Analysis of Automobile Rim on Strain criteria

S.S. Pote[1], R.S. Nimse[2], A.S. Kshatriya[3], G.D. Shinde[4]

[1]*Ass. Prof. Department of Mechanical Engineering, SIEM, Nashik 422213 (M.S.), India.*
[2,3,4]*Student, Department of Mechanical Engineering, SIEM, Nashik 422213 (M.S.), India.*

Abstract – *In this work the Multi Objective Analysis approach for Replacement of Spot welds by Arc weld on Automobile Steel Wheel Rim using Finite Element Analysis Method will be studied. Spot welded rim must pass certain tests like Weld Strength Test (WST), Dynamic Cornering Fatigue Test (DCFT) and Radial Fatigue Test (RFT). Here we studies the multiple objective for analysis of the rim like in Weld Strength test a shear force is applied on the spot weld using Universal Testing Machine. In dynamic cornering fatigue Test a moment is applied on the rim as specified by the company standards. In Radial Fatigue Test Influence of Tire pressure and vehicle load are studied. We choose three parameters for optimization namely, Length of Arc, Angle of Arc & Thickness of Arc. Finite Element Analysis Models are developed for the tests mentioned. Experiments of three tests will be carry out. FEA results and Experimental results will be compared for best combinations of selected parameters. Also, in DCFT by use of Finite Element Model we will study the pattern of stresses on the rim as well as the disc. In RFT using FEA method we will study influence of maximum tire air pressure and maximum loading conditions.*

Key Words: **Steel Rim, Strain Gauges, CFT, RFT.**

1.INTRODUCTION

Automobile rim is one of the critical part of vehicle during designing as it has to bear all the vehicle load. Till date Steel Rims are most preferred due to its low manufacturing cost. The conventional method uses Spot-weld method for joining the two parts of Steel Rim, but due to certain disadvantages of Spot-welding method now a days Arc-welding is also used. Wheels can be looked upon as safety related components. Consequently, fatigue performance and state of stress distribution in the rim, under various loading conditions, is a subject of concern. Furthermore, a comprehensive study of performance of the rotating wheel continues to receive significant importance as increased emphasis is laid on decreasing weight by either using lightweight materials or using materials of thin gauge. Although the loads applied on the rotating wheel are complex in nature and the resultant state of stress is usually high, the weight of the rotating body continues to remain as one of the most significant requirements desiring attention. This has necessitated the emergence and use of cast aluminum alloys in both existing and emerging rim designs. Lightweight rims made from a lightweight aluminum alloy are increasingly popular. Further, end-users consider the nature of the rotating wheel on their vehicle as a symbol of status. The sustained drive to reduce fuel consumption provided the impetus for car manufacturers to make rapid strides in altering traditional vehicle designs. Research efforts have found that a smooth outer wheel surface facilitates a reduction in air resistance. The conjoint influence of inflation pressure and radial load on stress and concomitant displacement distribution in the rim of a rotating body, the wheel, is studied. The influence of circumferential angle on stress and displacement distribution is also examined. Influence of tire inflation pressure on performance of the rotating body is rationalized.

1.1 Problem Statement

In traditional welding method wheel parts are joined with the help of spot welding. Production time required for wheel parts i.e. wheel hub and wheel rim is few seconds and time required for joining two parts by spot welding is few minutes therefore welding process requires much more time compare to production time which increases holding time on production line. Presently 24 spots are used to join disc and rim on spot weld machine having 3 stations. One cycle of 3 spots requires 6-7 seconds. There are 8 cycles of welding and 7 time indexing. One indexing requires and each indexing takes 4-5 seconds. Loading and unloading time is near about 3 seconds. So for production of one rim will require near about 94 seconds. Other operations need maximum 35 seconds. Now there is a bottleneck at this stage of production, in a shift with other operations 900 to 960 rims are produced but at welding machine 300 to 306 rims are produced. To cope up with this problem 2 Welding machines are installed still the maximum production of rims is 612 rims per shift. So this thesis aims at replacing spot welding method by arc welding method, which will increase the production rate as compare to spot weld arc weld requires much lesser welding time. If we can replace the spot weld by arc weld by keeping the same weld strength then much more time can be saved.

1.2 Objective

a. The objective of theorist to replace tempted by arc weld without affecting the fatigue life of a wheel to improve productivity.

b. Stress distribution on rims varies from one region to another. Based on this type of FEA analysis, we have to decide as to which parts are critical, then, can strengthen those zones.

c. Prediction of stresses into rim under dynamic conditions using FEA.

d. Development of finite element analysis model of wheel rim to get a better understanding of the influences of stress condition on the mechanisms of the crack initiation and propagation in steel wheel.

1.3 Scope

As seen earlier due to spot welding production time of automobile wheel increases drastically. We know that as compare to spot welding method arc welding requires much lesser time to complete operation, therefore if spot welding is replaced by arc weld production time required will be less. Hence this thesis has large scope in near future.

2. Methodology

Approach from problem definition to solution implementation. Following is the methodology of this project work.

a. Finite element method
b. Experimental method.

Before starting analysis, it is important to understand current process. Literature review has been carried out to understand past work carried out in field stress analysis of wheel rim. To validate the solution following methodology is adopted. Experimental study of proposed solution is planned as per following tests.

a. Finite element analysis of 24 spot weld rim for cornering fatigue test
b. First 3D model is designed with the help of CATIA and this model then imported in ANSYS after that Finite element analysis is carried out to find the fatigue life of a wheel for the cornering fatigue test.
c. Experimentation- Cornering fatigue test.
d. A cornering fatigue test is carried out on cornering fatigue test rig as per the steps mentioned in standard procedure and fatigue life of 24 weld spot rim is determined.
e. The FEA result of 24 weld spot rims is compare with Cornering fatigue test result and validation is done.
f. After that different combination of spot weld was proposed. The different combination of arc weld was 100 %, 50 %, 40%, 35% weld and Finite element analysis for proposed scheme were carried out. Then Finite element analysis of proposed scheme was compared with standards and solution was obtained.
g. FEA analysis of obtained solution for static pressure test.
h. After obtaining solution the proposed combination was checked for static pressure and radial load. Analysis was carried out for proposed work under different combination of pressure and radial loads to check the effect of pressure and radial load on the rim.
i. An experiment is carried out on 24 spot weld rim under the different combination of tire air pressure and radial load to know the effect of pressure and radial load on the rim.

After experimental study conclusions were drawn and solution is selected for to improve the productivity of the wheel rim and to reduce the manufacturing time.

3. Finite Element Analysis of Arc Weld Rim

The Finite Element Method produces numerous synchronous arithmetical mathematical statements, which are created and understood in FEA bundle. The FEM is utilized for anxiety examination is an effective and investigation device. FEM or FEA has wide degree in outlining and examination field structure mechanical to electrical. FEA gives an answer for the errand of showing so as to foresee disappointment because of obscure burdens issue zones in a material and permitting planners to see the greater part of the hypothetical hassles within. FEA comprises of a PC model of a material or configuration that is focused and broke down for particular results. Itis utilized as a part of new item plan, and existing item refinement. . If there should arise occurrence of auxiliary disappointment, FEA might be utilized to decide the outline changes to meet the new condition. FEA utilizes a mind boggling arrangement of focuses called hubs, which make a framework called a cross section. This cross section is modified to contain the material and auxiliary properties, which characterize how the structure will respond to certain stacking.

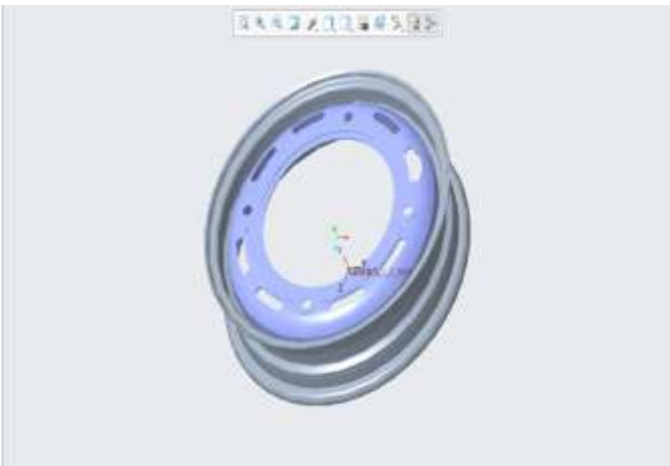

Fig -1: Wheel Rim Assembly

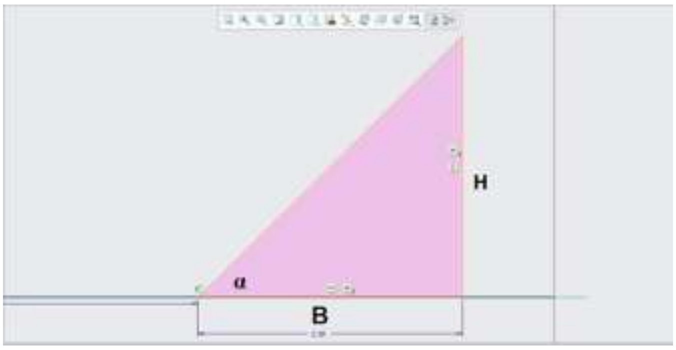

Fig -2: 2-D Drawing of Weld

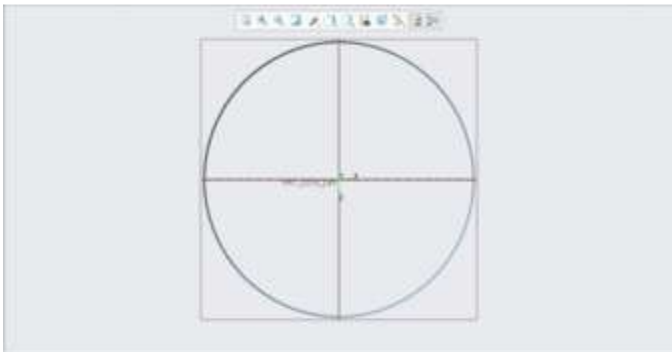

Fig -3: Generated Arc Weld with 100 % Circumferential Length

Fig -4: Exploded View of Wheel Rim and Weld Assembly

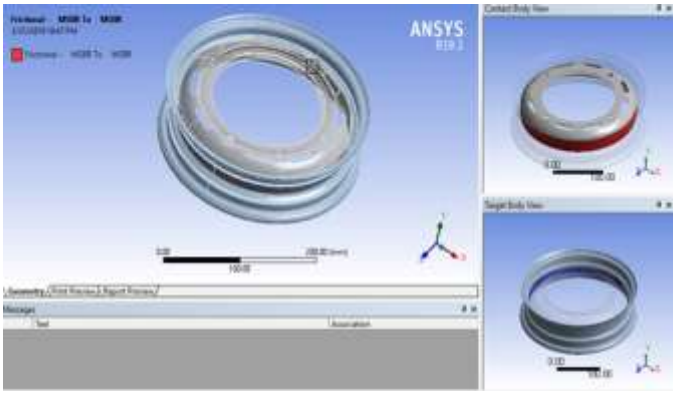

Fig -5: Frictional Contact between Circumference of Disc and Rim

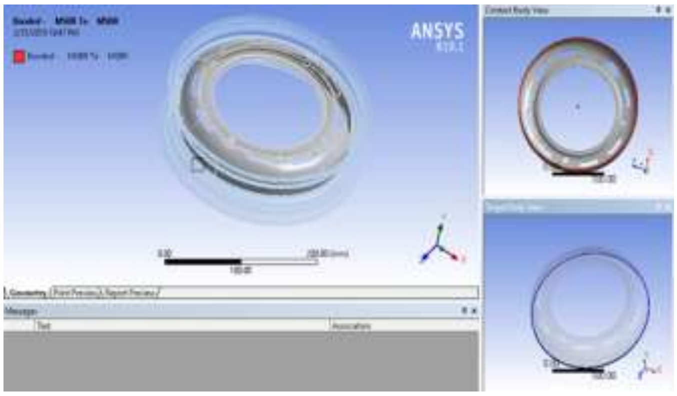

Fig -6: Bonded Contact between Weld and Circumferential Edge of Disc

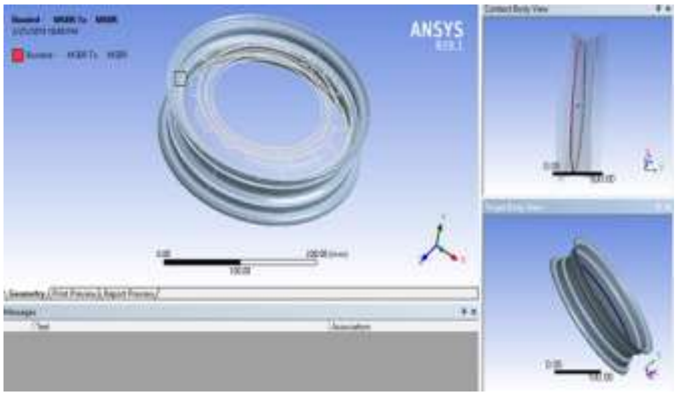

Fig -7: Bonded Contact between Weld and Circumference of Rim

ANSYS Workbench conveys numerous new potential outcomes to the ANSYS AUTODYN client as far as CAD geometry import, complex geometry era, lattice and convenience. To supplement the essentially improved model era abilities, a scope of new solver, material demonstrating and post-handling highlights empower bigger recreations to be illuminated in a speedier time. ANSYS Inc created and looks after ANSYS, a broadly useful limited component displaying bundle for numerically tackling static/dynamic auxiliary investigation (both straight and nonlinear), liquid and warmth exchange issues and additionally electromagnetic and acoustic issues.

3.1 Boundary Conditions

The boundary conditions applied for wheel rim are shown in following figure 3.3. The boundary conditions are explained with the help of a loading. In a cornering fatigue test constant bending moment is applied on a wheel rim. In this set up rim without tire is fitted on a rotating table and four bolts are fitted in a bolt holes.

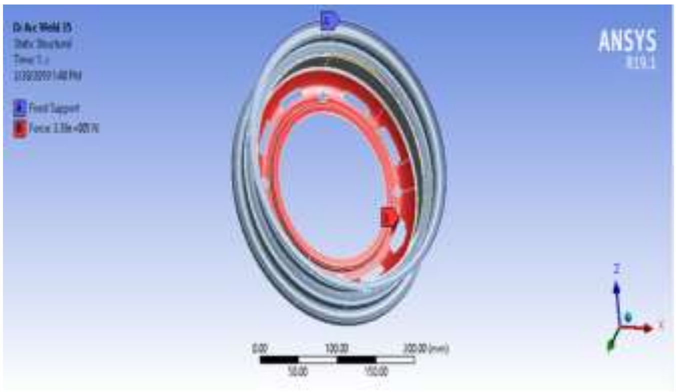

Fig -8: Weld Strength Test Boundary Conditions

Weld analysis of arc weld with 100% weld
We will start Analyzing the Rim for Weld strength at h = 2.3 mm and b = 2.3 mm. If this experiment fails all the remaining experiments for various weld percentage will get fail because we are using 100 % weld so other weld percentages will be weaker than above used. If passed we will go for the next experiment with lesser weld percentage. Force: As in UTM a force of 237.5 kN is applied. (As specified by Manufacturer)

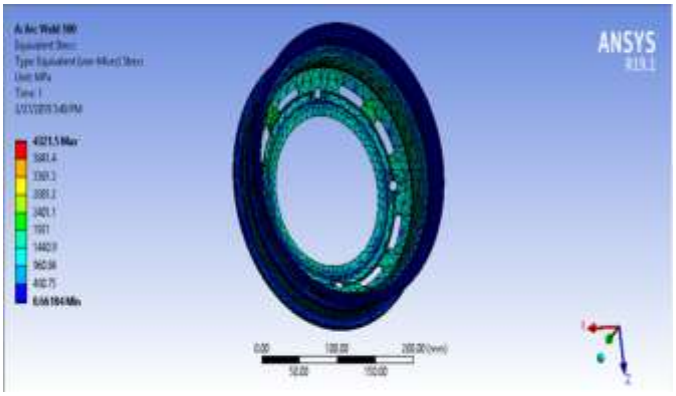

Fig -9: Stress Results for Weld Analysis with 100 % Weld

From above plots Maximum strain is 0.032488; maximum equivalent stress is 4.3252×109 N/mm2 Maximum shear stress is 2.165×109 N/mm2. This do not exceeds the minimum limit indicating the rim will be safe. So we go for the 50 % weld.
Similarly following above stated procedure we have tested the rim for 40% and 35% respectively.

3.2 Finite Element Analysis of Fatigue Mechanism

The basic feature that underlies all the specific fatigue failure mechanisms is the existence of repeated or cyclic stresses at some point of the component. This could be considered the basic definition of fatigue. The cyclic stress sensor strains give origin to damage accumulation until it develops into a crack that finally leads to failure of the component. Keeping in mind the basic assumption for a fatigue failure, different definitions will be provided for the specific fatigue failure mechanisms. The different fatigue failure mechanisms are essentially related to the way those cyclic stresses arise in a specific point of the component, or to the cause of the stresses. Sometimes they are also related to the existence of other concurrent or synergistic damaging mechanisms such as wear or corrosion.

Table-1: FEA Results

Circumferential Weld %	Maximum Strain mm/mm	Maximum Equivalent Stress ((Von-Mises) in MPa	Maximum Shear Stress in MPa
100%	0.032291	4321.5	2163
50%	0.055152	7163.9	3604.8
40%	0.04745	6298.9	3177.7
35%	0.047606	6440.3	3262.7

3.2.1 Radial Fatigue Test

The total weight of a car is balanced with a vertical reaction force from the road through the tire. This load constantly compresses the wheel radially. While the car is running, the radial load becomes a cyclic load with the rotation of the wheel. Hence, the evaluation of wheel fatigue strength under radial load is an important performance characteristic for structural integrity. According to the SAE specification, a wheel should maintain structural integrity without any cracks or plastic deformation for more than 4x rotations under a radial load.

Under a radial load, the strength of the rim usually determines the fatigue life of a wheel, so the stress evaluation is mainly focused on the radial load. The contact area is modeled by one element with the summed thickness of the disc and the rim.

In an actual wheel, since a radial load is applied to the wheel on the bead seats with the tire, the distributed pressure is loaded directly on the bead seats of the model.

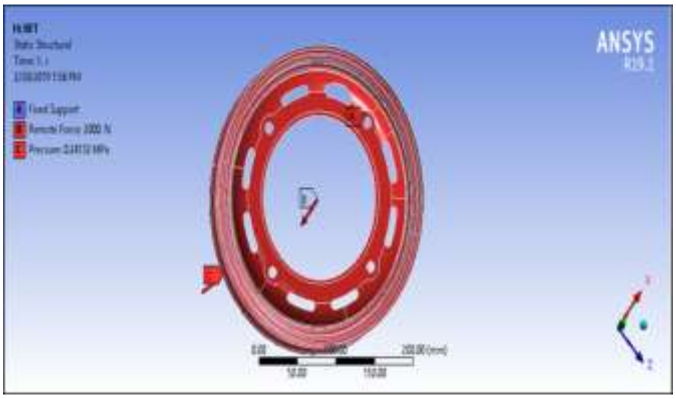

Fig -10: Boundary Conditions for RFT

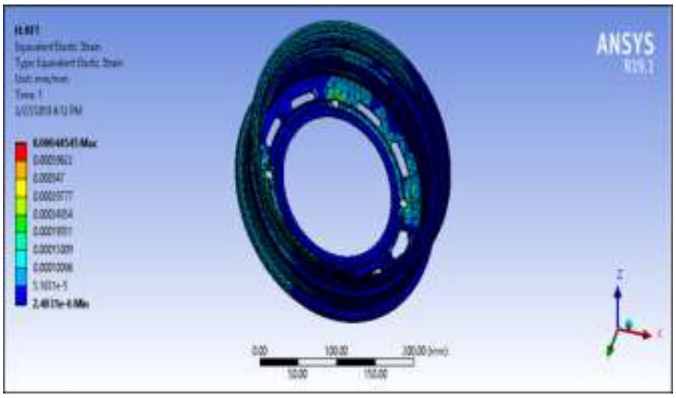

Fig -11: Strain Results of RFT

3.2.2 Cornering Fatigue Test

In the FEA model, loading and boundary conditions were set up similarly to those in the bench test. The wheel was constrained around flange edge of the rim and loaded with a constant force at the end of the shaft. The load shaft and wheel were connected by bolts. Due to the main concern being wheel deformation, the load shaft in the FEA analysis was defined as a rigid body, using tie connection with wheel. J area under the wheel rim was under full constraints. To simulate the cycle, there were 2 load cases and wheel responses were calculated respectively. The direction at 0° gave the positive direction of x axis from the original direction of cyclical loading force in the simulation. The applications of moments are at 2 geometries: 1) Mounting Holes 1) Rim Disc And as in actual test rim is fixed we will Fixed Support to Rim Structure.

Fig -12: Boundary Conditions of CFT (Bolt as Fixed Support)

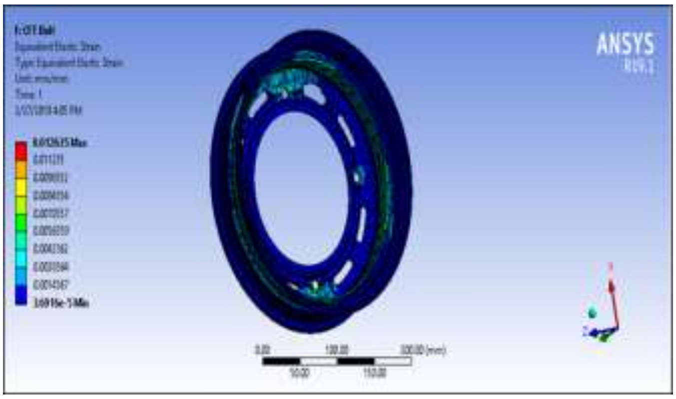

Fig -13: Strain Result of CFT (Bolt as Fixed Support)

4. CONCLUSION

As mentioned above we have carried FEA weld analysis in ANSYS. The test has been carried for the various percentage values of weld length with respect to the circumference of disc of wheel rim. The percentage values of weld length for which the weld analysis has been done are 100 %, 50 %, 40 % and 35 %. The value of maximum permissible tensile stress specified by manufacturer is 7.4803×1010 N/m2 approximately. The actual values of maximum tensile stress developed during FEA of weld are 4.3252×109 N/m2 for 100 % weld length, 7.0936×109 N/m2 for 50 % weld length, 6.2769×109 N/m2 for 40 % weld length & 6.4352×109 N/m2 for 35% weld. From above values it can be concluded that the values of stress developed during FEA are within the specified limit. For the validation of results strain is selected as criteria. The strain developed during FEA of rim for CFT and RFT are 1.3416e-003 and 2.2974e-005 respectively. This shows that the 'Spot Weld' may be replaced by 'Arc Weld'.

REFERENCES

[1] 1. Susheel S. Pote, Prof. R. A. Kathar, Nilesh Phalke, R. B.Patil, Spot Weld Optimization in Wheel Rim, IJIRMF, 2016, 2 (8), pp. 146 – 150.

[2] 2. R. A. Kathar, R. A. Kathar, N. A. Kharche, Optimization of Spot Weld in Steel Component using FEA Technique, IJIIRD, 2017, 1 (2), pp. 127 – 132.

[3] 3. Pirotr Tarasiuk, Krzysztof Lech Molski, Andrzej Szymaniuk, Fatigue Designing of Welded Agricultural Wheels, Eksploatacja i Niezawodnosc – Maintenance and Reliability, 2013, 15 (2), pp. 123 – 128.

[4] 4. Dr. D. R. Prajapati, Daman Vir Singh Cheema, Optimization of Weld Crack Expansion Defect of Wheel Rims by using Taguchi Approach: A Case Study, IJIRSET, 2013, 2 (8), pp. 3437 – 3446.

[5] 5. Manugonda Babu et al, Modeling Analysis of Automotive Wheel Rim, IJIRSET, 2016, 5 (4), pp. 6192 – 6199.

[6] 6. Sushant K. Bawne, Prof. Y. L. Yenarkar, Optimization of Car Rim, IJERA, 2015, 5 (10) Part – 2, pp. 1 – 8.

[7] 7. Devarasiddappa. D, Automotive Applications of Welding Technology – A Study, IJMER, 2014, 4 (9), pp. 13 – 19.