International Journal of Future Generation Communication and Networking
Vol. 13, No. 3s, (2020), pp. 1268–1276

FEA of Aluminium Based Alloyed Wheel Rim of Passenger Vehicle

Awanit Kumar*[1], S K Panigrahi[2],

[1]*PG Student ,Mechanical Engineering Department, Defence Institute of Advanced Technology, Pune, India*

[2]*Professor, Mechanical Engineering Department, Defence Institute of Advanced Technology,Pune,India*

Abstract

In this paper, Finite element analysis (FEA) of a passenger vehicle wheel rim made of Aluminium alloy has been carried out to simulate the load conditions and boundary conditions of 'Dynamic Radial Fatigue Test'. The aim is to find how able the wheel structure is to take the above mentioned loads and hence to predict the critical locations having maximum stress and deformation respectively and whether the structure is failing or safe under these loads. Static structural analysis has been done with two different wheel disc thicknesses to see the effect on stresses developed. The paper presents every step involved in FEA in detail and stress and deformation distribution throughout the wheel structure has been shown in result section through different plots. The solid modelling of the wheel rim is carried out in SOLIDWORKS 2016 and finite element static structural analysis in ANSYS Workbench 16.0 software.

Keywords*— Finite element analysis , Wheel Rim, Dynamic radial fatigue test, Critical locations, Stress distribution.*

I. INTRODUCTION

A wheel rim, being a safety component of a passenger vehicle, it becomes very essential to design it with utmost care so that it can be able to withstand any possible load condition on road. During last 20 years, design or structural analysis based on simulating various wheel tests such as Dynamic radial fatigue test, Dynamic cornering fatigue test & Impact test has become more popular and relevant. This is due to the fact that every newly developed wheel has to pass all these tests legally before coming in to the production. Hence, if in the design or analysis phase itself the loading conditions pertaining to these tests can be taken care of, the probability of failure of realised prototype in these tests can be minimised. The structural integrity or strength of the wheel can be determined by these test methods. Numerous studies have been made on simulating each of these wheel tests and finding stress distribution using finite element analysis

J Sterns et al. [2] used finite element method for analysing stress and displacement distributions in an aluminium alloy automotive rim-tyre combination unit subject to the conjoint influence of inflation pressure and radial load. P. Ramamurty Raju et al. [3] carried out finite element analysis to generate S-N curve for aluminium alloy (Al) A356.2-T6 and estimation of fatigue life of a car wheel under radial fatigue load. Also, M.M. Topac et al. [1] conducted Dynamic radial fatigue tests of a newly designed heavy commercial vehicle steel wheel till the failure and also validated the test results through linear static finite element analysis.

Xiaofeng Wang and Xiaoge Zhang [4] simulated the dynamic cornering fatigue test of a steel car wheel rim by using the linear transient dynamic finite element analysis and the local strain approach. Rakesh B. Thakare et al. [20] described a model for prediction of fatigue failures of steel disc wheels, and compares the prediction using simulation results with experimental test data.

Chia-Lung Chang and Shao-Huei Yang [6] presented a study in which nonlinear dynamic finite element was used to simulate the SAE wheel impact test. Muhammet Cerit [12] performed simulation of impact test for a cast aluminium alloy wheel using 3D explicit finite element analysis.

In view of the above discussion, this paper has an objective to carry out Finite Element Analysis of a 12 inch wheel rim (of wheel disc type) made of Aluminium alloy to simulate loading and boundary conditions used in dynamic radial fatigue wheel test and to find out the maximum von misses stress and total deformation and to predict the factor of safety.

In Dynamic radial fatigue wheel test, the wheel assembled with tyre is positioned against a rotating drum. Radial test loads are applied to evaluate the stresses in the tyre and rim. Due to this applied load, a contact pressure is developed between tyre and drum. A typical schematic test setup of the dynamic radial fatigue test is shown in the Figure1.

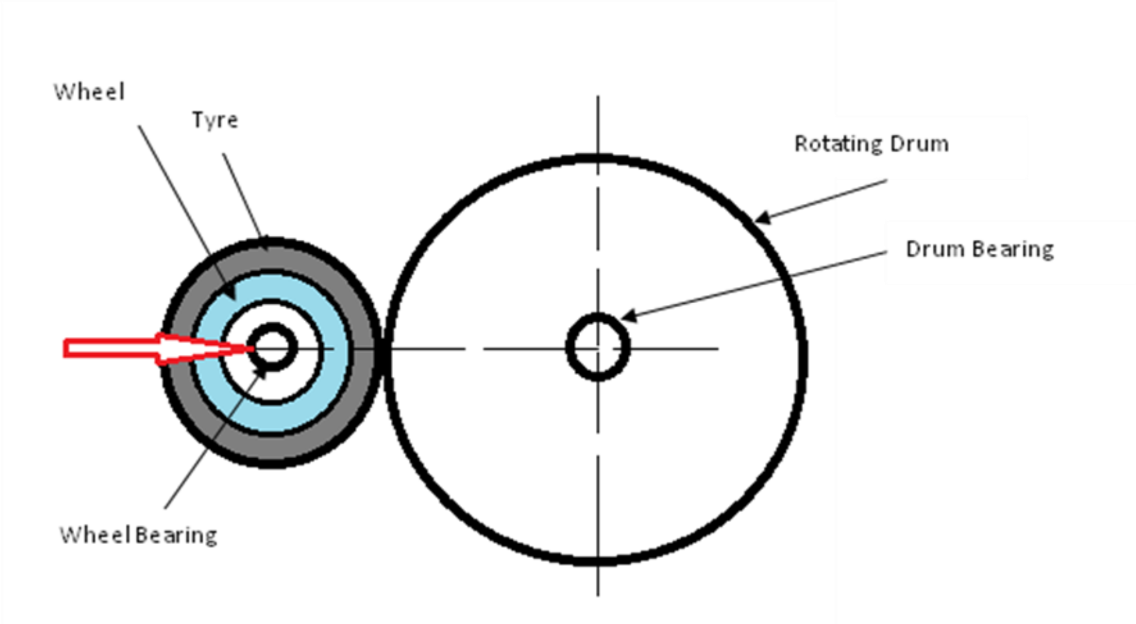

Fig. 1 Schematic test setup of the Dynamic Radial Fatigue Test

II. SOLID MODELLING AND FINITE ELEMENT ANALYSIS OF WHEEL RIM

Some important dimensional information of the wheel rim are given in Table 1.

TABLE 1
DIMENSIONAL INFORMATION OF WHEEL RIM

Rim Dia	Flange Height	Rim width	Rim size designation	Tyre size designation
12 inch	0.68 inch	4 inch	4J	145/80 R12

A. *Solid modelling*

A full scale solid model of a 12 inch car wheel rim was prepared in Soldworks-16 software with wheel disc thickness as 4 mm and rim thickness 3.6 mm as shown in Figure 2. The wheel structure has two subparts – Rim and Wheel disc.

ISSN: 2233-7857 IJFGCN

Fig. 2 Solid model of a 12 inch car wheel rim

This solid model was imported in .STEP format in Design Modeller platform of ANSYS Workbench.

B. Material Specification

Aluminium Alloy A356.2 was used as material for analysis of wheel structure and its properties specified in Engineering Data section of static structural module of ANSYS Workbench are presented in Table 2.

TABLE 2
PROPERTIES OF ALUMINIUM ALLOY A356.2

SR. NO.	**PROPERTY**	**VALUES**
1.	Density (g / cm ^{3}) , ρ	2.7
2.	Young's Modulus of Elasticity (GPa) , E	72.4
3.	Poisson's Ratio , μ	0.33
4.	Yield Strength (MPa) , σ_y	230
5.	Ultimate strength (MPa) , σ_u	250

C. Meshing

This is one of the most important pre-processing activities in finite element analysis which should be done with utmost care and correctness. A quality meshing ensures proper discritisation of the problem domain which can offer the converging approximate solution while solving by a solver if the rest of the things have been done correctly. After importing the solid model of wheel rim, it was opened in Mechanical platform in 'Model' section of Static structural module. While trying for auto meshing, it was resulting in to Tetrahedral elements mainly, which is not efficient meshing as number of elements and nodes are much larger. Hexahedral dominant or Brick meshing which is known as much efficient meshing , was not becoming possible due to very curvy and complex nature of the wheel rim structure. Therefore, to make it possible it was necessary to divide the whole structure in to smaller sectors so that parts may become ready for accepting Hexa meshing.

Hence, by creating new planes at 30° intervals and by using these planes and 'create slice' tool in Design Modeller platform, both rim and wheel disc parts of solid model were sliced into 12 sectors. After that by using 'Form new part' option, a new part was created using these 24 sectors. Solid model after slicing operation is shown in Figure 3.

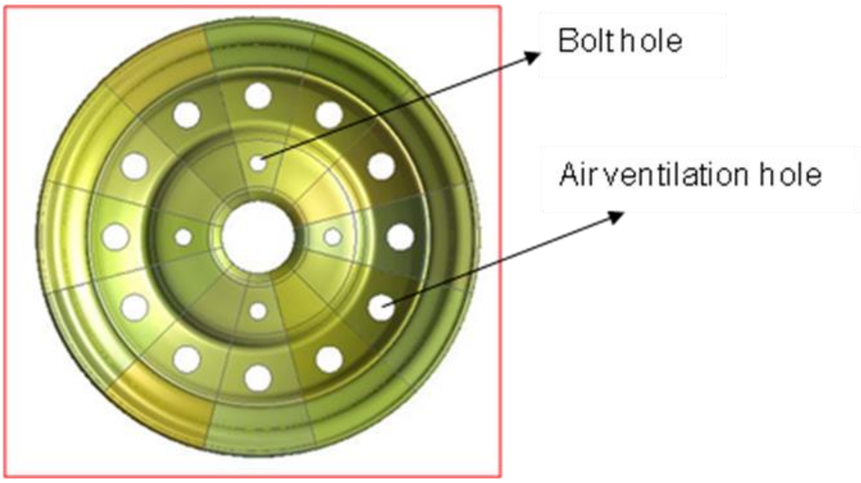

Fig. 3 Solid model after slicing operation

One of the 12 sectors of rim and wheel disc are shown in figures 4 and 5 respectively.

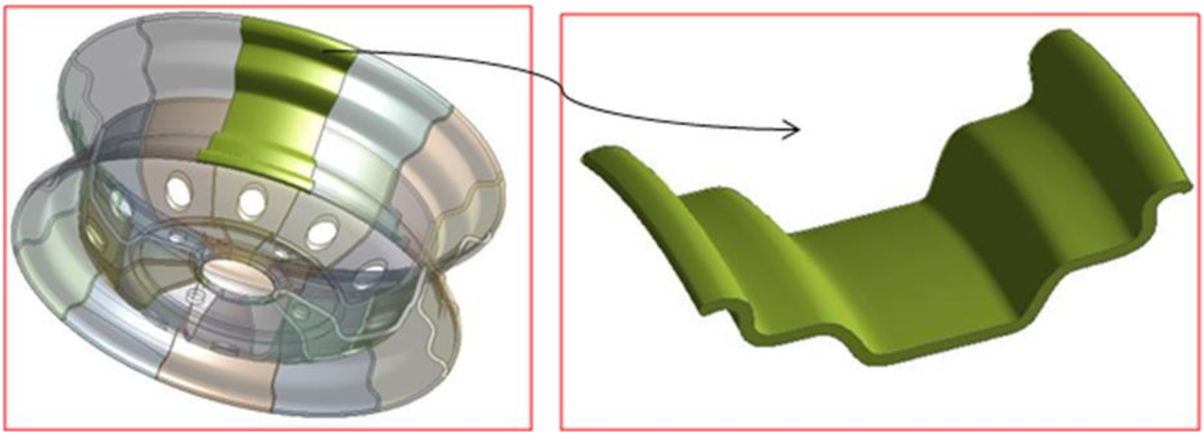

Fig. 4 One of the 12 sectors of Rim

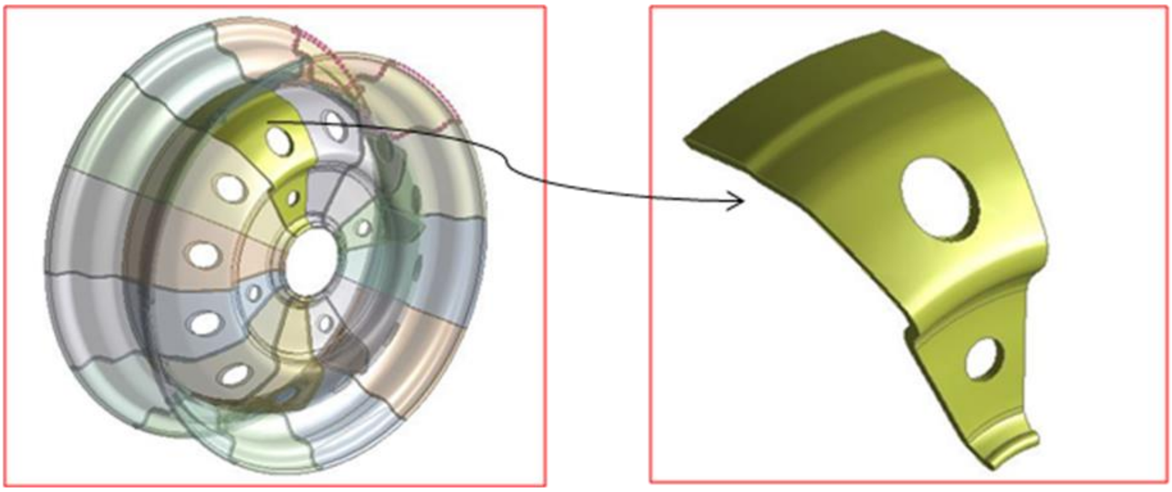

Fig. 5 One of the 12 sectors of Wheel disc

After making the solid model of car wheel rim suitable for Hexa dominant meshing, various options for meshing were chosen properly and by using 'multizone' method and taking element size of 2 mm, all 24 sectors were meshed with mostly "Hex20" elements and some small regions with

“Wed15” elements. With this element size of 2 mm, the mesh was quite fine which was checked with convergence test of meshing.
Number of elements and nodes were 2,18,546 and 10,82,336
Final meshing with element size of 2 mm is shown in Figure 6.

Fig. 6 Meshing of whole wheel structure

D. Boundary conditions and loading setup

As this paper simulates loading and boundary conditions used in radial fatigue wheel test, hence, the boundary conditions and loadings on solid model of 12 inch car wheel rim has been employed similar as M.M. Topac et. al. [1] have applied.

1) Boundary conditions: With the help of “insert Fixed Support” tool command, fixed support boundary conditions were applied at the nut - wheel disc and wheel disc - Brake drum contact surfaces as shown in Figure 7.

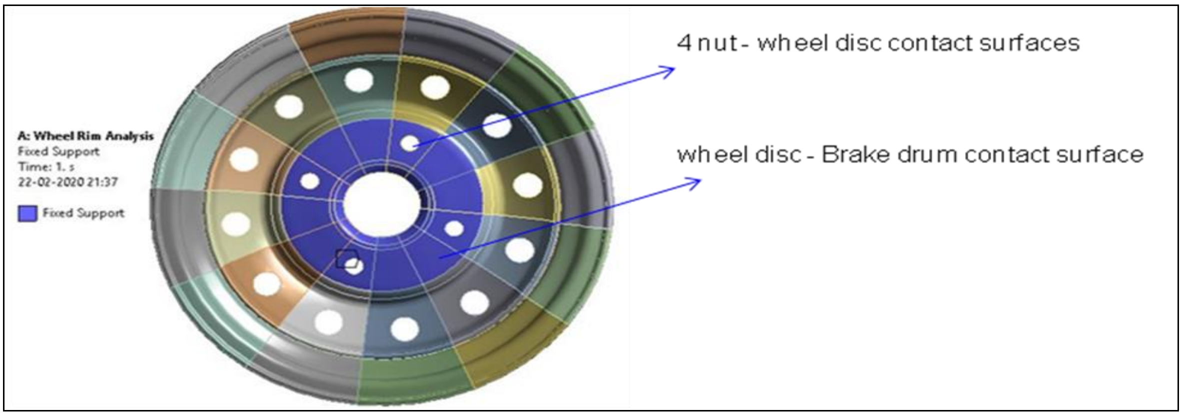

Fig. 7 Fixed Support – 4 nut –wheel disc contact surfaces and wheel disc - Brake drum contact surfaces

2) Loading: Along with the inflation pressure on outer rim surface (in contact with tyre), a radial load ‘F’ is also applied in order to simulate radial loading in actual radial fatigue wheel test.

Since the selected rim falls under tyre size designation 145/80 R12 and rim contour size 4J, maximum cold inflation pressure as per the standard AIS- 044 (Part 2) to be applied on outer circumferential surface of rim flange is equal to 240 kPa (or 0.24 MPa) as shown in Figure 8.
The radial load F is determined as $F = F_v \cdot k$,
Where, F_v is nominal design load of the wheel. The accelerated test load factor k is given as 2.2 by EUWA (Association of European Wheel Manufacturers) and SAE (Society of Automotive Engineers)
For the wheel rim considered in this project, Maximum nominal load as per the standard AIS- 044 (Part 2) is 375 kg
Hence, Radial load, F = 375 x 2.2 kgf = 825 kgf =825 x 9.81 N = 8085 N

ISSN: 2233-7857 IJFGCN

Therefore, radial load F = 8100 N is considered for analysis.

Now, on the basis of angular location of application of radial load, only two different possibilities are there which is illustrated in Figure 9.

Fig. 8 Pressure applied on outer circumferential surface of rim

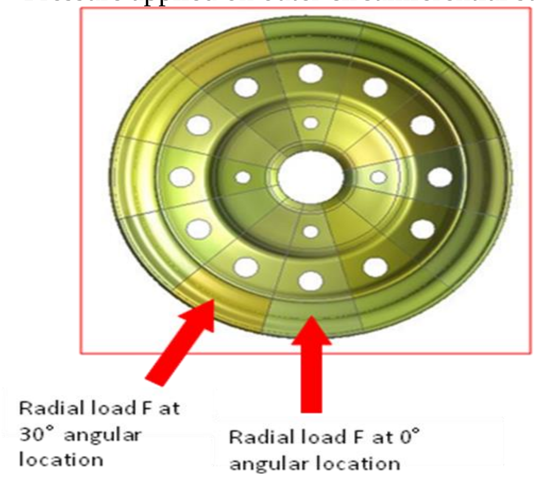

Fig. 9 Angular locations of application of radial load F

Looking at the symmetrical positioning of bolt holes and air ventilation holes on wheel disc, it can be clearly seen that if the radial load F is applied on any sector (out of 12 sectors) other than the two shown in above figure, it will have same configuration as one of these two. Hence, analysis for these two configurations will take care of full rotation of the wheel rim during radial fatigue wheel test.

Hence, following two cases will be considered for analysis:

- case – 1, in which radial load F is applied at 0° angular location
- case – 2, in which radial load F is applied at 30° angular location

3) Scheme of applying radial load F: The radial load F was exerted on tyre bead seat – rim contacts i.e. at two symmetrical locations as shown in figure. Magnitude of force at each of these two locations is F/2 i.e. 4050 N. In the Figure 10, these forces are shown with the names 'Force' and 'Force2' respectively.

ISSN: 2233-7857 IJFGCN

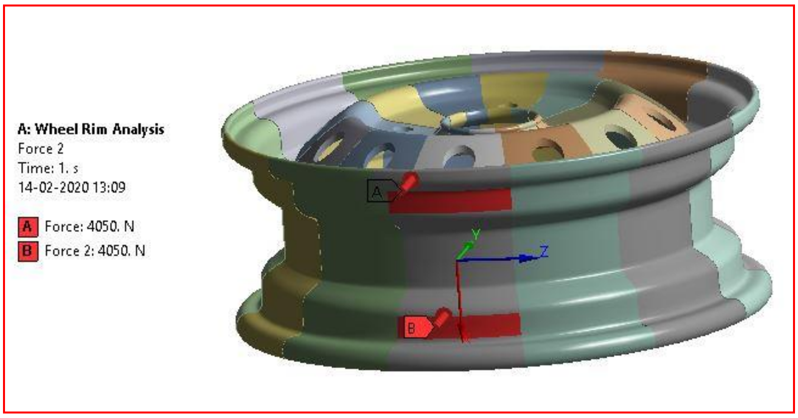

Fig. 10 Radial force F equally distributed as ‘Force’ and ‘Force2’ at two locations

4) Solution: After having all the pre-processing activities completed, finite element model was solved and solutions in terms of von misses stress and total deformation were obtained.

Solutions were obtained for following two load cases

- Load case – 1 (Inflation Pressure + Radial load load F applied at 0° angular location)
- Load case – 2 (Inflation Pressure + Radial load load F applied at 30° angular location)

III. RESULTS, CONCLUSIONS AND FUTURE SCOPE

Analysis were conducted for two different wheel disc thicknesses 4 mm and 1.8 mm keeping outer rim thickness same as 3.6 mm.

A. Results

Results obtained through Finite Element Analysis in above mentioned load cases are summarised in Table 3.

TABLE 3
SUMMARY OF RESULT

From above Table, we can clearly see that load case – 1 is more critical than load case-2 because maximum stress and maximum deformation are on higher side in this case.

Stress and deformation plots for wheel disc thickness 4 mm in load case-1 are shown in Figure 11 and for wheel disc thickness 1.8 mm in load case-1 are shown in Figure 12.

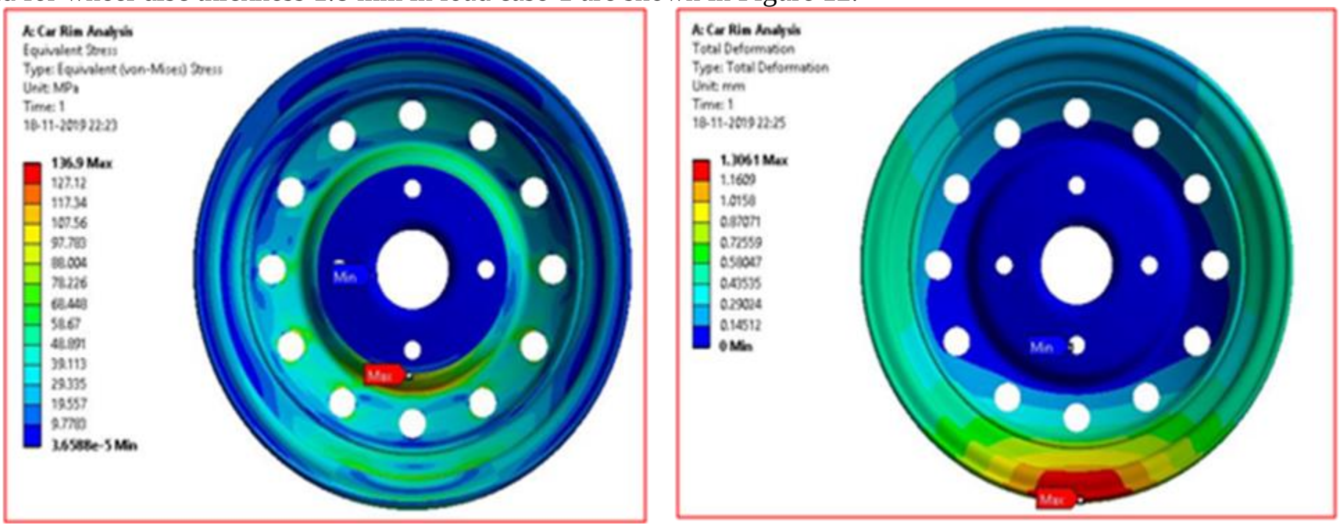

Fig. 11 Von-misses Stress (MPa) and Total deformation (mm) for 4 mm thick wheel disc in load case-1

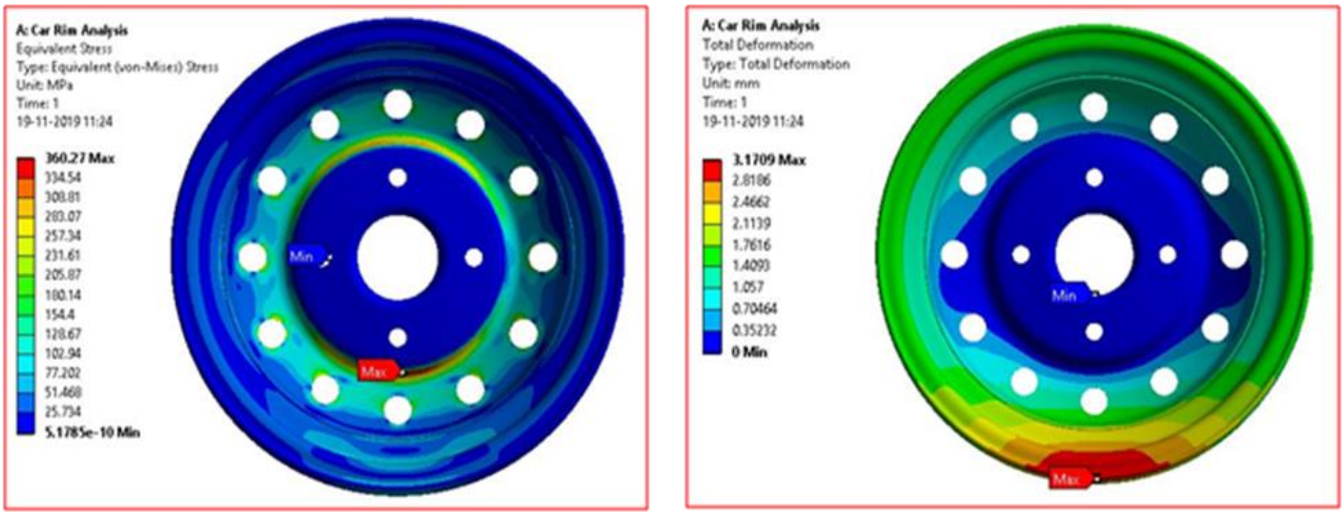

Fig. 12 Von-misses Stress (MPa) and Total deformation (mm) for 1.8 mm thick wheel disc in load case-1

B. *Conclusion & Future scope*

Yield strength of the material used for analysis i.e. Aluminium Alloy A356.2 , is 230 MPa. For wheel disc thickness 1.8 mm, the maximum Von-Misses Stress in load case -1 is 360.27 MPa which is higher than yield strength. Hence, in this case the rim structure is failing. But, for Wheel Disc thickness 3 mm, the Maximum Von-Misses Stress in load case -2a is 136.9 MPa which is well within the Yield strength and Factor of Safety is 1.68. Also, Total deformation is 1.3 mm which is quite less and acceptable.

By increasing the thickness of wheel disc and outer rim further, factor of safety can be increased and hence higher Fatigue life of wheel rim can be achieved in dynamic radial fatigue test.

The wheel rim can further be analysed against load conditions in Dynamic cornering fatigue test & Impact test and then the design of wheel rim can be finalised keeping in view the results obtained in all the three cases.

ACKNOWLEDGMENT

The authors gratefully acknowledge their colleagues in Defence Institute of Advanced Technology, Pune for their support and encouragement throughout this research work.

REFERENCES

[1] M.M. Topaç, S. Ercan, N.S. Kuralay, “Fatigue life prediction of a heavy vehicle steel wheel under radial loads by using finite element analysis” Engineering Failure Analysis 20 (2012) 67-79.
[2] Stearns J, Srivatsan TS, Prakash A, Lam PC, “Modeling the mechanical response of an aluminum alloy automotive rim”. Mater Sci Eng, A 2004;366(2):262–8.
[3] Raju PR, Satyanarayana B, Ramji K, Babu KS, “ Evaluation of fatigue life of aluminum alloy wheels under radial loads”. Engineering Failure Analysis 2007;14(5):791–800.
[4] Xiaofeng Wang, Xiaoge Zhang, Simulation of dynamic cornering fatigue test of a steel passenger car wheel. International Journal of Fatigue 2010;32:434–42.
[5] Liangmo Wang , Yufa Chen, Chenzhi Wang, Qingzheng Wang, “Fatigue life analysis of aluminum wheels by simulation of rotary fatigue test” Journal of Mechanical Engineering Volume 1 Pages 31-39(2011)
[6] Chia-Lung Chang, Shao-Huei Yang. “Simulation of wheel impact test using finite element method” Engineering Failure Analysis 16(2009) 1711–1719.
[7] Grubisic V, Fischer G. Procedure for optimal lightweight design and durability testing of wheels. International Journal of Vehicle Design1984;5:659–71.
[8] Carboni M, Beretta S, Finzi A. Defects and in-service fatigue life of truck wheels. Engineering Failure Analysis 2003;10:45–57.

[9] Firat M, Kozan R, Ozsoy M, Mete OH. Numerical modelling and simulation of wheel radial fatigue tests. Engineering Failure Analysis 2009;16:1533–41.
[10] B Raghupathi et al. "Design and analysis of car wheel rim using fem technique" journal International Journal of Computer Science information and Engg Technologies (IJCSIET) Vol. 3, Series 4, Issue. 4, Dec. 2014
[11] 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.
[12] Muhammet Cerit. "Numerical simulation of dynamic side impact test for an aluminium alloy wheel" Scientific Research and Essays (SRE) Volume 5, ISSN 1992-2248 (Sept. 2010), Page No. 2694-2701
[13] Carvalho CP, Voorwald HJC, Lopes CE. Automotive wheels- an approach for structural analysis and fatigue life prediction. SAE tecnical papers. SAE 2001-01-4053.
[14] Hsu YL, Wang SG, Liu TC. Prediction of fatigue failure of aluminum disc wheel using the failure probability contour based on historical test data. J Chin Inst Industrial Eng 2004;21(6):551–8.
[15] Fischer G, Grubisic V. Cast aluminum wheels and buses – testing and evaluation. SAE Technical Paper Series 841705; 1985;6.1051–6.1062.
[16] Riesner M, I.'DeVries R. Finite element analysis and structural optimization of vehicle wheels. SAE technical paper, No. 830133; 1983.
[17] Karandikar H, Fuchs W. Fatigue life prediction for wheels by simulation of the rotating bending test. SAE technical paper, No. 900147; 1990.
[18] Jaspreet Singh et al. "Static analysis of alloy wheel using ANSYS15.0" International Journal of Research in Engineering and Technology (IJRET).ISSN No. 2319-1163,Volume 4, Issue 7 (July 2015)
[19] H. Akbulut, "On optimization of a car rim using finite element method" reported in the journal of Finite Elements in Analysis and Design 39 (2003) 433–443.
[20] Rakesh B. Thakare, "Stress Analysis in Automobile Wheel Rim by using cornering fatigue test under different conditions"-IJARIIE ISSN(O)-2395-4396 (2017)
[21] "Theory of Ground Vehicles", Third Edition, J. Y. Wong.
[22] "Introduction to FEM in Engineering" Fourth Edition T R Chandrupatla, Ashok D Belegundu.
[23] ANSYS Workbench help files.