International Journal of Advance Research in Science and Engineering
Vol. No.5, Special Issue No. (02), March 2016
www.ijarse.com
IJARSE
ISSN 2319 - 8354

REWIEW OF FINITE ELEMENT ANALYSIS OF VEHICLE RIM

Rakesh B. Thakare [1], Prof.Tushar Y.Badgujar [2]

[1]P.G.Student, [2] Professor &Guide (L.G.N. Sapkal College of Engg., Nashik)

ABSTRACT

In this Paper a general concept of the FEM analysis is also shown through the realproject on vehicle rim. FEA consists of a computer model of a material or design that is analyzed for specific results. Modifying an existing product or structure is utilized to qualify the product or structure for a new service condition. In case of structural failure, FEA may be used to help determine the design modifications to meet the new condition. Structural analysis covers static analysis, modal analysis, harmonic analysis, transient dynamics, Eigen value problems.

Keywords: Finite Element Methods (FEM), Finite Element Analysis (FEA), Radial stress, Rim

I. INTRODUCTION

As part of technological improvement, comfort and safety have become essential demands of human beings. This comes not only from market-oriented competition but also from legislation that may seek some certain standards. Cars, important favours of technological development, are widely used in daily life. It seems that mankind no longer lives without them. Therefore, these wonderful machines should be safe and economical so that people could use them safely and more people could purchase them. Since rims, on which cars move, are the most vital elements in a vehicle, they must be designed carefully. The rim type examined in this study has some trouble when touching any curb or entering a sharp curve. The rims manufactured by various methods are made of either steel or cast aluminium alloys. In particular, rims made of aluminium casting alloys are more preferred because of the weight and the cost effectiveness.

II. FINITE ELEMENT ANALYSIS

FEA has been developed to an incredible precision. There are generally two types of analysis that are used in industry: 2-D modeling, and 3-D modeling. While 2-D modeling conserves simplicity and allows the analysis to be run on a relatively normal computer, it tends to yield less accurate results. 3-D modeling, however, produces more accurate results while sacrificing the ability to run on all but the fastest computers effectively.

FEA has advantages over most other numerical analysis methods, including versatility and physical appeal.

FEA is applicable to any field problem: heat transfer, stress analysis, magnetic fields and so on. There is also no geometric restriction. The body or region analyzed may have any shape.

Boundary conditions and loading are not restricted.

Material properties are not restricted to isotropy and may change from one element to another or even within the element.

Components that have different behaviors, and different mathematical descriptions can be combined. Thus a single FE model might contain bar, beam, plate, cable and friction element.

FE structure closely resembles the actual body or region to be analyzed.

The approximation is easily improved by grading the mesh so that more elements appear where field gradients are high and more resolution is required.

III. STRUCTURAL ANALYSIS OF RIM

Static analysis calculates the effects of steady loading conditions on a structure, while ignoring inertia and damping effects, such as those caused by time-varying loads. A static analysis, however, includes steady inertia loads (such as gravity and rotational velocity), and time-varying loads that can be approximated as static equivalent loads (such as the static equivalent wind and seismic loads commonly defined in many building codes).

IV. LOADS IN A STRUCTURAL ANALYSIS

Static analysis is used to determine the displacements, stresses, strains, and forces in structures or components caused by loads that do not induce significant inertia and damping effects. Steady loading and response conditions are assumed; that is, the loads and the structure's response are assumed to vary slowly with respect to time.

The kinds of loading that can be applied in a static analysis include

- Externally applied forces and pressures.
- Steady-state inertial forces (such as gravity or rotational velocity).
- Imposed (non-zero) displacements.
- Temperatures (for thermal strain).

V. GENERAL STEPS INVOLVED FATIGUE ANALYSIS USING FEA SOFTWARE

Wheel Specification

Provide proper dimensions of rim as per design and specification.

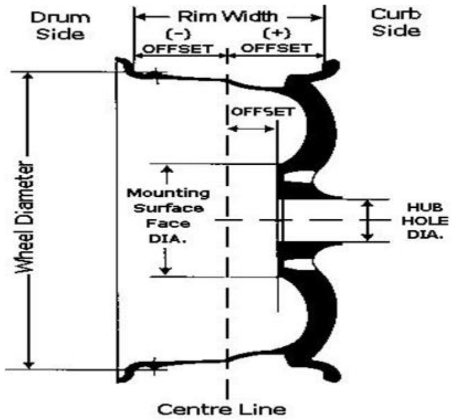


Fig.1.RIM SPECIFICATIONS

Model Description

The 3-diemensional modal of the wheel can be created in CATIA or PRO-E and the file can exported in the IGES (international graphics exchange specification) format into ANSYS. The 3-diemesional model example is shown below.

Meshing of the Wheel

The meshing was performed using the mesh generate option in the ansys workbench. The process of generating a mesh of nodes and elements consists of three general steps.

3.1 Set the element attributes.

3.2 Set mesh controls (optional).

3.3 Meshing model.

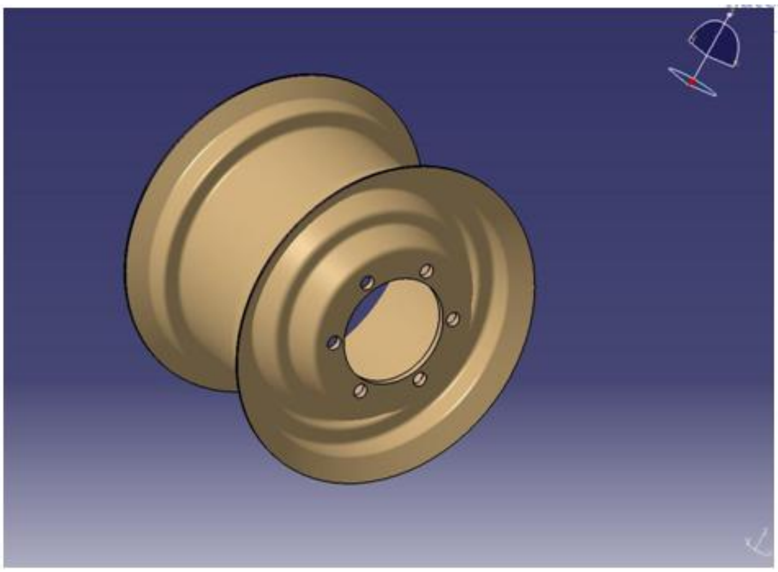


Fig.2.3-D Model of RIM (an example)

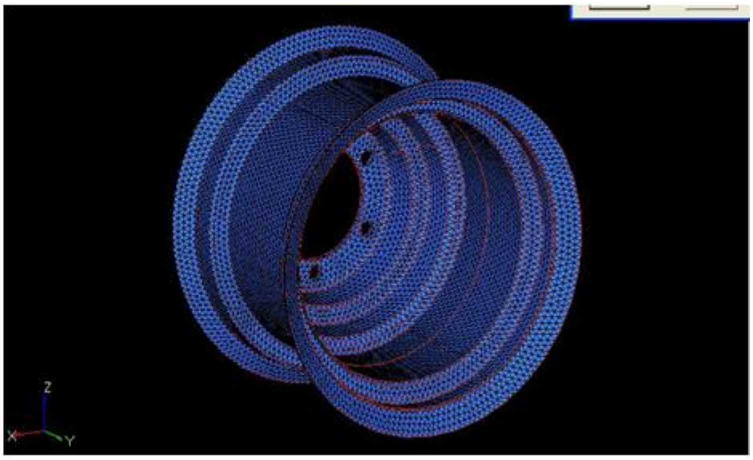


Fig.3 MESHING OF RIM (an example)

IJARSE
ISSN 2319 - 8354

4. ANALYSIS: Fatigue analysis is used to determine the life, safety and damage of anycomponent. The present work involves the determination of the life, safety factor and damage of alloy wheel and corresponding deformation, shear stress and alternative stress.

In ansys work bench we had added tools of static analysis and fatigue tool.

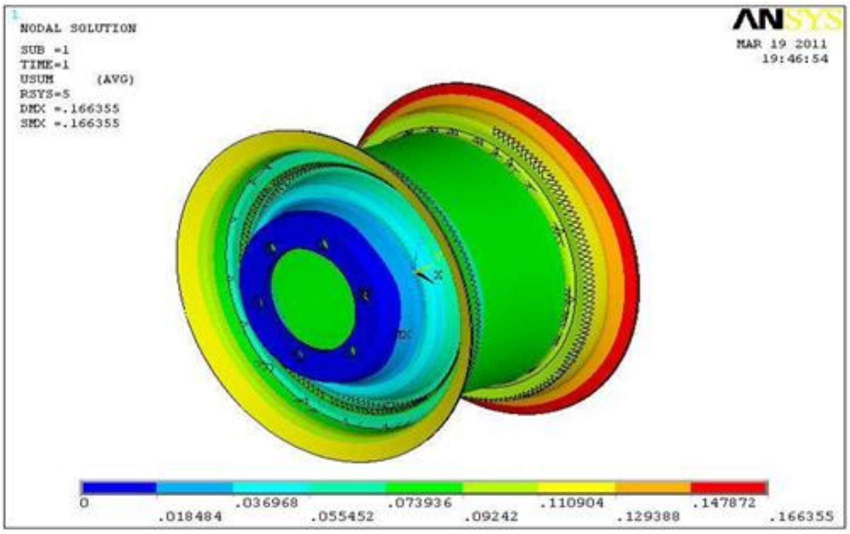


Fig.4 AN EXAMPLE OF ANALTSIS OF RIM

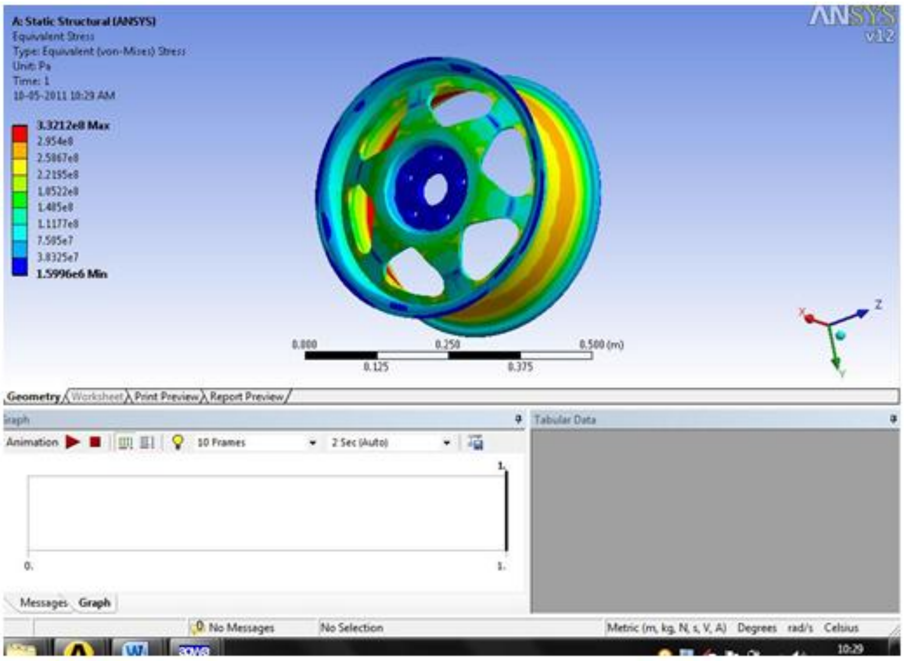


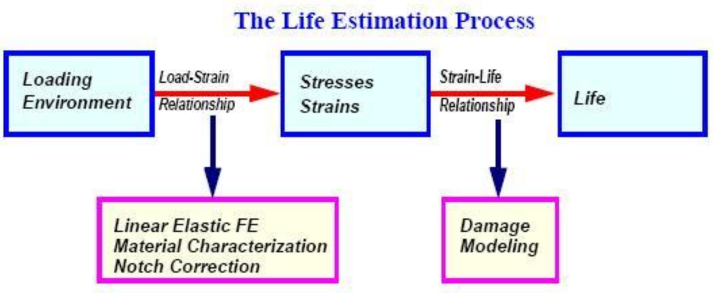
Fig.5.AN EXAMPLE OF FATIGUE ANALYSIS OF RIM

5. Life Estimation Process

The life estimation process really centers around two major relationships.

5.1 The first relation is that of the loading environment to the stresses and strains in the component or model. This load-strain or load-stress relation is determined using finite element modeling and running linear elastic FE analysis. It is dependent on the characterization of the material properties and in some instances requires that a notch correction procedure take place. For the purposes of this discussion a notch correction is simply a way to compensate for plasticity from a linear FE analysis.

5.2 The second relation is that of the stresses or stains to the life of the component or model. This is accomplished by using damage modeling. Each fatigue life method has its own techniques to determine and sum damage which shall be explained as you progress through the example problems.



5.3 The fatigue analysis is carried out in MSC fatigue tool .The von-misses stresses from ANSYS(.rst file format) is imported to the MSCfatigue and find the number of cycles to failures of crankshaft for forged steel and sintered aluminium

VI. CONCLUSIONS

The FEM analysis of the wheel under various loads showed the exact location of the failure of the rim. The laboratory test showed the failure of the aluminum wheel by crack initiation, the present analysis confirmed the flaw of the design of the wheel.

REFERENCES

[1] C. BagcB, J.C. Wong, A.P. Singh, Irregular polar elements for stress analysis of complex systems having contact surfaces rotating members and interference _ts, Comput. Struct. 14 (3–4) (1981) 289–318.

[2] D.R.J. Owen, J.A. Figuearias, Anisotropic elasto-plastic _nite element analysis of thick and thin plates and shells, Internat. J. Numer. Methods Eng. 19 (1983) 541–566.

[3] Y.Q. Guo, et al., Recent developments on the analysis and optimum design of sheet metal forming parts using a simpli_ed inverse approach, Comput. Struct. 78 (2000) 133–148. 307-320.

[4] P.R. Raju, B. Satyanarayana, K. Ramji, K. Suresh Babu. Engineering Failure Analysis. 2007, 14, 791–800.

[5] G. Fischer, V. Grubisic. Cast Aluminum Wheels And Buses – Testing and Evaluation. SAE Technical Paper Series 841705, 1985, 6.1051–6.1062.

[6] G.Laslaz and P.Laty, "Gas Porosity and Metal Cleanliness in Aluminum Casting Alloys", in

[7] AFS Transactions, 40(1991), 83-90

[8] W.Mendenhall and T.Sincich, Statistics For Engineering and The Sciences, (Mac Millan Publishing, New York, 1992), p.476.

[9] J.G. Conley, J.Huang, J.Asada and K.Akiba, "Modeling the effects of cooling rate, hydrogen content, grain modifier on microporosity formation in Al A356 alloys", in Materials Science& Engineering, A285 (2000), 49-55

[10] R.C.Atwood, S.Sridhar, W. Zhang and P.D. Lee, "Diffusion-Controlled Growth Of Hydrogen Pores In Aluminium-Silicon Castings: In Situ Observation and Modelling", in ActaMaterialia, 48(2000), 405-417

[11] J.P.Anson and J.E.Gruzleski, "The Quantitative Discrimination Between Shrinkage and Gas Microporosity In Cast Aluminum Alloys Using Spatial Data Analysis", in MaterialsCharacterization, 335(1999), 319