Impact Analysis of Car Alloy Wheel Rim using Finite Element Analysis

Sahil Bandral
M. Tech. (Machine Design),
Department of Mechanical Engineering,
NIT, Kurukshetra

Satnam Singh
Assistant Professor, Department of Mechanical Engineering, NIT,
Kurukshetra

***Abstract*:- The wheel rim is one of the most critical components of the vehicle. Therefore, a number of tests have to be performed on it so that, it could meet the safety requirement. But physical testing and the inspection of the wheels during their development is very costly and time-consuming. So, finite element analysis (FEA) has developed as a major tool for their analysis. In this paper, we studied the simulation of 90-degree impact test for a cast aluminium wheel using 3-D explicit finite element analysis. Modelling of four different wheel rim models having the different number of spokes along with striker had been done using CATIA P3V5-6R2017 software and analysis was done using ANSYS15.0. The analysis results are presented as a function of time and the maximum value of equivalent Von Mises stress on the wheel rim is calculated for each model and compared. A typical alloy wheel of Swift Dzire car is chosen for the study.**

Keywords- Explicit dynamic analysis, Aluminium alloy wheel, CATIA, ANSYS

1. INTRODUCTION

Each component of the vehicle has its own relative importance but the wheel is one of the most crucial components of the vehicle which can't be ignored. The number of static and dynamic loads acts on the wheel while the vehicle is in operation. Therefore designing and manufacturing of wheel should be done carefully so that it could be used safely and economically. Style, weight, manufacturability and performance are major technical concerns while designing a vehicle [4]. The wheels are generally of two types, pressed steel wheels and cast alloy wheels. Pressed steel are generally having more weight than cast alloy wheels. It leads to increase in inertia of the vehicle which in turn affects the braking system and also unnecessary increases the fuel consumption. Nowadays, wheel rims made of alloys having stylish appearance are commonly used. Alloy wheels are generally made of aluminium or magnesium alloys. The problem with magnesium alloys is that they are very much prone to fire that's why they are rarely used [5]. Aluminium alloy wheels are having excellent features of lightness, extra corrosive resistance etc that's why they generally preferred [7].

The numbers of tests have to be performed on the wheel rim before it can be used in the original working conditions. These tests include rotary bending test, radial fatigue test and impact test. Impact tests are of two types- 13-degree impact test and 90-degree impact test. In case of 13-degree impact test, the wheel is mounted with its axis at an angle of 13 degrees to the vertical and in case of 90-degree impact test; the wheel is mounted with its axis at an angle of 90 degrees to the vertical.

It is very expensive to do impact test during development and testing phase of the wheel. So FEA has been developed as a major tool to overcome this problem. In this paper, the work involved was the simulation of 90-degree impact test with the absence of inflated tyre which is very helpful in reducing both time and cost during manufacturing and design of wheel rims. Four rims having the different number of spokes along with striker were modelled in CATIA P3V5-6R2017 and analysis is done in ANSYS15.0 [12-13]. The objective was to find the maximum value von mises stress in rim under the dynamic conditions when the striker strikes the wheel rim for each model and results were then compared to get the optimum design

Chia lung chang et al.(2009) compare the FEA results of impact test with experimental results and concluded that during the impact test the total plastic approach can be used to predict the wheel fracture[3]. Muhammad Cerit(2010) simulated the impact test to investigate the stress and displacement distribution during wheel impact test. From the results, it was concluded that impact test was the highly non-linear phenomenon and the maximum stress was in lug region of the wheel [4].

Satoshi Ishikawa et al.(2014) performed the 13-degree and 90-degree impact test using finite element analysis and then the analytical results were compared with experimental results. It was concluded that experimental results were in good agreement with analytical results [9]. H.Zainuddi et al.(2016) performed the dynamic impact test on a wheel rim using finite element analysis. The author performed both 13- degree and 90-degree impact test on rims made of different materials such as aluminium, magnesium and stainless steel in the absence of inflated tyre and impact energy absorbed in each case was correlated. It was concluded that 90-degree impact test was superior to the 13-degree impact test and the correlation of both the impact test showed that aluminium had the best percentage of deviation [10].

2. MATERIAL PROPERTIES

The wheel is one of the most critical components of the vehicle. It should be light in weight and its strength should be high so as to withstand rough loads and harsh environment. Moreover, its manufacturing cost should be low without affecting the safety of the passenger. The typical chemical composition of the material for aluminium alloy and mechanical properties of the aluminium alloy are shown below:-

Table2.1: Chemical Composition of aluminium Alloy

Copper	Manganese	Silicon	Iron	Zinc	Aluminium	Others
0.25 %	0.35%	6.5 to7.5 %	0.6 %	0.35 %	90 to 92%	0.05 %

Table 2.2: The mechanical properties of aluminium alloy

Density (kg/m^3)	Yield Strength (MPa)	Ultimate Strength (MPa)	Young's Modulus (GPa)	Poisson's Ratio
2770	280	310	71	0.33

The striker used in impact analysis was modelled as an elastic material using structural steel material properties. Mechanical properties of structural steel are shown in table 2.3:-

Table 2.3: The mechanical properties of structural steel

Density (kg/m^3)	Yield Strength (MPa)	Ultimate Strength (MPa)	Young's Modulus (GPa)	Poisson's Ratio
7800	250	480	200	0.3

3. MODELLING AND EXPLICIT FINITE ELEMENT ANALYSIS

The modelling of the wheel was carried out in CATIA, and then it was converted into STP format. It was then imported to ANSYS and its finite element analysis was done in ANSYS. The diameter of the rim is taken as 390mm and width of the rim is taken as 165 mm. The dimensions of striker used are 200mm×200mm×200mm [12-13].

The loading and boundary conditions were set up similar to those in laboratory test and were suggested by the society of automobile engineers (SAE). The free dropping of the striker should be above 101.6 mm as suggested by SAE[9]. So we assume the free dropping distance as 150 mm and the initial velocity which was assumed the free dropping of 150mm was applied to the striker under the explicit dynamic conditions. Initial speed and initial mass were selected as 1700mm/sec and 70 kg respectively. The option of CONTACT PAIR with friction conditions was invoked between wheel rim and striker. The striker was constrained to move only in the Z direction and the wheel is assumed to be fixed from the central hub position.

It is important to determine time step in the explicit finite element analysis and the automatic incremental time step in software was used. The end time selected was 1000μs. Aluminium being the ductile material will yield when the stress reaches the yield point. So, the equivalent stresses are considered to determine whether the material yields or not.

Experimentally it is not easy to find the wheel impact response during contact due to very small impact time. However, the numerical simulation may be capable of providing direct observations during the entire impact test. Explicit finite element analysis can record the results for the time interval of microseconds.

4. RESULTS AND DISCUSSION

The initial conditions, boundary conditions and the loading conditions were kept same in all the four cases. The finite element analysis results obtained for Von Mises stresses from these loading conditions shown in fig.1 to fig.4

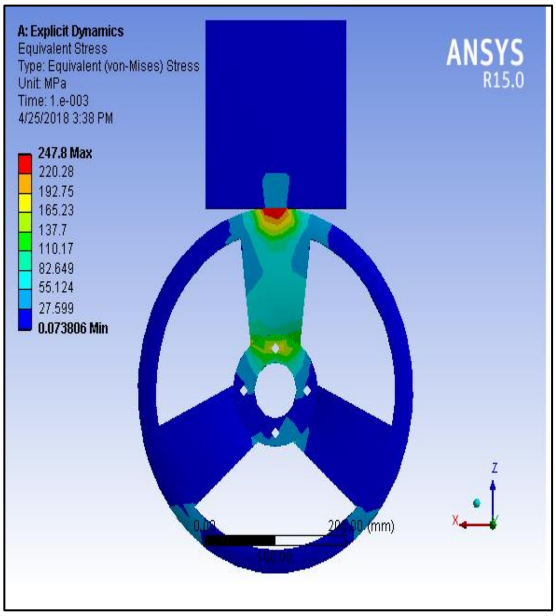

Fig. 1 Stress variation for 3 spoke wheel

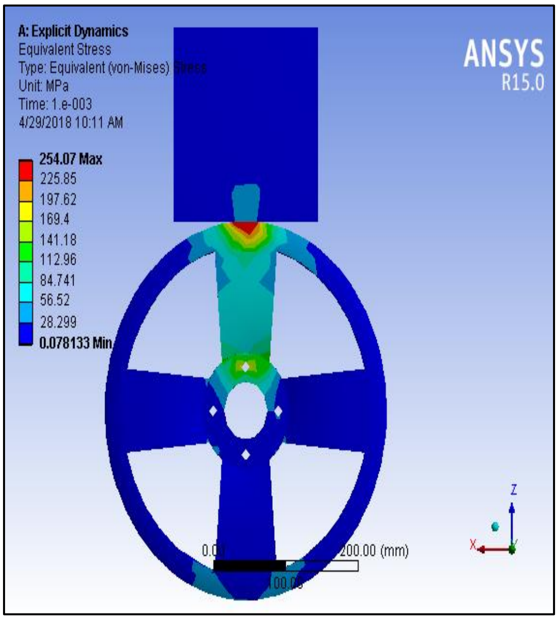

Fig. 2 Stress variation for 4 spoke wheel

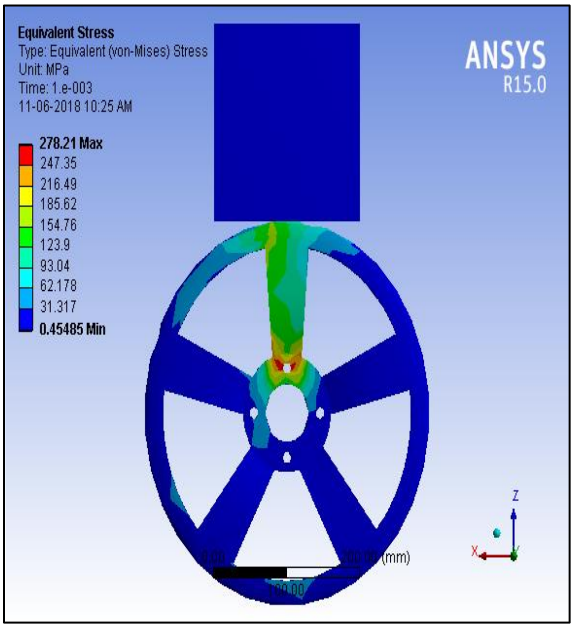

Fig. 3 Stress variation for 5 spoke wheel

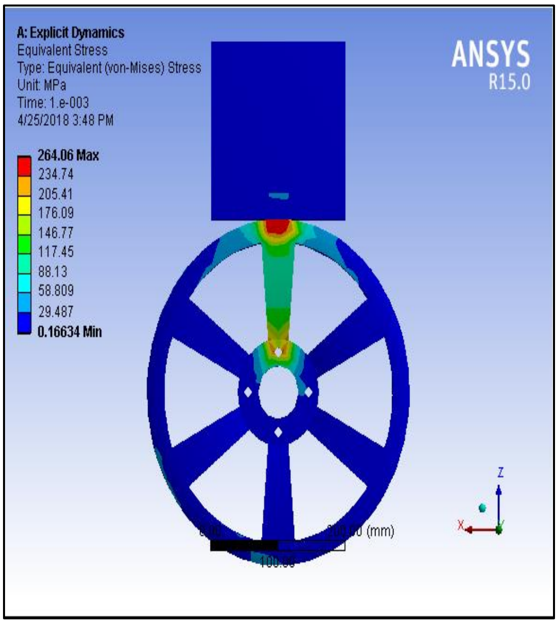

Fig. 4 Stress variation for 6 spoke wheel

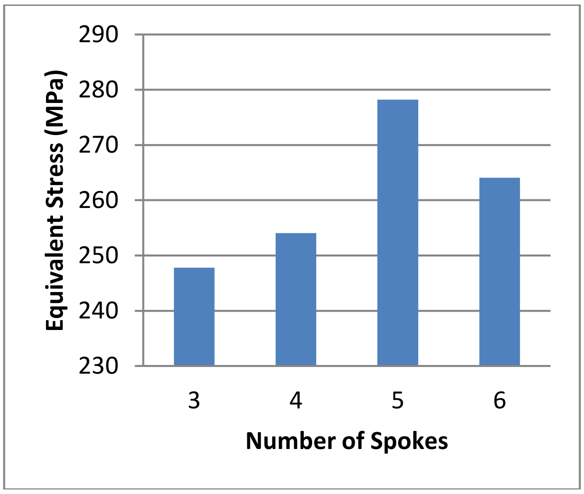

Fig. 5 Equivalent stress Vs Number of spokes

It can be observed that the maximum stress location is at the point where striker hits the rim. In the four design of wheel as modeled, the stresses developed are highest in 5 spoke wheel and lowest in 3 spoke wheel.

Fig. 6 shows the variation of stress in wheel with time from 0s to 1000 μs

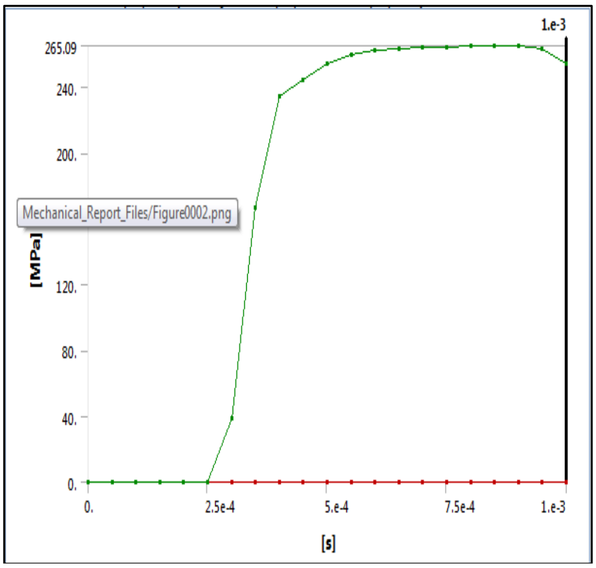

Fig. 6 Variation of stress with time

From the above fig.6 it is observed that stress remains zero up to 250μs from the start time of analysis that means there was no impact till 250μs and after 250μs stress starts increasing and is maximum near the end of the analysis.

5. CONCLUSION

The dynamic response of a wheel rim during impact test is a highly non-linear phenomenon. In this analysis, impact test was performed on four wheel rims having the different number of spokes. The maximum equivalent stress comes out to be 247.8 MPa in 3 spoke wheel. If the numbers of spokes are increased, the maximum equivalent stress starts increasing and the value of maximum equivalent stress comes out to be approximately 278 MPa for 5 spoke wheel and 264 for 6 spoke wheel. So it is concluded that 3 spoke wheel is safest of all if the results of impact test are to be considered.

From the figures 1, 2, 3 and 4, we can observe that impact takes place on that part of the periphery of the rim which is above the spoke. The width and thickness of spokes in case of 3 spoke wheel is larger than other configurations, therefore stress generation is less in 3 spoke wheel in comparison to other configurations. The stress generation in 5 spoke wheel is more than 6 spoke wheel, this is due to the reason that in case of 6 spoke wheel, a spoke is directly in the line of the impact on the lower side of the hub which resisted the stress generation.

6. REFERENCES

[1] W. Altenhof, W. Ames,Strain rate effect for aluminium and magnesium alloys in finite element simulations of steering wheel armature impact test", Fatique Fract Engng Mater Struct (2002)25, 1149-1156
[2] J. Stearns, T.S. Srivatsan, A. Prakash, P.C. Lam,"Modeling the mechanical response of an aluminum alloy automotive rim, Materials Science and Engineering A366 (2004), pp. 262-268
[3] Chia-Lung Chang, Shao-Huei Yang Simulation of wheel impact test using finite element method" Engineering Failure Analysis 16 (2009), pp. 1711–1719
[4] Muhammad Cerit, Numerical simulation of dynamic side impact test for an aluminium alloy wheel, Scientific Research and Essays Vol. 5(18), pp. 2694-2701, 2010
[5] S Vikranth Deepak, C Naresh1 and Syed Altaf Hussain,"Modelling and analysis of alloy wheel for four wheeler vehicle", ISSN 2278 – 0149 Vol. 1, No. 3, October 2012
[6] P. Meghashyam, S. Girivardhan Naidu and N. Sayed Baba, Design and Analysis of Wheel Rim using CATIA & ANSYS, International Journal of Application or Innovation in Engineering & Management (IJAIEM). Volume 2, Issue 8, August 2013 ISSN 2319 – 4847
[7] McMahon. M, Trade of motor mechanics: Wheel and Types, SOLAS 2013 Module 7. 2013
[8] T. Siva Prasad,T. Krishnaiah, J. Md. Iliyas, M.Jayapal Reddy, A Review on Modeling and Analysis of Car Wheel Rim using CATIA & ANSYS, International Journal of Innovative Science and Modern Engineering (IJISME) ISSN: 2319-6386, Volume-2, Issue-6, May 2014
[9] Satoshi ISHIKAWA, Yasuki SAKAI, Norimasa NOSAKA, Application of Impact Analysis for Aluminum Wheel with Inflated Tire, Simulia Community Conference (2014)
[10] Chauhan MR, Kotwal G, Majge, A. Numerical simulation of tire and wheel assembly impact test using finite element method. SAE Technical Paper; 2015.
[11] H. Zainuddin, M. B. Ali, N. B. Muhammad Said and K. A Zakaria1, Correlation between vertical wheel impact energy with lateral wheel impact energy: A finite element analysis approach, International Journal of Automotive and Mechanical Engineering (IJAME) ISSN: 2229-8649 (Print); ISSN: 2180-1606 (Online); Volume 13, Issue 3 pp. 3574 - 3583, December 2016
[12] ANSYS Theory Reference. Ansys Release 15.0, ANSYS Inc.
[13] CATIA V5R17. For engineers and designers by Prof.Sham Tickoo