"HENRI COANDA"
AIR FORCE ACADEMY
ROMANIA

GERMANY

"GENERAL M.R. STEFANIK"
ARMED FORCES ACADEMY
SLOVAK REPUBLIC

INTERNATIONAL CONFERENCE of SCIENTIFIC PAPER
AFASES 2011
Brasov, 26-28 May 2011

APPROACHES CONCERNING THE OPTIMIZING OF SUPPORTING STEEL FRAME OF A WHEEL LOADER ENGINE BONNET

Ionut SERBAN*, Gheorghe BOBESCU*, Cornel ARAMA**

*"Transylvania" University, Brasov, Romania
**"Henri Coandă" Air Force Academy, Brasov, Romania

Abstract: *For developing high-quality and cost-efficient products, it is important to evaluate and compare system level performance for different configurations early in the development process. This paper will present the development and optimization for a steel frame system using a combined CAD/FEM program. In this study, a detailed steel frame model is designed and analysed. The issues to be improved are the vibrations of the outer parts of the bonnet. The negative effect consists in acoustical disturbance, mechanical stress, material fatigue and oscillations. 3D model is analysed through finite elements analysis method and confirm a higher displacement at frame's extremities caused by an unstable dynamic behaviour as an effect of different excitation sources (combustion engine, loading, unloading, general balance of the machine). To demonstrate the applicability of the developed models, results of computer simulations are presented.*

Keywords: *optimization, structures, engine, design, automotive*

1. INTRODUCTION

The bonnet of the engine is a part of the wheel loader car body which is designed to cover the engine and other components placed in the engine compartment. One of the most important functions is to give a good optical impression as much as to protect the under-bonnet area. Noticing the existing technical solution, a very unpleasant effect is stated: the extremities of the bonnet oscillate, causing acoustical disturbance and probably, in the future, material damages. The effect of the vibrations generated by the excitation factors (forced vibrations) are described using a reduced dynamic model. The frame consists of rectangular profiles welding one another. The 3D spatial model (computer modelled) reproduces the real model and it has its characteristics: the density of mass, the geometry, the mechanical and thermal behaviour. After running the analysis procedure of the finite element, the responses of the system will give an image about the amplitudes of different mechanical (vibratory) characteristics. The model is developed using Pro/Engineer® environment. Further, based on the understanding provided by the model, an improved geometry will be presented.

2. THE MODEL

As shown in the Fig. 1, the locations where the oscillations act are indicated by the red arrows. Usually, the engine is placed at the bottom of the vehicles.

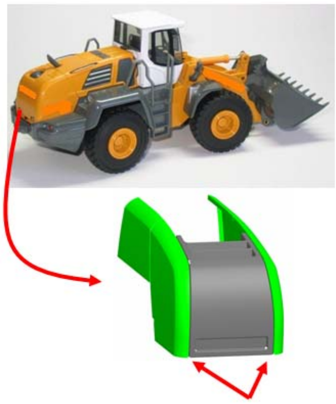

Fig. 1 – Wheel loader engine bonnet

The bonnet is fixed on the supporting steel frame using special bolts which are laminated into the glass fibre reinforced plastics. The conceptual design of the original model of this steel frame looks like in the Fig. 2.

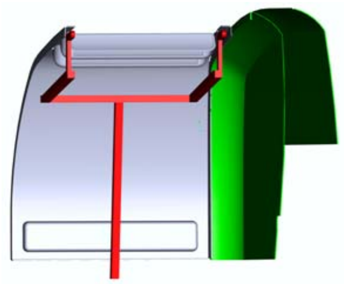

Fig 2 – Original version of steel frame

Further, the modelled assembly looks as in the Fig. 3. For the analysis, three surfaces of the frame will be considered as being fixed.

Fig. 3 – Final design of the steel frame's original version

As it can be observed, the corners of the steel frame are free and they build a lever arm with an increased vibrational behaviour. The reduced 3D model will be prepared for the analysis. As in Fig. 4, a modal analysis will be carried out in order to investigate the existing vibratory system.

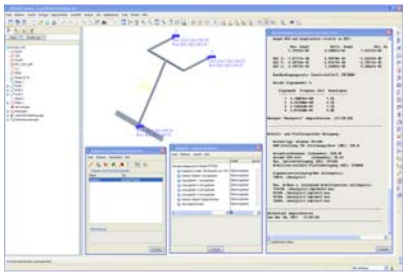

Fig 4 – Modal analysis of the existing frame

3. THEORETICAL ISSUES

The majority of structures can be made to resonate, i.e. to vibrate with excessive oscillatory motion. Resonant vibration is mainly caused by an interaction between the inertial and elastic properties of the materials within a structure. Resonance is often the cause of, or at least, a contributing factor to many of the vibration and noise related problems that occur in structures and operating machinery. To better understand any structural vibration problem, the resonant frequencies of a structure need to be identified and quantified.

In every development of a new or improved mechanical product, structural dynamics testing on product prototypes is used to assess its real dynamic behaviour.

Modes are inherent properties of a structure, and are determined by the material properties (mass, damping, and stiffness), and boundary conditions of the structure. Each mode is defined by a natural (modal or resonant) frequency, modal damping, and a mode shape.

A single-degree-of-freedom system is described by the following equation:

$$m\ddot{x}(t) + c\dot{x}(t) + kx(t) = f(t) \qquad (1)$$

with m the mass, c the damping coefficient, and k the stiffness.

Although very few practical structures could realistically be modeled by a single-degree-of-freedom system, the properties of such a system are important because those of a more complex multiple-degree-of-freedom system can always be represented as the linear superposition of a number of single-degree-of-freedom system characteristics (when the system is linear time-invariant).

Multiple-degree-of-freedom systems are described by the following equation:

$$\mathrm{M}\,\ddot{x}\,(t) + \mathrm{C}\,\dot{x}\,(t) + \mathrm{K}\,x\,\,(t) = \mathrm{f}\,(t) \qquad (2)$$

It is not the intent of this paper to deepen the theory of vibration but only to find out for the given system of how is his behaviour after applying a natural frequency analyses. After that, optimization approaches will deliver the new shape of the frames.

Due to the diversification of structural optimization problems, most structural optimization problems can be classified as size, shape and topology optimization. In this case, the main application of optimal design of steel structures is the shape optimization, in order to get a better vibrational stability.

The aim of the structural optimization is to determine the value of the design variable x that minimizes the objective function V(x). The objective function is accompanied by a various number of constraints (here g(x)).

min V(x) with:

$$x_1 \le x \le x_n$$

and

$$g(x) \le 0.$$

Generally, the optimization model consists of the objective and constraint functions and a numerical optimization algorithm that drives the optimization.

4. SIMULATION AND OPTIMIZATION

The existing shape solution of the steel frame is modelled with Pro/ENGINEER®. Material characteristics are assigned to the 3D model. All the unnecessary design details are removed from the model. After that, the FEM software Pro/Mechanica® is launched. In case of simple structures, ordinary calculation can be done manually by the specialised engineers. If the structure comprised a complicated form, the simplification procedure will conduct to a quite inexact model. In this approach the software was used because of the complicated spatially model in order to get a better approximation.

After modelling, the 3D model (existing version) gets a mesh structure. This mesh will be automatically generated by the software, as in Fig. 5:

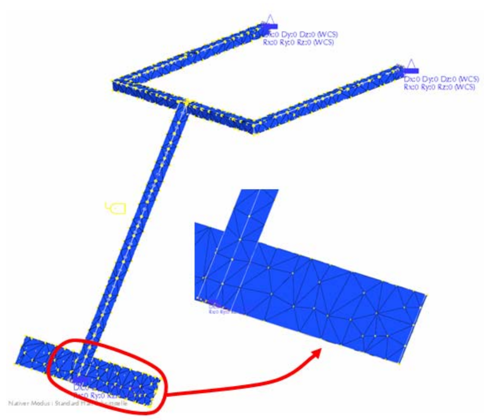

Fig. 5 – Mesh 3D model of the existing version

A modal analysis study is carried out. The window of the results is presented in the Fig. 6. The first and second vibration mode / frequencies (16.59 Hz and 20.27 Hz) are representative for the searched problem.

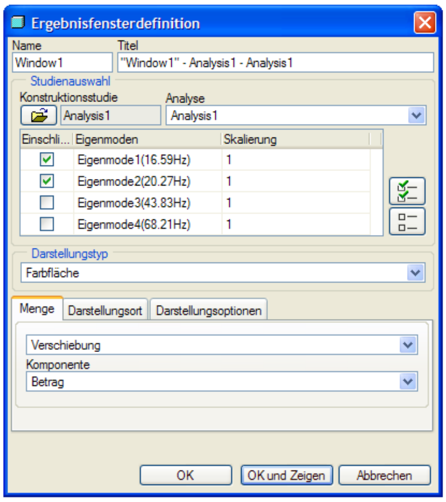

Fig. 6 – The window of the results with first and second vibration modes

The results of these vibratory movements will be exemplified in Fig. 7. Here, the extremities move much more as the rest of the frame.

As we can observe the "red" extremities, maximum displacement is here approx. 1.0009 x 10^{-1} mm.

The optimized shape is a result of the optimized algorithm and a few practical restrictions taking into account the placement of a few new components in the under-bonnet area. The meshed shape is presented in Fig. 8.

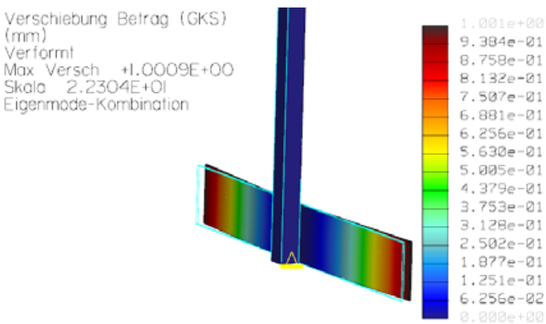

Fig. 7 – Results of modal analyse for the existing model

Fig. 8 – Mesh 3D model of the optimized version

As in the Fig. 9, a new modal analyse is running with Pro/Mechanica® and the results are shown in Fig. 10. Under the first vibratory node and a natural frequency of about 43.44 Hz, the behaviour of the structure in the studied region is much better than the original one. It can also be noticed that the value of the first natural frequencies is about 2.5 times higher as compared to the original version. The maximum displacement is now about 6.25 x 10^{-2} mm, which means that the level of the vibrations and noises is on the lowest level.

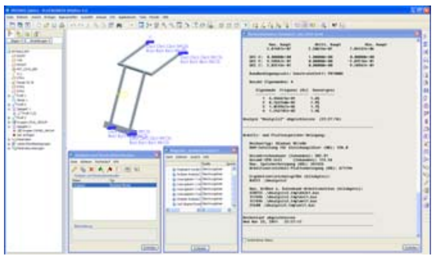

Fig. 9 – Modal analyses of the optimized frame

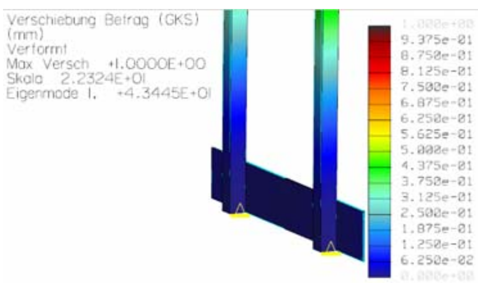

Fig. 10 – Results of modal analyse for the optimized model

5. CONCLUSIONS

Using the FEM and modal analyse, together with a shape optimization algorithm, a new shape can be designed, analysed and optimized. This application can also find applicability in the military field, for example different components of the military vehicles such as the armour plate, the vehicle body and so on, could receive a better design and a low level of noises.

BIBLIOGRAPHY

[1] Bobescu, G. s.a. (2000) *Motoare pentru automobile si tractoare*, Vol. II, Editura "Tehnica", Chişinău;

[2] Schumacher, A. (2005) *Optimierung mechanischer Strukturen - Grundlagen und industrielle Anwendungen*, Springer Verlag, Berlin;

[3] Serban, I. s.a. (2010) *Considerations about optimized cad design application for the automotive and engine's mechanical structures*, CONAT Paper, Transilvania University of Brasov.