitation force of the impact hammer.

Table 1 is showing the comparison between the obtained natural frequencies of the numerical model and experimental model of simplified structure.

Table1. Comparison of natural frequencies obtained from the numerical model and the experimental simplified structure

| Natural Frequency (Hz) | Numerical Model ($\omega_A$) | Experimental Model ($\omega_E$) | Variation $|(\omega A - \omega E)|/\omega E$ (%) |
|---|---|---|---|
| 1ª | 36.25 | 36.33 | 0.22 |
| 2ª | 55.56 | 54.50 | 1.94 |
| 3ª | 60.74 | 57.81 | 5.06 |
| 4ª | 71.11 | 71.02 | 0.13 |
| 5ª | 76.94 | 77.63 | 0.89 |

The results show the identification of the model from natural frequencies of the input and system output relationships. The results showed that the biggest difference between the numerical and experimental model was only 5.06%, which shows the good relationship between the two methods used. The difference may have been caused by disregard of welds together as masses concentrated in the numerical model. Nevertheless, the proximity of the natural frequencies allowed validate the model structure and increased confidence to perform modal analysis in a more complex structure such as the chassis of the UFVbaja prototype.

### 4.3 Modal Analysis Theoretical Cage

The cage UFVbaja vehicle is made up of steel tubes and profiles 1020, connected by metal welds. The main steel tube used has 31.75 mm outside diameter and 1.60 mm wall thickness and the total mass of the chassis was of 35.50 kilograms. The analytical model of the vehicle cage was made using the Finite Element Method because it is a complex geometry. The cage was designed using SolidWorks®. The Figure 2 is presenting the chassis designed the software.

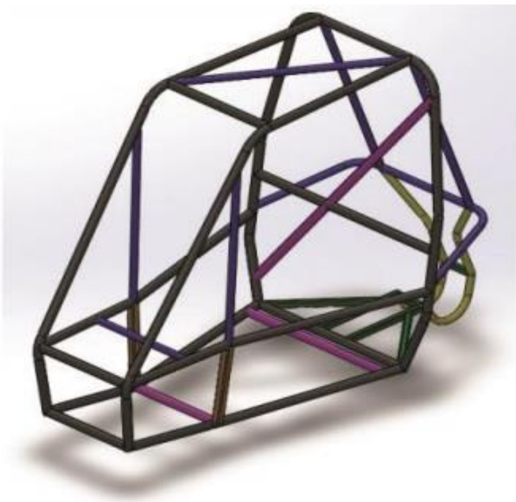

Figure 2. Cage designed using SolidWorks®

Modeling was performed on the computer program ANSYS Multiphisics® and mechanical properties of the material used being shown in Tab. 2.

Table 2. Mechanical properties used in modeling
(*Fonte: http://www.matweb.com*)

| Mechanical properties of Steel AISI 1020 | |
|---|---|
| Density (kg/m³) | 7850 |
| Modulus of Elasticity (GPa) | 200 |
| Poisson’s Ratio | 0.30 |

The mesh was generated using type *PIPE 16* elements, which is a uniaxial tube element capable of traction-compression loads, torsion and bending and has six degrees of freedom in their two nodes. This type of element has symmetrical behavior and the input geometric data required for modeling are the same as in the prototype, with the external diameter of 31.75 mm and a wall thickness of 1.60 mm. The definition of the size of the mesh element was based on susceptibility testing in which it assessed the variation of the results of the first natural frequency and processing time depending on the degree of refinement. As the element size is reduced, the amount of interaction of calculations increases proportionally with the increase of number of elements and nodes requiring greater computational processing time. So it is always necessary to have a limited number of elements. Table 3 shows that from the 0.01 mm equal element size there is no significant change in the result, indicating the convergence of the mesh. This means that the accuracy achieved is the best possible and the generated mesh resulting in 2772 nodes and 2803 elements.

Table3. Mesh sensitivity test

| Size element (mm) | First natural frequency (Hz) |
|---|---|
| 0.5 | 53.731 |
| 0.4 | 53.721 |
| 0.2 | 53.618 |
| 0.1 | 53.414 |
| 0.05 | 53.355 |
| 0.01 | **53.334** |
| 0.005 | 53.333 |
| 0.001 | 53.333 |

The elements of welds in the pipe fittings, holders of other subsystems soldiers were not considered as concentrated mass elements. For simulation, the boundary condition used was free-free condition in which there is no restriction on movement.

### 4.4 Determination of the Cage Measuring Points

The excitation points, the impact hammer, and monitoring, by uniaxial accelerometers, previously established based on the vibration intensity of the computational model. Some points had no way to get away from the welded joints due to the large amount of solder points presented by the cage. All measuring and excitation points are strategically positioned so that they could describe the dynamic behavior of the structure. In total twelve points were selected for monitoring and in accordance with the vibration intensities indicated by computer simulations, the points focused on the front, rear and top of the chassis. Each natural frequency has its respective modal form. Figure 3 is showing the vibration intensities indicated in computer simulations for each mode shape.

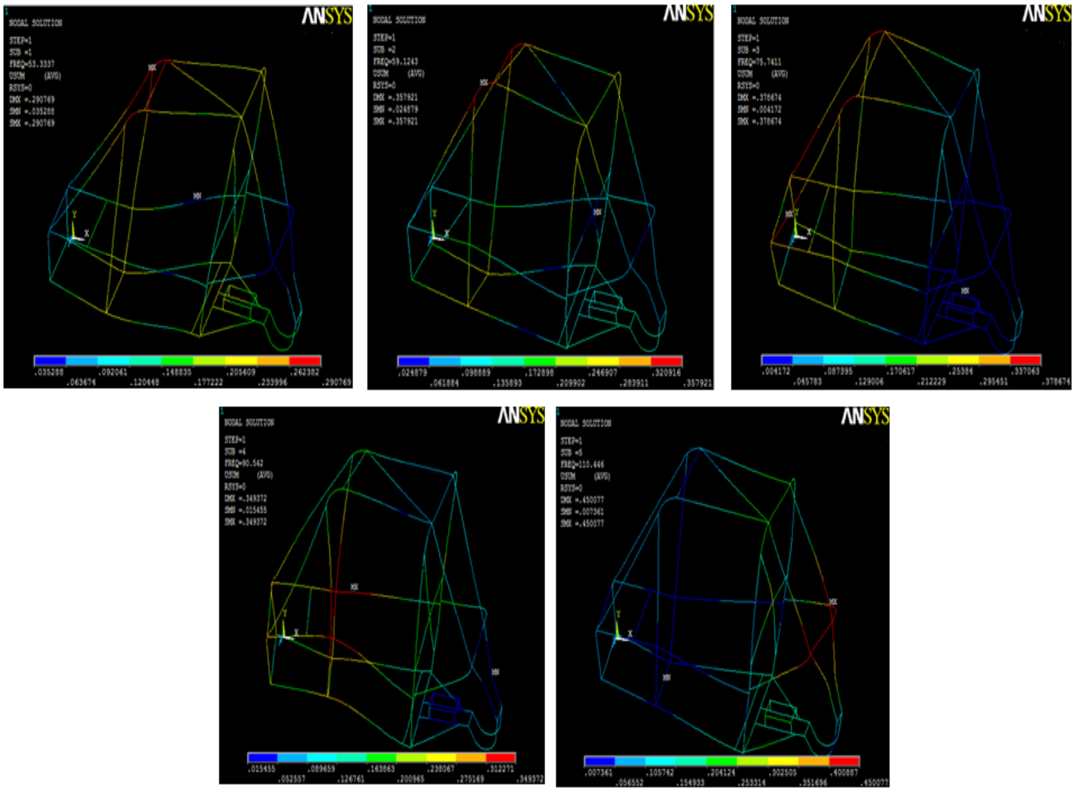

Figure3. Vibration intensity indicated by the computer simulation for the first five mode shapes

### 4.5 Experimental Modal Analysis of the Cage

In this experiment the support condition used will be free-free condition for this, the structure was suspended using flexible cords. The support condition must be well defined, for this assembly may negatively influence the identification of the dynamic properties of the structure. The structure was suspended a few feet off the ground and in a balanced manner to carry out the experimental test. Accelerometers were also fixed in the structure rigidly and the impulsive excitement caused by the impact hammer, was made in the same direction for measuring accelerometers. The Figure 4 is showing the suspension of the cage with the use of flexible cords and the excitement conducted by impact hammer in the same direction accelerometer marked in red.

Figure4. Test with experimental thriller cage for flexible cords

The Cage response was monitored by the two accelerometers by the method input / double-output and the sample rate used was 1000 Hz. We chose to study the first 5 natural ways of the cage because they are close to the speed range engine and in real conditions the modes of lower frequencies are usually the most important. The frequency range used in these experimental tests was 0-200 Hz, including the first five vibration modes. The natural frequencies were obtained through the peaks found in Response Function Frequency (FRF) graph.

## 5. RESULTS AND DISCUSSION

Through Finite Element Method, the first five natural frequencies for free-free status were determined for cage. Intended to validate the mathematical models, experimental tests were conducted on the same boundary conditions used in the computer simulations. Table 4 is showing the comparison between the obtained natural frequencies of the numerical model and the experimental design of the cage

Table4. Comparison of natural frequencies obtained by numerical and experimental model of the cage of the vehicle off-road

| Natural Frequency (Hz) | Numerical Model ($\omega_A$) | Experimental Model ($\omega_E$) | Variation $\lvert(\omega A - \omega E)\rvert/\omega E$ (%) |
|---|---|---|---|
| 1ª | 53.33 | 46.25 | 13.29 |
| 2ª | 59.12 | 64.41 | 8.95 |
| 3ª | 75.74 | 79.28 | 4.67 |
| 4ª | 90.54 | 110.66 | 22.22 |
| 5ª | 110.45 | 132.13 | 19.63 |

The results showed a difference between the numerical and experimental models. The maximum error observed in identifying natural frequencies shows the difficulty in analyzing the dynamic behavior of a complex structure. However, it is important to note that both the second as the third natural frequency similar values between the numerical and experimental models.

The difference between the values found in the numerical and experimental analysis can be credited to the simplification of computer modeling. It notes that the welds were not considered and during the test, the cage of the vehicle had the anti-flame cockpit wall and welded components such as support of management and the tank that may have influenced the result.

The results also allowed the observation that the first two natural frequencies, 53.33 and 59.12 Hz, matching the engine operating range from baja. According to the UFVbaja team, the engine speed range is 1700 rpm (28.33 Hz) to 4000 rpm (66.66 Hz). The vehicle engine is the primary source of excitation frequency and the natural frequency when the cage coincides with the excitation frequency resonance phenomenon occurs, which generates excessive vibration in the vehicle chassis and contributes to the discomfort of the driver. Thus it is recommended to work with the engine speed of the vehicle outside these frequency bands in order to avoid the phenomenon of resonance.

Other studies also developed used the modal analysis technique in free-free status for the natural frequency of a given structure. Borges (2006) performed modal analysis in baja chassis developed by Unesp – Ilha Solteira. And Renuke (2012) conducted modal analysis on the chassis of a commercial car.

Table 5 is showing a comparison of five the first natural frequency of the numerical model with other work. The chassis of the vehicles off-road type baja were designed and developed in accordance with Regulation BAJA SAE, yet have different constructive and geometrical characteristics. It can be seen that numerical model was studied next natural frequencies of the numerical model developed by Borges (2006).

Table5. Comparison of natural frequencies obtained by the numerical model to other developed works

| Natural Frequency (Hz) | Numerical Model ($\omega_{A1}$) | Numerical Model Borges ($\omega_{A2}$) | Numerical Model Ghriem ($\omega_{A3}$) |
|---|---|---|---|
| 1ª | 53.33 | 64.45 | 42.42 |
| 2ª | 59.12 | 72.37 | 49.26 |
| 3ª | 75.74 | 77.31 | 51.32 |
| 4ª | 90.54 | 84.15 | 66.11 |
| 5ª | 110.45 | 95.76 | 77.04 |

## 6. CONCLUSION

This work presents the application of two methods of modal analysis for two different models. It can be concluded that the Finite Element Method and the method used in experimental tests were effective to validate the simplified model. As for the cage, the results may be regarded as satisfactory, since the first natural frequency had a very low error.

The finite element modeling of the cage still requires research and development in the review of its features to be accurate in the solutions. For the development of the next prototype, the chassis of the vehicle must be dimensioned so that the natural frequencies of the cage are outside the engine speed range. That is, the first natural frequency must be higher than 67 Hz.

## 7. ACKNOWLEDGEMENT

The authors of the work thank the FAPEMIG - Foundation for Research Support of the State of Minas Gerais and the CNPq - National Council of Scientific and Technological Development, Brazil - for the support for accomplishment of this research.

## 8. REFERENCES

Borges, A. S., 2006. “Análise modal baseada apenas na resposta”. Dissertação (Mestrado em Engenharia Mecânica), Universidade Estadual Paulista, Ilha Solteira, SP.

Ewins, D.J.; “Modal Testing: Theory, Practice and Application”. Second Edition, Research Studies Press LTD, 2000.

Liu, G.R.; QUEK, S.S.; “The Finite Element Method: A Practical Course”. Singapura, 2003, p. 1 – 35, 2003.

Renuke, P., 2012. “International Journal of Engineering Research and Applications”. November – December. 2012 <http://www.ijera.com/papers/Vol2_issue6/EK26955959.pdf>.

Soeiro, N. S., 2001. “Análise modal experimental”. Grupo de Vibrações e Acústica – UFPA. 02. Maio. 2013 <http://www.ufpa.br/gva/Apostilas/ANALISE%20EXPERIMENTAL.pdf>.