International Journal of Engineering Science Invention
ISSN (Online): 2319 – 6734, ISSN (Print): 2319 – 6726
www.ijesi.org Volume 3 Issue 9 ǁ September 2014 ǁ PP.18-23

Static And Modal Analysis Of Rotating Wheel Rim Using Ansys

Jeetendra Kumar Chakrawarti
Industrial Training Institute,
Jaitpur, distric Shahdol (M.P), India

ABSTRACT: *This paper provides an analysis of a rotating wheel at different velocities on a rigid surface. Finite Element model is developed in ANSYS 14.0 to study the stress distribution and Free Vibration behavior for rotating wheels through Modal analysis which combines accuracy and substantially reduces computational effort.*

KEYWORDS : *Finite Element, modal analysis, ANSYS, Free Vibration*

I. INTRODUCTION

Now days, research undertaken in the field of finite element analysis is proven to be a successful tool in predicting the behavior of frame structures. Tire vibration modes as a characterization of tire dynamics are used in a broad range of Tire/vehicle system simulations, ranging from low-frequency vehicle handling to higher frequency structure-borne and air-borne noise simulations. The modal parameters are used either to define a complete modal tire model or to calibrate internal parameters of a tire model [1]. The essence of car wheel rim provides a firm base on which to fit the tire. Its dimensions, shape should be suitable to adequately accommodate the particular tire required for the vehicle [2]. The purpose of the car wheel rim provides a firm base on which to fit the tire. Its dimensions, shape should be suitable to adequately accommodate the particular tire required for the vehicle [3]. At higher rotational speeds, the inertia effects of the rotating parts must be consistently represented in order to accurately predict the rotor behavior and to decrease possibility of failure [4]. Alloy wheels made up of composite materials will reduce the unstrung weight of a vehicle compared to one fitted with standard aluminum alloy wheels. The benefit of reduced unstrung weight is more precise handling and reduction in fuel consumption. [5].

MODELING & METHODOLOGY: During rotation of a wheel at variable velocities types of stresses, vibration and impact are observed, to determine these effects a ANSYS is used for simulation. Analyzing these results we can improve a wheel strength and durability.

II. FINITE ELEMENT METHOD

Finite element analysis (FEA) consists of computer model of a material or design that is stressed and analyzed for specific results. It is used in new product design, and existing product refinement. In other words, FEA is a numerical method to find out an approximate solution for variables in a problem which is difficult to obtain analytically. The calculation of potential design changes such as temperature, buckling and deflection are usually complicated. A numerical method that is able to solve these engineering problems is known as the finite element analysis. In case of structural models failure, FEA may used to help determine the design modifications to meet the new condition. The concept of the finite element analysis is solving a continuum by a discrete model. It is done by dividing the problem into small several elements. Each element is in simple geometry and this is easier to be analyzed than the actual problem or the real structure. Each element is then applied with known physical laws. The equation which is formed by each element or parameters then will combined to form a global equation. The new equation can be used to solve the field variables such as displacement, buckling, temperature and so on.

Working of finite element method: FEA uses a complex system of points called nodes which make a grid called a mesh. This mesh is programmed to contain the material and structural properties which define how the structure will react to certain loading conditions. Nodes are assigned at a certain density throughout the material depending on the anticipated stress levels of a particular area. Regions which will receive large amounts of stress usually have a higher node density than those which experience little or no stress. Points of interest may consist of: fracture point of previously tested material, fillets, corners, complex detail, and high stress areas. The mesh acts like a spider web in that from each node, there extends a mesh element to each of the adjacent nodes. This web of vectors is what carries the material properties to the object, creating many elements. The basic methodology used by ANSYS in performing the simulation is:

- Modelling of wheel rim
- Analysis of rotating wheel in fem (ANSYS)
- Pre Processing
- Post processing
- Solution
- Results

Geometry: Modeling was performed on workbench platform under the Geometry section of ANSYS 14.0. Structural Steel material was applied after that to the model. Fig. 1 shows the model of wheel rim.

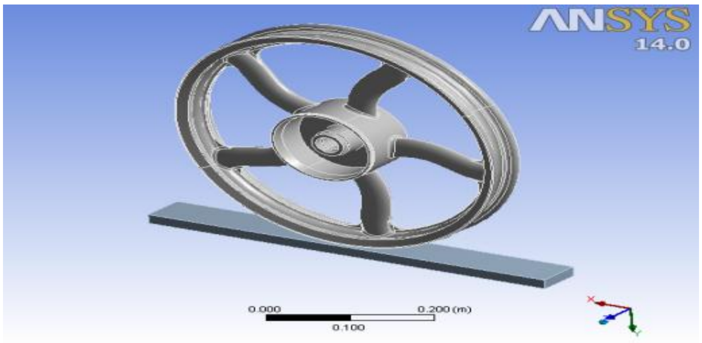

Figure 1: Isotropic Elasticity of structural Steel

TABLE I. PROPERTIES OF STRUCTURAL STEEL

Property	*Value*
Young's Modulus Pa	2.00E+11
Poisson's Ratio	0.3
Bulk Modulus Pa	1.67E+11
Shear Modulus Pa	7.69E+10

Meshing: The important requirement of the FEM is the need to split the solution domain (model geometry) into simply shaped sub domains called 'finite elements'. This is a discretization process commonly called meshing and element are called finite because of their finite, rather than infinitesimally small size having infinite numbers of degree of freedom. Thus the continuous model with an infinite number of degrees of freedom (DOF) is approximately by a discretized FE model with a finite DOF. This allows the reasonably simple polynomial functions to be used to approximate the field variables in each element. Meshing the model geometry also discretized the original continuous APDL are can be known as tools to help designers perform parametric analyses in which simulation software automatically solves for entire ranges of specified variables and generates displays that enable users to readily spot trends and identify an optimal design ANSYS Workbench platform is one of the most efficient ways of deal with geometric parameters which enables parameters of the CAD model to be driven directly from simulation. for a rainbow of colors' (rainbow network flow or RNF) incorporated.

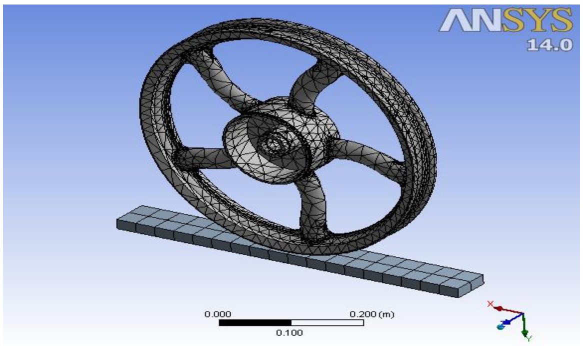

Figure 2: Meshing
Number of nodes generated – 40196
Number of Elements - 22186

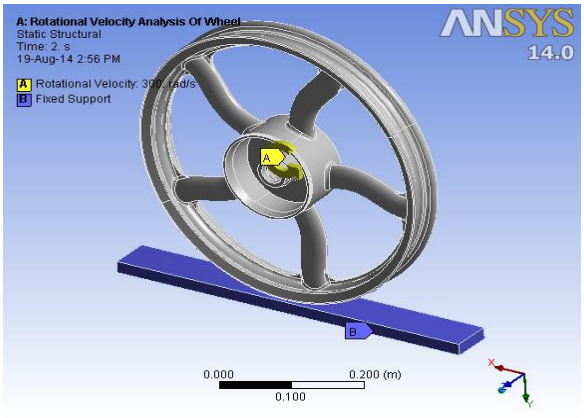

Figure 3: Defining Loads and support

Velocity of 100 rad/sec to 600 rad/sec was applied at the time step of 1 second to 6 seconds at the centre of the rim. A rigid support was provided to act as road on which rim resting.

SOLUTION

- Static Structural Analysis

  Static structural analysis was performed and Total deformation and Principle stress were analyzed.

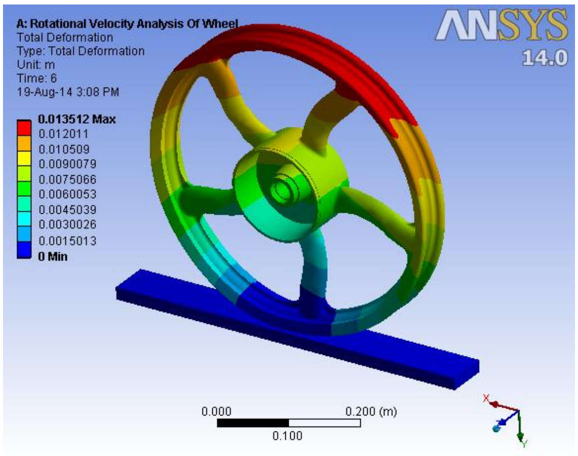

Figure 4: Total Deformation

The maximum deformation was found out to be .0135 m at top surface of the rim. Fig. 4 shows the deformation in contour.

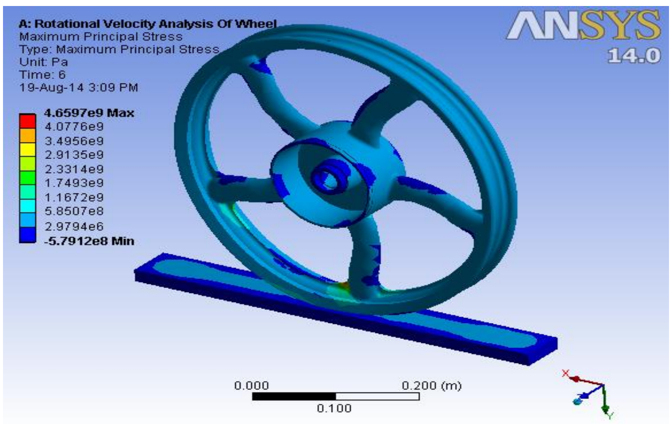

Figure 5: Principle Stress

The maximum principle stress was found out to be 4.66 x 10^3 Mpa at the joint of spokes and rim which can be seen in Fig. 5.

- Modal Analysis

Modal analysis was performed in ANSYS to study the free vibration behavior of the rim.

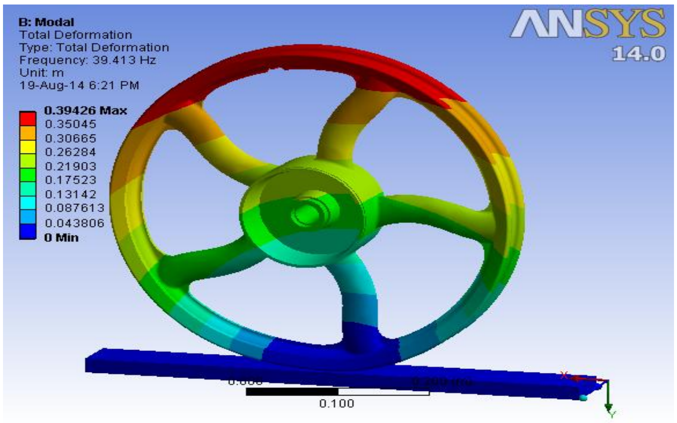

Figure 6: First mode deformation

TABLE II. MODE FREQUENCIES

Mode	*Frequency[Hz]*
1	39.413
2	81.387
3	224.86
4	437.3
5	586.94
6	742.66

Table II displays the first six mode frequencies. From Fig.6 we can see the maximum deformation at first mode is 0.394m which is at a frequency of 39.413 Hz. Our concerns are first three modes. Hence to avoid catastrophic failure these three frequencies should be avoided.

III. RESULTS :

Fig.7 to Fig.8 shows a total deformation, principle stresse and modes of vibration in variable frequencies in ansys with respect to 1 seconds to 6 seconds time steps and 100 rad/sec-600 rad/sec velocity.

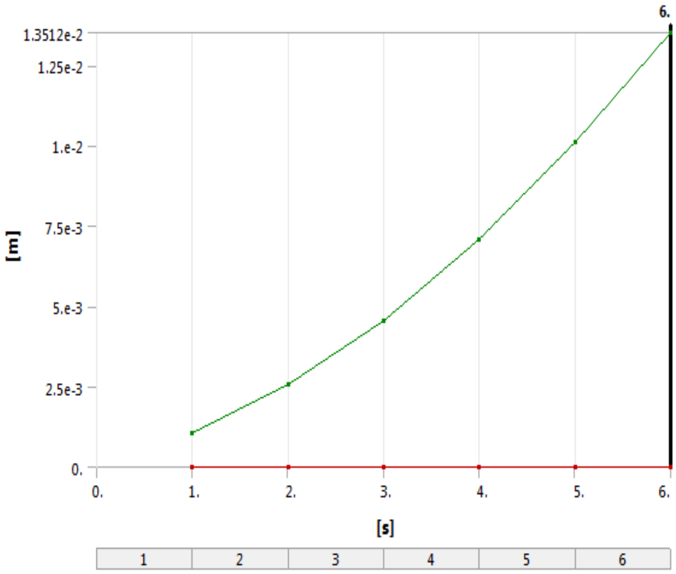

Figure 7: Total Deformation

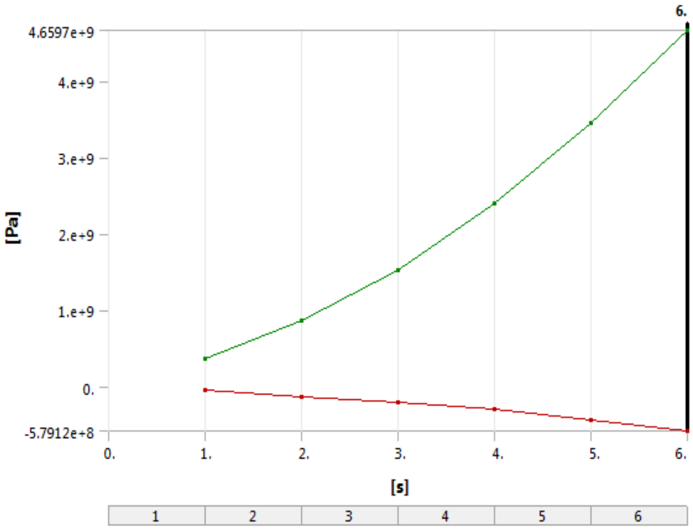

Figure 8: Principle Stress

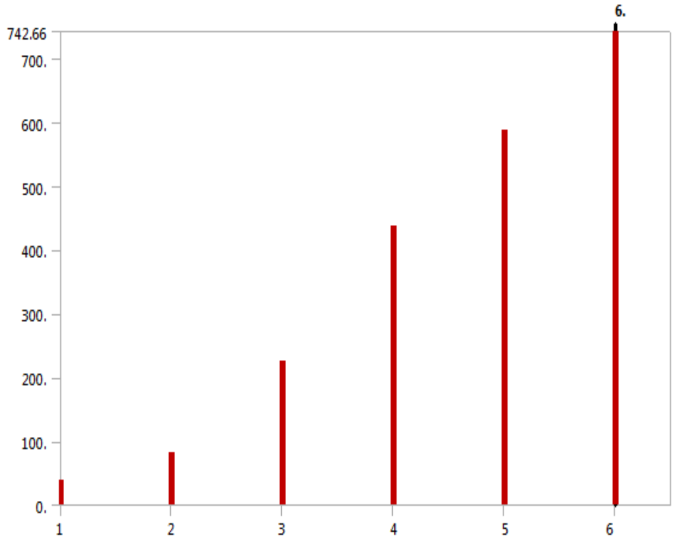

Figure 9: Modal frequency

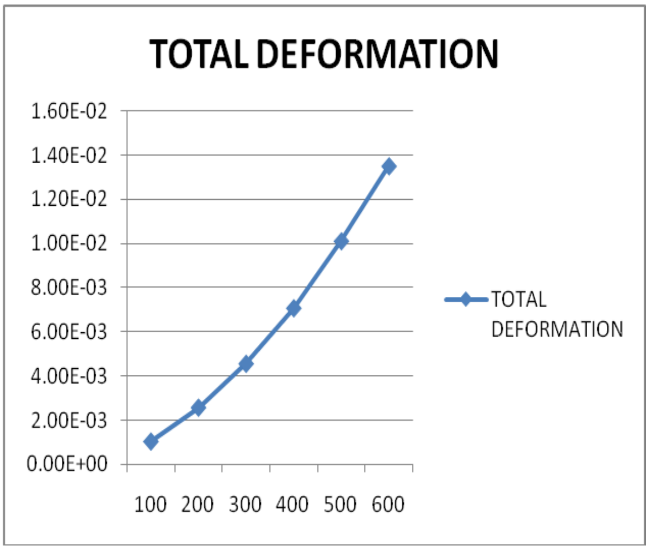

Figure 10: Total Deformation

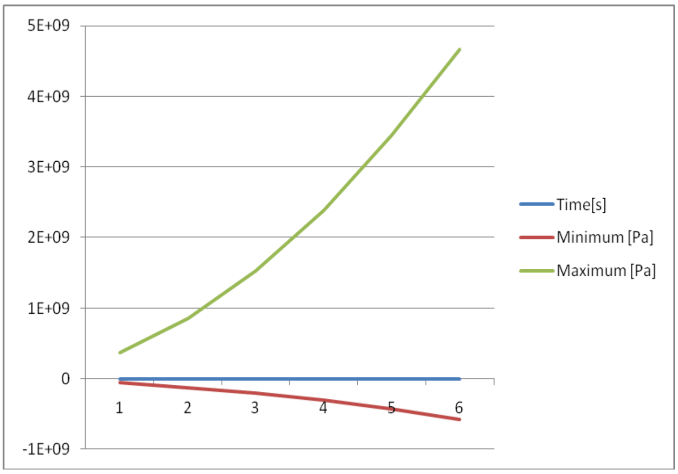

Figure 11: Max and Min Principle Stresses

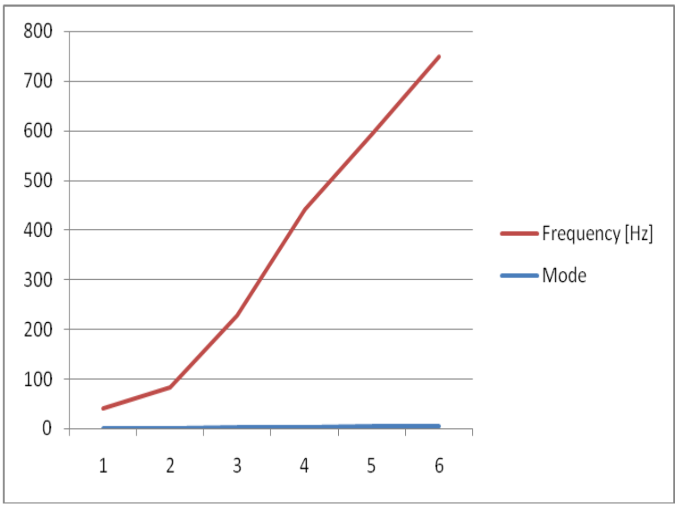

Figure 12: Modes of vibration in different frequencies & Steps

IV. CONCLUSION

The application of the Finite Element Modeling to research related work of wheel rim has been quite numerous, starting just after the maturity of the Finite Element Method as a general tool of analysis. From the result section we can see the various parameters which were analyzed and conclude that ANSYS can be used to save the laborious and time consuming work in the design process of wheel rim, meanwhile it can also be used to do any change in the design if required seeing the analysis results to save money in building up the prototype. It can also help us to predict the behavior under free vibration and get the natural frequencies of the model, which can be avoided to prevent the model from any catastrophic failure.

REFERENCES

[1] Peter Kindt, Filip De Coninck, Paul Sas and Wim Desmet, "Experimental modal analysis of radial tires Under different boundary conditions," The thirteenth international congresson sound and vibration, Vienna,Austria, July 2-6, 2006.
[2] T. Siva Prasad,T. Krishnaiah, J. Md. Iliyas and M.Jayapal Reddy, " A review on modelling and analysis of car wheel Rim using CATIA and ANSYS,"(IJISME), ISSN: 2319-6386, Volume-2, Issue-6, May 2014.
[3] P. Meghashyam, S. Girivardhan Naidu and N. Sayed Baba, "Design and Analysis of Wheel Rim using CATIA & ANSYS," IJIEM, Volume 2, Issue 8, August 2013.
[4] M. Varun Kumar and B. Ashiwini Kumar, "Model Analysis of Axially Symmetric Linear Rotating Structures", IJEAT, ISSN: 2249-8958, Volume-2, Issue-4, April 2013.
[5] Saurabh M Paropate1 and Sameer J Deshmukh, "modelling and analysis of a motorcycle wheel rim," Vol. 2, No. 3, July 2013