International Journal of Applied Engineering Research
ISSN 0973-4562 Volume 10, Number 8 (2015) pp. 20875-20885
© Research India Publications
http://www.ripublication.com

An Investigation of Tire Tread Material Effect on Auto Wheel Impact Strength Using FEA and Experimentation

Balaji Sasane, Shahid Tamboli, Ravi Sekhar*
Symbiosis Institute of Technology (SIT),
Symbiosis International University (SIU),
Lavale, Pune - 412 115, Maharashtra State, India
**Email: ravi.sekhar@sitpune.edu.in*

Abstract

Impact strength consideration is a critical part of any tyre design procedure. This paper is concerned with a comparison of four different tire tread materials for performance under impact loading. Finite element analysis was conducted to obtain displacements and stress distributions of these four kinds of treads under impact loading between two wheel spokes. Experiments were conducted on wheels having four kinds of tread materials on an impact test rig. Experimental results validate the FEA predictions of tyre deflections and stresses. A comparison of results indicates that the Mat III rubber material is most suitable for the given tyre application.

Introduction

Safety is one of the most important criteria in vehicle design. Some structural parts of the vehicles are designed for plastic collapse in the event of a collision to absorb the impact [1]. In such scenarios, the tyre-road interaction is also of prime significance. Therefore, tyres must be designed to withstand high impact loads to ensure passenger safety during collisions. According to Schoor et al [2], a lot of accidents occur due to tyre deficiencies.

A vehicle tyre serves four functions, to carry the vehicle weight, to accelerate and brake the vehicle using traction, to facilitate steering and to cushion the vehicle from road surface imperfections [3]. A tyre is a difficult structure to analyse, as it is made up of parallel layers of different materials. This analysis becomes especially complicated due to the large impact loads acting on layers of tyre materials with non-linear behaviours [4-7]. Therefore, such complex analysis of a tyre can only be done in full detail by a numerical procedure like the finite element method [8].

Ghoreishy [9] developed a non-linear finite element model for a biased tire under varying conditions of inflation pressures and contact loads. They simulated this model NSTAR using Lagrangian formulation. Tan et al [10] conducted experiments for deformation behavior of front wheel-tyre assembly of a motorcycle subjected to frontal impact loading. Parameters such as impact speed, impact mass, inflation pressure level of the tyre, contact geometry of the striker and offset distance of impact location from the axle were varied during the experiments. It was found that the deformation of the wheel-tyre assembly is well correlated to the dissipated impact energy. Reid et al [11] simulated tyre behaviour during crash of a vehicle. They developed a new tire taking into account specifications like tread, sidewall, steel beads, steel belts and plies. This new tire model was simulated for several applications including impacting a curb, driving over rocks and landing on a culvert grate to demonstrate its effectiveness. Neves et al [12] developed a wheel-tyre impact test rig to understand the various mechanisms of tire compound failure. This test comprised of a round indentor dropped upon pressurized tyre specimens with different impact energies. Test results were validated through FEA analysis. Chang et al [13] conducted similar experiments to show that total plastic work approach can be used to predict the wheel fracture during the impact test. Kumar and Meher [14] predicted the plastic strain of Aluminium alloy wheels under impact loads. They found that cracks appear at plastic strains greater than 4 %. They also predicted the tyre locations vulnerable to failure and predicted the minimum required thickness of Al alloy wheel (5.9mm) for safety under impact loads. Srikanta and Raju [15] optimized a car alloy wheel for car by simulating different geometrical shapes with different materials. They found that the zamak material along with honey-comb structure performed better as compared to other materials.

In the present work, four kinds of tire tread materials were tested for performance under impact loading using finite element analysis.

Finite Element Modelling

Geometric Modeling

In this study, a general-purpose commercial explicit finite element code, HyperMesh-LS-Dyna was utilized to conduct the dynamic impact test simulations. The CAD model of tire and wheel used in this study was constructed based on the actual geometry of a bias auto wheel. A 3-D solid model consisting of three components, namely tire, wheel and impactor was firstly modeled in CatiaV5 software (Fig. 1) using exact dimensions from the experimental setup (Fig.15).

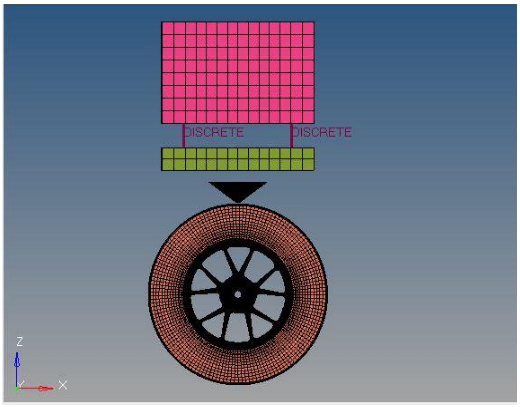

Figure 1: FEA Model Setup

Meshing

The CAD geometry model in IGES format was imported in HyperMesh for the preparation of FE model. 1-D beam elements were used to discretize steel cords of the tire. Thereafter, the impactor was modeled using 8-node hexahedral elements.

Tyre and Rim Meshing

The pneumatic tire consists primarily of rubber materials and reinforced rubber composites. Therefore, it was modeled as an assembly of three-dimensional Mooney-Rivlin hyperelastic solid elements for the rubber material, two-dimensional fiber-reinforced layered membrane elements for reinforced rubber composites, and one-dimensional beam elements for the two beads. Finite element models of rim and tire are as shown in Fig 2 and 3, respectively. Rim is composed of tetra mesh with 37006 nodes and 124056 elements. Tire consists of hexahedral meshing with 31030 nodes and 27115 elements.

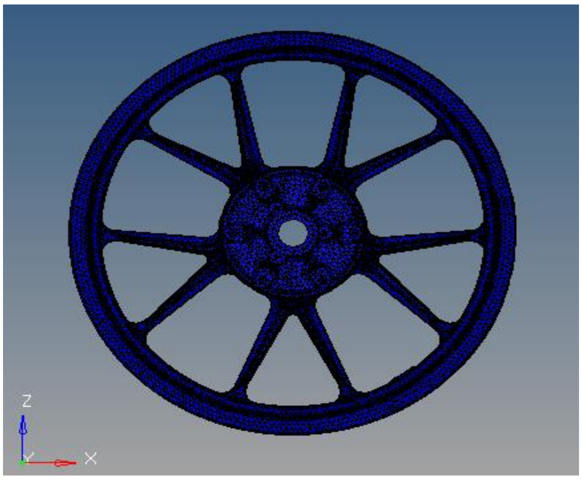

Figure 2: Finite Element Model of Rim

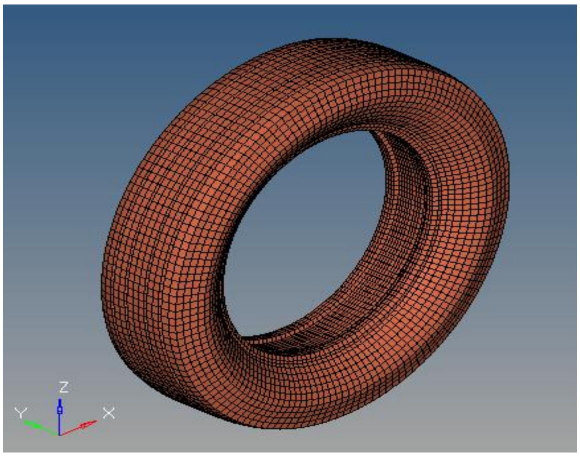

Figure 3: Finite Element Model of Tire

Meshing of Tread and Bead Fillers

To accommodate the complex and variable geometric features of tire and wheel, two layers of 8-node linear hexahedral elements were used to mesh the tread portion of the tire.

Since the tread and the bead fillers are made of relatively thicker rubber material and experience shear stresses and sudden changes of curvature during service, three-dimensional solid elements are required to model those parts in the tire. A three-dimensional model of solid parts is created by rotating the whole cross-section one full rotation about the tire axis.

Tire-Rim Contacts Definition

Tire and rim contacts were defined appropriately to assure proper mounting and fit of the tire to the rim. As the tire is inflated, the tire model sits tightly inside of the rim and is constrained by the tire-rim contact. The tire contact parts with the rim, called rim strips, and rim contact are extremely important because pressurized air is sealed by the contact (Fig. 4).

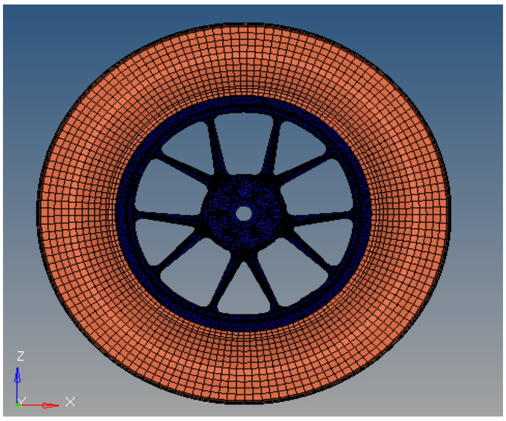

Figure 4: Tire-Rim Assembly

Tire Pressure Modeling

Pressure in tires was modeled by defining a pressurized control volume, referred to as an airbag in LS-DYNA.

Material Properties

This work analyzes the impact performance of four different tread materials (rubbers). These tread material properties are given in Table 1. Other components' properties are as follows.

Sidewall Material

The sidewall rubber was modeled using an elastic material, with a density of 1.10E-06 kg/mm^3, a Young's modulus of 0.03 GPa, and a poison's ratio of 0.45.

Steel beads

To model the steel beads that run near the inner portion of the tire, resultant beam elements were used. The bead material was modeled using an elastic material, with a density of 7.86E-06 kg/mm^3, a Young's modulus of 200 GPa, and a Poisson's ratio of 0.28.

Steel belts

Each steel belts cable consists of seven individual wires similar to the steel bead. Each wire has approximately a 0.35 mm diameter. There are two layers of these cables in the tire. Each layer has 64 cables in it and the cables are spaced approximately 2.5 mm apart.

Boundary Conditions and Loads

The wheel centre was constrained for all six degrees of freedom. The movement of the impact load was restricted in all directions except for vertically downward direction. Main load (480 kg) and auxiliary load (40 kg) were applied to simulate car and suspension system weights respectively.

Table 1: Tread Material Properties

Material	Density (Tons/mm^3)	Elasticity Modulus (MPa)	Poison's Ratio
MAT-I	1.0E-14	1.82	0.49
MAT-II	1.004E-13	30	0.45
MAT-III	1.004E-13	11.6	0.495
MAT-IV	1.004e-15	100	-

Results and Discussions

Figures 5-12 show the analysis results (displacements and Von Mises stresses) of impact tests on the four kinds of tread materials. From these figures it can be seen that among all the four materials, material III shows lower variation of deflection per unit of load. The decreasing order of variation of deflection per unit load is MAT-IV, MAT-II, MAT-I and MAT-III.

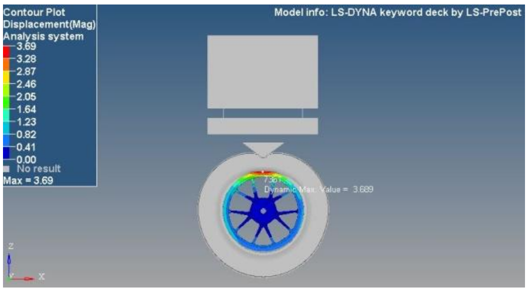

Figure 5: MAT-I displacements

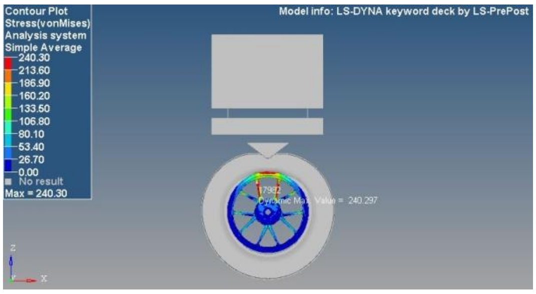

Figure 6: MAT-I Stress Distribution

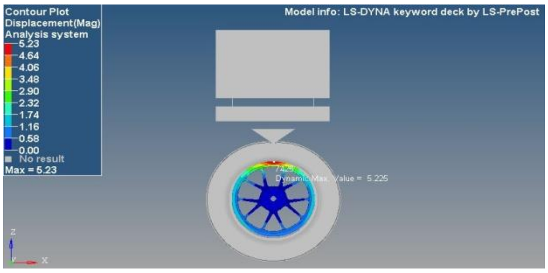

Figure 7: MAT-II Displacements

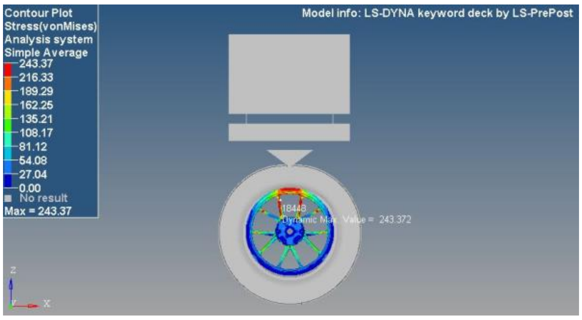

Figure 8: MAT-II Stress Distribution

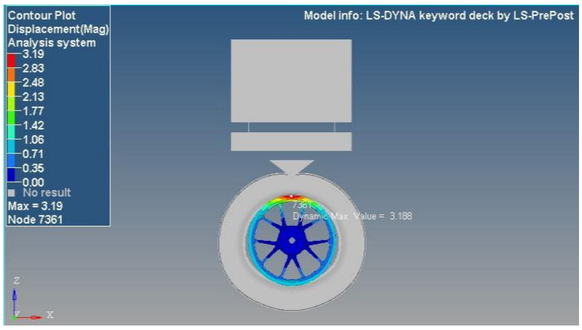

Figure 9: MAT-III Displacements

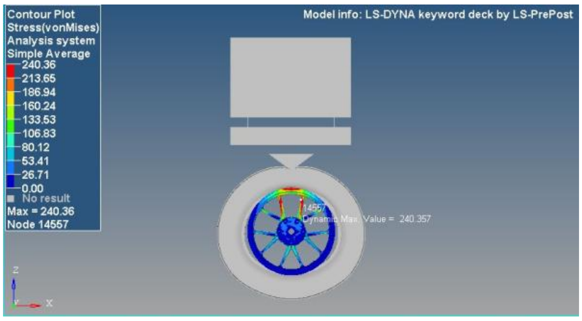

Figure 10: MAT-III Stress Distribution

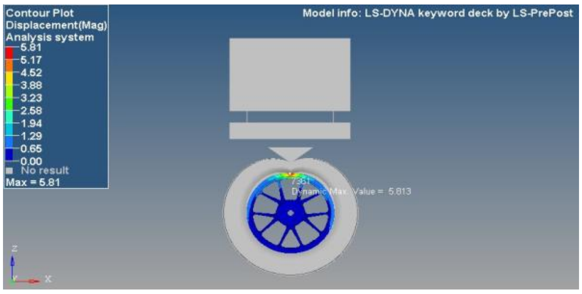

Figure 11: MAT-IV Displacements

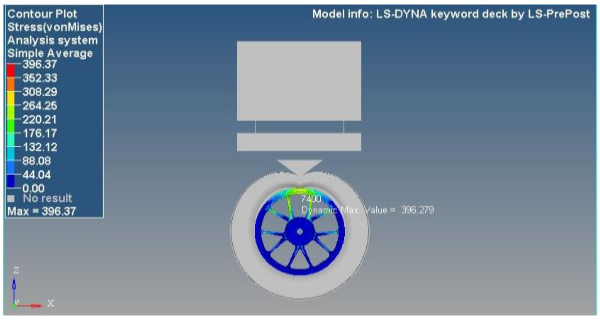

Figure 12: MAT-IV Stress Distribution

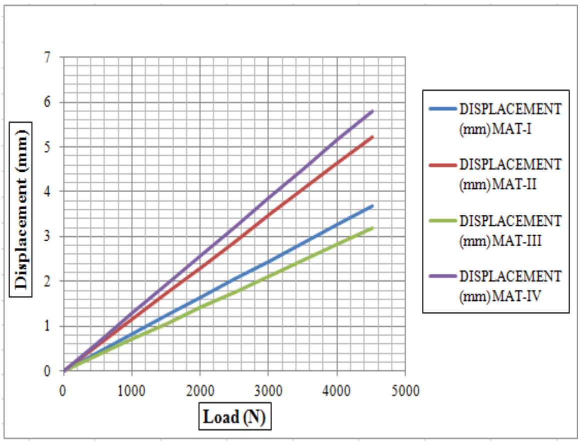

Figure 13: Rim Displacement Plots Comparisons

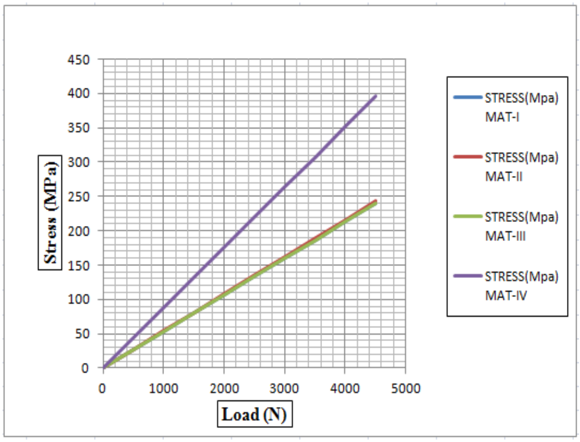

Figure 14: Rim Stress Plots Comparisons

Figures 13 and 14 show comparisons of displacements and stress behaviors of the four tread materials. It is clear from these plots that MAT-III material is most desirable among all materials under current study for impact loads applied between wheel spokes.

Experimental Validations

The experimental setup (impact test rig) consisted of main weight, auxiliary weight with “V” shape indenter, quick return mechanism, stand, guide and frame. Impact load consisted of main and auxiliary weights with indenter and spring as shown in Fig.15. Main weight consisted of a 420 Kg mass, whereas the auxiliary weight consisted of a 40 Kg mass. The distance between main weight and auxiliary weights was 100 mm, separated by a coil spring of stiffness 304 N/mm. The guide was used in setup to allow the movement of the impact load in vertically downward direction only. Wheel was fixed on the stand and falling height of the impact load was 400mm.

The obtained experimental results matched closely with the simulated results (Tables 2 and 3).

Table 2: Comparison of Experimental and FEA results (displacements)

Material	Displacements (mm)		% Error
	Experimental	FEA	
MAT-I	4.03	3.69	8.43
MAT-II	6.25	5.23	16.32
MAT-III	3.65	3.19	12.6
MAT-IV	6.67	5.81	12.89

Table 3: Comparison of Experimental and FEA results (stresses)

Material	Stresses (MPa)		% Error
	Experimental	FEA	
MAT-I	243.5	240.3	2.65
MAT-II	256.8	243.37	5.23
MAT-III	247.4	240.3	2.87
MAT-IV	400.6	396.37	1.05

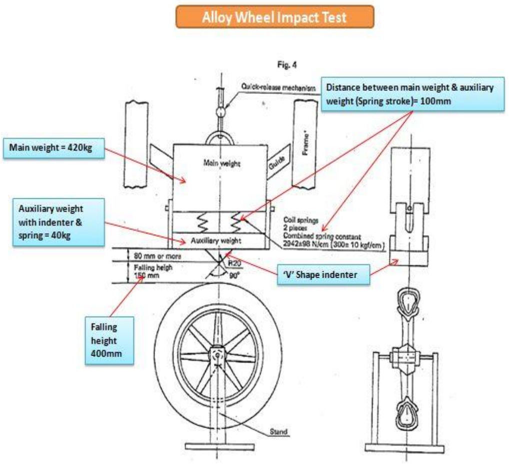

Figure 15: Schematic of The Impact Test Rig

Conclusions

This paper reported a comparative analysis of different tread materials for impact load performance of a biased auto tire. Detailed FEA simulations were conducted to obtain stress and displacements of the wheel under identical impact loads. Validation experiments were performed on the wheels using an impact testing rig. Results indicate the superiority of the Mat III rubber as compared to other 3 materials to give least displacements and stresses. Therefore, this material may be recommended for application in the said tires for optimum performance.

References

[1] Karagiozova D, Alves M. Dynamic elastic-plastic buckling of structural elements: a review. Applied Mechanics Reviews 2008; 61(4).

[2] van Schoor O, van Niekerk JL, Grobbelaar B. Mechanical failures as a contributing cause to motor vehicle accidents – South Africa. Accident Analysis and Prevention 2001;33:713–21.

[3] Tonuk E, Unlusoy Y S, Prediction of automobile tire cornering force characteristics by finite element modeling and analysis, Computers and Structures, 2001, 79, 1219-1232.

[4] Helnwein P, Liu CH, Meschke G, Mang HA. A new 3-D finite element model for cord-reinforced rubber composites – Application to analysis of automobile tyres. Finite Elements in Analysis and Design 1993;4:1–16.

[5] Yang LM, Shim VPW, Lim CT. A visco-hyperelastic approach to modeling the constitutive behaviour of rubber. International Journal of Impact Engineering 2000;24:545–60.

[6] Rao S, Daniel IM, Gdoutos EE. Mechanical properties and failure behaviour of cord/rubber composites. Applied Composite Materials 2004;11:353–75.

[7] Zamzamdeh M, Negarestani MA. 3D tyre/road interaction simulation by a developed model (ABAQUS code). ASIMMOD. Chiang Mai (Thailand) 2007.

[8] Neves R R V, Micheli G B, Alves M, An experimental and numerical investigation on tyre impact, International Journal of Impact Engineering, 2010, 37, 685-693.

[9] Ghoreishy H R, Finite element analysis of a 6.45-15 bias tire under contact load, Iranian Polymer Journal (Iran), 2000.

[10] Tan K S,Wong S V,Radin Umar R S,Hamouda A M S,Gupta N K, An experimental study of deformation behaviour of motor-cycle front wheel-tyre assembly under frontal impact loading, International Journal Of Impact Engineering 2006;32:1554-72.

[11] Reid J D, Boesch D A, Bielenberg R W, Detailed tire modeling for crash applications, ICrash, 2006.

[12] Neves R R V,Santiago R C, Alves M, Numerical solution of wheel-tire assembly subjected to impact loading. International Congress of Mechanical Engineering (Brazil), 2009.

[13] Chang Chai-Lung, Yang Shao-Huei, Simulation of wheel impact using finite element method, Engineering Failure Analysis, 2009;16:1711-19.

[14] Kumar Ch P V Ravi, Meher R Satya, Topology optimization of aluminum alloy wheel. International Journal of Modern Engineering Research, 2013, 1548-53.

[15] Srikanta B. Anusha, Raju P. Veera, Geometrical and material optimization of alloy wheel for four wheeler, International Journal of Research and Innovation, 2012, 1401-02.