International Research Journal of Education and Technology

Peer Reviewed Journal

ISSN 2581-7795

Study on Structural Analysis of Automated Wheel Rim using Ansys Workbench

Mukund Fartade[1], P.H.Jain[2], V.V.Mane[3]

[1]M-Tech Student Dept. Of Mechanical Engineering College of Engg. Dharashiv, Maharashtra, India
[2]Professor Dept. Of Mechanical Engineering, College of Engg. Dharashiv, Maharashtra, India
[3]Princaipal College of Engineering, Dharashiv, Maharashtra, India

---***---

Abstract - *The vehicle may be towed without the engine but it is not possible without the wheels. The design and analysis of wheels have drastically improved for the last two decades. Research works have shown the possibility of using a wide range of materials that gives out maximum efficiency for the users. Alloy wheels are practically the mixtures of two or more different metals. Their unique properties over the conventional ones have made them the most favoured. This project involves the design and analysis of an existing car model. The design and the composition of wheel are aimed towards increasing the strength and other parameters that are involved in the failure of these wheels. An alloy wheel is selected for this purpose and its composition is to be varied for getting the best results and comparing them with the existing composition and properties. Finite element analysis (FEA) is carried out by simulating the test conditions to analyze stress distribution and fatigue life of the aluminum alloy wheel rim of passenger car. Experimental analyses are carried out by radial fatigue testing machine for evaluation of fatigue life under influence of camber angle. The test indicates that integrating FEA and nominal stress method is a good and efficient method to predict alloy wheels fatigue life.*

Key Words: Catia, Ansys, Static Structural analysis, Alloy wheels etc

1. INTRODUCTION

The wheel is one of the most significant discoveries of old times. The wheel has developed from nothing more than an oversized bearing to a fully integral part of any modern transportation vehicle. Wheel is an important structural member of the vehicular suspension system that supports the static and dynamic loads encountered during vehicle operation. A vehicle can be towed without engine or its operation but it cannot be done without a wheel. Moreover a wheel is the member or a crucial component of an automobile, which carries the whole weight of the vehicle including engine, body and all other components that it houses. The wheels thereby distribute the vehicle load to the road surface and also propel the vehicle forward by rotation of these wheels. The wheels also translate or slide through the linear axis which steers the vehicle and provides directional control. The wheels must bear the weight and retain their structural shape during sudden shocks and impact loads that act upon them. Modern day wheels are mostly alloys of aluminium, magnesium, titanium etc. Most sport cars that run on sport tracks that are well maintained also use carbon fibre wheels.

1.1 Objectives

The main objective of the project is to perform a static structural analysis in ANSYS, for an alloy wheel of an automobile (car wheel).The wheel design is performed in CATIA as a 3D model and then converted it to STEP/IGES file there by imported to ANSYS to perform analysis. In modern day's wheels are mostly made up of aluminum and magnesium alloys.

In this project we are taking three materials namely Titanium Alloy, Al alloy (AlSi7Mg0.3) and Mg alloy (AM60B).

The main objective of this project is to find out better or best material by conducting static structural analysis individually to each material and compare the results

2. LITERATURE REVIEW

Dr. S. Nallusamy, et.al [1] during a part of research a static and fatigue analysis of aluminum alloy wheel A356.0 was carried out using FEA package. The 3-D model was imported from CATIA into ANSYS using the appropriate format. The distributions of displacements and stresses for the specified pressure, static load, and angular velocities have been determined using static analysis.

Sumit Agarwal, et.al. [2] They compare equivalent stress, Total deformation and equivalent elastic strain in 19-inch wheel rim of MG Gloster by Finite element method. SOLIDWORKS 2019, a 3-D modelling programme, was used to model the rim, while ANSYS R2 2020 was used to analyse its static structure.

K. Srinivasa Rao, et.al. [3] they analysed the existing car model wheel rim is drawn in the design software Creo 2.0 and various loads and forces are theoretically calculated and applied on the model and analysed through Ansys (17.0) software. the fatigue analysis which includes the product life, the damage factor and safety factors are determined and all this analysis is done by taking three different designs of the same model of car (Volkswagen polo 1.0 TSI) and rim materials (aluminium A356-T6, magnesium, MgAm60, titanium, 6Al4V) are changed in each case to know the best design and best material for a particular type of loading.

G. Ashok kumar, et.al. [4] The wheel rim is designed by using modeling software CATIAv5R18. Later this CATIA

ISSN 2581-7795

model is imported to ANSYS for analysis work. ANSYS software is the latest software used for simulating the different forces, pressure acting on the component and also for calculating and viewing the results. ANSYS static analysis work is carried outby considered two different materials namely aluminum and forged steel and their relative performances have been observed respectively.

2.1 Work scope following the literature review:

We learned from the previous review that aluminium alloy is the most often used material, and the above reviews also support this claim. To complete the analysis and obtain the results, we will use Titanium and Magnesium Alloy materials on the wheel, with the alloy serving as the basis material.

3. CALCULATIONS

The total load acting on the wheels and the load acting on each wheels is calculated, we will apply this load as downward force on seathbed after design is completed.

Load to be applied: Gross weight of the vehicle on wheels: 2095 kg.

$$1665 + (5x70) + 80 = 2095 \text{ Kg}$$

Where,

Weight of the Car: 1665 Kg

No. of person inside the each of weight is 70*5=350Kg

Therefore, Total Load on the wheels: 2095 Kg.

Load acting on each wheel: 2095/4 = 523.75 Kg or **5137.98 N**

Pressure load acting on the wheel: Consider 35 psi of air pressure acting on the outer surface of the wheel.

$$35 \text{ psi} = 35 \times 0.4535 \times 9.81 / (25.4)^2 = 0.241 \text{ N/mm}^2$$

Stages of Speeds that are considered:- The wheel taken at various speeds for analysis are as follows:

A) 80 Kmph.B) 120 Kmph C) 160 Kmph

The linear velocity of wheel is to be converted into angular velocity (rad/s) so,

$$v = r * w,$$
$$w = v / r$$

Where,

r = Radius of wheel = 246.518 mm

w = Angular velocity,

v = Linear velocity.

A) v = 80 Kmph = 22.22 m/s.
r = 246.518 mm = 0.2465 m
w = v/r = 22.22 / 0.2465 = 90.15 rad /s.

B) v = 120 Kmph = 33.33 m/s
r = 246.518 mm = 0.2465 m.
w = v / r = 33.33 / 0.2465 = 135.21 rad /s.

C) v = 160 Kmph = 44.44 m/s.
r = 246.518 mm = 0.2465 m.
w = v / r = 44.44 / 0.2465 = 180.28 rad / s.

3.1 Boundary Conditions:

1. Five PCD holes (i, e.), 120x5, constrained for 5 (UX, UY, UZ, RX, RZ) degree of freedom, (Rotation in Y direction is set free).
2. Force: Downward force applied on seat bed.
3. Analysis type: Static Structural Analysis.
4. Boundary Conditions applied are Radial load and Pressure load.
5. Rotational Velocities at different speeds are also applied:

i) V = 80 Kmph = 90.15 rad / s.
ii) V = 120 kmph = 135.21 rad/s.
iii) V = 160 kmph = 180.28 rad/s.

4. 3D MODELLING OF WHEEL

CATIA is 3D CAD modelling software used to design, visualize, and test product ideas. It allows you to create product prototypes that accurately simulate the weight, stress, friction, driving loads, and much more of products and their components in a simulated 3D environment. It enables the Creation of 3D parts, 3D sketches, sheet metal, composites and moulded for or tooling Parts up to the definition of mechanical assemblies.

Step 1: Part Design Module:

Fig 1 Alloy Wheel in CATIA

Step 2: Drafting of Wheel:

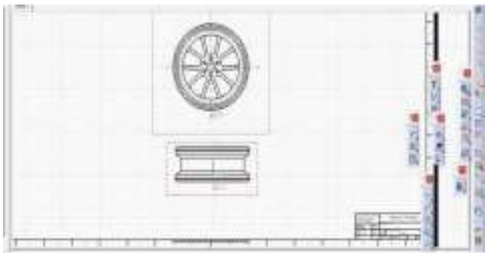


Fig 2 Drafting of Alloy Wheel in CATIA

4. ANALYSIS OF WHEEL RIM

Ansys provides a common platform for product development, from design concept to final-stage testing and validation.

Finite-element method (FEM) is a good choice for the analysis of sheet metal processes since it helps in eliminating the need for time-consuming experiments tooptimize the process parameters such as sheet metal thickness, the material of the sheet, punch fillet and percentage clearance. The FEM simulations are increasingly used for invesgating and optimizing the punching process.

4.1 Procedure in Ansys:

- Step 1: Start on ANSYS Workbench Project.
- Step 2: Create a Static - Structural Analysis System.
- Step 3: Add a new material.
- Step 4: Insert Geometry.
- Step 5: Create a Profile Sketch to Customize units.

International Research Journal of Education and Technology

Peer Reviewed Journal

ISSN 2581-7795

- Step 6: Create an Extruded Body.
- Step 7: Launch the Static - Structural Analysis System.
- Step 8: Generate Mesh.
- Step 9: Apply Boundary Conditions.
- Step 10: Solve and Retrieve the Results.

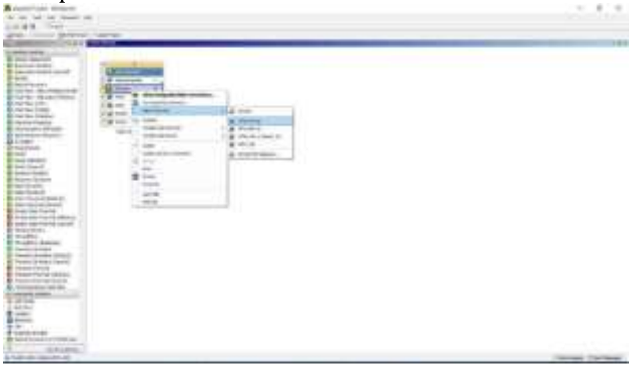


Fig 3 Geometry Selection

4.2 Mesh Generation:

Meshing is usually the first step done while doing analysis on the wheels. Meshing is done for reducing the time and effort required to get the actual results or the structural analysis of the wheels.

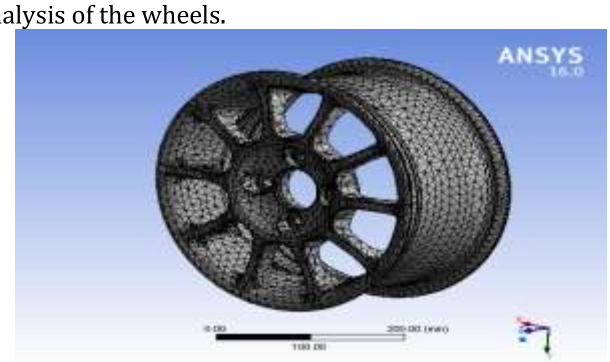


Fig 4 Mesh Generation

4.3 Applying Boundary Conditions:

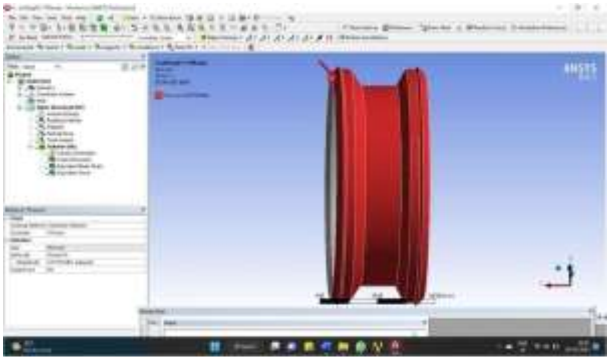


Fig 5 Radial pressure at 40Psi (0.27579Mpa)

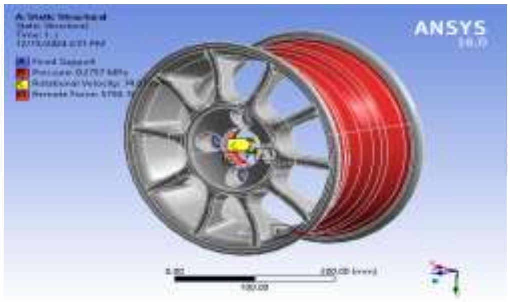


Fig 6 Remote force of 5138N

Obtained Result for three different material at three different speeds

1. Deformations of aluminium alloy

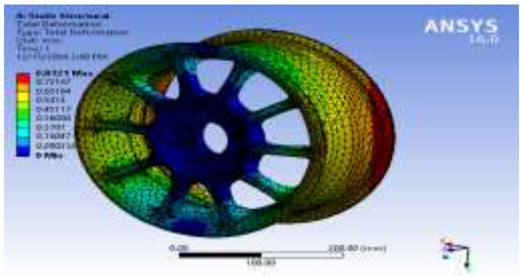


Fig.7 Total Deformation at 80kmph

Same results are generated at 120 & 160 Kmph

2. Equivalent (von-misses) stresses of aluminium alloy

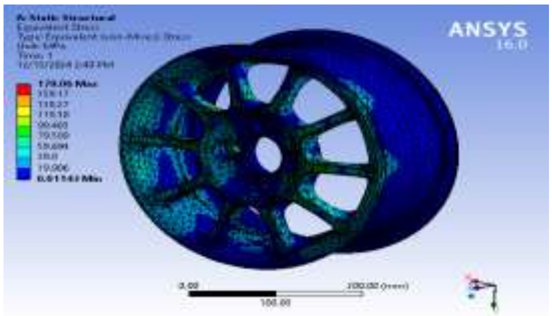


Fig.8 Equivalent (Von-Mises) stress at 80kmph

Same results are generated at 120 & 160 Kmph

3. Equivalent elastic strains of aluminium alloy

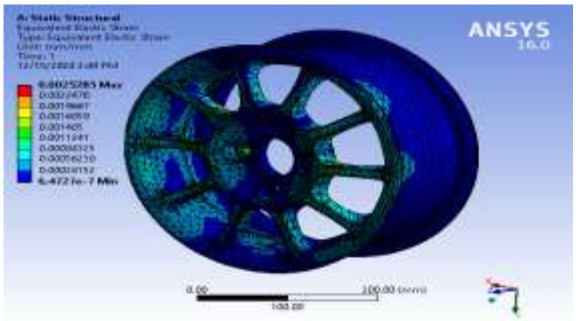


Fig.9 Equivalent Elastic Strain at 80kmph

Same results are generated at 120 & 160 Kmph

Same procedure followed for other two materials.

International Research Journal of Education and Technology

Peer Reviewed Journal

ISSN 2581-7795

5. Result & Discussion

Analysis is done on imported design from CATIA V5 using static structural and modal analysis workbench under required condition, on different alloy materials. The results are obtained and compared between different materials under same parametric conditions.

5.1 TOTAL DEFORMATION:

Material	Total deformation in mm at 80kmph	Total deformation in mm at 120kmph	Total deformation in mm at 160kmph
Aluminium alloy	0.8121	0.81353	0.81553
Magnesium alloy	1.2802	1.2817	1.2838
Titanium Alloy	0.6008	0.60258	0.60507

5.2 EQUIVALENT (von-mises) STRESSES:

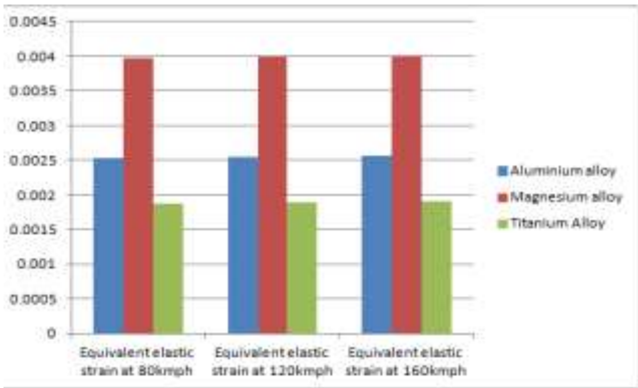
Material	Equivalent (von-mises) stress in Mpa at 80kmph	Equivalent (von-mises) stress in Mpa at 120kmph	Equivalent (von-mises) stress in Mpa at 160kmph
Aluminium alloy	179.06	179.98	181.27
Magnesium alloy	178.63	179.23	180.07
Titanium Alloy	179.28	180.81	182.96

5.3 EQUIVALENT ELASTIC STRAINS:

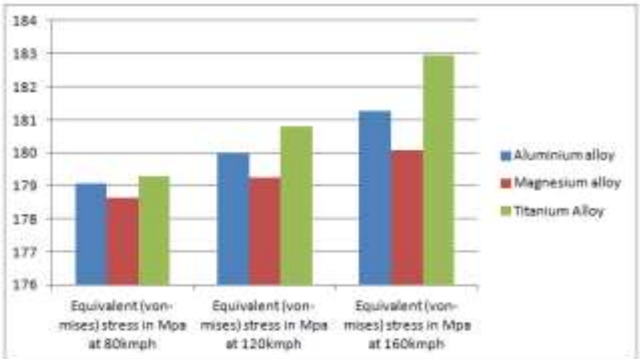
Material	Equivalent elastic strain at 80kmph	Equivalent elastic strain at 120kmph	Equivalent elastic strain at 160kmph
Aluminium alloy	0.0025285	0.0025414	0.0025596
Magnesium alloy	0.0039794	0.0039927	0.0040113
Titanium Alloy	0.001872	0.0018879	0.0019103

The study was done for three different type of materials i.e. Aluminium alloy (AlSi5Mg0.3), magnesium alloy (AM60B), Titanium Alloy. This material was choosen as wheels are generally fabricated of high strength materials. Further the analysis was carried out to get results i.e. total deformation, equivalent elastic strain and equivalent (von-mises) stress

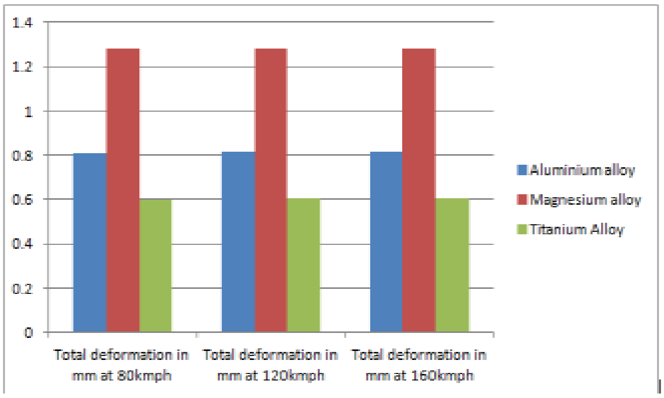
.



Graph.1 Equivalent elastic strain at different speeds for different materials



Graph.2 Equivalent (von-mises) stress at different speeds for different materials



Graph.3 Total deformation at different speeds for different materials

As the analysis is performed, and the result graphs are drawn, By looking at the graph 6.1 we can see that Titanium Alloy has less Strain developed than other two materials. Also the Aluminum alloys are almost similar.

From the graph.2 of equivalent (von-mises) Stress we can see that Titanium has developed higher stress where Aluminium alloy and magnesium lesser and similar to each other.

From the graph 3 of Total Deformation, it is clearly shows that Titanium Alloy has less deformation far from Aluminum and Magnesium Alloy.

6. CONCLUSION

1. The current work that is done in this project is the consideration of these Aluminum alloys and Magnesium alloys as they are greatly used, and also an alloy called titanium, that has the similar properties like strong, hard, light weight.
2. Titanium alloy exhibited better result with a maximum deformation of 0.60507 mm and max.Equivalent elastic strain with 0.0019103 and with maximum value of equivalent stress of 182.96 Mpa for same loading conditions.
3. Where as, Aluminum alloy AlSi7Mg0.3 becomes the second better material and deviates slightly from the above both alloys.
4. Hence from the analysis of the project we can conclude that rather than Titanium, Aluminium and Magnesium alloys can be used in case of durability as Magnesium alloy can fail before Titanium and Aluminum alloys.

REFERENCES

[1] Dr .S. Nallusamy, N. Manikanda Prabu, K. Balakannan, Dr. Gautam Majumdar“Analysis of static stress in an alloy wheel of the passenger car” International Journal of Engineering Research in Africa Vol. 16 (2015) pp 17-25 doi:10.4028/www.scientific.net/JERA.16.17.

[2] Sumit Agarwal, et.al. “Modelling and static analysis of automotive wheel rim using different materials” FASET 2020.

[3] K. Srinivasa Rao, et.al. “Design and analysis of alloy wheels” International journal of Engineering and Technology (IRJET) volume: 04 Issue: 06| June 2017.

[4] G. Ashok kumar, et.al. “Design and Analysis of Alloy wheel rim using CATIA and ANSYS” ” International journal of Engineering and Technology (IRJET) volume: 03 Issue: 12| Dec 2016.

[5] Dr. M. S. Hebbal, et.al. “Static structural, Modal & Harmonic Analysis of Alloy car wheel rim using ANSYS workbench” International journal of Engineering and Technology (IRJET) volume: 08 Issue: 07| July 2019.

[6] Jaspreet Singh, Siddhartha Saha, “Static Analysis of Alloy wheel using ANSYS15.0” International journal of Engineering and Technology (IRJET)

[7] Gamachisa Mitiku Tadesse, “Modelling and Analysis of car wheel” International journal of Engineering and Technology (IRJET) volume: 04 Issue: 02| July 2017. Rajesh Dhakar, Ruchika Saini “Static analysis of Four-wheeler Alloy wheel using ANSYS workbench” International journal of Engineering and Technology (IRJET) volume: 06 Issue: 06| June 2019.

[8] Vaddi Naveen, “Modelling and Static Analysis Alloy wheel using different materials” International journal of Engineering and Technology (IRJET) volume: 05 Issue: 06| June 2018.

[9] Mr. CHINTAPALLI SHEKHAR, et.al. “Design and Structural Analysis of car alloy wheel using with various materials” International journal of advance scientific research and engineering trends” volume: 05 Issue: 07| July 2020.