FINITE ELEMENT MODELING AND ANALYSIS OF APASSENGER CAR TYRE

S. Muthuselvi[1],Dr.kishore brahma [2], M. Ramesh Kumar[3]
[1,3]Assistant Professor, Department of Mechanical Engineering
[,2]Professor, Department of Aeronauticall Engineering
Dhanalakshmi Srinivasan College of Engineering and Technology, Mamallapuram

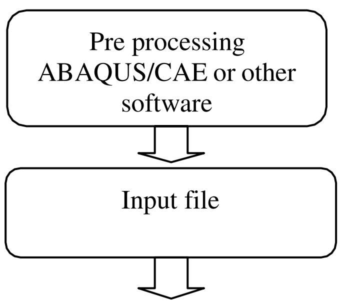
Abstract: Tyre analyses done through experimental methods takes more time as well as cost and also thedevelopment of the new tyre take longer time. To avoid this tidious tyre development process its behaviour can be predicted through Finite Element analysis. The main objective of the work is to perform finite element analysis of the tyre using Abaqus software. For the analysis I had chosen 165/70R 14 81S passenger car tyre and Mooney rivilin material model is used For my analysis. I had used frequency linear perturbation procedure and subspace Eigen solver with the maximum frequency of 20.68 Hz considering the tyre rotates at a speed of 120 Km/hr totally 4000 iteration has been used for this analysis. Since tyre is in torus in shape Tetrahedral element can only used for meshing.From this analysis we can find six different modes of shape of the tyre and maximum displacement at different modes. In static and dynamic analysis I have planned to find out the stress developed in bead part, body ply and tread part.

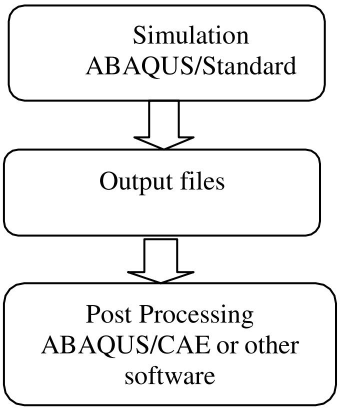
INTRODUCTION

The pneumatic tyres are the most widely used composites structure in commercial use and have functioned as an integral part of wheeled – vehicle system. The dominant use of pneumatic rubber tires in vehicle is attributed to their variousproperties, such as ability to support and guide the vehicle and to provide adequate traction and braking with good durability under widely varying tire-road interactions and loading conditions. The ride quality, handling, and directional and stability performance requirement of a vehicle thus strongly rely upon properties and tread design of the tyres. The characteristics of tyres, however, are not easily predictable and comprehensible due to complexity of tyre structure and lack of effective analysis methods.

ABAQUS PROCEDURE

A complete ABAQUS/Standard analysis usually consists of three distinct stages: pre-processing, simulation, and post processing. These three stages are linked together by files as shown in Figure





PREPROCESSING

We had constructs a model of the part to be analysed in which the geometry is divided into a number of discrete sub regions, or "elements," connected at discrete points called nodes." Certain of these nodes will have fixed displacements, and others will have prescribed loads. These models can be extremely time consuming to prepare, and commercial codes vie with one another to have the most user-friendly graphical "pre-processor" to assist in this rather tedious chore. Some of these pre-processors can overlay a mesh on a pre- existing CAD file, so that finite element analysis can be done conveniently as part of the computerized drafting-and-design process.

ANALYSIS

The dataset prepared by the pre- processor is used as input to the Finite element code itself, which constructs and solves a system of linear or nonlinear algebraic equations

Kijuj = fi

where **u** and **f** are the displacements and externally applied forces at the nodal points. The formation of the **K** matrix is dependent on the type of problem being attacked, and this module will outline the approach for truss and linear elastic stress analyses. Commercial codes may have very large element libraries, with elements appropriate to a wide range of problem types. One of FEA's principal advantages is that many problem types can be addressed with the same code, merely by specifying the appropriate element types from the library.

POST PROCESSING

In the earlier days of Finite Element analysis, the user would pore through reams of numbers generated by the code, listing displacements and stresses at discrete positions within the model. It is easy to miss important trends and hot spots this way, and modern codes use graphical displays to assist in visualizing the results. A typical postprocessor display overlay coloured contours representing stress levels on the model, showing a full-field picture similar to that of photo elastic or moiré experimental results.

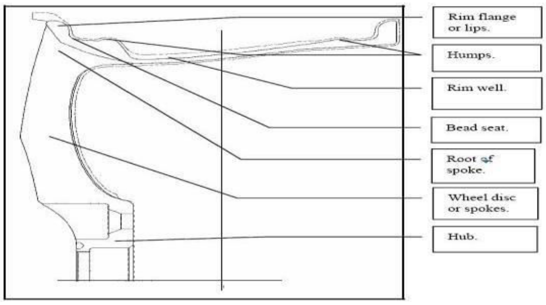
The operation of a specific code is usually detailed in the documentation accompanying the software, and vendors of the more expensive codes will often offer workshops or training sessions as well to help users learn the intricacies of code operation. One problem users may have even after this training is that the code tends to be a "black box" whose inner workings are not understood. In this module we will outlinethe principles underlying most current finite element stress analysis codes, limiting the discussion to linear elastic analysis for now. Understanding this theory helps dissipate the black-box syndrome, and also serves to summarize the analytical foundations of solid mechanics.

MODAL ANALYSIS

Modal analysis is the study of the dynamic properties of structuresunder vibrational excitation.The goal of modal analysis in structural mechanics is to determine the natural mode shapes and frequencies of an object or structure during free vibration.Detailed modal analysis will determines the fundamental vibration mode shapes and corresponding frequencies.

WHEEL MODEL

WHEEL ASSEMBLY



3D VIEW OF RIM

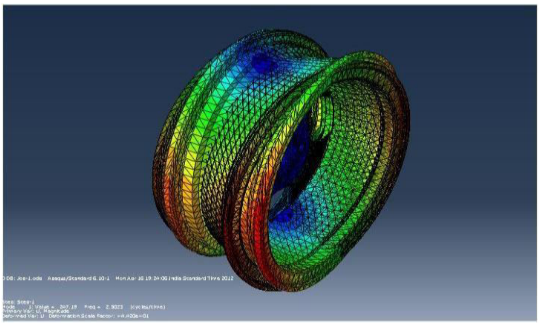
MATERIAL PROPERTIES

Young's Modulus	6.895Gpa
Yield Strength	20.7Mpa
Poisson ratio	0.32
Ultimate strength	257.95Mpa
Hardness (BHN)	80

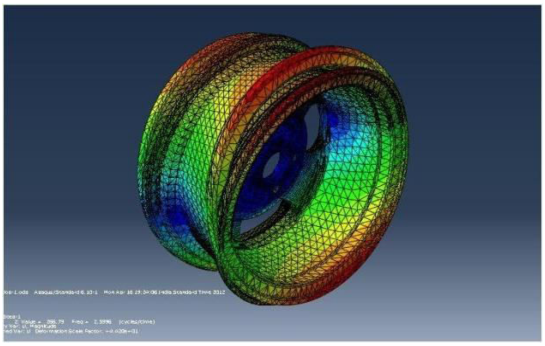
MODAL ANALYSIS

The deformation of the wheel due to varying frequency as shown in the figure. Results show that the displacement varies from 0.0954 mm to
1.145 mm.

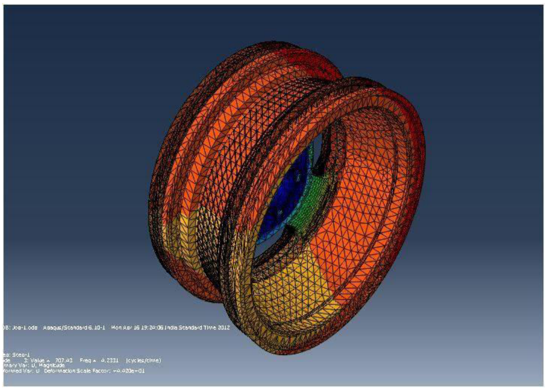
Mode 1



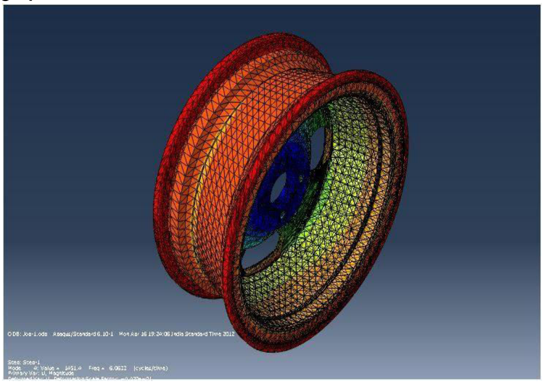
Mode 2



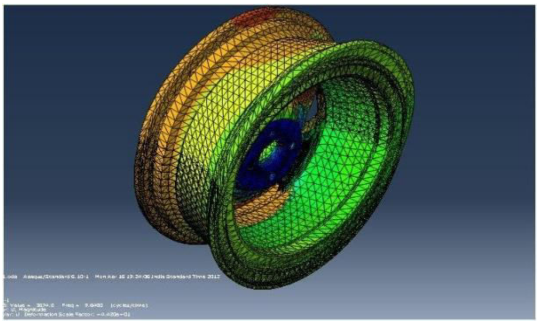
Mode 3



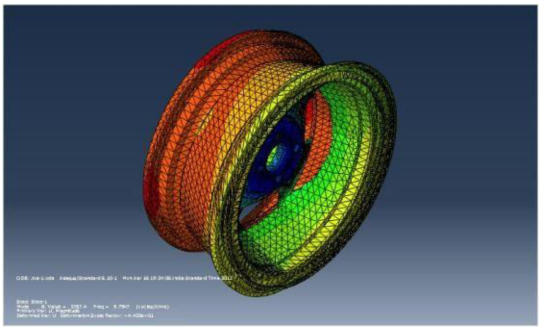
Mode 4



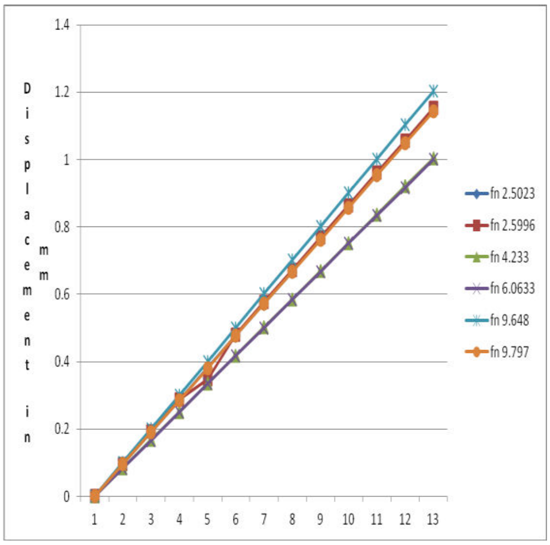
Mode 5



Mode 6



Result

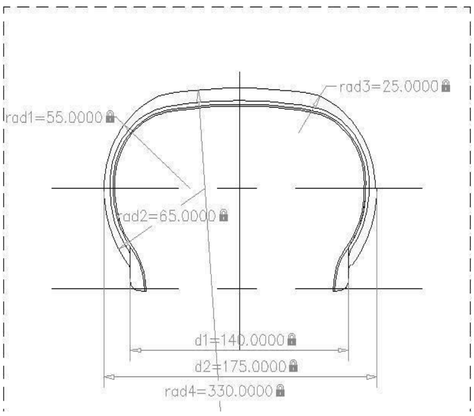


Highest frequency range =9.648Hz
Maximum displacement=1.144mm

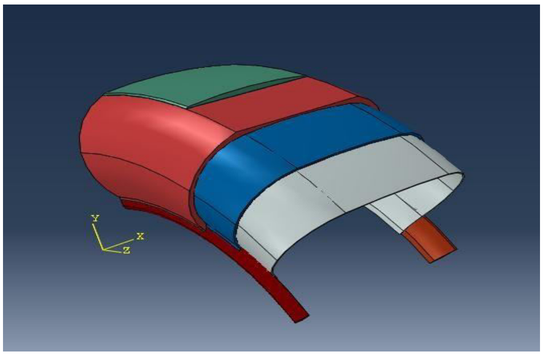
TYRE MODEL

2D & 3D MODELING OF TYRE
The 2D & 3D modeling has been createdwith AUTOCAD software. We had chosen 165/70R 14 81S passenger car tyre for analysis.

Tyre Designation:
165- Nominal section width70- Aspect ratio
R- Radial construction14- Nominal rim dia 81- Load rating
S- Speed rating2D VIEW



ASSEMBLED 3D CUT SECTIONAL VIEW OFTYRE



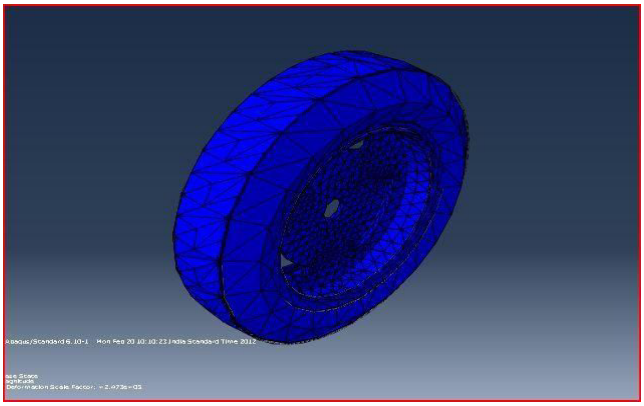
Material properties of tyre components(TONUK1998)

Tire section	Young's modulusE	Posion ratio µ	Density
Body ply	3.74 MPa	0.4	1250
Tread ply	3.25 MPa	0.4	1250
Rim	6.895 GPa	0.32	1250

MODAL ANALYSIS OF TYRE

For this analysis we had used linear perturbation solving method. And considering tyre rotates maximum at a speed of 140 Km/hr so that the maximum frequency ranges up to 17.98cycle/min. The tyre and rim part are connected with surface to surface contact and contact definition is created between the tyre and rim part. In this analysis the wheel hub has been arrested with all degree freedom considered the rim has been connected with vehicle. Each and every parts of the tyre are meshed separately with 30 nodes TED element (tetrahedral element).

ASSEMBLED AND MESHED 3D VIEW OFTYRE



CONCULTION

Following the described objective, the state of stress and the mechanical response of the aluminium alloyed automobile rim has beenestablished. In future with this result the tyre has to be incorporated with the rim part . And the mooney rivlin constants for our own tread formulation has to be find and the values shouldbe incorporated in our model and modal analysis, static and dynamic analysis has to be perform to find six different mode of shapes of tyre and stress, strain acts on the tyre.

Tire section	C_{10}	C_{01}
Bead filler	14.14 MPa	21.26 MPa
Sidewall	171.8 KPa	830.3 KPa
Tread	806.1 KPa	1.805 MPa

REFERENCES

D. Manas*, M. Manas, M. Staněk, V. Pata “Wear of tyre treads”, Journal of Achievements in Materials and Manufacturing Engineering (2009)
Mir Hamid Reza Ghoreishy “A State of the Art Review of the Finite Element Modelingof Rolling Tyres” , Iranian Polymer Journal (2008)
U. Suripa, A. Chaikittiratana “Finite element stress and strain analysis of a solid tyre” Journal of Achievements in Materials and Manufacturing Engineering(2008)
N.KorunoviC, M.TrajanoviC,M.StojkoviC “FEA of tyres subjected to static loading” Journal of the Serbian Society for Computational Mechanics(2007)
J.R. Cho, K.W. Kim, W.S. Yoo, S.I. Hong “Mesh generation considering detailed tread blocks for reliable 3D tire analysis” Advances in Engineering Software(2004)
Chang-Ro Lee and Jeong-Won Kim, John
O. Hallquist, Yuan Zhang and Akbar D. Farahani “Validation of a FEA Tire Model for Vehicle Dynamic Analysis and FullVehicle Real Time Proving GroundSimulations”(2004)
Wan-Suk Yo, Jeong-Hyun Sohn, Kwang- Suk Kim,Jeong-Han Lee “Tire Models for Ride Analysis and Modification of Road Profiles to Compensate Nonlinear Characteristics of Tires”(2003)
Rao, K.V.N. Kumar, R.K Bohara, B.C Mouli, C.P, “Transient Finite Element Analysis of Tire Dynamic Behavior”, TireScience and Technology(2003)
Yong-Joe Kim and J. Stuart Bolton “Hybrid Two-Dimensional Finite Element Model of Tyres"(2002)
Mir HamidReza Ghoreishy “FiniteElement Analysis of a 6 .45-14 Bias Tire Under Contact Load” Iranian PolymerJournal (2001)
David Roylance “Finite ElementAnalysis”(2001)
Rao, K.V.N Kumar, R.K Bohara, B.C Mouli, C.P, “Dynamic Analysis Of Tyre Cornering Behavior Using Explicit Finite Element Code”(2002)
Tonuk, "Computer Simulation of Dynamic Behavior of Pneumatic Tires", Ph. D. Thesis, Mechanical Engineering Department, MiddleEast Technical University, Ankara(1998)

Bosi, C., and Garagnani, G.L., "Fatigue Properties of a cast Aluminum Alloy for Rims of Car Wheels"
Chang,C.L. and Young, S.H., "Finite Element Simulation of Wheel Impact Test" Volume 28 Issue 2- 2008
Currie, A., and Wilson, B., "Finite Element Analysis of an Automotive Wheel - A Case History," National Conference Publication - Institution of Engineers, Australia, pp.16-20, 1981.
Keer, S.C., Russell, D.L. "FE-Based Wheel Fatigue Analysis Using MSC. FARIGUE" 1999
Kirkner, D.I., Spencer, B.F., and Schudt,E.E., "Determination of tire – wheel interfacepressure distribution for aircraft wheels,"AIAA/ASME Structures structural dynamics and materials conference, pp. 415-423, 1993.
Liangmo Wang, Yufa Chen "Fatigue Life Analysis Wheels By Simulation of Rotary Fatigue Test" 2010
Morita, Y., Kawashima, H., Ishihara, K., and Komatsu, H., "Finite Element Analysis of a Car Wheel," Sumitomo Metals, Vol. 39, No.3, pp. 245-263, 1987.