Scientific Journal of Impact Factor (SJIF): 4.72
e-ISSN (O): 2348-4470
p-ISSN (P): 2348-6406

International Journal of Advance Engineering and Research Development

Volume 4, Issue 7, July -2017

# ANALYSIS OF INFLATED STRUCTURES: A CASE STUDY ONANLYSIS OF AUTOMOBILE TYRE USING ANSYS

Pravin Menariya[1], Akash Shelar[2], Prof. G. N. Kotwal[3]

[1] *M.Tech. [Design Engg.] Department of Mechanical Engineering, Vishwakarma Institute of Technology, Pune*
[2] *M.Tech. [Design Engg.] Department of Mechanical Engineering, Vishwakarma Institute of Technology, Pune*
[3] *Asst. Professor.Department of Mechanical Engineering, Vishwakarma Institute of Technology, Pune*

**Abstract —** *This paper highlights the various performing characteristics of inflatable structures. The brief information about some research paper is presented in this review. The classification of inflated structures with respect to their use in specific area is shown in this paper. To study the characteristics regarding analysis of inflated structures, the studies on the Tires have been analyzed. To study the aspects of analysis of pre-stressed inflated structures, one Tire model is analyzed for Tire- Road contact analysis.*

***Keywords****-application; inflatable; Ansys; structures; Non-linear*

## I. INTRODUCTION

Inflatable structures are characterized by being made from a relatively thin skin which, unless pre-stressed, has very little capacity to bear compressive loads. In order to provide some more general knowledge of this family of structures, an overview is given of the various applications and related technologies. Three distinctive fields of applications have seen some renewed interest over the past decade. They are: civil, aeronautical and space applications. In civil engineering the pre-stressing of the skin is achieved by either a pressure differential over the skin or by loads applied to the edges. The latter method has wide-spread application in tent structures and therefore a majority of the research is focused on patterning and the effect of loads such as wind on the skin. In aeronautical engineering inflatable structures can be found in applications such as airships and inflatable wings. Equipping unmanned small aircraft with inflatable wings may reduce the weight and may be beneficial over rigid systems in term of energy consumption during wing deployment. A number of research projects have been undertaken since the mid nineteen-eighties in relation to using inflatable technology in space. Some of these projects have resulted in actual flight experiments. In 1996, a 14m diameter inflatable antenna was successfully deployed in space and in 2000 an inflatable re-entry vehicle was tested. Typical research areas are: wrinkling behavior; static and dynamic mechanical behavior; surface accuracy; and deployment modeling and control.

## II. LITERATURE REVIEW

[1] In this paper by Bernd Kroplin has briefly introduced to the fundamentals of inflated structures, their potential and their limitations. Geometry and Shapes are investigated. Additionally, high and low pressure within structural differences and their typical pressure control problems are investigated. A brief morphological sketch of inflated structures is presented, which has opened a room for further development in the future.

The variability in construction can be classified as follows:

1. Permanent membrane structures with external pretension

 

***Figure 1 Cooling tower Schmehausen***

***Figure 2 Inflated reentry capsule***

2. Permanent and temporary membrane structures with inflated walls (Figure 2)

3. Structures with full gas volume

***Figure 3 Tennis court cover***

4. Buoyancy structures

***Figure 4 High altitude balloon***

5. Interesting combinations

***Figure 5 Air Chain Stuttgart***

[2] Tiberiu Giurgiu, Florina Ciortan, and Cristina Pupaza dealt with static and dynamic behavior by modeling and simulation of the radial tires for civil emergency vehicles or military armored vehicles. The tire is made up of a complex composite structure consisting rubber, textile-cords and steel-cords. Complex models of tire were developed in CATIA v.5 and square block road surface were considered with tire contact. Static and transient dynamic behavior of the models is obtained with the help of The Finite Element Method and ANSYS software. The presented work was aimed to increase the mobility and the safety of the vehicles. Hyperelasticity related to materials for which stresses are derived from total strains using a strain energy density function. Odgen material model was used to describe the nonlinear strain behavior of the tire in this work. Two rubber-type materials, available in ANSYS material library were used for the tire, and Structural Steel for the road surface. To obtain the optimum parameters of working condition, a first homogeneous model of the tire was used. It was used without any steel insertion and with a smooth tread surface, without ribs and grooves. In the next phase to achieve performance as close as possible to the real tire, by using steel-cords and beads in different configurations more complex models were realized in the CAD system,

[3]Another research by Wang Qingchao, Song Wanqing, Liang Jiankai is aimed to analyze and get a further understanding of the dynamic characteristics of the tire by means of computing modal analysis. On the basis of the previous studies a tire modal testing system was built on. They also investigated to study how the amplitude of exciting force and loading changes affect the tire modal test, which gives a foundation for implication of the tire modal test into practical use. Besides, the relations of experimental modal and computing modal will also be analyzed. The analysis and application of the tire modal plays significant role for a solid foundation with medium-high frequency tire mechanical model. This is being established by their research group. This was done using HYPERMESH and ANSYS for setting up the finite element model of the tire and to analyze the tire structural dynamics. Which lays a foundation for setting, recognizing the correctness of experimental modal parameter as well as studying the relationship between calculating modal and experimental modal. And this work also studies the law of tire’s dynamic characteristics and its reason.

[4]In another research paper by Krunal Girase, Mr. Navneet Patil , Dr. Dheeraj Deshmukh worked in Comparative Stress Analysis using Ansys 12 software is used for different tires of same material with different structured threads in the same operating condition. In this work, for stress analysis different structured threads are used as Plane threaded tire, line threaded tire and Cross threaded tire. Operating conditions are kept constant as 300RPM, Plane concrete road profile, pressure on all tires are same and rubber material is used for all three tires. To evaluate this two geometries are modeled in solid-edge tool as three different tires and the road profile. In Ansys 12 Load and rotation is provided on tire models while road profile is fixed. For Plane threaded tire as compared to line threaded and cross threaded tire minimum stress as well as minimum deformation was observed during the simulation analysis

After summarizing above studies about the inflated structure, we have carried out static analysis of inflated tire against road surface.

 

## III. ANALYSIS OF INFLATED TYRE

Hydrostatic fluid elements are well suited for calculating fluid volume and pressure for various problems involving fluid solid interaction. Modeling with a hydrostatic fluid allows us to study the changes in fluid behavior when it is contained within a solid upon which various loads are applied.

### 3.1. Problem Description.

A 3D tire model is inflated and pressed against the road surface. The tire is modeled with hyperelastic material and reinforcing elements. The air inside is modeled with a hydrostatic fluid element, and the pressure, volume and density is monitored as loads are applied to the tire.

### 3.2.1. Tire Modeling.

To simulate actual conditions, the tire dimensions approximate those of a P215/65R16 car/minivan tire. An incompressible hyperelastic material model is used for the tire. Reinforcing elements are embedded into the solid elements to model steel reinforcing within the tire structure.

Steps involved in modeling the problem are:

1. Model the Tire as a Solid
2. Model the Air Inside of the Tire
3. Model the Tire Reinforcing
4. Model the Tire Rim
5. Model Tire Contact with the Road Surface

### 3.2.2. Element Details.

HSFLD242 Element Description

HSFLD242 is used to model fluids that are fully enclosed by solids (containing vessels). The hydrostatic fluid element is well suited for calculating fluid volume and pressure for coupled problems involving fluid-solid interaction. The pressure in the fluid volume is assumed to be uniform (no pressure gradients), so sloshing effects cannot be included. Temperature effects and compressibility may be included, but fluid viscosity cannot be included.

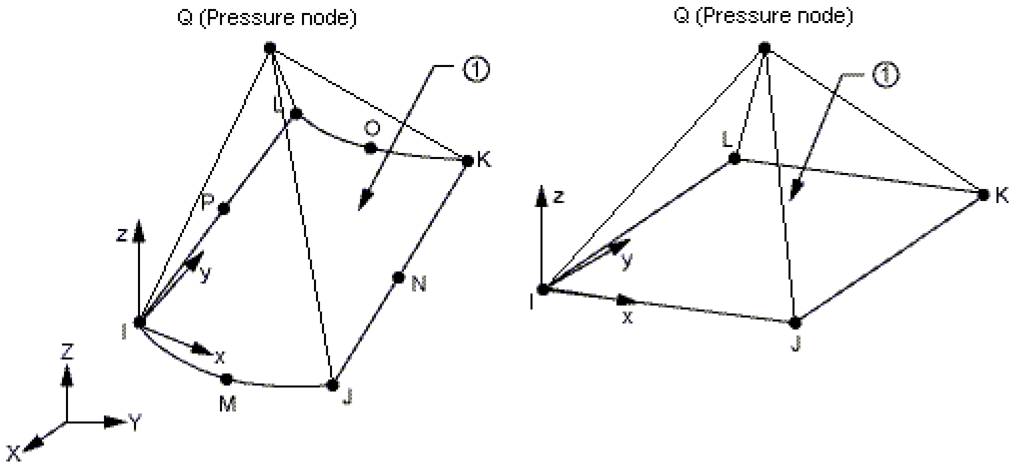


*Figure 6 HSFLD242 Element*

### 3.3. Material Properties.

*Table 1. Tire Material*

| Tire Rubber | |
|---|---|
| Young's Modulus | 6894.8 MPa |
| Poisson's Ratio | 0.5 |
| Density | 2.67E-9 Kg/mm3 |

 

***Table 2. Fluid Material Properties***

| Internal Air | |
|---|---|
| Initial Density | 1.225e-12 Kg/mm3 |
| Reference Temperature | 20˚C |

***Table 3. Reinforcing Material Properties***

| Reinforcing in contact with road surface | |
|---|---|
| Young's Modulus | 2.0E5 MPa |
| Poisson's Ratio | 0.3 |
| Reinforcing in side wall | |
| Young's Modulus | 2.0E4 MPa |
| Poisson's Ratio | 0.3 |

### 3.4. Model Definition.

The tire is modeled with SOLID186 solid elements by rotating a 2D meshed area around an axis defined by two key points. The air inside of the tire is modeled with HSFLD242 hydrostatic fluid elements. The elements are generated (ESURF) with a pressure node over the solid elements to enclose the air. Because the fluid elements are pyramid shaped with common vertices at each pressure node, the fluid elements cover some undesired volumes as well. So that fluid elements exist only in the region where air should be present, fluid elements having a negative volume in the undesired region are used. This technique involves manually generating fluid elements with the nodes ordered in a clockwise direction.

Tire reinforcing is modeled with reinforcing element REINF265. Different material models are used to define the reinforcing in the road contact area and the side walls. A rim is modeled as a rigid body using a multipoint constraint (MPC) algorithm (rigid constraints). Node to surface contact elements (CONTA175) are used for the rim nodes and a target element (TARGE170) is used to define a pilot node at the center of the tire Surface to surface contact pair is modeled between the road and tire. The CONTA174 element is used to model the part of the tire's surface which comes in contact with the road. The road is modeled as a rigid target (TSHAP,QUAD).

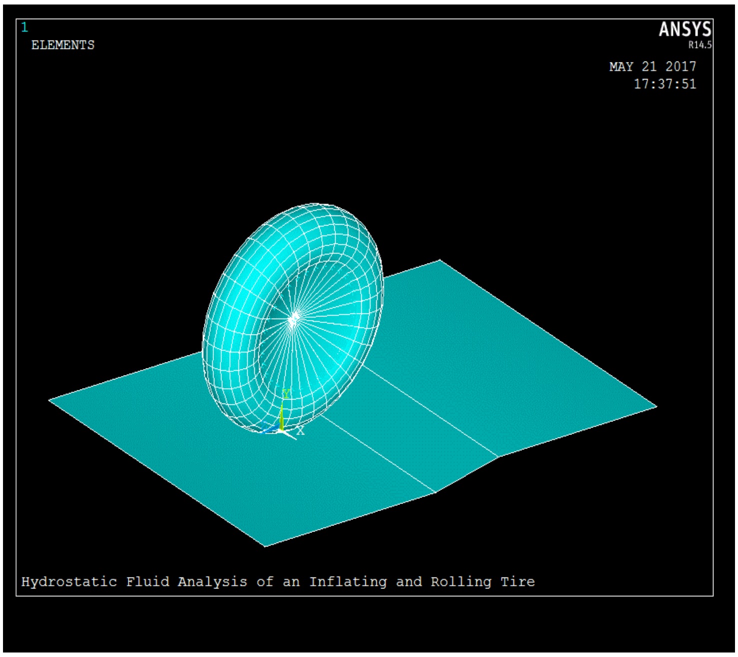


***Figure 7 Modeling of tire-road surface***

 

## IV. RESULTS

Model of tire is created and simulated in Ansys APDL. The tire is inflated to 36 psi and a mass of 1 ton is added at the axle to simulate the portion of the vehicle's mass resting on that axle.

The analysis occurs over three load steps, as follows:

1. Gravity load is applied and reference temperature of $20^0$ is set for the air.
2. Tire is inflated for required pressure.
3. Tire is moved down to make contact with the road surface.

After applying the above conditions the model is solved for deformation and stresses. The deformed shape and stress plots of the model are shown in the figure below.

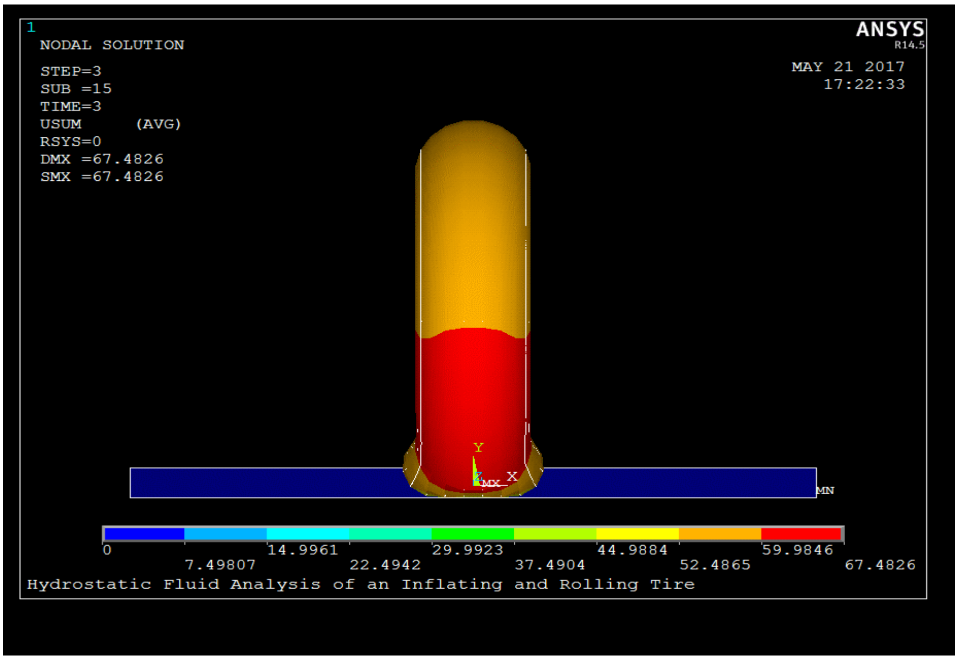


*Figure 8 Deformation Plot for 30 psi internal pressure*

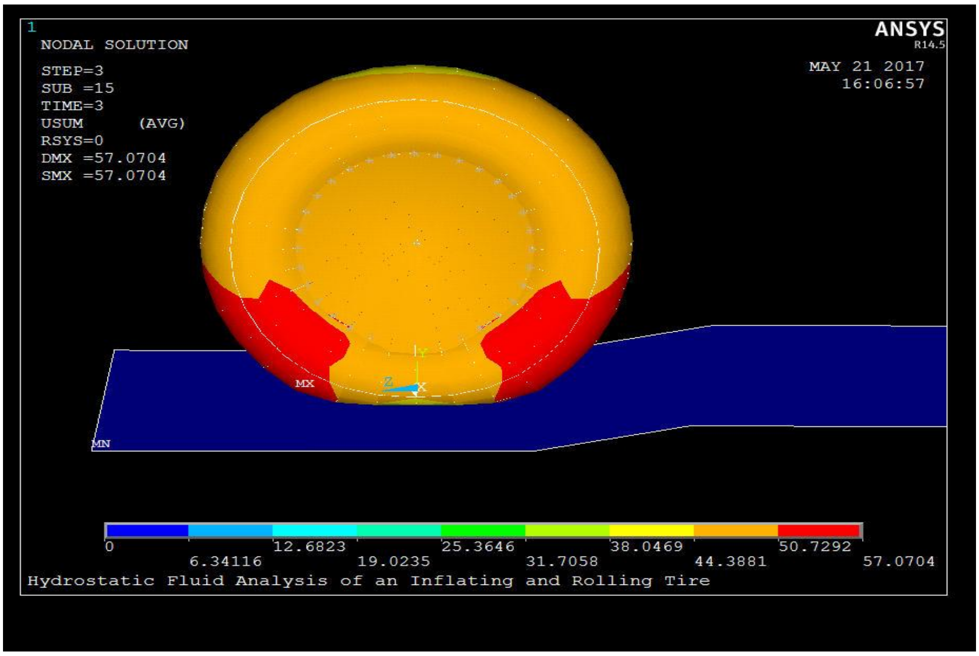


*Figure 9 Deformation Plot for 36 psi internal pressure*

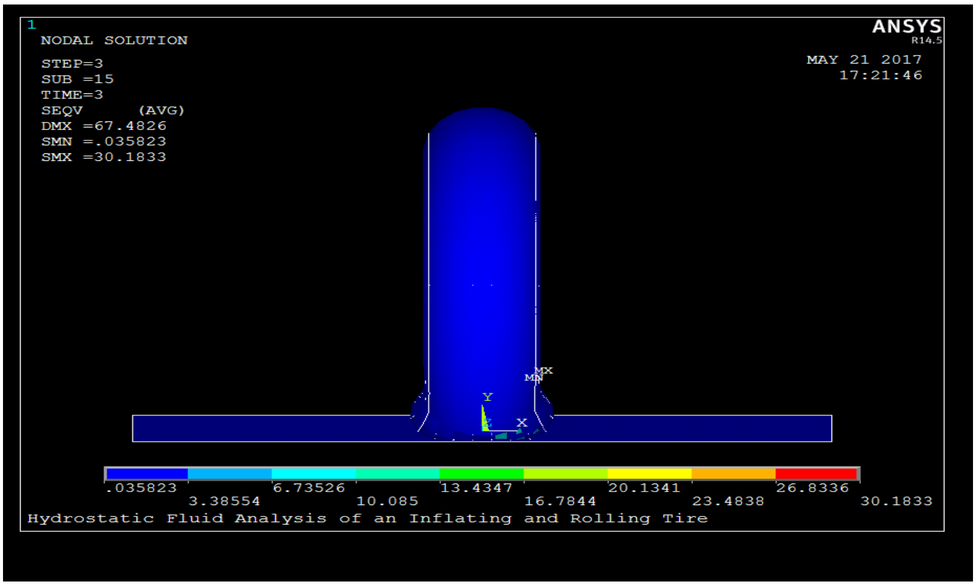


***Figure 10 Stress Plot for 30 psi internal pressure***

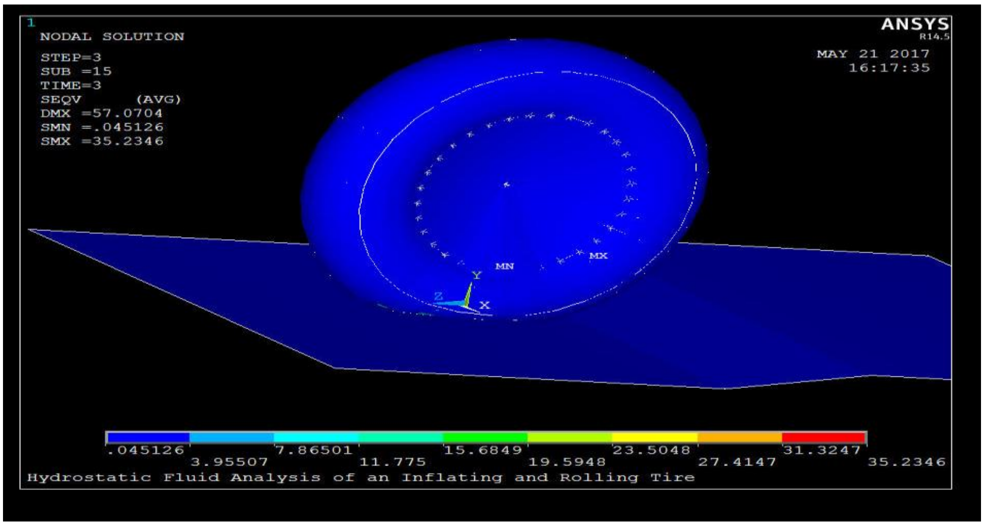


***Figure 11 Stress Plot for 36 psi internal pressure***

***Table 4 Deflection and Stress values***

| Result | Inflate Pressure (psi) | Gravity Load (Ton) | Deflection (mm) | Stress (MPa) |
|---|---|---|---|---|
| 1 | 30 | 1 | 67.48 | 30.1833 |
| 2 | 36 | 1 | 57.0704 | 35.2346 |

## V. CONCLUSION

We have studied several papers related to inflatable structures, their classification and respective applications. We have studied various researches about the analysis of the Tire structure as example of inflated structure that contain modal analysis, dynamic behavior of the tires and comparative analysis with different thread orientation. We have modeled and analyzed one model of tire- road contact stresses and deformation incorporated between the tire and the road surface in ANSYS 14.5. The inflated or deployable structures are hot topics in today's innovating science topics. It has multiple applications in space craft as landing equipment. As it is lighter in weight, can be shortened in volume and providing relatively good strength these aspects has made it one of the most innovative topic. We have studied the basic information regarding the different deployable structures.

## REFERENCES

[1] Bernd Kroplin, “Inflated Membrane Structures on the Ground, in the Air and in Space - A Classification”, E. Oñate and B. Kröplin (eds.), Textile Composites and Inflatable Structures,2005, 213–220.

[2] Tiberiu Giurgiu, Florina Ciortan, Cristina Pupaza,”Static and Transient Analysis of Radial Tires Using ANSYS” University POLITEHNICA of Bucharest, ISBN: 978-1-61804-186-9, pp 148- 158.

[3] Wang Qingchao, Song Wanqing, Liang Jiankai, “Tire Modal Analysis of ANSYS and Hypermesh Under Co-simulation”, ISSN: 2331-0642, 2015; 3(5): 37-43

[4] Krunal Girase, Mr. Navneet Patil, Dr. Dheeraj Deshmukh, “Comparative Stress Analysis using Ansys Software for Different Tires Performance Evaluation”, IJSSBT, Vol. 3, No. 2, June 2015.

[5] Ansys Mechanical APDL user guide,release 14.5, Southpointe, November, 2013.

[6] Bergstrand B Eggren E Jakobsen P Penttinen O Stigberg B Oberg G:. “Tillstandsmt -ning drift och underhall. Nordiska Vagtekniska Frbundet”, 1996. Utskott 41: Drift och Underhall. Borlnge.

 