DESIGN AND ANALYSIS OF ALLOY WHEEL FOR MULTI-PURPOSE VEHICLE

Dhiraj Bhargav. A. S[1], A. S. Jayaram[2], Chandan. R[3]

[1]Pursuing M-Tech in Machine design from Dr. Ambedkar Institute of Technology, Bengaluru, Karnataka.
[2]Associate Professor, Dept. of Mechanical Engg, Dr. Ambedkar Institute of Technology, Bengaluru, Karnataka.
[3]Assistant Professor, Dept. of Mechanical Engg, Dr. Ambedkar Institute of Technology, Bengaluru, Karnataka.

---***---

Abstract - *In the automobile industry, alloy wheels are those wheels which are made of an alloy of aluminum or magnesium. They are different from steel wheels as they are much lighter, which in turn improves the steering and the speed of the car. Alloy wheels also have reduced un sprung weight of the vehicle when compared with the steel wheels which results in precise steering along with reduction in fuel consumption. Here the term 'MUV' stands for Multi-Utility Vehicle and these types of vehicles can carry more number of passengers and have large cargo space. It's a challenge to design wheels for such robust load requirements. Here Magnesium AZ80 material is selected for the alloy wheels. This material is selected due to their superior mechanical properties, durability and light weight. Design analysis is carried out for a 18 inch alloy wheel and design is evaluated for cornering test, radial test and impact test. Further stress distribution is studied. Boundary conditions are set and the model is imported to ABAQUS and using tetrahedral elements and further linear analysis for radial and cornering test is carried out using ABAQUS/standard solver and results are studied. Further dynamic analysis for the impact test is done using ABAQUS/Explicit and analysis is studied based on the results obtained*

Key Words: Stress Analysis; ABAQUS; Wheel rim.

1. INTRODUCTION

Alloy wheel is a term generally used for wheels which are made from nonferrous alloys. Vehicle handling can be improved if the wheels are light. This is possible as the un sprung mass reduces which in turn allows suspension to follow the same stretch of land more closely hence traction increases. Fuel consumption also reduces due to the reduction in overall mass of the vehicle. The open wheel design along with better heat conduction helps to dissipate all the heat from the brakes, which in turn improves braking performance. Compared to steel wheels, alloy wheels are more expensive to produce. Magnesium alloy wheels generally referred to as "Mag wheels" were the first die-cast produced wheels. In the early 1960's aluminum alloys were used instead of magnesium wheels as it had more advantages when compared with the later.

Lately pure magnesium wheels are no longer produced and can only be seen in vintage cars as it would cause many problems. It was susceptible to corrosion, pitting and cracking. Magnesium generally is hard to ignite if it is in bulk but pure magnesium wheels can be ignited by continuous scraping of the wheel on the surface of the road which in turn leads to puncture. Further developments were made on alloys of magnesium to negotiate all these problems.

1.1 STATIC AND DYNAMIC STRESS ANALYSIS

Static Stress Analysis

Linear static stress analysis of an alloy wheel is carried out using ABAQUS software. Initially the component is meshed in Hyper Mesh and is imported to ABAQUS. Hyper mesh is a pre-processor and post- processor, ABAQUS is the solver. Static stress analysis is performed to determine the maximum stresses induced in the alloy wheel structure to identify maximum compression in the structure.

Dynamic Stress Analysis

Dynamic stress analysis is regarded as one of the most powerful situation techniques which can be subjected to complex engineering systems. This analysis helps to carry out impact of transient loads or can be used to eliminate potential noise and vibration problems. This analysis can be quite expensive and carrying out dynamic analysis at design stage can easily reduce the expense of rig testing on the material. The failure that occurs in the process can be quite damaging and by studying and evaluating the given structure at design stage can easily avoid mistakes which can be quite expensive. Here dynamic stress analysis of an alloy wheel is carried out using ABAQUS software. Impact test of an alloy wheel is carried out and results are studied based on the impact load.

1.2 Types of wheel rims

Wheel Dimensions

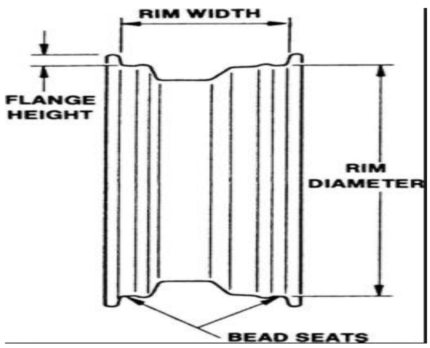

Fig-1: Wheel dimensions for a typical light alloy wheel.

W = width in inches .

D = diameter in inches.

Code number is given to represent the wheel and is given in following sequence.

- Width.
- A letter identifying the rim profile.
- Diameter.

Example, code number 6.50 J-14 and 3 D-16 are the representation of wheels as in terms of codes and can be shown in below table:

Table-1.1 Code numbers to represent dimensions of each wheel

Width	Rim type	Diameter d
6.5	J	14”
3	D	16”

Each individual wheel of same dimension may have different offset that is represented by ‘C’. the distance from the mounting surface to the wheel rim’s center line is called as an offset and it may be either negative or positive in nature. Identical offsets must be fitted to a vehicle at all times.

2. Geometric Model of an alloy wheel

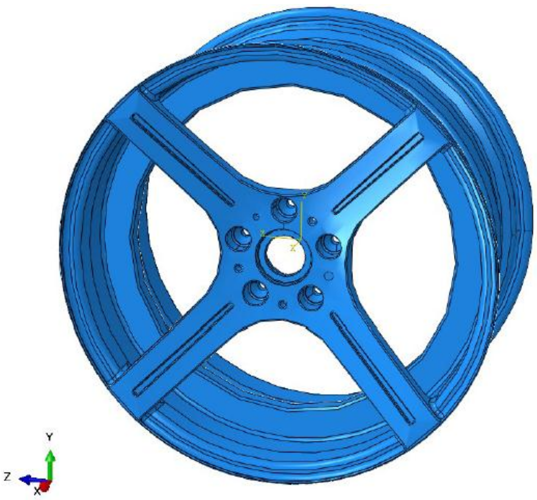

Fig-2: Geometric modelling of an alloy wheel.

The geometric model shown in previous section is discredited using tetrahedral element as shown above the figure. These elements are selected because of their ability to capture the geometry of any complex model.

2.1 Material Properties

The material properties required for the current analysis are density, young’s modulus, Poisson’s ratio and tensile yield strength

- Density (kg/m^3) =1335
- Young’s modulus(Gpa) =44.8
- Poisson’s ratio =0.35
- Tensile yield strength(Mpa) =275
- Compressive yield strength(Mpa) =240
- Ultimate tensile yield strength(Mpa) =380

2.2 Meshing

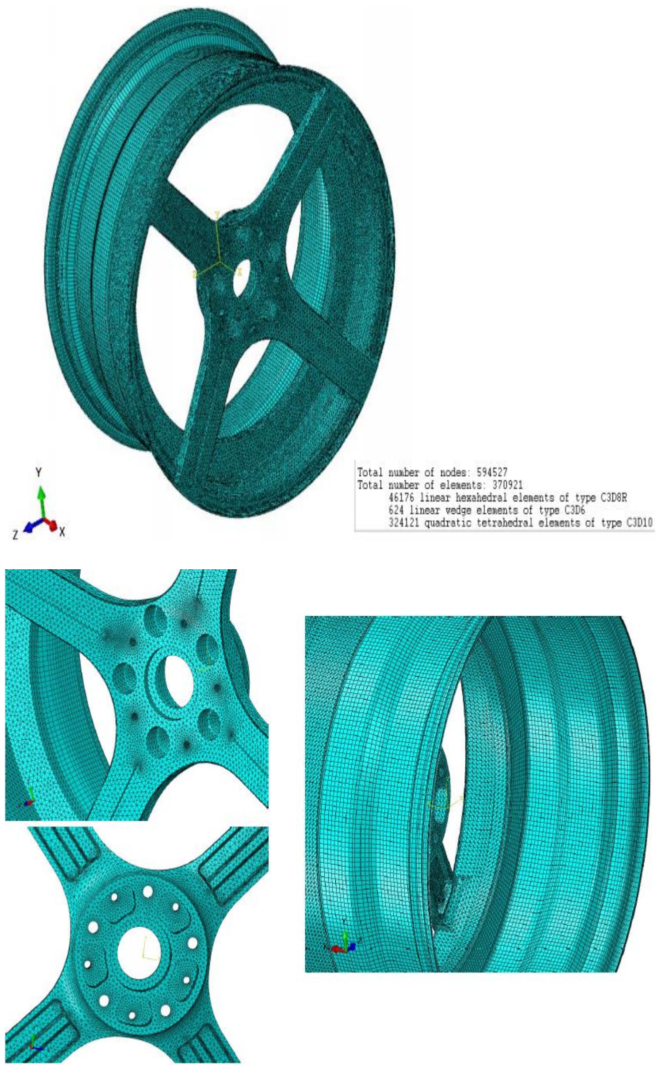

Fig-3: Meshing of alloy wheel.

2.2 Loads and Boundary Conditions:

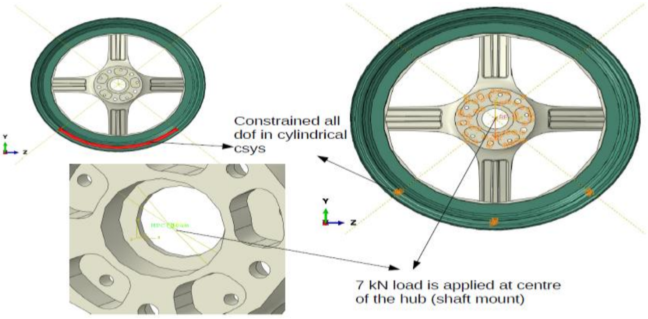

Fig-4: boundary condition applied to alloy wheels.

3 Results and discussion

3.1 Radial test

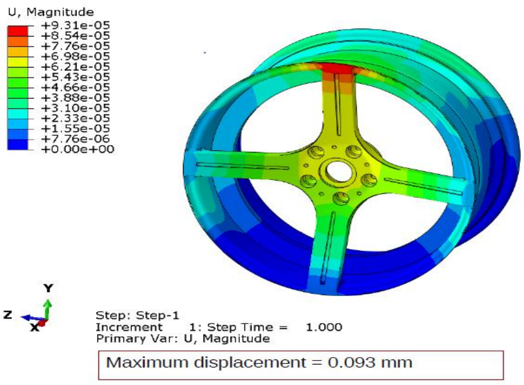

Fig-5: Static test analysis - Radial test.

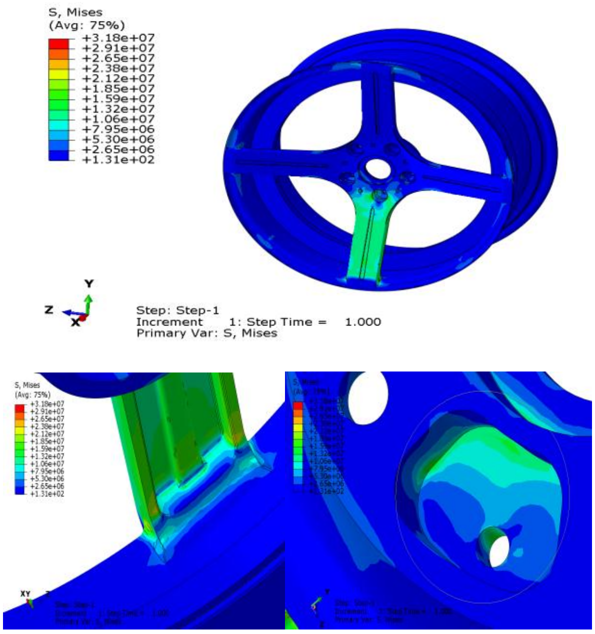

Fig-6: Elemental stress plot- Radial test

From figure 5 and 6, results were obtained and the value obtained is much less than the yield strength value of the particular material. Hence we can say that the alloy wheel is safe under the load applied and is well within the maximum allowable stress value.

3.2 Corner test:

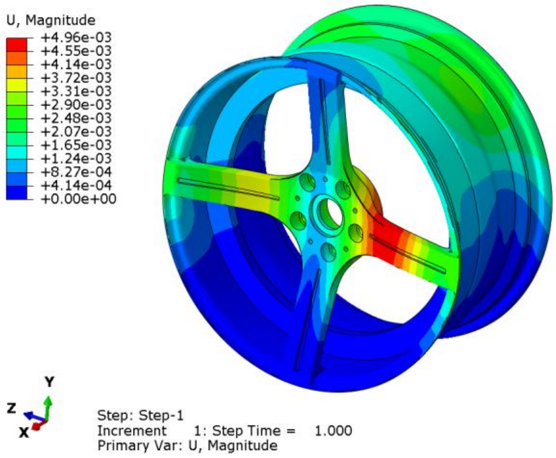

Fig-7: Maximum displacement - Corner test.

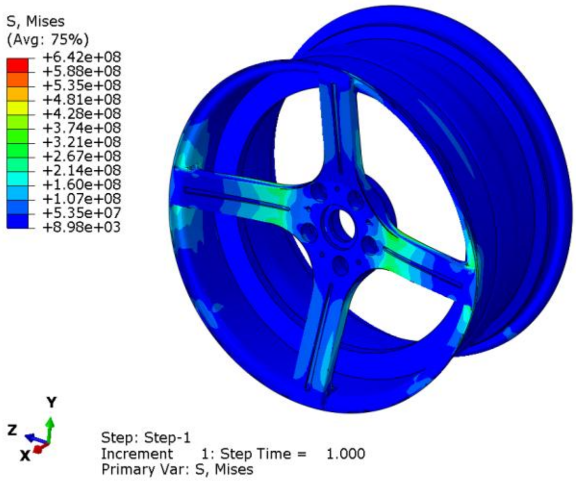

Fig-8: Von misses stress- Corner test

A maximum stress of 642Mpa was observed which was more than two times the ultimate limit for the given material which was around 340Mpa.

3.3 Impact test

The impact analysis shows a maximum stress within 380 Mpa which is surface stress. The average stress in spoke cross section is about 130 Mpa which is well within the limits. A rigid body is made to impact with the wheel which is tilted at an angle of 15 degrees. The impact analysis shows a maximum stress within 380 Mpa which is surface stress. The average stress in spoke cross section is about 130 Mpa which is well within the limits. A rigid body is made to impact with the wheel which is tilted at an angle of 15 degrees

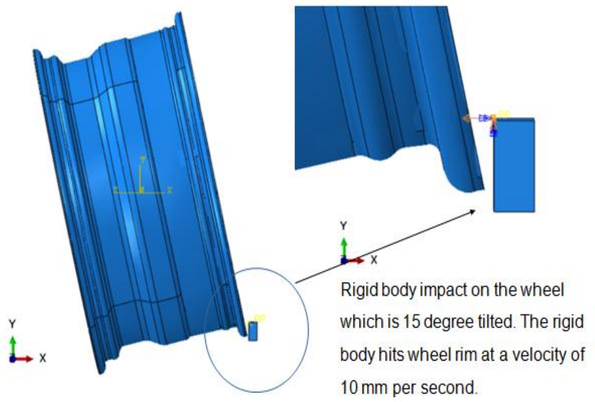

Fig-9: Impact test

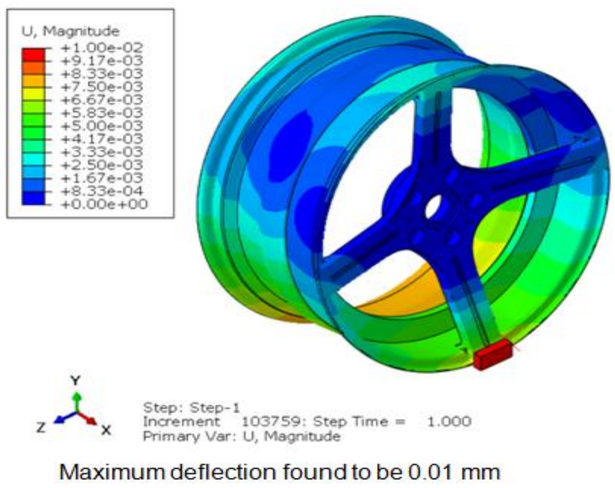

Fig-10: Maximum deflection obtained for impact test analysis

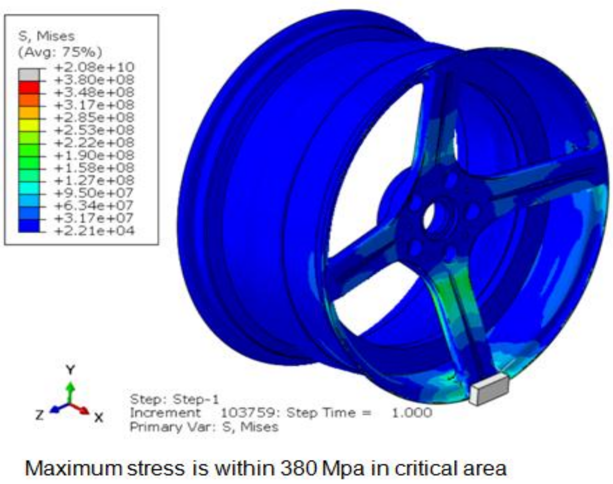

Fig-11: Maximum stress for impact stress

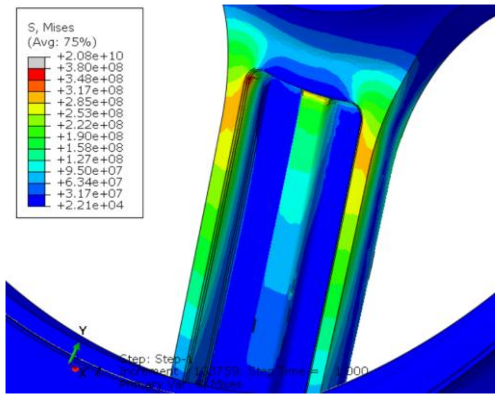

Fig-12: impact test analysis

3. CONCLUSION

The analysis of Alloy wheel of a MUV consisting of different scenario is done. The analysis results shows up stress levels within the required levels for radial and impact tests. However the corner tests failed due to high stress and propagating at the spoke which is not safe. The radial test analysis showed a low displacement and stress value of 0.093 mm and 31.8 Mpa respectively. This confers to the requirement of the standards and is well within the limits. The cornering test analysis shows a maximum deflection of 4.96 mm and stress of 642 Mpa which propagates. This is more than two times the ultimate limit of 380 Mpa. So further investigations are necessary to avoid failure. The impact analysis shows a maximum stress within 380 Mpa which is surfacial stress. The average stress in spoke cross section in about 130 Mpa which is well within the limits.

REFERENCES

[1] V. Karthi, N.Ramanan, J.Justin Maria Hillary "Design and analysis of sports bike wheel spokes". Volume 3, Special Issue 2, April 2014, IJIRSET, an ISO 3297: 2007 Certified Organization

[2] Sourav Das "Design and Weight Optimization of Aluminium Alloy Wheel", (CAE Analyst) Altair Engineering India Pvt Ltd, Bangalore. International Journal of Scientific and Research Publications, Volume 4, Issue 6, June 2014

[3] Praveen. S, Dayanand. H, Sangangouda.P, "Static Analysis of Alloy Wheel Using FEA".

[4] Rajarethinam P, Periasamy K made modification of design and analysis of motor cycle wheel spokes. IJMER.

[5] K. Srinivasa Rao," Design and analysis of alloy wheels" Volume: 04 Issue: 06 | June -2017 IRJET.