Optimisation of Wheel by Wheel Disc Flow Forming Technology

Dhondiram Pawar[1], Vidyasagar Bajaj[2]

[1]*ME Student, Department of Mechanical Engineering, Alard College of Engineering & Management, Pune, India*

[2]*Head of Department, Department of Mechanical Engineering, Sinhgad Academy of Engineering, Kondhwa. Pune, India*

***Abstract*—The objective of this project is to reduce the weight of a wheel without affecting its primary functions such as fitment, air flow and fatigue life, the design is made with the concept of variable thickness, means reducing thickness wherever the stresses are low and also addressing the failures in the field. However, prototype manufacturing makes the validation process laborious, costly and time consuming. Finite Element Method (FEM) has emerged as a resourceful tool for analyzing various components under a variety of operating conditions. It is being used to predict the critical points bearing the highest stress in a wheel. This project focuses on the design of variable thickness disc and validation of the same using FEM and Rig testing. The analysis will be carried out on existing and proposed designs and the salient features were compared.**

***Index Terms*— Variable thickness; Prototype; Validationn; Rig testingo;FEM.**

I. INTRODUCTION

The wheel is one of those safety critical components in a vehicle, whose performance in the field as aspired by the end user is of vital importance. Over the past few years there has been a significant improvement in wheel design and its testing methods with real time testing condition using road load data. In this context and foreseeing the upcoming design requirements of the product it calls for weight reduction of the same without affecting its performance in the field. Weight reduction without compensating the performance of the wheel is a major challenge confronted by the automotive wheel manufacturers. Also validating the design becomes important before the product enters the market for its usage. The aforementioned weight reduction concern can be met through design change implementation by identifying the regions of higher stress and lower stress and material can saved where there is lower stress and where the load on the wheel has minimal impact. Experimental Stress Analysis (ESA) is becoming increasingly crucial for the validation of any new design of the wheel. One can know the exact stresses acting at a point on the wheel during its operation with the help of ESA, making it very helpful for further up gradation in design.

II. NEED OF RESEARCH

Validating any physical or functional prototype evolves as time consuming and expensive process as it requires a physical product or a replica to be made. Whereas validating a virtual product through Finite Element Analysis stands as a powerful tool for the same purpose as reliable as physical or functional prototype testing. Stress analysis using FEA can be very effective, especially when validating new designs as it is time efficient and development of prototypes can be avoided. Moreover it will give the understanding on Stress pattern that will help in redesign and values engineer the produced component. There still exists a significant variation in the stress results obtained by FEA and that measured by the conventional stress analysis using strain gauges. There is still considerable doubt over the FEA stress results, especially when it comes to the prediction of the life of the wheel. In the above context the result from FEA will be validated against ESA.

Objectives of the study

To Optimize the Existing Wheel rim for Weight reduction without compromising on the Fatigue strength and durability. To determine the New designed wheel rim fatigue strength in comparison with existing wheel rim Design & to validate the performance of the new wheel rim physical validation. Study of Stress Analysis comparison of Existing design & proposed design with the aid of ESA & FEA.

Methodology

The stresses in the existing wheel and the new wheel using Finite Element Analysis (FEA) were compared and the new design was validated. For the stress measurements, real time loading conditions based on road load data were used to stimulate the realistic conditions, the validation was carried out through Experimental Stress Analysis for the wheel physical model. The methodology of ESA evaluates stress pattern on a particular region of interest. Also the strain associated with the same, the reference states the experimental methodology to study stress distribution in wheel. The test to study the fatigue occurring in the wheel due to cornering (lateral) load called the Cornering Fatigue Test was simulated. Simulation of Bi-axial test using the dedicated software package for the same.

III. LITERATURE REVIEW

A. Review of Papers

S.Chaitanya and B.V.Ramana Murty presented paper on Mass Optimization of Automobile Wheel Rim [1]. It is based on optimization of wheel rim by changing the material with polymer matrix composite and taking the stress and weight into account. PEEK with 30%carbon reinforced is suggested as best material to replace the aluminium.

Sourav Das presented paper on Design and Weight Optimization of Aluminium Alloy Wheel [2]. 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. R.Muthuraj,R.Badrinarayanan and T.Sundararajan presented paper on Improvement in the wheel Design using realistic Loading Conditions–FEA and Experimental Stress Comparison.[3] The present study focuses on the validation of a new design of forged aluminum alloy wheel using ESA and FEM by comparing with the existing design. The analyses using both the methods were compared and correlated using Required Fatigue Strength (RFS) calculation.

R. Muthuraj and Dr.T.Sundararajan presented paper on the Forged Hybrid Wheel for Commercial Vehicles, A Robust Design for Augmented Product Service and Performance [4]. Forged Hybrid Wheel was developed to be used with tube type tires. This wheel overcomes the investment on tires, since the existing tube type tires can be used while replacement, and offers all the advantages of aluminum usage.As a consequence of reduced weight the fuel economy increases on an average 3% to 5%. Higher emissivity of the material of the product results in dissipating heat generated during service faster (almost double that of the steel) thereby improving the tire life still further. Having in view the cost difference between aluminum and steel, the pay back for the investment can calculated to less than a year. Not only with the tires but also this product does a cooler service with other mating parts thereby improving their life and performance. .

Haruo Nagatani & Tsuyoshi Niwa presented paper on Application of Topology Optimization and Shape Optimization for Development of Hub-Bearing Lightening [5] The shape optimization analysis executed for lightening hub bearings for reduced weight of cars. Shape optimization was centered on mechanical strength. However, it needs to set targets for hub bearing rigidity. Therefore, multi-face optimization, which attempt to promote further lightening while maintaining sufficient bearing rigidity. So that it can establish hub bearing technology to cope with the needs of various car manufacturers. Topology optimization analysis is essentially linear analysis, and does not offer sufficiently accurate results for a boundary nonlinear problem. For this reason, high-precision nonlinear analysis is necessary to verify the results obtained from topology optimization.

B. Outcome of Review papers

Many experiments were performed to know the FEM Analysis of Wheel rim, Optimization of Automobile component using topology optimization. Correlation of FEA result with ESA result. In none of the papers talks about Wheel rim optimization with variable thickness disc is mention. There is a clear scope for further analysis.

Fig.1 Wheel Rim 3D View

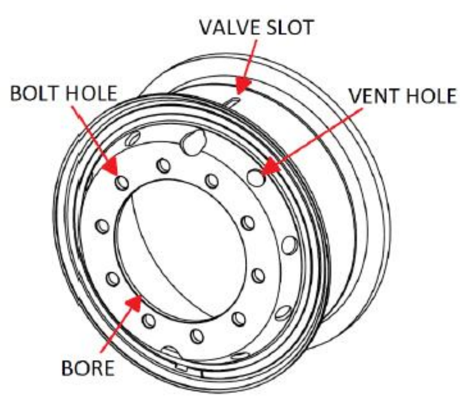

Fig.2 Wheel Rim Mounting Nomenclature

IV. OPTIMISATION OF WHEEL RIMO

There are significant and addressable advantages moving towards light weight wheels, the first and the foremost thing being the fuel economy of the vehicle, reduced vehicle un-sprung mass, reduced driver fatigue and from the view of the manufacturer – reduced raw material consumption and reduced cost play a significant role. Data from the field showed that the majority of the failures occur at the nave region of the wheel than the regions near vent holes and other areas. This study focuses on Design change implementation in the region identified to have failure and reduce the material in the regions having lower stress values. This project focuses on the Finite Element Analysis (FEA) of a new design wheel that has increased thickness at the region identified to have higher stress and gradual reduction of material thickness at lower stressed regions. The modification in the design is shown in the Figure 4, Figure 4 (a) shows the initially developed disc with uniform thickness (UT) and Figure 4 (b) shows the modified design of variable thickness (VT) in the disc. The present investigation is to reduce the localized high stress points by minor dimensional change pave way to reduce weight of the components without compromising their fatigue performance.

The following were the design changes incorporated to reduce the product weight and simultaneously increase the product performance:

- Increase in nave thickness to avoid crack initiation in the region due to fretting
- Variable disc thickness to reduce weight and cost of the product and increased cold working effect on the product
- Increased brake drum clearance through variable thickness.
- Rim Flow Forming
- Flange Shape Optimization

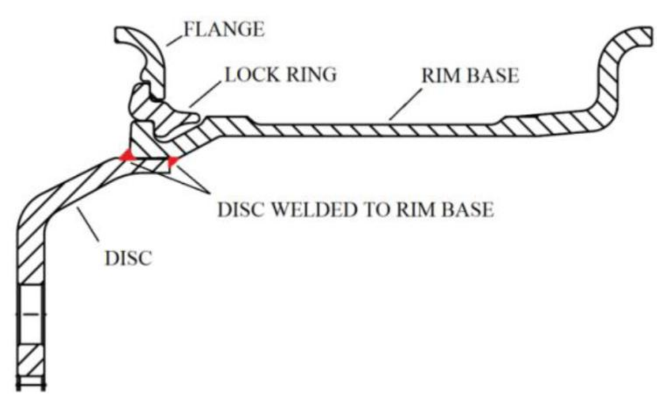

Figure-3 Wheel Rim Section View

Figure-4 (a) Uniform Thickness (UT) Disc Design

Figure-4 (b) Variable Thickness (VT) Disc Design

V. EXPERIMENT

A. CFT Simulation.

The wheel loading during and due to cornering and lateral load was simulated for required test conditions and results were obtained. For the stress measurement using FEA in wheels, the displacement at each node is evaluated according to the initial and the boundary conditions, from which the strain can be calculated and then, using Hooke's law, the stress at that point can be calculated using the Young's modulus of the material of the wheel. The FEA results show that the variable thickness (VT) disc had lower stress value than the regular wheel of uniform thickness (UT). For the same analysis the uniform thickness (UT) disc had a stress value of 170 MPa whereas the variable thickness (VT) disc had a stress value of 162 MPa. It is interesting to note that the peak stress is high at the vent hole region for existing products with the thickness of 11 mm. By reducing the thickness to 10mm and changing the disc profile, the stress level can be bought down by 8-10 %. This is achievable due to an advanced manufacturing technique called the flow form / spinning process. This process facilitated the vent hole position in a single plane with adequate ventilation area and the reduction in stress was also noticed.

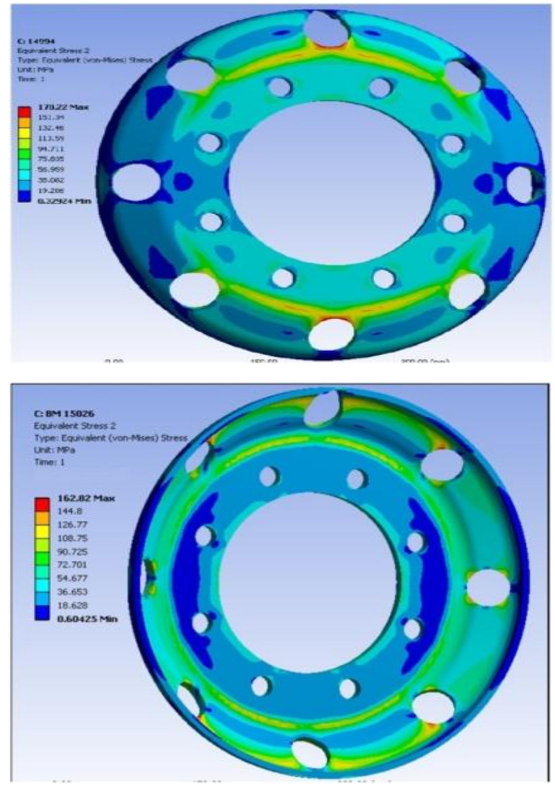

Fig 5: (a) CFT Stress distribution in UT Disc design

(b) CFT Stress distribution in VT Disc Design

VONMISES STRESS (Mpa)	2300Kg	2600Kg	3000Kg	3500Kg	YIELD STRENGTH (Mpa)
UT	130	150	170	197	380
VT	125	143	162	189	380

Table 1: Comparison of FEA Results

B. Bi-Axial Wheel Fatigue Test simulation

The two types of fatigue loads acting on the wheel i.e. cornering and radial were simulated and the results obtained were used to calculate the required fatigue strength of the wheel. For the present study, the FEA was carried out using a general purpose finite element soft simulation technique. Then, the virtual 3D wheel model was then pre processed to carry out analysis. The nave constraint is applied in the axial direction Supports were fixed at the bolt holes and the wheel bore so that all the degrees of freedom of the mounting face are constrained. The simulator software was used to load the wheel automatically, according to the tire that is used. This software was used to generate the load on the wheel according to the tire size 8.25 R 20 and Inflation pressure of 825KPa was applied. Rated Wheel Load of 3000 kg was applied at various discrete regions as required over the entire circumference. The nodes on which the load will be acting were then selected meticulously and the program was run. The realistic load for the required road condition for the above said load is incorporated in the software with the aid of built-In S-N curve for the material.

Fig 5 (a) is showing the stress plot of Uniform thickness (UT) disc design at 30KN load & Fig 5 (b) is showing the stress plot of Variable thickness (VT) disc design at 30KN load.

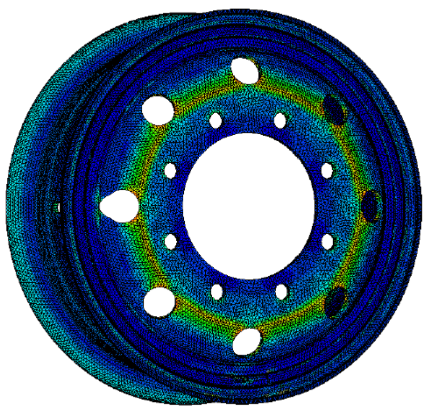

Fig 5: (a) Bi-Axial Stress distribution in Uniform thickness Disc design

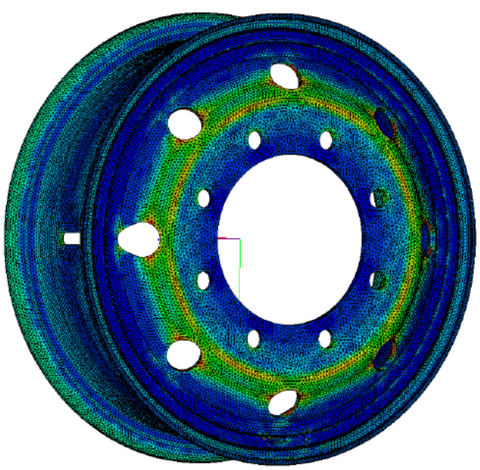

Fig 5 :(b) Bi-Axial Stress distribution in variable Thickness Disc Design

C. Experimental Stress Analysis

The Experimental Stress analysis was done for both the wheels namely Uniform Thickness (UT) and Variable Thickness (VT) design. The strain gauges were fixed in both UT and VT wheels, over the disc region of the wheel as shown in figures 6.

Initially, the wheel surfaces were polished along the disc region, where the strain gauges are to be mounted, with the emery paper (Grade 120 and then 100) to make the surface smooth. The polished surface was cleaned with a phosphoric acid based conditioner (M-prep Conditioner A), followed by neutralizer, which is comprised of ammonia water. The gauge attachment positions were marked such that there were 3 positions, Nave region, Access hole & Bolt hole, on the disc of the wheel (refer fig 6). The Uniaxial strain gauges (KYOWA Strain gages) were taken and placed on the marked lines by perfectly aligning them on the lines by checking their centers. They were bonded to the wheel using Anabond 202, a cyanoacrylate adhesive. Terminals were used and soldered to handle the wires easily as the length of the gage wires was not sufficient. The gauge resistance and the insulation were tested using gauge insulation tester to ensure that the gauge is properly fixed onto the wheel. Moisture proofing is done wherever necessary. The tyre was mounted on to the wheel very carefully from the rear side to avoid any damage to the strain gauges attached. MRF tyre with 8.25R20 was used for this study.

Picture showing Strain gauge located on disc

For stress analysis

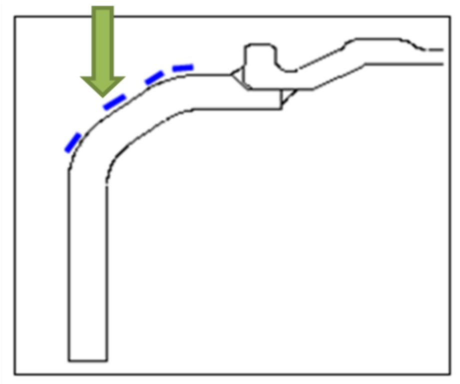

Fig 6: Cross section of Disc with Strain gauge positions.

The terminals of the strain gauges from the wheel were connected to the measurement system. The micro strain in the strain gauges was measured using Strainsmart software. Required calibrations were done before the measurement of strain. The tyre pressure was initially maintained at 825 KPa and the strain values were recorded at loads 30 kN, and an angles of every 60 degree. The average microstrains obtained across all the positions along the circumference is converted as stress values were obtained using the Young's modulus of Steel.

VI. RESULT & DISCUSSION

The ESA results for the uniform thickness (UT) & variable thickness (VT) wheel were obtained after taking the average

of the strain values obtained at the three different region.

In this chapter result of FEA & ESA is compared for three different region of wheel rim i.e. Disc region, Access hole region & bolt hole region.

All the Figure shows the graph plotted for the stresses at the different region (disc, access hole & bolt hole) from the microstrains obtained at 825KPa tyre pressure at loads 30 kN at every 60°rotations from reference to strain gauge location.

A) Below are the stress results observed in the vicinity of Disc region.

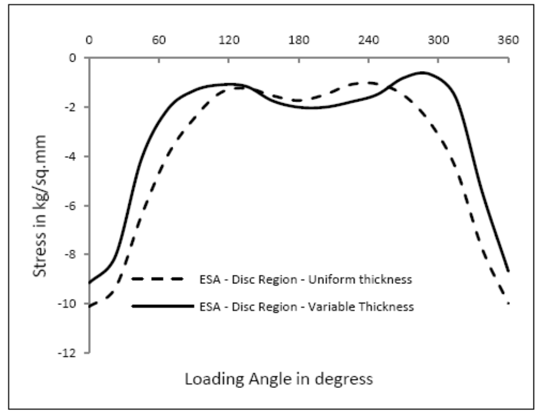

Fig 7: Stress distribution as per ESA in both UT & VT wheel

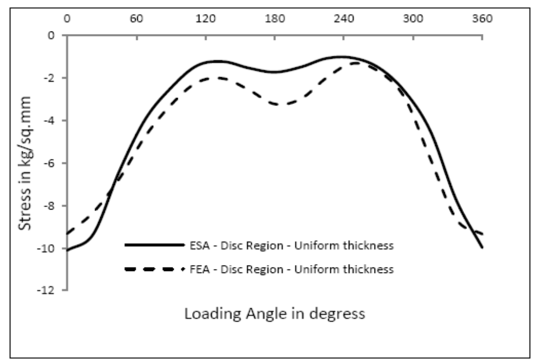

Fig 8: Stress distribution as per ESA & FEA for UT wheel

Figure 8 shows the correlation between the stress values obtained by Experimental Stress Analysis and Finite Element Analysis along the disc region of the uniform thickness UT disc wheel. Figure 9 shows the same correlation for the variable thickness VT disc wheels.

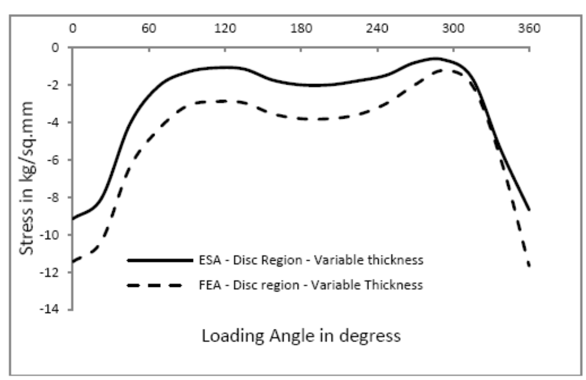

Fig 9: Stress distribution as per ESA & FEA for VT wheel

Both the experimental and simulated results show that the modified and improved variable thickness disc is capable of performing as intended with reduced weight and lowering the risk of crack initiation at the nave region.

B) Below are the stress results observed in the vicinity of access hole region.

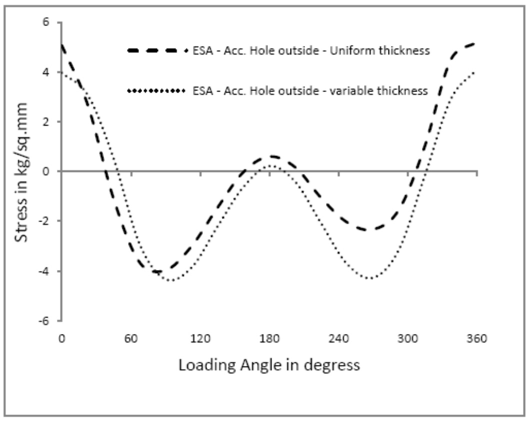

Fig 10: ESA result in the vicinity of access hole region.

It was observed that the strain values obtained using ESA for the wheel with the variable thickness (VT) disc design were significantly lower when compared to the uniform thickness (UT) disc wheel profile in access hole region.

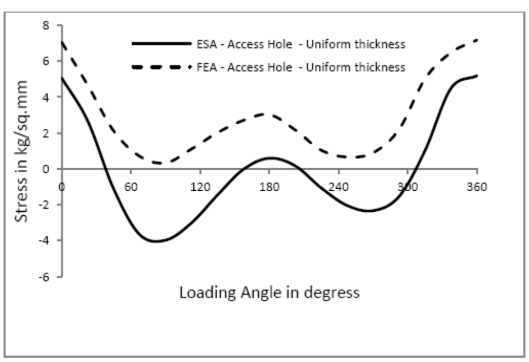

Fig 11: Stress distribution as per ESA & FEA for UT wheel at access hole region.

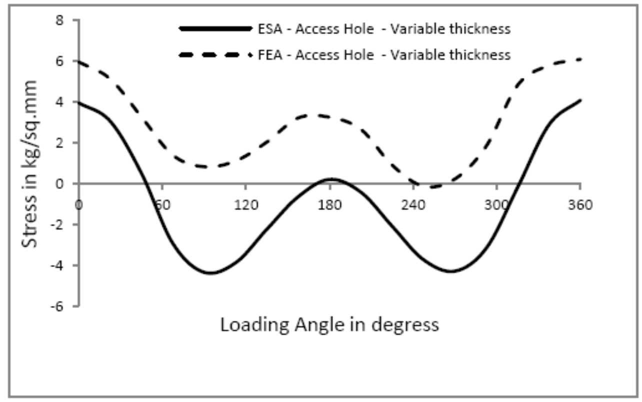

Fig 12: Stress distribution as per ESA & FEA for VT wheel at access hole region.

The stresses at the vicity of the vent hole region are at limits and the above plots show that the crack initiation from the vent hole region is significantly reduced in the improved version.

C) Below are the results observed in the bolt hole region.

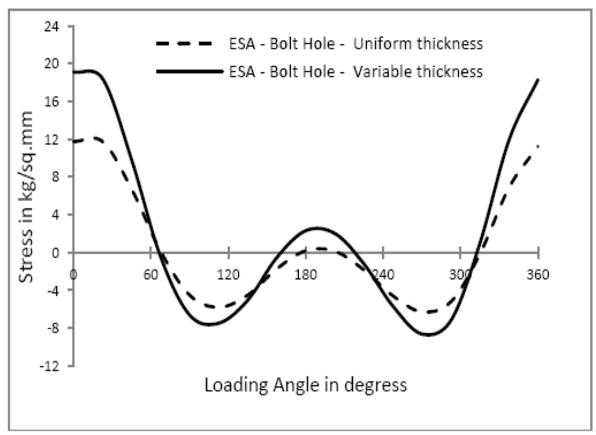

Fig 13: ESA result in the vicinity of bolt hole region

It is clear from the figures that the stress distribution along the wheel followed the same pattern in both the wheels. However, there is no significant change in stress amplitude level due to design modification.

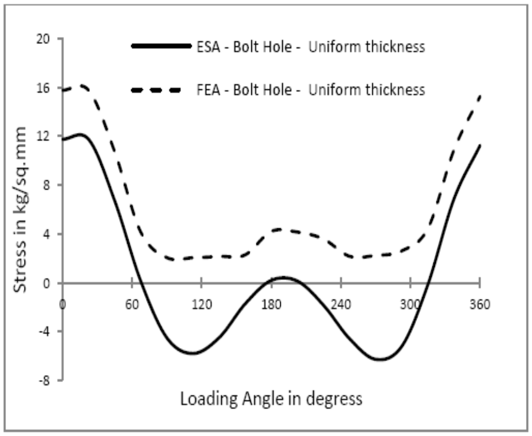

Fig 14: Stress distribution as per ESA & FEA for UT wheel at access hole region.

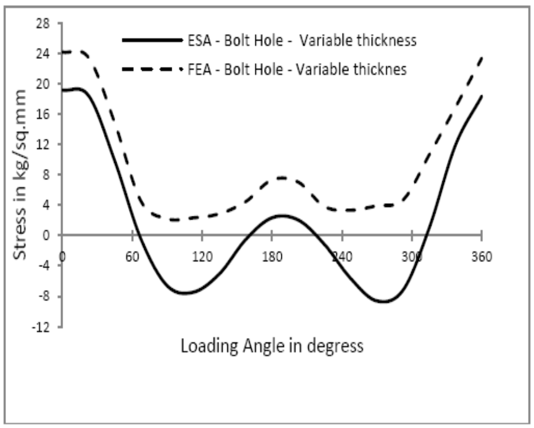

Fig 15: Stress distribution as per ESA & FEA for VT wheel at bolt hole region.

For one revolution of the wheel the stress pattern in the bolt

hole region is observed to be same between FEA and ESA.

D. RFS(Required Fatigue Strength)Calculation

To study the required fatigue strength of the wheel mathematical model used to relate radial load of a rotating body to its material endurance. The fatigue endurance limit of the new design is evaluated at around 260 MPa for Disc and 200 MPa for rim, considering their Ultimate Tensile Strength (UTS) to be 510 MPa and 400 MPa respectively. Exceeding this limit would detoriate the fatigue performance of the component. To have most robust component the required fatigue strength of the component should be well below this limit. This means the principle stress on the localized region should be as minimum as possible. Also the stress amplitude should be minimum. Following equation (Goodman's equation) is used to calculate the RFS (σf.) of the all above iteration. The realistic loading conditions of the wheel can be simulated to study the performance in field by the use of the equation

$$\frac{\sigma a}{\sigma f} + \frac{\sigma m}{\sigma UTS} = 1$$

Where,

σa - Stress Amplitude

σm - Mean Stress

σUTS - Ultimate tensile strength

σf - Required fatigue strength

Maximum RFS value (MPa) Comparison						
	DISC		ACCESS HOLE REGION		BOLT HOLE REGION	
	UT	VT	UT	VT	UT	VT
ESA	117.5	108.8	89.3	79.6	73.2	65.2
FEA	120.6	104.9	95.6	83.6	88.6	92.1

From the results of ESA & FEA, it was observed that the RFS value of the Wheel with the variable thickness (VT) disc design would perform as intended and will further facilitate increased loading application. From all the major peaks stress region it is revealed that the proposed design has RFS value much lower than their endurance limit

VII. CONCLUSION

The stress values for the wheel with the variable thickness disc design were much lower (5-10%) when compared to the earlier design. The design was hence successful in reducing the stress acting on wheels. FEA using road load data and tire model closely predict the experimental stress and the fatigue performance of the product. FEA results & ESA results were in good agreement to each other. It was found that Stress in the variable thickness disc is comparable with the uniform thickness disc. But in variable thickness disc will improve strength in nave region, vent hole region & Bolt hole region with reducing the weight & cost of Wheel rim. In this design 15% weight reduction achieved without compromising the performance of wheel rim.

ACKNOWLEDGMENT

I gratefully acknowledge Mechanical engineering department of ACEM, Pune for technical support and providing the research facilities. I would also like to thank to Dr. S.B. Padwal, Principal (ACEM, Pune) and Prof. V.R. Bajaj, HOD (Mechanical department) for their help and dedication toward my research and related research, also my friends for their directly & indirectly help, support and excellent cooperation.

REFERENCES

[1] S.Chaitanya and B.V.Ramana Murty "Mass Optimization of Automobile Wheel Rim", IJETT, 2015, 26 (3), pp 169-173.

[2] Sourav Das "Design and Weight Optimization of Aluminium Alloy Wheel", IJOSRP, 2014, 4(6), pp1-12

[3] R. Muthuraj, R.Badrinarayanan and T.Sundararajan, "Improvement in the wheel Design using realistic Loading Conditions – FEA and Experimental Stress Comparison", SAE Technical Paper Series 2011-28-0106

[4] R. Muthuraj and Dr.T.Sundararajan, "The Forged Hybrid Wheel for Commercial Vehicles, A Robust Design for Augmented Product Service and Performance", SAE Technical Paper Series 2015-26-0068.

[5] Haruo Nagatani & Tsuyoshi Niwa "Application of Topology Optimization and Shape Optimization for Development of Hub-Bearing Lightening", TN Technical Review 73, 2015, pp14-19

Reference Standard

[6] IS: 9438-Performance Requirements and Methods of test for Wheels/Rims for Trucks and Buses standard, 2000.

Reference Book

[7] Reference Manual –ITTAC, 2014.