American Journal of Engineering Research (AJER)
e-ISSN: 2320-0847 p-ISSN : 2320-0936
Volume-10, Issue-03, pp-95-104
www.ajer.org
Research Paper Open Access

Computer Aided Design and Simulation of Aluminium 6082 as HDV Wheel

Christopher E. Nabodi[1], Tolumoye J. Tuaweri[2], Alexander N. Okpala[3]
[1]*M.Eng. graduate, Department of Mechanical Engineering, Niger Delta University, Wilberforce Island, Bayelsa State, Nigeria*
[2]*Department of Mechanical Engineering, Nigeria Maritime University, Okerenkoko, Delta State, Nigeria*
[3]*Department of Mechanical Engineering, Niger Delta University, Wilberforce Island, Bayelsa State, Nigeria*

ABSTRACT: *Light weight for fuel efficiency is a crucial requirement in the design of automotive parts and components. Improved performance of heavy-duty vehicles resulting from lighter weight of the wheels is a promising achievement, though without comprising the engineering and safety requirements for standard wheel design. In this research, characteristics of aluminum 6082 material for a heavy-duty automobile wheel weres investigated. The dimensions of existing steel wheel were measured and used as control data. Modeling and simulation (stress, fatigue and impact) analysis was conducted using commercial Computer Aided Engineering (CAE) software. SolidWorks and ANSYS were utilized for solid modeling and finite element analysis (FEA) respectively. Performance evaluations were carried out based on SAE and EUWA (2006) outlines. Load and pressure distribution on the proposed wheel were kept at a constant tyre pressure of 4MPa on wheel surface while a radial load of 329.3KN was applied round the drop centre. The stress analysis results showed that maximum stresses occurred at the rim flange of the wheel. Fatigue analysis showed that the wheel has an infinite life 1.0 ×106 cycles. The S-N curve generated for the wheel material showed that, it has a fatigue life of about 9.0 × 10^3 cycles. Also, the wheel suffered maximum deformation on the circumference of the rim flange. Impact test conducted on the wheel to investigate its crashworthiness revealed that, while the wheel performed well for the stress and fatigue analysis with a 2MPa tyre pressure, it performed differently under a pressure of 4MPa. Besides, the impact test revealed that the wheel has poor crashworthiness with maximum deformation of 0.07m.*

KEYWORDS: *Rim material, Wheel, Stress analysis, Fatigue-test, Aluminium 6082.*

Date of Submission: 27-02-2021 Date of acceptance: 12-03-2021

I. INTRODUCTION

Wheels are among the critical components in an automobile. Their design must meet both the styling appearance and engineering functions. Also, the wheel must be durable enough to withstand rough and harsh environments. The weight of a wheel and its manufacturing cost should be minimized without compromising the safety requirements. Wheel design and development uses three main wheel tests (rotating bending test, radial fatigue test and impact test) to evaluate a prototype wheel for various fatigue and durability considerations [1]. In this work, the evaluation of aluminum 6082 wheel is carried out following the recommended material/component design standards as specified by [1] and [2]. The automobile is made up of about 15,000 separate parts be it passenger car, light or heavy duty vehicles (HDVs) [3]. These are categorized into six main systems which include; the power plant, power train, steering system, braking system, body and the suspension system. However, in most cases the wheel assembly is considered as a major sub-assembly of the suspension system due to their proximity in connection settings [3]. The wheel functions to connect the body of the vehicle and the tyre with the ground and also help to resist the strains created during turning and as well transmit the driving torque for propelling and breaking torque for retarding [4]. In other words, during service condition the wheel is subject to cyclic induced stresses caused by radial loads which are generated by various road irregularities. In the automotive industry the wheel assembly is termed ''Safety and critical components'' due to its position and function in the vehicle suspension system [5]. As such, any wheel design that fails prematurely during service would jeopardize safety of the driver, passengers and cargos. Overtime, vehicles have been built body-on-frame with most of its components made of steel material due to its effective strength and high durability, which accounts for excellent performance capabilities, though with relatively heavy weight. Recent materials innovation programs are centered on developing light weight materials for the manufacture of both light and heavy duty vehicles components [6].

A reduction in the gross weight of a vehicle by any means possible automatically means a need for reduction in the weight of the wheel since the wheel bears the gross weight of the vehicle. The need therefore to optimize the weight of existing steel wheel is crucial. However, optimization of the automobile wheel is tasking due to the varieties of design variables, and standard test conditions that any newly manufactured wheel must meet [6].

In order to ensure that all newly manufactured wheels perform up to expected standard, the society of automotive engineers [2] and the European wheel manufactures association [1] recommends three major wheel prototype durability tests. These three tests include; the rotary cornering/bending test, the dynamic radial fatigue test and impact test. The rotary cornering test simulates cornering induced loads by applying a constant rotating bending moment to the wheel. The dynamic radial fatigue test simulates the radial loading of the wheel during service by applying a radial load against a constantly rotating drum. The impact test is established to evaluate the impact damage on the wheel when it hits a curb or pavement [7]. According to Saeed (2008) [8]; ANSYS is a comprehensive general purpose finite element computer program that was first released in 1971 that contains more than 100,000 lines of code. ANSYS is capable of performing static, dynamic, heat transfer, fluid flow and electromagnetism analyses. The current version of ANSYS has undergone different modifications which include: multiple windows, a graphical user interface (GUI), pull – down menus, dialogue boxes and a tool bar. A special version of this software known as ANSYS WORKBENCH which contains the aforementioned modifications was used in this work for the FEA static stress and fatigue analysis of both the steel and aluminum wheels (Siva et al, 2014) [9], used PRO – E to model a solid spoke wheel with varying number of spokes by utilizing suitable features of the PRO – E package. A 2D profile of the circular pattern for the spokes of the wheel was created by selecting the pattern tool from the sketcher menu. A spoke profile was then drawn in a similar manner and the blend tool was utilized to define, section, direction and depth. The hub part was created through the provisions of the extrude tool from which the internal sketch option was choosen with appropriate plane where the hub profile was created with circular option. According to Satyanarayana and Sambaia (2012) [10], in their project created a 3-D mathematical model of a wheel using CATIA and the file was exported to the ANSYS workbench environment for analysis through the IGES format. Meshing was done in the ANSYS environment with 10-node tetrahedral structural solid element. The total number of nodes and elements used were 212319 and 117243 respectively. A pressure of 2.653MPa was applied on the rim and it was observed that the maximum and minimum shear stresses were 48.195 and -48.241MPa respectively at the hub. The equivalent Von-mises stresses were 163.97 and 0.038MPa respectively at the hub Ridha (1976) [11], proposed a finite element stress analysis of automotive wheels which could be applied to both the rim and to the entire wheel body. The rim cross-section was modeled by an interconnected mesh of fine triangular elements. The displacements of each element at the nodal points were calculated and then the strains and finally the stress distribution were derived. The stiffness matrix of a constant strain triangular element for axisymmetric problems was formulated alongside the modifications for non-axisymmetric problems. He concluded from the analysis that the largest principle stresses were located in the regions of sharp edges in the rim's contour. He also highlighted on the effects of increasing the width of the rim in order to reduce the stress levels Meghashyam (2013) [12]; using CATIA, selected an appropriate plane on which the 2D rim profile was drawn in vertical axis. This profile was then revolved with respect to the y – axis to obtain a 3D model of the rim. By selecting the face of the wheel the required design is drawn on the surface is removed by using POCKET operation. Circular pattern tool was utilized to draw the latter design round the entire face of the wheel through rotation. Finally, the edge fillet option side edges are made filleted for final finishing. Janardhan (2014) [13], during part of their project on radial fatigue analysis of an alloy wheel, drafted a 2D profile of the wheel using the MDT package. Same was then imported into the ANSYS modeling and FEA package through the IGES format, where the 3D model of the wheel was generated and analyzed. Emmanuel and Ebughni (2014) [14], in their project prepared a computer aided mathematical model of an alloy rim using of solidworks software. This was then imported into the ANSYS FEA package. The traditional stress – life method of fatigue analysis was employed which in this project involved the Wohler S – N diagram to deduce the corresponding fatigue life at a given stress level. It was observed that fatigue crack initiation regions on the wheel are subjected to stress concentration and that fatigue crack occurs at the most stress concentrated regions of the wheel spokes and air ventilation holes which are the critical regions of the wheel. The maximum Von – mises stress occurred at the wheel forks. The predicted failure locations are identical to the actual crack initiation regions and are consistent with other reports. Wang and Zhang (2010) [15], used the ABAQUS software to build a static load finite element model of aluminum wheels for simulating the rotary fatigue test. The equivalent stress amplitude was calculated based on the popular nominal stress method which considers the effects of mean load, size, fatigue notch, surface finish and scatter factors. The fatigue life of aluminum wheels was predicted by using the equivalent stress amplitude and aluminum alloy wheel S-N curve. The results from the aluminum wheel rotary fatigue bench test showed that the failed the test and its crack initiation was around the hub bolt hole area that agreed with the simulation. Chang and Yang (2009) [16], In their project, simulated the SAE wheel impact test, using a nonlinear dynamic finite element analysis. A striker with a specified weight and known linear dimensions was allowed to drop from a known height to make impact on a tyre-wheel assembly that was mounted at an angle of 13^{o} to the vertical plane with the edge of the striker in line with the outer radius of the rim with two attached simulated strain gages to the disc to measure the strain of the wheel during impact. They found that the dynamic response of a wheel during impact is highly nonlinear. They went further to say that nonlinear dynamic finite element with reasonable mesh size and time step can reliably calculate the dynamic response of the wheel at impact. In conclusion, they said that the simulated results in this work show that the total plastic work can be effectively employed as a fracture criterion to predict a wheel fracture of forged aluminium wheel during impact test Mehmet et al, (2010) [17]. In his project created a finite element model of a wheel and carried out a numerical simulation of dynamic side impact test for aluminium alloy wheel by allowing a striker with an impact face of 125mm width and 375mm long to make an impact on the rim with tyre that was mounted with its axis at an angle of 13 degrees ($\pm$ 1 degree) to the vertical. It was found that, the maximum displacement of the wheel occurred at the flange and that maximum stress occurred in the lug region of the wheel. He further commented that, the moment generated by the striker is highest with respect to an axis passing through lug region and concluded that, non linear simulation can be very useful in the optimization phase in the design of the wheel.

A workshop hosted by the Vehicles Technology office (V.T.O.) of the U.S. Department of Energy in March 2013 in Dearborn, Michigan, provided very interesting target metrics for developing lightweight materials for structures and propulsion materials for more efficient powertrain systems between the periods of (2020 – 2050). One of the key target metrics for weight reduction for various systems in heavy duty vehicles are shown in the Table 1

Table 1.0 Targets for weight reductions for systems in heavy duty trucks 2020 – 2050 [6]

	2020	2025	2030	2040	2050
Class 8 Tractor Component Group					
Wheel and Tires	10%	20%	20%	25%	25%
Chassis/Frame	3%	10%	10%	20%	20%
Drivetrain & Suspension	3%	5%	10%	15%	20%
MISc. Accessories/S	5%	15%	25%	30%	35%
Truck Body Structure	15%	35%	45%	55%	60%
Powertrain	5%	10%	15%	15%	20%
Total Class 8 HDV	7%	16%	22%	27%	31%
Trailer (53ft) Component Group					
Wheel and Tires	10%	20%	20%	25%	25%
Chassis/Frame	3%	10%	10%	20%	20%
Suspension	3%	5%	10%	15%	20%
Box/Other	5%	10%	15%	20%	25%
Total Trailer	3%	9%	13%	19%	23%
Truck and Trailer Combined Totals	5.0%	13.2%	18.0%	23.6%	23%

SOURCE: U.S department of energy (DOE, 2013)

II. MATERIALS AND METHOD

The design of the heavy-duty wheel utilized an aluminum alloy material as a possible replacement for an existing steel wheel. The wheel was designed with a zero offset, which means the disc was centrally located within the inner portion of the rim. The dimensions of the new aluminum design were obtained from the existing steel wheel through measurements as shown in Table 4.0. The thickness of the wheel was obtained through calculations using hoop stress equation according to Kurhmi and Gupta (2010) [18] as follows:

Table 2.0. Relationships between thin wall stress components and other wheel dimensions

$\sigma_c = \frac{p \times d}{2t_r}$	(1)
$\sigma_l = \frac{p \times d}{4t_r}$	(2)
$\sigma_c = 0.8 \times \sigma_y$	(3)

Where σ_c is the circumferential or hoop stress in the material; σ_l is the longitudinal stress in the material and σ_y is the yield strength of the material as obtained from the mechanical properties of the material, shown in Table 2.0

The design utilized the "thin wall theory" after comparing the diameter of the wheel design and the calculated thickness of the wheel.

The remaining dimensions of the wheel such as rim, width, size of bolt circle diameter, size of ventilation holes etc were obtained from the manual measurements taken while other more confidential once such as, bead seat angles, drop centre, back spacing, etc. were obtained from standard values of available data from Hayes Lermmez International Inc as shown in Table 4.0. *Hayes Lermmez International Inc. is a leading worldwide producer of aluminum and steel wheels for passenger cars and light trucks, and of steel wheels for commercial trucks and trailers. The company has operation with business, sales offices and manufacturing facilities located in 13 countries around the world* [19].

Solid Works software was employed in solid modeling of the wheel using data in Table 4.0 A 2D profile of the rim was first created; thereafter the disc part of the wheel was integrated into the rim profile to produce a 3D model of the wheel. See Figures 2 and 3. Finite element method (FEM) was employed in performing stress, fatigue and impact analysis using ANSYS, with the material properties in Table 3.0 As a primary requirement of the finite element method; the wheel was first meshed up to a relevance level of 100 after fixing the bolt holes, Hub diameter and the brake drum region. A uniformly distributed pressure of 4MPa was applied on 22 faces of the rim of the wheel as well as a radial load of 329.3 kN was applied in the y-direction via the central part of the rim. The equivalent Von-Mises stress, the total deformation, the factor of safety, the fatigue strength, fatigue life, the impact energy, the maximum momentum, the maximum kinetic energy etc. where all evaluated.

Table 3.0 Mechanical properties of EN AW-AlSi1MgMn (6082) Alloy (ASM Aerospace Specification Metal Inc, (2015)

Structural	Values
Young's Modulus	68.9GPa
Poisson's Ratio	0.33
Density	2.7g/cm^3
Thermal Expansion	0.0000252/^{o}C
Tensile Yield Strength	276MPa
ETensile Ultimate Strength	310MPa
Elongation at break	12%
Fatigue Strength	96.5
Thermal conductivity	167 W/mK
Specific Heat Capacity	896J/kg^{o}C
Resistivity	3.99e-006 Ohm-cm

Table 4.0 Summary of dimensions for the wheel modeling in this study

S/N	Specification	Value
1	Rim width	264mm
2	Rim diameter	570mm
3	Pitch circle diameter (PCD)	310mm
4	Centre bore diameter (CBD)	490mm
5	Bolt diameter	26mm
6	Ventilation hole	Non circular
7	Rim thickness	5.2
8	Disc thickness	10
9	Offset	0mm
10	Back spacing	124mm
11	Bead seat	5^o tapered
12	Number of bolt holes	7
13	Number of ventilation holes	6
14	Material	EN AW-AlSi1MgMn (6082)
15	Intended manufacturing process	Flow forming

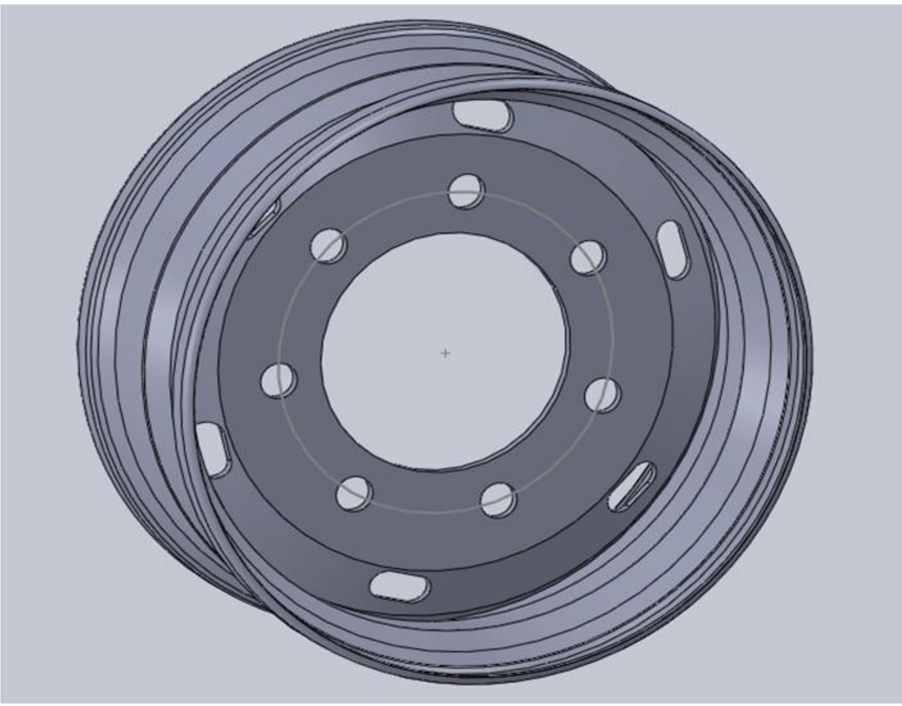

Figure 2.0 3D Solid Model of the wheel

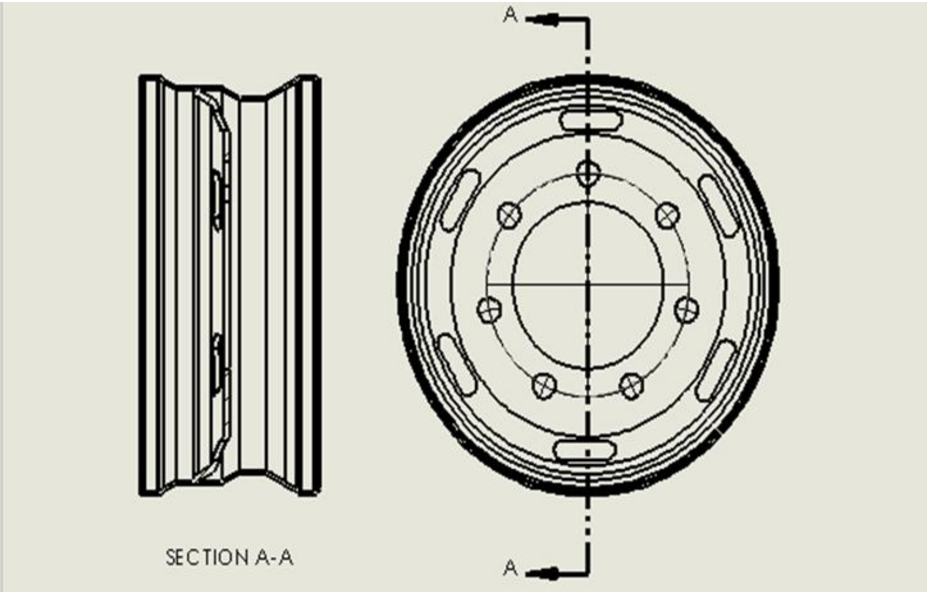

Figure 3.A Section view of the wheel

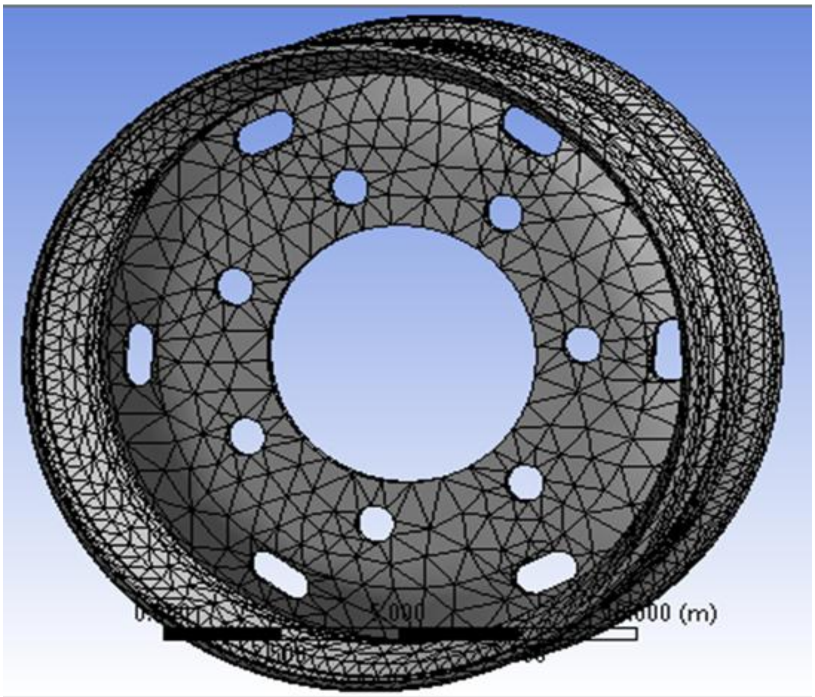

Figure 4. Meshed wheel with triangular elements for Stress and Fatigue analysis

III. RESULTS AND DISCUSSION

The static stress analysis in Figures 5.0 and 6.0 shows no deformation occurring at entire disc region of the wheel while a maximum deformation of 0.0352m occurred round the periphery of the inner flange of the rim as summarized in the stress analysis results Table 4.

This could be attributed to the high loading conditions that were applied directly on the rim and not on the disc of the wheel. The same stress analysis result revealed that the minimum equivalent Von-Mises stress was 181.88Pa and it occurred on the disc region of the wheel; while a maximum Von-Mises stress of 271.09MPa occurred at both the inner and outer peripheries of the rim flanges. Furthermore, the stress analysis also revealed that the entire surface area of the wheel had minimum safety, with a factor of 0.31797 as can be seen in figure 5c.

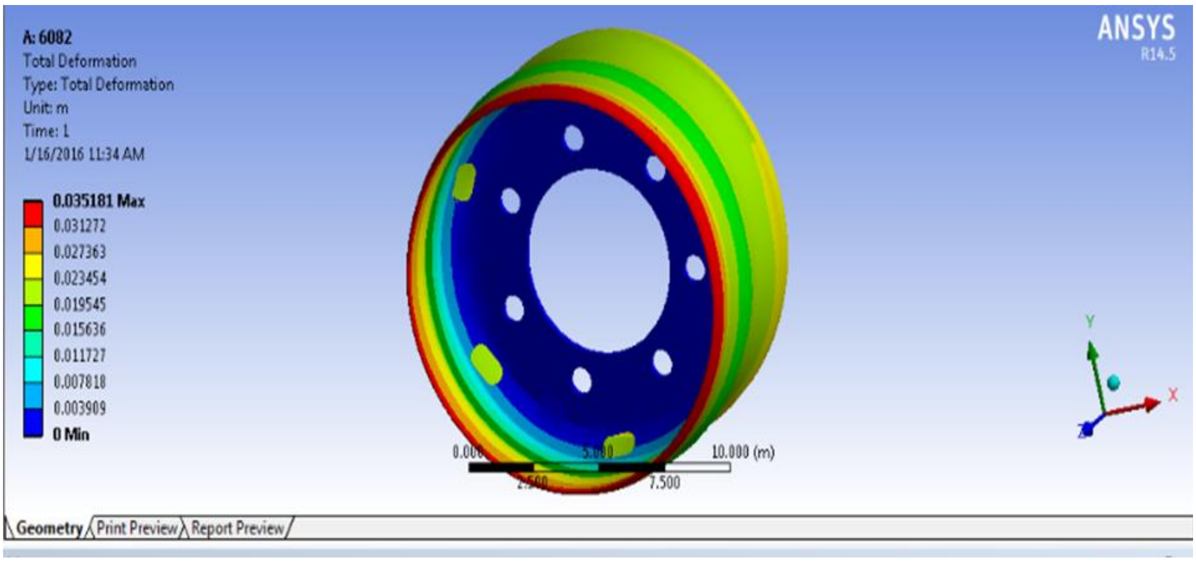

Figure 5.0 Total deformation of the wheel under maximum design load and pressure in ANSYS

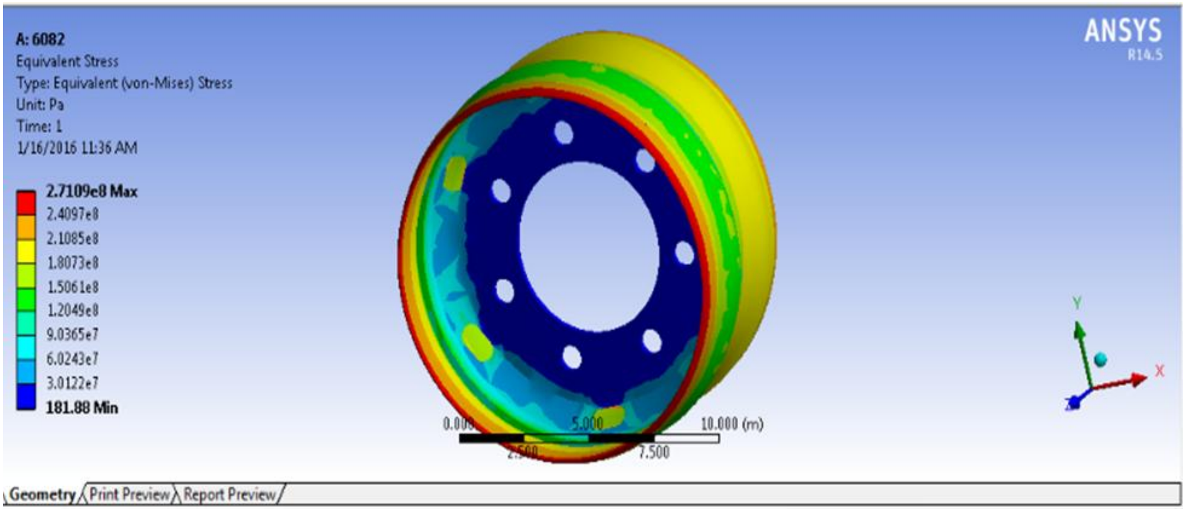

Figure 6.0 Equivalent (Von-Mises) stress distribution on the wheel under maximum design load and pressure in ANSYS

Table 5.0 Summary of stress analysis results

Analysis Parameter	Maximum	Minimum
Equivalent Von-Misses Stress	271.09MPa	181.88Pa
Total Deformation	0.0352m	0m
Region	Rim flange	Entire disc region
Factor of Safety	15(Disc region)	0.31797 (Rim flange)

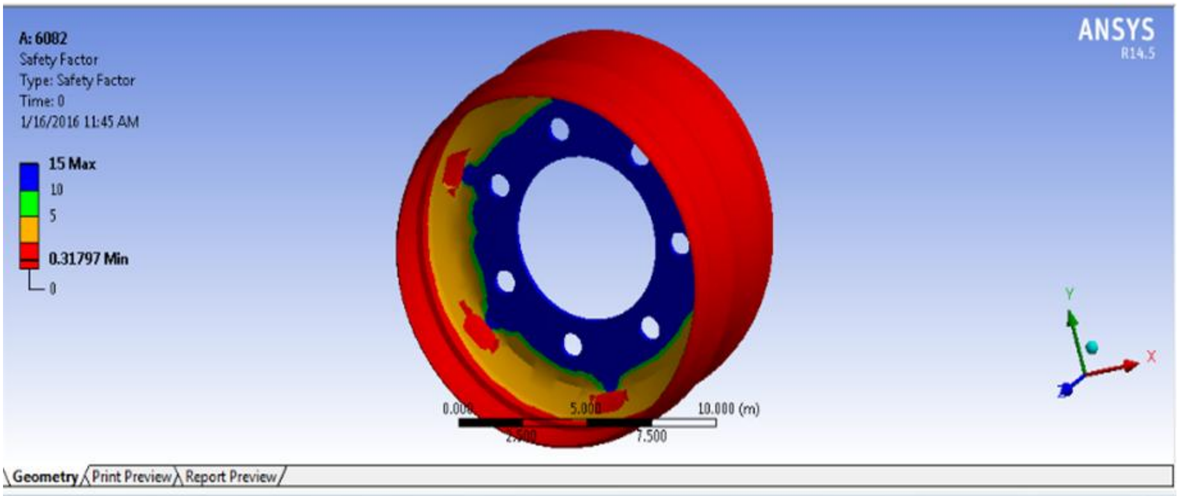

Figure 7.0 Safety factor distribution on the wheel under maximum design load and pressure in ANSYS

The simulation of the radial fatigue test as depicted in Figure 6.0 showed that under the given loading conditions; the S – N curve indicates that increase in the alternating stress leads to a corresponding decrease in the life (in cycles) of the wheel and the curve runs smoothly from the maximum alternating stress value to the maximum life of the wheel, and it clearly indicates that the designed wheel has fatigue strength of about 271.09MPa and the corresponding fatigue life of about 9.0×10^3 cycles which is quite far less than the expected fatigue life result for an aluminum alloy wheel.

However; this is accounted for by a number of extraneous factors notably; due to design geometry and extremely elevated loading conditions. This is confirmed when the loading conditions were markedly reduced, with air tyre pressure at 2MPa and the radial load at 82331N (82.33kN) which is ideal for the wheel as standardized by [2] and [1]. The fatigue life of the wheel was recorded at 1.0672×10^5 cycles which falls within the expected fatigue life range for aluminum alloyed wheels [20]. The analysis also revealed that the wheel has an infinite life of 1.0×10^6 cycles as shown in Table 6.0.

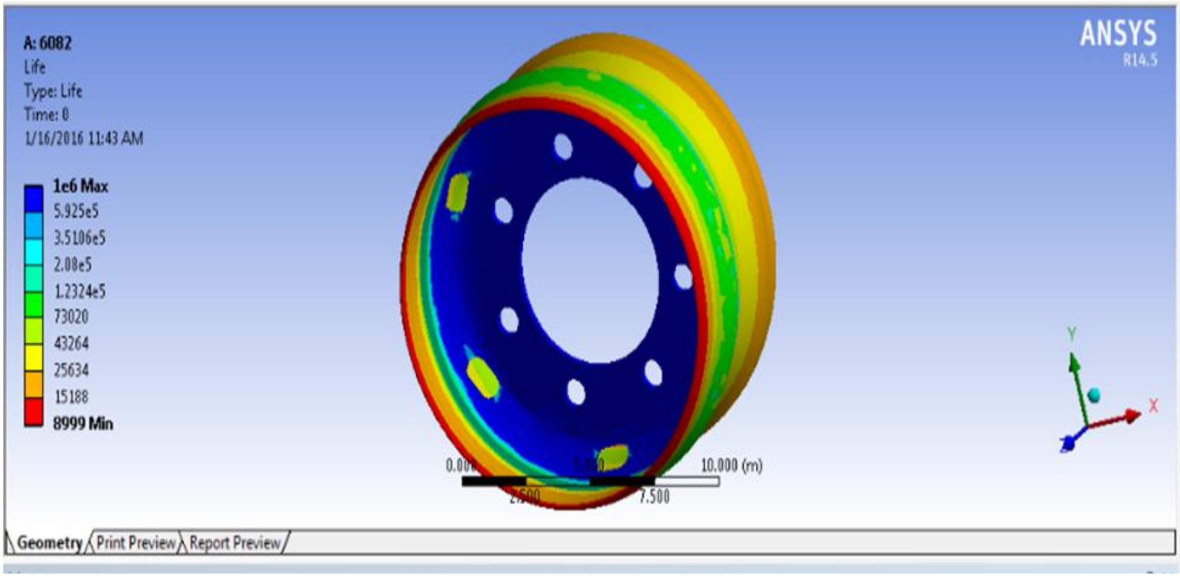

Figure 6.0 Wheel life distribution under maximum loading condition (271.09MPa alternating stress)

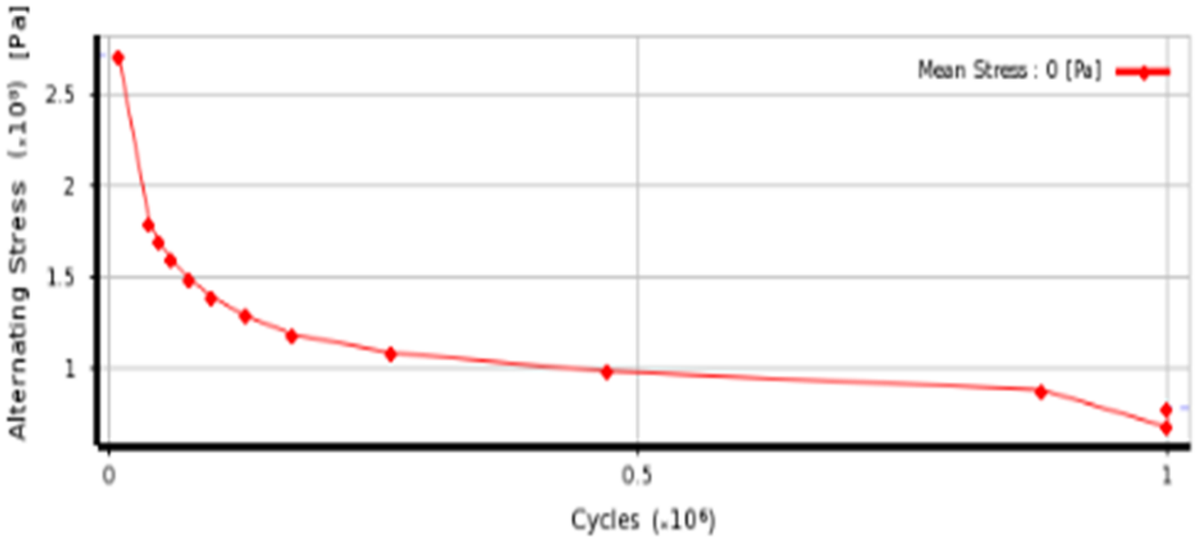

Figure 7.0 S – N Curve for the wheel displayed on a linear plot in ANSYS

Table 6.0 Summary of fatigue test results

Analysis Parameters	Values	Region
Fatigue Strength	271.09MPa	Rim flange
Fatigue Life	9.0×103 Cycles	Rim flange
Infinite Life	1.0×106 Cycles	Entire disc region

Finally, the impact test results shows the adaptation of the new design wheel to varying loading condition at impact, leaves the wheel with a maximum deformation of about 0.07m. This represents a very big dent on the rim, while the material is safe as per the Von-Mises stress result.

In each of the curves, a nonlinear relationship exists between the momentum, total energy and life of the wheel in (cycles) as these were the parameters used in analyzing the impact behavior of the wheel as depicted in figure 8.0 and figure 9.0 respectively. The plots revealed that while the wheel life increased steadily from minimum to maximum; there is an irregular behaviour of the momentum and energy of the wheel in service. Figure 8.0 (i.e. momentum plot) also reveals that the maximum momentum of the wheel in the x - direction is 1.17×106Ns and the corresponding impulse of the wheel in the same x – direction is 9.58×10^6 Ns. The maximum kinetic and internal energies of the wheel are indicated in Figure 9.0 as 4.181×10^9J and 2.09×10^9J respectively.

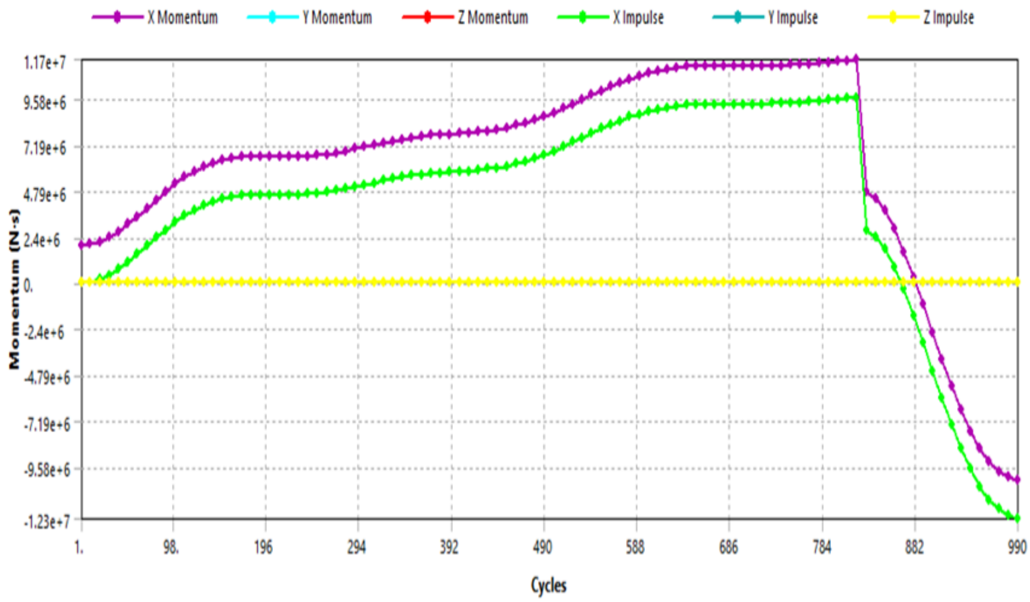

Figure 8.0 Wheel model momentum summary

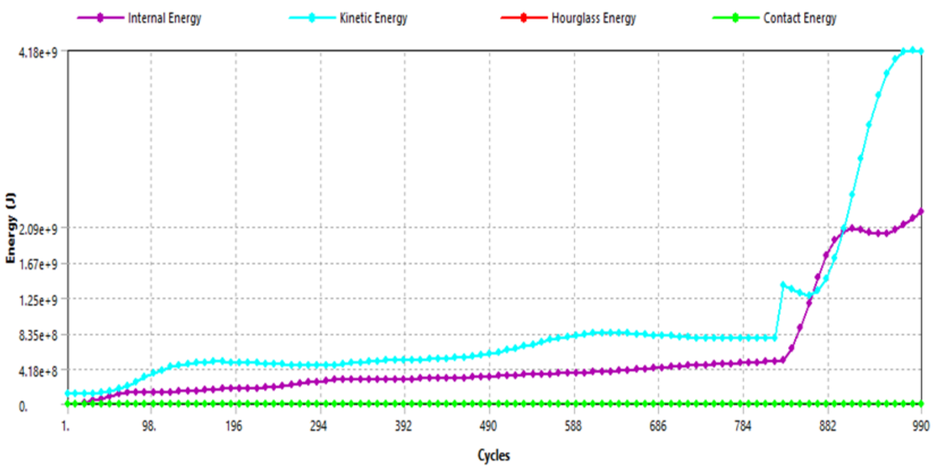

Figure 9.0 Wheel model energy summary

IV. CONCLUSION

The results of this research made it quite evident that the nature of wheel design, especially the choice of offset used, as well as other wheel dimensions and the choice of wheel material and loading conditions are collectively responsible for the overall performance of the wheel.

The importance of designing the wheel to conform to existing wheels or set standards cannot be overemphasized. The finite element method was employed in the analysis of the performance of this wheel through the help of computer aided algorithm and it was discovered that the maximum stress occurred round the peripheral path of the inner and the outer rim flanges under the given load conditions of 82331N and 4MPa. However, this area of maximum stress concentration does not conform to most others due to differences in the design of the wheel. However, when ideal loading conditions were applied, the wheel provided acceptable results that correlate with previous works.

Generally, computer simulation is a very handy tool in the manufacturing of wheels as the number of prototypes needed before mass production maybe greatly reduced, by estimating the right values of the desired design parameter before such will be replicated in the actual design.

REFRENCES

[1]. SAE J328 Revised. Wheels-Passenger car and truck performance requirements and test procedures (19940.

[2]. Test requirements for truck steel wheels. EUWA standards, ES 3.11. EUWA-Association of European Wheel Manufacturers (20060.

[3]. Crouse and Anglin, (2008).

[4]. Giri N.K. Automobile technology: Second edition (2008).

[5]. Carvalho C.P. Voorwald HJC, Lopes CE. Automotive wheels- an approach for structural analysis and fatigue life prediction. SAE tecnical papers.SAE 2001-01-4053 (2001).

[6]. Vehicle Technology Office. Final Report on Trucks and Heavy-Duty Vehicles Technical Requirements and Gaps for Light Weight and Propulsion Materials (Dearbon Michigan) (2013).

[7]. Carboni M, Beretta S, Finzi A. Defects and in-service fatigue life of truck wheels.EngFail Anal; 10: pp.45–57 (2003).

[8]. Saeed Moaveni. Finite element analysis: theory and application with ANSYS. 3rd edition, Pearson Prentice Hall, New Jersey (2008).

[9]. Siva Prasad T, Krishnaiah T, 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 (2014).

[10]. Satyanarayana N.& Sambaiah Ch. Fatigue Analysis of Aluminum Alloy Wheel Under Radial Load. International Journal of Mechanical and Industrial Engineering (IJMIE), ISSN No. 2231 – 6477, Vol-2, Issue-1,(2012).

[11]. Ridha, R.A. Finite Element Stress Analysis of Automotive Wheels, Society of Automotive Engineers, 760085 (1976).

[12]. Meghashyam P. Design and Analysis of Wheel Rim using CATIA & ANSYS. International Journal of Application or Innovation in Engineering & Management (IJAIEM) Web Site: www.ijaiem.org Email: editor@ijaiem.org, editorijaiem@gmail.comVolume 2, Issue 8, ISSN 2319 – 4847 (2013).

[13]. Janardhan J. Radial Fatigue Analysis of An Alloy Wheel Int. Journal of Engineering Research and Applications www.ijera.com ISSN : 2248-9622, Vol. 4, Issue 12(Part 6), December 2014, pp.253-258 (2014).

[14]. Emmanuel M. Adigio and Ebughni O. N. Computer Aided Design and Simulation of Radial Fatigue Test of Automobile Rim Using ANSYS IOSR Journal of Mechanical and Civil Engineering (IOSR-JMCE) e-ISSN: 2278-1684, p-ISSN: 2320-334X, Volume 11, Issue 1 Ver. IV. pp 68-73 (2014).

[15]. Wang, X. and Zhang, X. Simulation of dynamic cornering fatigue Test of a steel passenger car wheel. International journal of Fatigue, 32 p.434 – 442 (2010).

[16]. Chang C.L and Yang S.H, (2009). Finite element simulation of wheel impact test. Journal of Achievement in Materials and Manufacturing Engineering, vol. 28 ISSUE 2 June 2008.

[17]. Mehmet F., Recep K., Murat O., and Hamdi M.O. Numerical modelling and simulation of wheel radial fatigue tests. Engineering Failure Analysis 16, pp.1533 – 1541 (2010).

[18]. Kurhmi R.S. and Gupta J.K. (2010). Machine design: Fourteenth edition. pp 152 – 225.

[19]. Hayes Lemmerz, (2012). *Wheel overview*. [online]. Available at < http://www.hayes-lemmerz.com/wheels_overview.php> (2015).

[20]. Liangmo W. Yufa C.W. and Qingzheng W. Fatigue Life Analysis of Aluminum Wheels by Simulation of Rotary Fatigue Test.

[21]. Journal of Mechanical Engineering 57(2011)1, 31-39 Paper received: 08.04.2009 DOI:10.5545/sv-jme.2009.046 (2010).

Christopher E. Nabodi, et. al. "Computer Aided Design and Simulation of Aluminium 6082 as HDV Wheel."*American Journal of Engineering Research (AJER)*, vol. 10(3), 2021, pp. 95-104.