ISSN 2250-3153

Design and Weight Optimization of Aluminium Alloy Wheel

Sourav Das, (CAE Analyst)
Altair Engineering India Pvt Ltd, Bangalore

Abstract- This paper deals with the design of aluminum alloy wheel for automobile application which is carried out paying special reference to optimization of the mass of the wheel. The Finite Element analysis it shows that the optimized mass of the wheel rim could be reduced to around 50% as compared to the existing solid disc type Al alloy wheel. The FE analysis shows that the stress generated in the optimized component is well below the actual yield stress of the Al alloy. The Fatigue life estimation by finite element analysis, under radial fatigue load condition, is carried out to analyze the stress distribution and resulted displacement in the alloy wheels. S-N curve of the component depicts that the endurance limit is 90 MPa which is well below the yield stress of the material and safe for the application. The FE analysis indicated that even after a fatigue cycle of 10^{20}, the damage on the wheel is found only 0.2%.

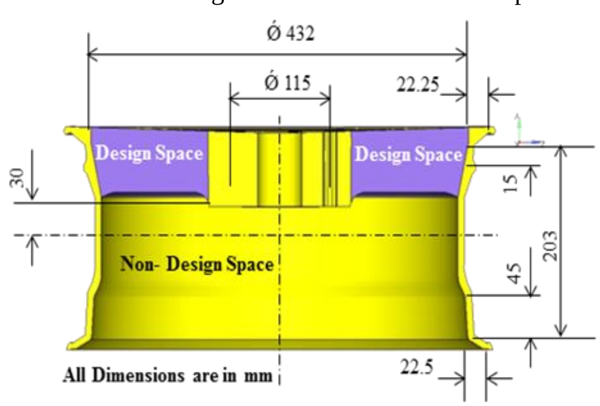
Index Terms- AlSi7Mg0.3, wheel rim, Design optimization, stress analysis, weight optimization, fatigue analysis.

I. Introduction

A sport utility vehicle or suburban utility vehicle (SUV) is similar to a station wagon or estate car, and are usually equipped with four-wheeled drive for on- and-off road ability. Automobile Wheels are classified into many types based on their complexity / simplicity and their material strength to withstand worst loading conditions. In the case of heavy loading condition steel wheels (density: 7.8 g/cc) are preferred and for medium and low load condition Al (density: 2.7 g/cc) and Mg (Density: 1.54 g/cc) alloy wheels are suggested essential for aesthetic look. However, in any type of wheel, the basic construction is consisted of a rim, a hub, spokes/arms/wires and tires. Various wheel specifications used for design are PCD, height, offset distance, bead width, humps, drop centre etc. Casting process such as low and high pressure die casting is used widely to make the wheels. Forming processes such as forging, extrusion etc are also being used for making the wheels. A new extrusion process has been developed recently for making automobile wheel out of AZ80 Mg alloy (1). Conducting various tests such as radial fatigue, impact and bending fatigue confirm that AZ80 Mg alloy can meet application requirement of wheel in automobile (1). Additionally, casting and forging processes have been used for the manufacture of Mg alloy automobile wheel (2-5). It has been mentioned that the most accepted procedure for car wheel is to pass through the tests such as radial and cornering fatigue test (6). The recent introduction of alloy wheel for car, which has more complicated design and shape than a regular shape, needs prediction of fatigue life by analytical methods rather than a regular test. Limited research has been carried out on the analysis of wheel disc using finite element analysis (7-9). Ramamurty et. al. (10) have studied the fatigue life of aluminium alloy wheels under radial loads and reported that the predicted fatigue life of wheel is found to be in close agreement with the experimental observations. Gope (11) has reported that minimum of three specimens are needed to predict the fatigue life using log normal distribution. Wang et.al. have (12) analysed the fatigue life by finite element simulation. ABACUS Software was used for building the static load finite element model. The results of Al alloy wheel rotary fatigue bench test showed that the wheel failed and the crack initiated around the bolt hole area which is closely agreed with the prediction by simulation. It was also reported that during the assembly of wheel disc, considerable amount of stress is developed in the component and alters the mean stress value. Guo et.al (13) have reported that inclusion of clamp load improves the prediction of the critical stress area and fatigue life. In the previous study, it is observed that in most of the cases fatigue life estimation and prediction of suitability of alloy for wheel disc is carried out; however no attempt has been made for mass optimization and design of alloy wheel. Hence, in the present investigation an attempt has been made to analyse the alloy wheel from a solid disc shape to an improved design which resulted into use of less requirement of mass of material with improved design. The objective of this paper is to design an aluminium alloy wheel by meeting all the design standards. In this paper, the area between the rim and the hub is considered for optimization. Topology Optimization has been carried in 5 cyclic cases where the loading conditions are similar for every 72°. This new optimized design is analyzed under radial, bending and lateral loads to determine the stresses induced in static condition of the wheel of automobile. The succeeded model is used to evaluate to determine its life period under radial loading condition.

CAD Design of Wheel

The CAD design of wheel is prepared based on the standard nomenclature at the outer and the hub region of the wheel. Figure 1 shows the CAD design of the wheel rim before optimization



ISSN 2250-3153

Fig. 1 CAD design of rim before optimization

Material used

In the present investigation Al-Si (B.S.: LM25 alloy) is used. The alloy mainly consisted of 6.5-7.0.0%Si, 0.3-0.4% Mg and rest is Aluminium. The properties of the alloy used is in heat treated (T6) condition. The properties of the alloy are shown in Table 1. The microstructure of the alloy in heat treated condition shows primary Al and near spherical eutectic Si. Fine precipitate of Mg_2Si is responsible for improved properties.

Table 1. Material properties of LM25 aluminum alloy in T6 condition (14)

material properties	**Magnitude with units**
Tensile Stress	230 MPa
Endurance Limit	56 MPa
Modulus of Elasticity	71 (GPa)
Shear Strength	120 MPa
Tensile Yield Stress	185 MPa
Compressive Yield Stress	185 MPa
Elongation (%)	4
Density	2.685 g/cm^3 at 20°C

II. Finite Element Analysis

The FE model is prepared for 36° of its circumference as the remaining part can reflected exactly. The required modifications can also be done in the same portion. Hexahedral and pentagonal elements are used for modeling. Finite element modeling and analysis is carried out using **Hypermesh**. Element size of 5 mm is used for meshing with 100118 elements and 100200 nodes. Figure 2 shows the 2D and 3D elements of the wheel rim and the cross section.

Fig 2: 2D and 3D element representation of Rim and its cross section

Some of the elements are deleted in between to maintain the average element length of 5mm. The FE model prepared for 36° of the rim is rotated completely as shown below.

Loading conditions

Optimization is done to reduce the material consumption hence to reduce the weight of the wheel. Hence the loading conditions were considered based on the automobile weight applied over it. Each wheel in an automobile will carry the load by distributing among them. This load is considered to be along the radial direction and applied it in optimizing the model for mass. Figure 3 shows typical radial Loading of 9000 N.

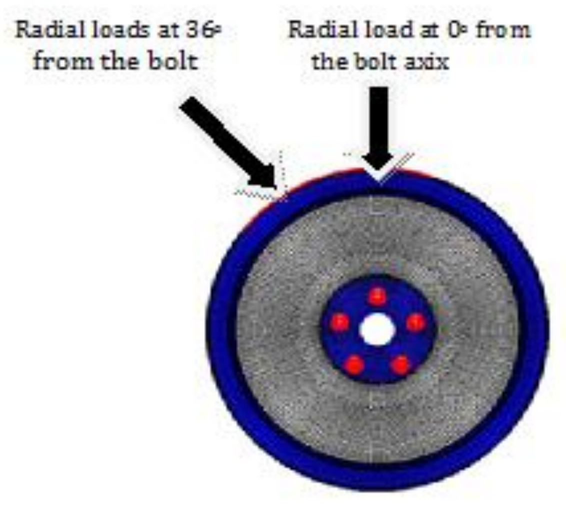


Fig. 3 Radial Loads with respect to Bolt Location
Model Setup for Optimization

The loads on the wheel are transferred by using RB3 elements. The space required optimizing and the standard design space is segregated. The model is optimized by using topology optimization design variables. Figure 4 shows the constraints and loading locations.

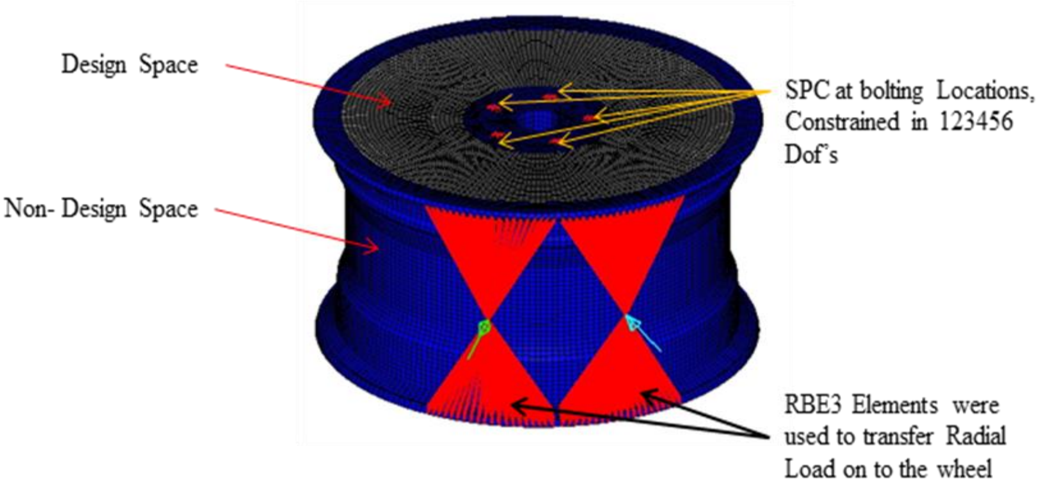
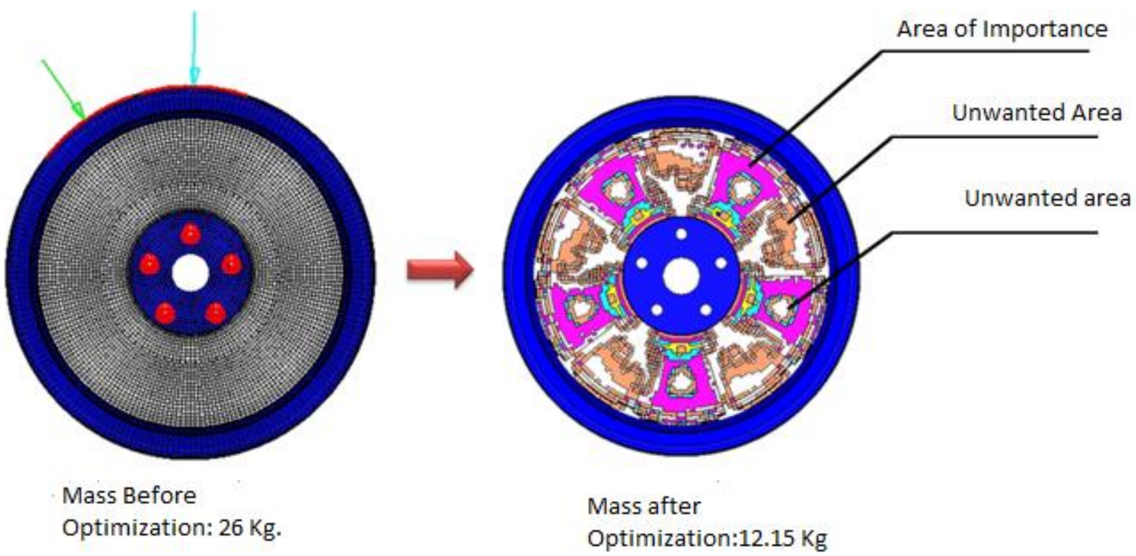


Fig. 4: Model Setup shows the constrains and loading location

Weight Optimization

In the recent days considerable efforts are being made to reduce the weight of the components which ultimately reduces the overall weight of the vehicle. It is observed that a proper design brings about useful shape to carry the load applied on the system distributed in a manner to sustain the applied load and which intern reduces the weight of the component**.**

Optimization of the wheel rim is done through Hypermesh – optimization solver. Optimization is carried out taking special reference to the minimum material requirement to sustain the stresses applied on the wheel during operation. Figure 5a shows the shape of the wheel rim before optimization and Fig. 5b shows the shape of the wheel rim after optimization. Figure 5c shows the actual shape of the wheel. It is observed that the mass of the wheel rim was 26 kg of Al alloy and after optimization the actual mass required for the wheel rim is reduced to 12.15 kg of Al alloy. This shows that there is a reduction of 13.85 kg of Al alloy for making the components. This exercise clearly indicates that a proper optimization of wheel rim considerably reduces the useful mass of Al alloy required to make the component. This clearly shows that a proper optimization may leads to a minimization of material use to 52%. Considering, the cost of Al alloy of Rs. 300 per Kg resulted in a saving of Rs. 4000/- approximately by using proper optimization technique.



(a) (b)

ISSN 2250-3153

(C) Actual view of the wheel after optimization and removing the unwanted material.

Fig.5 (a) Initial size and shape of wheel rim and (b) Optimized size and shape of Rim (c) Optimize view of the wheel rim after removing the unwanted material.

Boundary conditions

In order to do the Finite Element Simulation, one has to consider boundary conditions. In the present simulation following boundary conditions were used:

(i) Radial load of 8976 N was used
(ii) Lateral load of 4044 N was used
(iii) Bending load of 4488 N was used.

The wheel fixed to a vehicle experiences different loads due to its position. The loads applied on wheel are divided into six sub – cases and acting on the wheel as depicted in the Figure 6. All the bolt holes are constrained with rigid spiders and connected to the centre of the drive shaft. The driveshaft is represented with a single RB2 element.

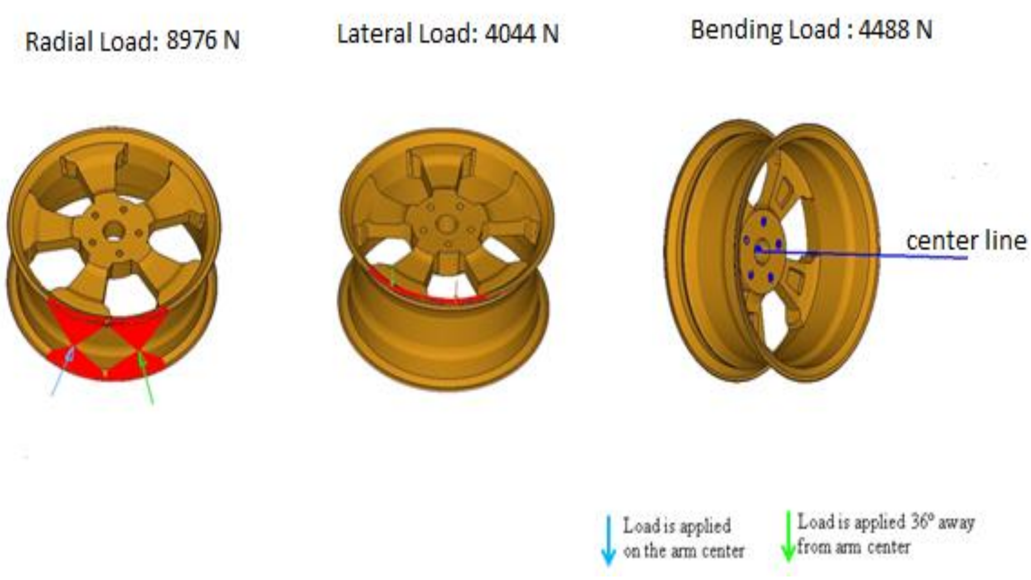
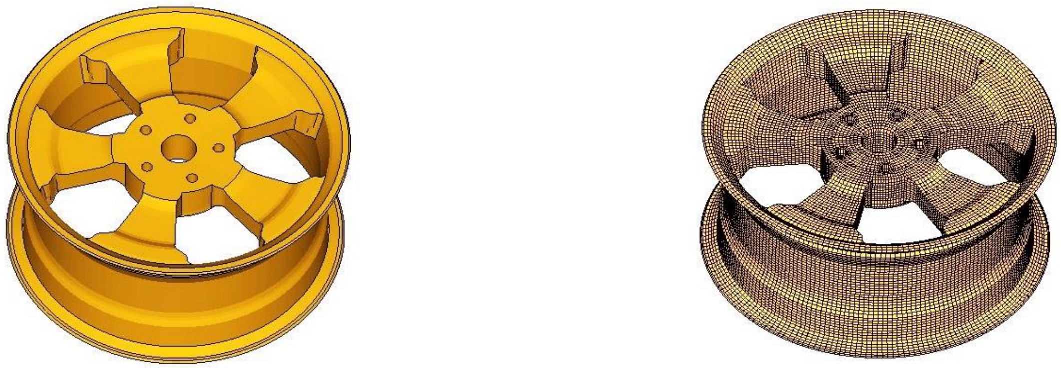


Fig. 6 Boundary conditions for applied load on the wheel

Static Analysis of New Design of wheel

On the basis of optimized results, CAD model was developed. Figure 7a shows the typical CAD model of the wheel rim. For carrying out the finite element analysis, mesh was developed using Hyper mesh. Figure 8b shows the mesh model of the optimized wheel rim. Based on the optimization result, the new design of cad and its FE model is created as shown below:



(a) (b)

Fig.7 (a) CAD model and optimized design of the wheel rim (b) Mesh model of the wheel rim for finite element analysis.

Static Analysis

Static analysis is carried out using two loads in each loading condition namely radial load, lateral load and bending load. The stress and displacement on the wheel at each loading condition were found out. Table 2 shows the stress experiences by the wheel rim and the displacement occurred due to stress on the material. It may be noted that the stress value is well below the yield stress of the Al alloy.

Table 2: Results of Static Analysis

LOAD TYPE	**STRESS (MPa)**	**DISPLACEMENT(mm)**	**Remarks**
Radial Load Case 1	94.56	0.73	ok
Radial Load Case 2	93.00	0.71	ok
Lateral Load Case 1	58.86	0.32	ok
Lateral Load Case 2	63.47	0.29	ok
Bending Load Case 1	34.33	0.394	ok
Bending Load Case 2	34.33	0.394	ok

Stress and Displacement contours

(i) Radial Load

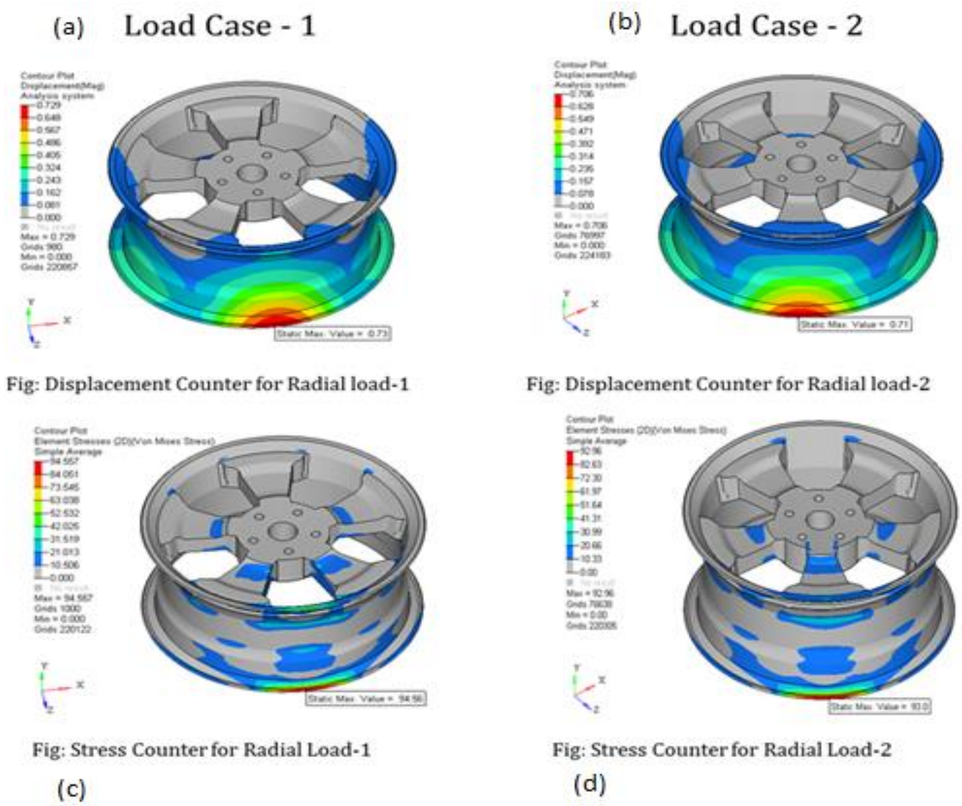


Fig. 8 Stress and displacement contours for the radial load applied on the wheel rim.

Figures 8 (a & b) show the displacement contours of the wheel rim under radial load condition. The stress distribution of wheel rim of passenger car under radial load condition is done to assess the vehicle condition mainly in off road field area and uneven road. It is noted from the Table 2 that a displacement of 0.73 is obtained by applying a radial load of around 94 MPa. The FE analysis could reveal that the rim flange has the maximum displacement. Figures 8 (c) & (d) show the stress profile by applying the radial load. The maximum stress felt by the wheel rim is around the rim flange area.

(ii) Lateral Load

Figures 9 (a&b) shows the displacement contours of the FE analysis of the wheel rim when applied lateral load. A displacement of 0.32 is noted by applying the lateral load. A stress of 58-62 MPa is experienced by applying the load. This value is much below the yield stress of the material. Figures 9c & 9d show the stress contour for the application of lateral load. It also shows that rim flange of the wheel felt maximum stress.

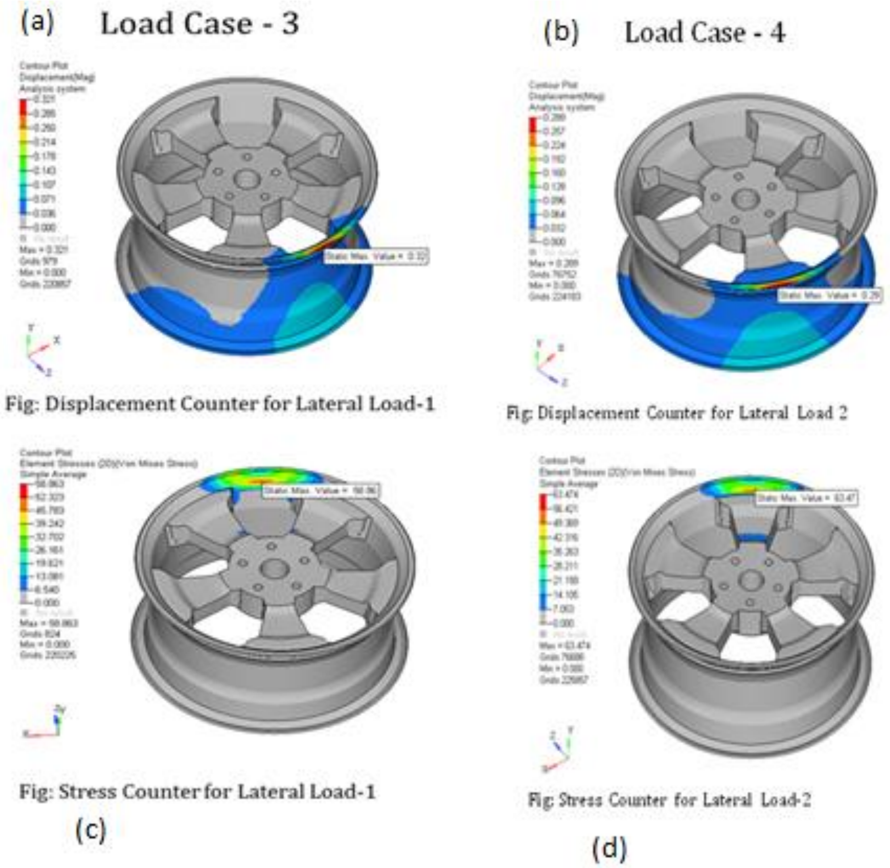


Fig. 9 Stress and Displacement contours of the FE analysis for Lateral Load

(iii) Bending Load

Figures 10 a&b show the displacement contours of the wheel obtained by FE analysis . The bending load applied is around 4488 N and the displacement obtained by FE analysis is 39 mm. which experiences near the sproket. The stress contour is showed in Figs. 10c & 10d. It is observed that due to bending load the maximum stress generated is 34 MPa at the sproket region. The stress value obtained is well below the yield stress of the Al alloy selected in the present study.

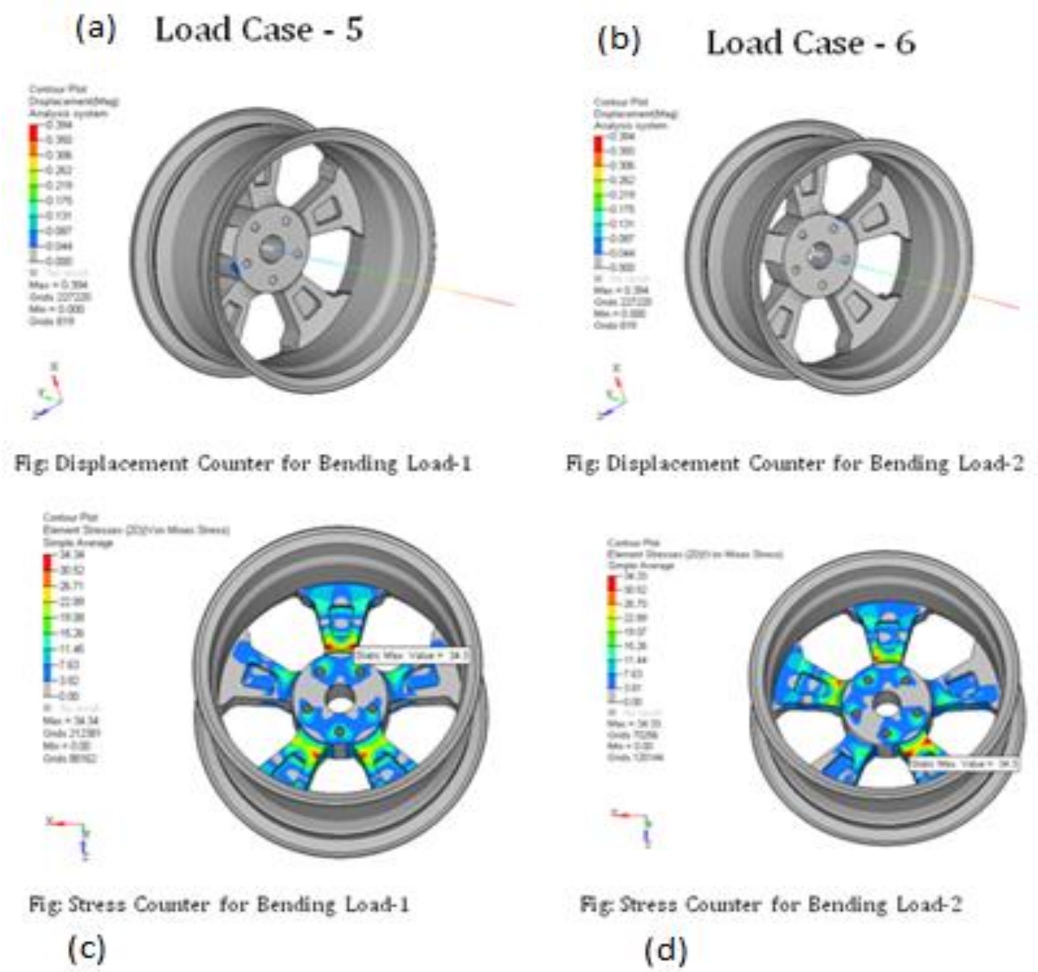


Fig. 10 Displacement contour and stress of the wheel due to Bending load

Fatigue analysis

Fatigue is one of the important material properties where the component subjected to cyclic loading. Due to cyclic loading, the materials undergone stress and nucleate micro cracks essentially on the surface of the component. It is understood that tensile stress is always assisted in nucleation and propagation of cracks and leads to failure. In the present investigation S-N curve (fatigue curve) is generated through FE analysis for the material used in the present investigation. It is noted from the Figure 11 that for high cycle fatigue test, of the material selected in the present study, the endurance stress is found 90 MPa which is well below the yield stress of the material.

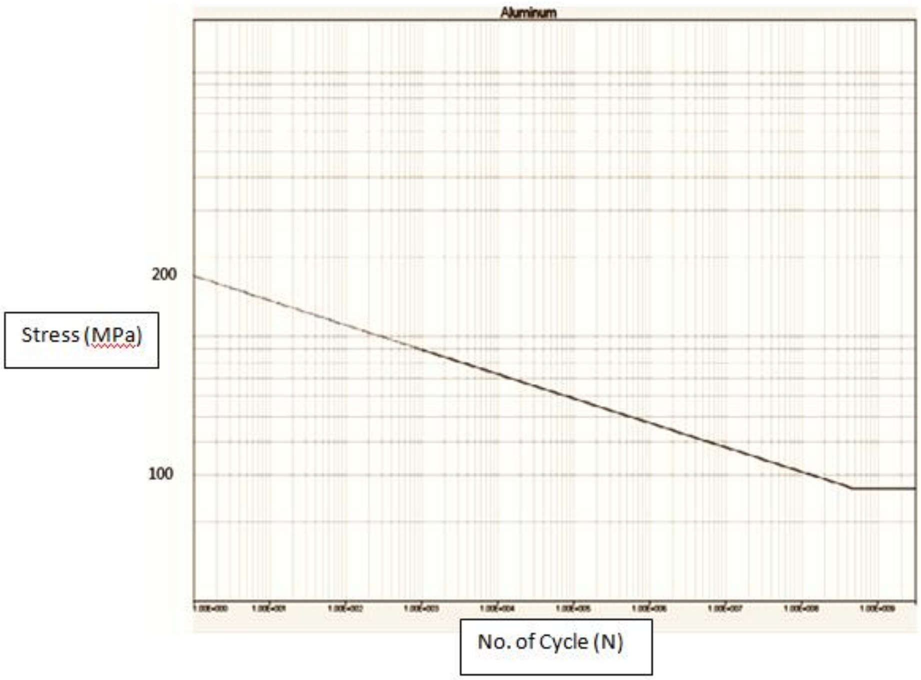


Fig. 11 S-N curve of Al alloy from FE Analysis

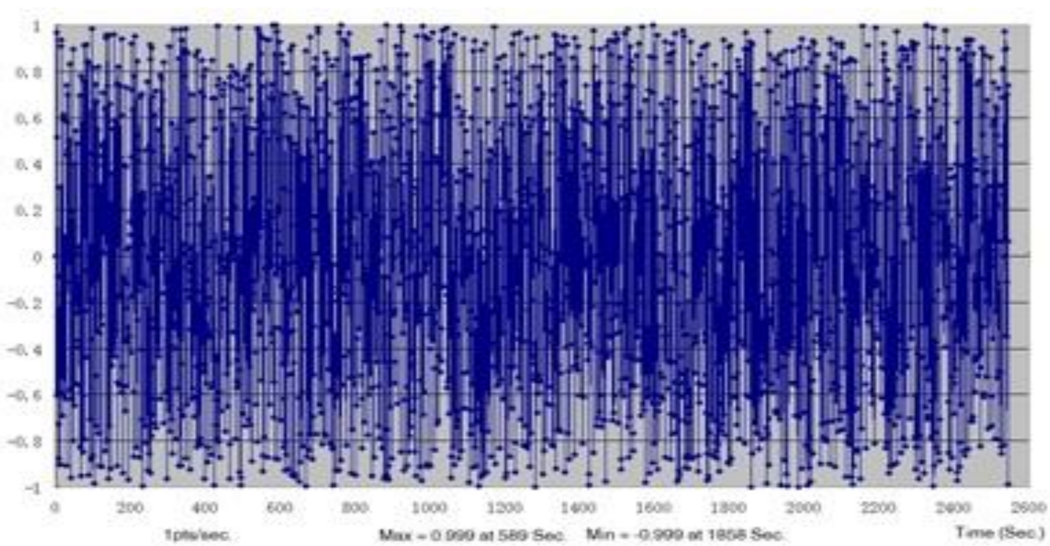


Fig: 12 Load -Time history from FE Analysis

The load-time history obtained from the FE analysis is shown in Fig. 12.

Damage Analysis

The development of micro-cracks on the wheel and damage under the stress factors which deform the material has been found out by FE analysis and shown in Fig. 13. It is noted that the probability of damage of the wheel is around the flange as shown by FE analysis. Based on the FE analysis one may mentioned that the dimensions of the flange section should be enough to sustain the stress. In order to save guard the failure of material one should look into the design aspects and also the strength properties. It is also noted that after a fatigue cycle of $1x10^{20}$ the damage is only 0.2%. This analysis indicated that even after a fatigue cycle of 10^{20}, the damage on the component is only 0.2%. Fig. 14 shows the FE analysis after life cycle. The fatigue analysis results obtained is shown in Table 3.

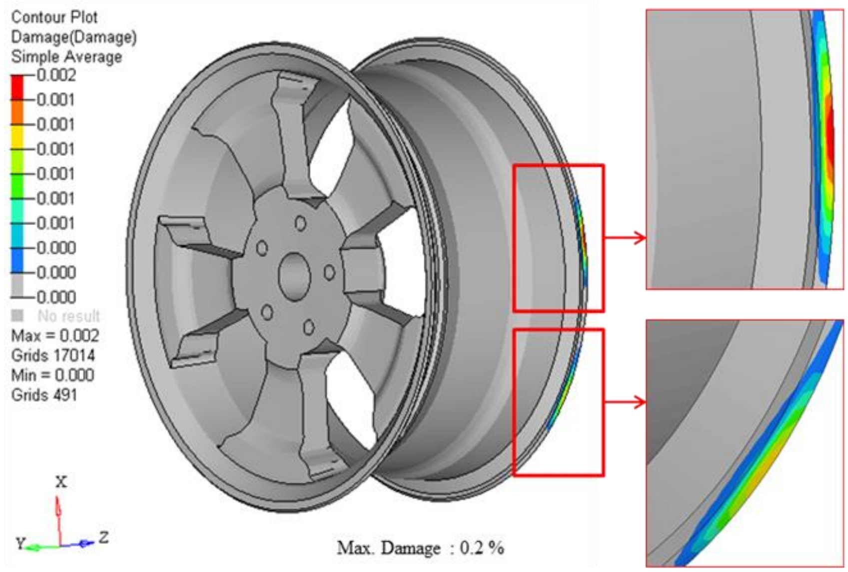


Fig 13 FE analysis shows the damage area which is 0.2%

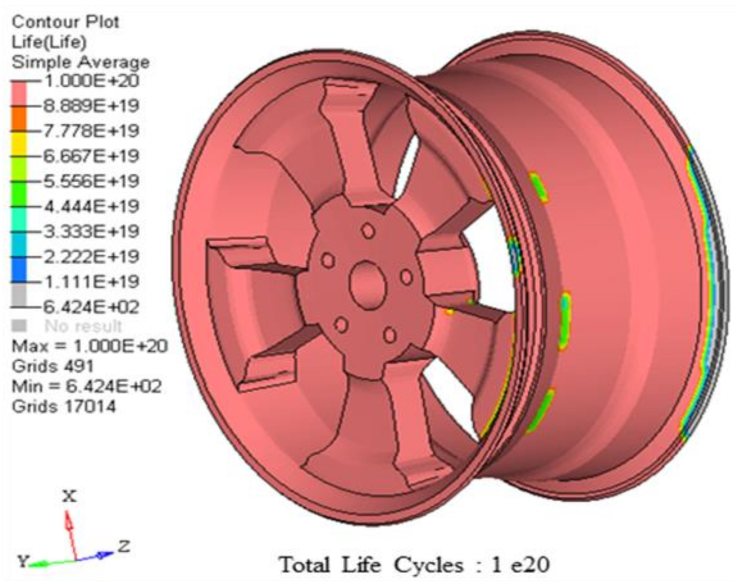


Fig: 14 Life Cycles of Component obtained from FE analysis.

Table 3 Fatigue analysis results of the component.

FATIGUE ANALYSIS RESULTS

Subcase	Element	Damage	Life
7	390942	2.067E-03	4.8387E+02
7	390841	1.978E-03	5.0546E+02
7	391031	1.913E-03	5.2267E+02
7	390411	1.700E-03	5.8814E+02
7	397018	1.608E-03	6.2182E+02

AVERAGE FATIGUE ANALYSIS RESULTS

Subcase	Damage Top 0.1%	Life Top 0.1%	Damage Top 1.0%	Life Top 1.0%	Damage Top 5.0%	Life Top 5.0%
7	1.178E-03	8.490E+02	2.035E-04	4.913E+03	4.080E-05	2.451E+04

III. Discussion

Alloy wheels are usually made of aluminum or magnesium alloys. The advantages of alloy wheel are lightweight, better heat conduction and excellent aesthetic appearance. Because of these interesting properties alloy wheel gaining popularities. Although Mg alloy wheels are developed in 1960 but due to lack of ductility, the alloy wheel lost their favour in vehicle. In the recent technological advancement, considerable attempts are being made to develop Al and Mg alloys to suit to requirement for alloy wheel. Further weight reduction of the wheel hub may be realized by design. Weight reduction is one of the essential criteria of a vesicle as far as the fuel efficiency is concerned. Use of light weight metals and alloys is now becoming the designer's choice. The existing components which are made of steel and cast iron are slowly phased out by replacing with Al and Mg alloys having comparable properties. Considerable research is now being done to enhance the strength properties of Al and Mg alloys so as to find them an ideal replacement of heavier counterpart. Al-Si (LM25) is one such cast alloy having comparable strength, fatigue, Young modulus, yield stress and other related properties and finding place to replace steel and cast iron. In the present study LM25 alloy is used as a potential material for automobile wheel rim application. Zhao et.al (15) have studied the A356 Al alloy for wheel application. A356 Al alloy is similar to the LM25 alloy which was studied in the present investigation. Their work has emphasized that a sound casting is desirable for wheel hub application. Casting defects such as porosity and inclusions resulted into deteriorate the properties and would not fit for wheel hub applications. CAD is developed using Hypermesh FEA software with standard wheel design norms. The raw design of rim (i.e., non design space) is optimized for mass compliance at cyclic loading conditions. Aluminum Alloy Grade - AlSi7Mg0.3 T6, often known as A354 (LM25) is used in this wheel design. The analysis result is found within the yield stress limit and found safe.

A cyclic radial load of 8976 N is applied for every 36 degrees (10 times) because 5 arms are assumed to be in between rim and hub. This load transfer is done using RBE3 element. The first load is applied exactly normal to the wheel and bolt axis and the second load is applied at 36° from the first load in normal direction to wheel axis.

Optistruct solver is used for computing this problem. The weight optimization analysis shows that there could be a way to reduce the weight of the wheel rim from 26 kg to 12.15 kg. This clearly indicates that there is an ample scope for weight reduction of the wheel rim by using right kind of analysis. A similar study showed that using lighter Mg alloy resulted in saving of energy to an extent of 11-15% the energy (16). While selecting the alloy one has to critically examine the properties and most importantly the elongation. Ravi Kumar and Satya Meher (17) have optimize the A356 Al alloy for wheel application using impact analysis and reported that Al alloy of strain value 4% is ideal for safe use. Strain value less than 4% may assist to crack nucleation and propagation of crack lead to failure of component. In the present investigation, the strain value is considered as 4%. Topology Optimization is carried out by changing the thickness of the rim of the Cast Aluminium Alloy Wheel until the value of the plastic strain is less than 4.0% and the optimized thickness is found 5.9 mm (17)

The new CAD design is developed from the optimized design by smoothing irregular features with fillets and edges considering area of importance available in the form of density range. FE model is build up from the new CAD and boundary conditions are setup for static analysis. To validate the design, model is setup for three loading conditions such as radial, lateral and bending loads. Since the model is cyclic for every 72 degrees (and 72deg of model is symmetric for 36deg), instead of applying load for 10 times, 2 load cases are applied at arm location and between two adjacent arms. The FE analysis confirms that the stress value experienced by the wheel in all load condition is well below the yield stress of the material. The model is checked for the life period under fatigue radial loads and the same loads are applied in static analysis also. In fatigue analysis, the input load is considered as only radial load. The radial load is checked for number of possible fatigue cycles. Cerit (18) has reported based on the simulated impact test that lug hole is the region of initiation of crack and failure. Satyanarayana and Sambaiah (19) have studied the fatigue study of Al alloy wheel under radial load and reported that rim is to be design properly as maximum load is experiencing at the rim area. The possibility of failure is high at wheel spokes. In the present investigation the model is checked for 2500 amplitudes for analysis and solving. The fatigue analysis results clearly show that material is sustained for 1e20 cycles. Maximum damage occurred till the end of the 1e20 cycles is 0.2%. Present study also inferred that spokes area is the most vulnerable for failure of the wheel hub due to fatigue.

IV. Conclusion

1. In the optimization of wheel rim, the wheel structure and its features are divided into two parts, namely design space and non design space. The non design space is the standard design and cannot be modified. The design space is the region for optimizing the weight and shape of the arms. The wheel design space is optimized in order to withstand the existing load of the vehicle with the factor of safety with a least quantity of material and manufacturing cost and losses. The five arm structure is the optimal output of the solver to withstand stresses. Conclusions traced out during the optimization and evaluating the life of the wheel are as follows:

- The weight of rim is optimized from 26 Kg to 12.15 Kg using topology method.
- The shape of the arm's cross section is made easier to manufacture and to distribute the stress induced in the rim.
- The optimized design is analyzed to withstand all the loading conditions acting upon it, such as:

- Radial load used is 8976 N and the maximum stress induced in the wheel is 94 MPa which is less than the yield stress of the material suggested i,e., 185MPa
- Lateral load used is 4044 N and the maximum stress induced in the wheel is 64 MPa which is less than the yield stress of the material i,e., 185MPa
- Bending load used is 4488 N and the maximum stress induced in the wheel is 35 MPa which is less than the yield stress of the material i,e., 185MPa

2. The damage region is found around the flange portion of the rim

3. The fatigue analysis results clearly show that material is sustained for 1e20 cycles.

Maximum damage occurred till the end of the 1e20 cycles is 0.2%.

References

[1] Wang Qiang, Zhang Zhi-min, Zhang Xing, Li Guo-jun, Trans. Nonferrous Met.Soc.China 20 (2010)599-603

[2] Wang Jian-hong, Long Si-Yuan, Cao Han-xue, Special Casting & Nonferrous alloys, 2004(5) 21-23

[3] Peng Ying-hong, Wang Ying-chun, Li Da-yong, J China Mechanical Engineering 2006 17(19) 2034-2037.

[4] Wu Zeng-chen, Long Si-yuan, Xu Shao-yong, j.Foundry 2005 54(9), 878-880

[5] Cai Suo-qi, Cui Er-xin, J Foundry Technology, 2001 (5) 8-10

[6] JIS D 4103, Japanese Industrial Standard, Disc Wheel for Automobiles, 1989.

[7] Grubisic V, Fischer G, SAE Technical Paper Series 830135; 1984: 1.508-1.525

[8] Hsu YL, Wang SG, Liu TC, J Chin Inst Industrial Eng 2004; 21 (6), 551-558

[9] Sunil N Yadav, NS Hanamapure, Int Jr. of Engg Sci, and Innovative Tech. Vol 2 Issue 5, Sept (2013) 213-239

[10] P. Ramamurty Raju, B. Satyanarayana, K. Ramji, K. Suresh Babu, Engineering Failure Analysis, 14 (2007)791-800.

[11] PC Gope, J Engg Mater Technol., 2002,124:421-7

[12] Liangmo Wang, Yufa Chen, Chenzhi Wang,Qingzheng Wang,J Mechanical Engg, 57 (2011)31-39

[13] Guo M, Bhandarkar R, Lin B, Society of Automotive Engg., 2004Warrendale, Pennsylvania.

[14] Light Alloys, I.J. Polmear, Edward Arnold Publishers15.

[15] Wei Min Zhao, Liang Zhang, Zhi Feng Wang, Hong Ji Yan, Advanced Materials Research vol. 189-193 (2011) 3862-3865.

[16] Jian Li, Hui Xue Sun, Shao Ming Sun, Xin Xin Liu , Applied Mechanics and Materials (Volume 456) October, 2013 ,65-68

[17] Ch.P.V. Ravi Kumar, R. Satya Meher, International Journal of Modern Engineering Research (IJMER) Vol. 3 , Issue.3 May – June 2013 pp 1548-1553

[18] M. Cerit, Scientific Research and Essays, Vol. 5(18) September 2010 pp. 2694-2701

[19] N. Satyanarayana & Ch.Sambaiah, International Journal of Mechanical and Industrial Engineering (IJMIE), ISSN No. 2231 –6477, Vol-2, Issue-1, 2012, pp1-6.

Authors

First Author – Sourav Das, **CAE Analyst**, Altair Engineering India Pvt Ltd, Bangalore. Email: souravds21@gmail.com