Static and Transient Analysis of Radial Tires Using ANSYS

TIBERIU GIURGIU[1], FLORINA CIORTAN[1], CRISTINA PUPAZA[2]
[1]Test, Evaluation and Scientific Research Weapons Systems Centre,
Military Equipment and Technologies Research Agency
16 Aeroportului Street, Clinceni, Ilfov, RO-077060
[2]Machines and Manufacturing Systems Department
University POLITEHNICA of Bucharest
313 Splaiul Independentei Avenue, Bucharest 6, RO-060042
ROMANIA
tgiurgiu@acttm.ro, fciortan@acttm.ro, cristinapupaza@yahoo.co.uk

Abstract: - The paper deals with modeling and simulation of the static and dynamic behavior of radial tires for civil emergency vehicles or military armored vehicles. The tire is a complex composite structure which consists of rubber, textile-cords and steel-cords. For the computational model knowledge regarding the macrostructure and microstructure of the tire, as well as experimental data is required. The Finite Element Method and ANSYS software were used to obtain the static and transient dynamic behavior of the models. The simulation results were compared with the imprint of the tire on the road surface.

Key-Words: - Tires, Rubber, Modeling, FEM, Simulation, Static, Transient, Experiments

1 Introduction

The main characteristics of emergency and military armored vehicles are: mobility, safety and availability.

Simulation procedures combined with experiments on contact tire-surface interaction enable the designer to improve both the construction of the tire and the control system, taking into account the wheel dynamics. Important problems to which structural analysis can give solutions are: tire inflation, the behavior of the tire when passing obstacles, the tire-ground contact pressure, tire behavior when crossing a trench and so on.

Most tire simulations with FEM were static analysis, because tire is one of most complex structures. A non-linear static and transient FEA analysis of a tire model was performed [1], simulating the radial and lateral static stiffness test conditions, dynamic free-drop test conditions and the rolling cornering stiffness, but the analysis didn't focused on the bed-rim interaction. Characteristics of the tire analysis by means of FEM codes were described in [2], as well. Using the implicit formulation, a steady-state cornering simulation was performed, requiring a fine mesh only in the contact region because of the formulation by moving reference frame technique.

The present research is focused on modeling and simulation of a special type of tire, used for emergency or military vehicles. An existing wheel configuration is analyzed in order to find improved design solutions. The wheel is designed not only to assure the mobility of the vehicle, but also to withstand to high stress levels during the vehicle's movement.

A solution for replacing the old tires is to reconfigure the existing rims, so that a run flat technology can be used [3]. The aim is to increase the mobility and the safety of the vehicles. This process involves preliminary simulation attempts, experiments and testing procedures for homologation.

2 Tire 3D Model

A pneumatic tire is a flexible structure of the shape of a toroid, filled with compressed air. The most important structural element of the tire is the housing. It is made up of flexible cord layers with high modulus of elasticity, encased in a matrix of low modulus rubber compounds. The cords are made of fabrics of natural, synthetic, or metallic composition, and are anchored around beads made of high tensile strength steel wires. The beads serve as a support for the housing and provide adequate seating of the tire on the rim (Fig. 1). The ingredients of the rubber compounds are selected to provide the tire specific properties.

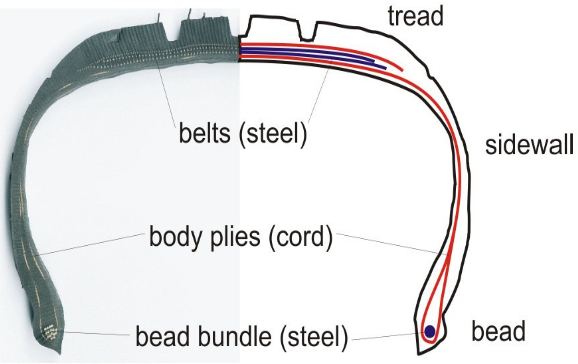

Fig. 1. Radial structure of the tire

Figure 2 and 3 shows three complex models, which were realized in CATIA v.5 using an emergency vehicle's tire and a military one [4]. The road surface was considered as a square block, in contact with the tire.

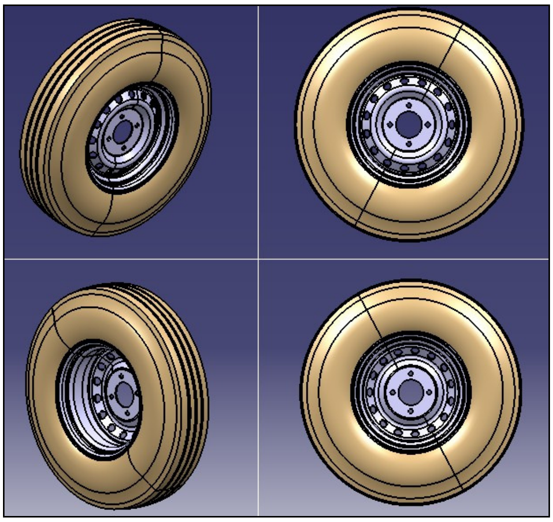

Fig. 2 CAD models of the tires

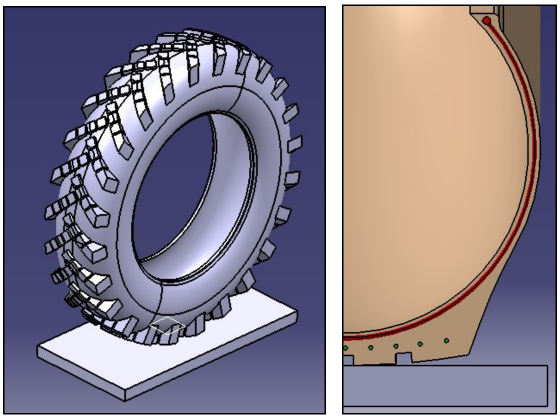

Fig. 3 CAD model. Detail

3 Hyperelastic material model

Hyperelasticity refers to materials whose stresses are derived from their total strains using a strain energy density function [5]. All straining is reversible and no permanent deformation occurs. Vulcanized rubber falls into this category and can generally be considered to be isotropic, nearly incompressible and strain rate independent. Many hyperelastic material models are actually available in advanced solvers. From these models the Odgen material model [6] was used to describe the non-linear strain behavior of the tire.

The Odgen material model assumes that the material behavior can be described by means of a strain energy density function, from which the stress-strain relationship can be derived.

The Ogden form of strain-energy potential W has the form [5]:

$$W = \sum_{i=1}^{N} \frac{\mu_i}{\alpha_i}\left(\bar{\lambda}_1^{\alpha_i} + \bar{\lambda}_2^{\alpha_i} + \bar{\lambda}_3^{\alpha_i} - 3\right) + \sum_{k=1}^{N} \frac{1}{d_k}(J-1)^k \quad (1)$$

where N is a constant, μ_i, α_i and d_k are material constants. J is the ratio of the deformed elastic volume over the reference (undeformed) volume of the material.

The Ogden material model usually provides the best approximation to a solution at larger strain levels. The applicable strain level can be up to 700 percent.

A higher N value can provide better fit the exact solution, however, it may cause numerical difficulty in fitting the material constants and also it requests to have enough data to cover the entire range of interest of the deformation. A value of N>3 is usually not recommended. Therefore N=3 was chosen.

The initial shear modulus, μ, is given as [5]:

$$\mu = \frac{1}{2}\sum_{i=1}^{N} \alpha_i \mu_i \quad (2)$$

The initial bulk modulus is:

$$k = \frac{2}{d_1} \quad (3)$$

4 Static analysis using ANSYS for emergency vehicle tires

Because the tire's geometry and structure is complex, the first step was to build a simple model, without any ribs or grooves, to import the model in the solver and to tune the computational parameters with the materials and the simulation environment.

A preliminary static analysis was performed, considering only the inflation *pressure*, the *displacement* or a *force* applied to the square block that represents the road surface, and a *fixed support* for the tire surface bonded to the rim. Due to the nonlinearity of the analysis, only a small sector of the tire was initially used.

The analysis took advantage of the two symmetry planes of the wheel, saving computing time. In order to determine the optimum computational parameters, a first homogeneous model of the tire was used, without any steel insertion and with a smooth tread surface, without ribs and grooves. Two rubber-type materials, available in ANSYS material library were used for the tire, and Structural Steel for the road surface. A comparison of the two models regarding the total deformation at 2 mm and 5 mm vertical displacement of the square block can be seen in Figure 4 and 5.

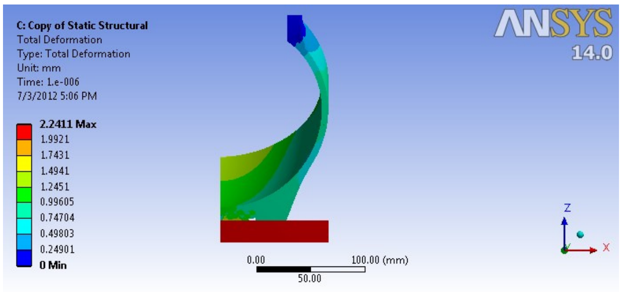

Fig. 4. Total deformation at 2 mm displacement

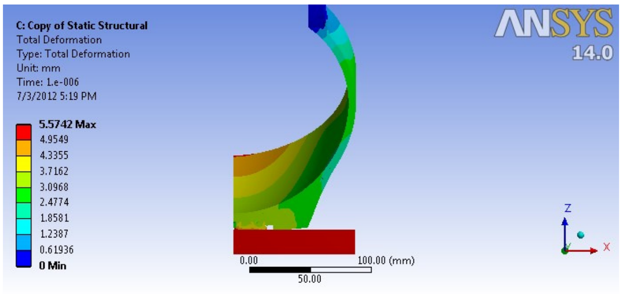

Fig. 5. Total deformation at 5 mm displacement

The next step was to get as close as possible to the real tire, so more complex models were realized in the CAD system, with steel-cords and beads, in different configurations. Because of the metallic insertions, additional conditions and parameters, such as frictional coefficients were introduced, as presented in the table below.

Table 1. Frictional coefficient values

Material 1	Material 2	Frictional coefficient	
		static	dynamic
Rubber	Asphalt		0.5 – 0.8
Steel	Steel	0.78	
Steel	Rubber	0.1 – 0.2	

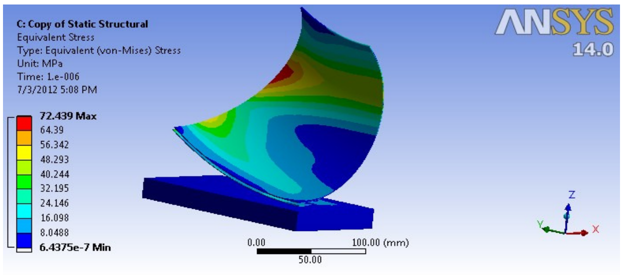

Fig. 6. The equivalent stress at 2 mm displacement

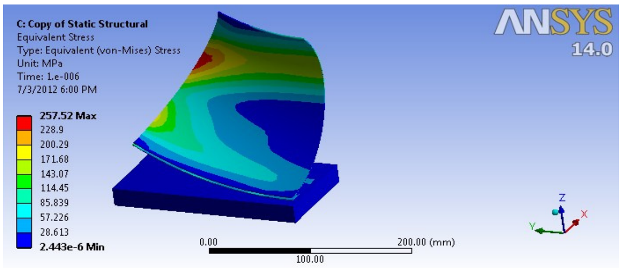

Fig. 7. The equivalent stress at 5 mm displacement

Figures 6 and 7 show the Equivalent Stress in the metallic layers, representing the steel-cords of the real tire, at 2mm and 5mm vertical displacement of the square block which represents the ground.

5 Transient analysis using ANSYS for emergency vehicles tires

The next stage of the simulation was a transient structural analysis. At this stage a shock loading was considered, simulating the pass over a 20 mm obstacle on the road surface [5], [6]. The loads and boundary conditions are mentioned in the Table 2.

Table 2. Transient Structural analysis parameters

Parameters	Values	Remarks
Gravitational acceleration	-9806.6 mm/s²	–Z direction
Fixed support	-	The surface in contact with the rim
Pressure	0.25 MPa	Equivalent to the real pressure in a tire
Frictionless support	-	Constraints imposed to the square block; displacement on Z axis
Frictionless support	-	
Initial displacement	20 mm	Z direction

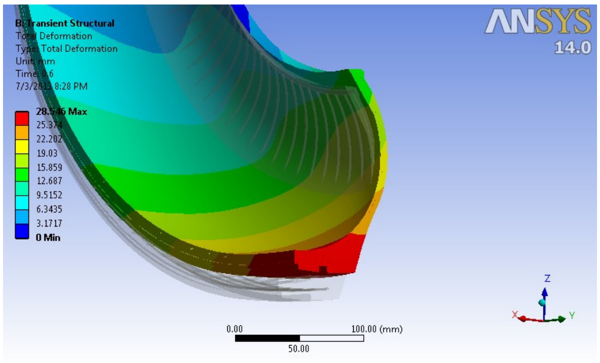

Fig. 8. Total deformation of the tire at 20mm

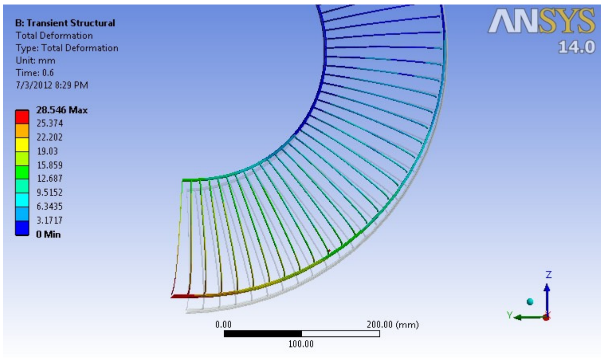

Fig. 9 Total deformation of the steel-cords at 20mm

In this case a fine mesh was generated, containing 270192 nodes and 151947 elements. Structural deformations are processed in Figures 8 and 9.

6 Simulation results for the entire model for an emergency vehicle tire

The tires on military armored vehicles have a more complex configuration than the civil ones. The complexity is required by the specific missions of this type of vehicles and the intense stress subjected by the tire during the movement on different types of terrain.

The meshed model used for this simulation contains 144320 nodes and 144320 elements, and was generated in ANSYS preprocessing system.

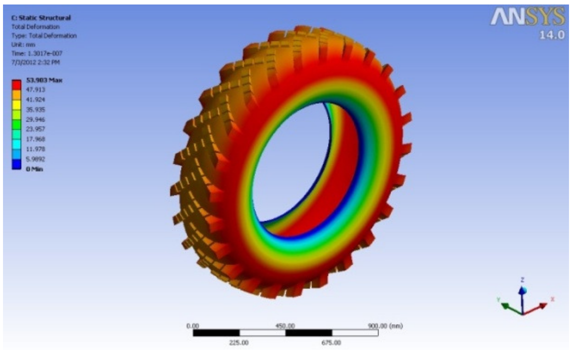

Fig. 10. Total deformation during tire inflation

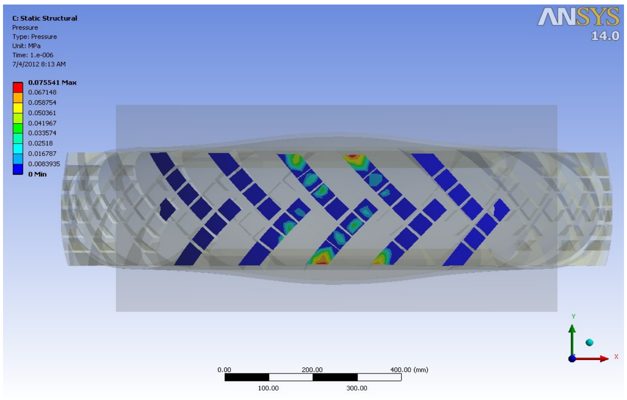

Fig. 11. Contact pressure on the ground

Figures 10 and 11 show the total deformation during tire inflation. Another problem that has to be considered for military vehicles is represented by the ground contact pressure (Fig. 11). This parameter is very important, as the vehicles have to cross different types of soil: mud, sand, snow, etc. A lower contact pressure on the ground provides better performances regarding vehicles mobility in all-terrain.

7 Conclusion

The quality of the transient analysis results were compared with experimental data.

Figure 12 shows the imprint on the ground of a military tire evaluated using ANSYS and Figure 13 presents the real print of the tire on a paper, achieved during the experiments. A good fit can be observed.

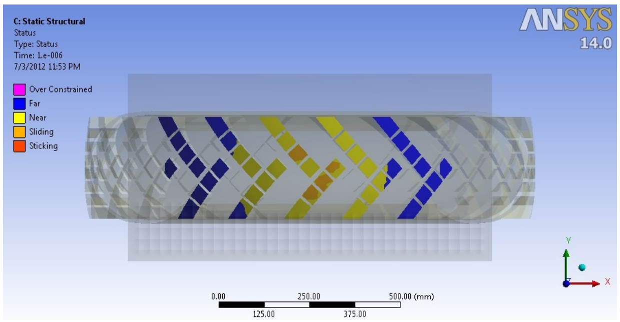

Fig. 12. Imprint of the tire generated in ANSYS

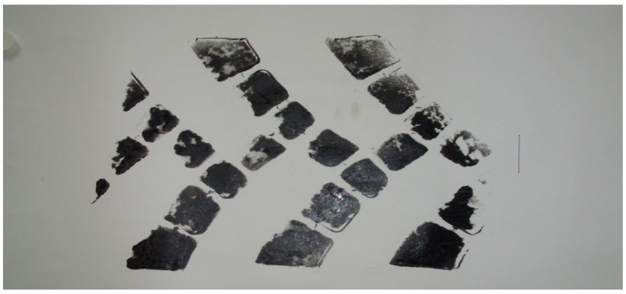

Fig. 13. Imprint on a piece of paper

This study is an initial simulation attempt in an improved design process of the military armored vehicles, in order to increase their mobility and safety. More experimental data will be further included in the simulation in order to obtain more realistic results and improved design solution.

References:

[1] Lee, C., Kim, J., Hallquist, J., Zhang, Y. et al. "Validation of a FEA Tire Model for Vehicle Dynamic Analysis and Full Vehicle Real Time Proving Ground Simulations," SAE Technical Paper 971100, 1997

[2] Kazuyuki Kabe, K., Koishi, M. “Tire cornering simulation using finite element analysis”, John Wiley & Sons, Inc. J Appl Polym Sci 78: p. 1566-1572, September 2000

[3] Giurgiu, T. "Modélisation et simulation du comportement des pneus a l'aide du logiciel ANSYS", Project. Master Conception Integree des Systemes Technologiques, “Politehnica” University of Bucharest, July 2012

[4] Giurgiu, T., Ciortan, F., Pupăză, C., "Tire Modeling using ANSYS", Engineering Numerical Modeling & Simulation, PRINTECH Publishing House, 2012

[5] ANSYS User’s Guide, SAS IP, Inc., 2011

[6] Ogden, R.W, Saccomandi, G., Sgura, I - Fitting hyperelastic models to experimental data. Computational Mechanics, 2004, http://persone.dii.unile.it/saccomandi/computationalmech.pdf

[7] Maia N.M.M., Silva J.M.M. Editors. Theoretical and Experimental Modal Analysis. Taunton Research Studies Press, 1997

[8] Iliescu M, Nuțu E, Georgescu L. “Finite Element Method Simulation and Rapid Prototyping”, 8th WSEAS International Conference POWER '08, p. 257-262, ISSN 1790-5117, Venice, Italy, November 21-23, 2008

[9] Marinescu D., Nicolescu A. “Gantry Robot Volumetric Error Evaluation using Analytical and FEM Modelling”, “Annals of DAAAM International Symposium“, Zadar, Croatia, 2010, ISSN 1726-9687, ISBN 978-3-901509-73-5, p. 1059-1060