Reducing Experimentation in optimization work using FEA Tool: A Case Study

Anil Pagar
Director
Invensyscad Solutions, Nashik,
Maharashtra, India
ajp@invensyscadsolutions.in

Susheel Pote
Education Head
Invensyscad Solutions, Nashik
Maharashtra, India
academic@invensyscadsolutions.in

Kunal Borase
Assistant TPO
Guru Gobind Singh Polytechnic,
Nashik, Maharashtra, India
kunal.borase@ggsf.edu.in

Abstract: Wheel optimization is a main point of concern in wheel industry. From wheel manufacturing point of view, wheel optimization is cost reduction. So, to reduce the manufacturing cost, wheel weight/ welds etc. must be minimized, without sacrificing mechanical performance. Wheel design and development is very time-consuming process as it requires lot of testing and design iterations before going to production. To minimize development time and to reduce the number of iterations of test is an important issue. To achieve these objectives, computer aided engineering (CAE)/Finite Element Analysis (FEA) is a useful tool and has been recently carried out to perform a wheel design modification. Wheel Rim is the part of automotive where it heavily undergoes both static loads as well as fatigue loads as Wheel Rim travels different road profile. It develops heavy stresses in rim so we have to find the critical stress point and we have to find for how many number cycles that the Wheel Rim is going to fail. In this case study, we are going through the optimization work done on spot welded rim to reduce the number of spots. During this work several parameters were finalized and experiments were designed, then the number of experiments were reduced by using Taguchi Approach, further using FEA software the actual experimentation is reduced which saves considerable time in optimization work. This practical approach is discussed in detail in this paper.

Keywords: Wheels, Finite Element Analysis, Optimization.

1. Introduction

Analysis of the rims consists of numerically analyzing the stress levels that rims experience during operating conditions. These stress levels will then serve as input parameters for a fatigue analysis of the rims to evaluate their respective fatigue life. Additionally, the load bearing capacity of the bolt pattern will be evaluated for conditions of severe loading. The finite element (FE) method is implemented for all rim analysis. The reliability of FEA approach is based on their previous experience in fatigue analysis studies .The magnitude of the static load and pressure contributes to increasing the stresses on the rim components. [1]

In this work Finite Element Analysis of Weld Test,Radial and Cornering Fatigue test using CAE software has been studied.

2. Objective

1. The objective of the work is that to reduce the number of spot weld without affecting the fatigue life of a wheel to improve productivity,
2. Stress distribution on rims varies from one region to another. Based on this type of FEM analysis, we have to decide as to which parts are critical, then, can strengthen those zones.
3. Prediction of stresses in to rim under dynamic conditions using FEA,
4. 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.

According to standards following are tests:

1.Weld Strength Test

2.Dynamic Cornering Fatigue Test.

3.Radial fatigue Test

3. Methodology

1. Testwise FEA
2. Actual Experimentation
3. Comparison of Results

4. Parameters

The parameters for Experimentation are selected as follow:

Parameters	1	2	3
Number of Spots	21	18	15
Diameter of Spots(in mm)	8.4	7.7	7.0
Plate thickness(in mm)	2.0	2.3	2.6

Table:1.1 Parameters considered for optimization

5. Optimization of Wheel rim

Multi-objective structure topology optimization with volume as constraint and considering static stiffness goals and dynamic vibration frequency target has become a hot topic in the research field of structure design ceptual design of aerodynamic missile's structure using topological optimization approaches. In this scheme, the compliance and eigen-frequency were regarded as static and dynamic optimization objectives, respectively. Multi-level sequential programming approach and the compromising programming method were conjoined to address the topology optimization problem. Currently, optimization approaches for wheel designs are concerned with size or shape optimizations while topology optimization methods are rarely applied. Only a few researchers have used topology optimization techniques to the design of wheels. Studied the mechanical design of wheel rims with novel application of the latest bidirectional evolutionary structural optimization method. [2]

6. Weld Strength Test

6.1 Weld analysis

In this work the effects of weld arrangement on the fatigue behavior of the multi-spot welded joints have been investigated via experimental and multiaxial fatigue analysis. To do so, three sets of the spot welded specimens with different arrangements were prepared and fatigue tests were conducted under the various cyclic loads.

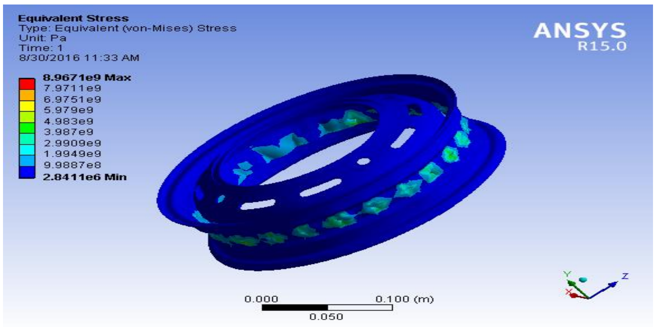

Fig.1 Equivalent Stress

6.2 Comparison of FEA Result and Experimental Results:

Ex pt. No.	Spot No	Spot Dia	Rim thk.	Equivalent Strain	Equivalent Stress	Shear Stress	Result
1.1	S1	D1	T1	0.60123	$1.19X10^{11}$	$2.96X10^{10}$	Fail
2.1	S1	D1	T2	0.07335	$08.96X10^{9}$	$2.72X10^{9}$	**Pass**
2.2	S1	D2		0.07952	$09.21X10^{9}$	$3.57X10^{9}$	**Pass**
2.3	S1	D3		0.50603	$01.0X10^{11}$	$2.31X10^{10}$	Fail
2.4	S2	D1		0.07991	$09.21X10^{9}$	$3.4 X10^{9}$	**Pass**
2.5	S2	D2		0.07994	$09.20X10^{9}$	$3.44 X10^{9}$	**Pass**
2.7	S3	D1		0.58970	$1.17X10^{11}$	$2.68X10^{10}$	Fail

Table 1. Outcome for Weld Test

7.Finite Element Analysis of Rim for Dynamic Cornering Fatigue Test

The Experimentation for Dynamic Cornering fatigue test is as follows:

Sr. No.	Spot No.	Spot Dia.
1	S1=21	D1=8.4 mm
2	S1=21	D2=7.7 mm
3	S2=18	D1=8.4 mm
4	S2=18	D2=7.7 mm

Table 2. DCFT Experimentation

The FEA of Rim for DCFT is carried out in following steps:

7.2 Loads and Boundary Conditions

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. [3]

The applications of moments are at 2 geometries:

1) Mounting Holes
1) Rim Disc

And as in actual test rim is fixed we will give Fixed Support to Rim Structure.

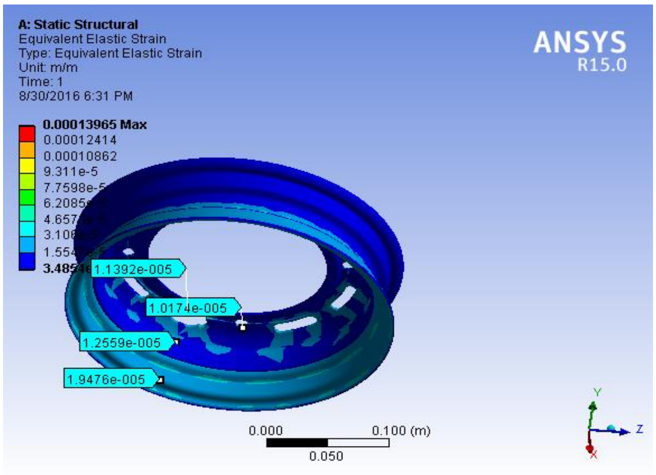

Fig 3. Elastic Strain on Rim 1

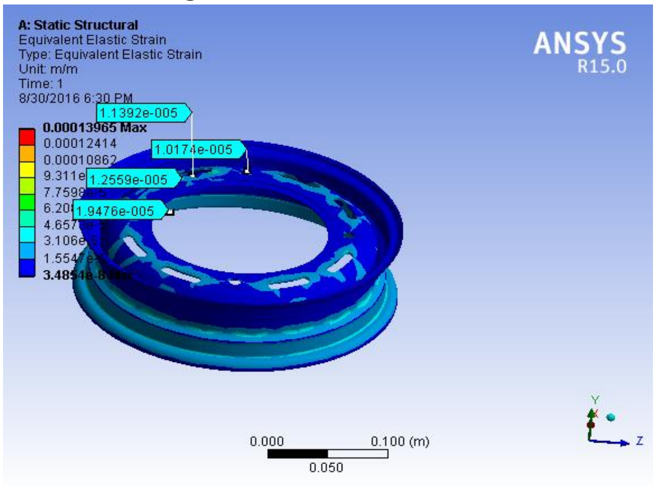

Fig.4. Elastic Strain on Rim 2

7.3Outcome of FEA of DCFT:

Sr. No.	Spot No.	Spot Dia.	Maximum Stress ($X10^7$ N/mm^2)	Result
1	S1=21	D1=8.4 mm	2.0060	**PASS**
2	S1=21	D2=7.7 mm	3.1648	FAIL
3	S2=18	D1=8.4 mm	2.0228	**PASS**
4	S2=18	D2=7.7 mm	3.2038	FAIL

Table 3. Outcome of FEA of DCFT

7.4 Result of Analysis using ANSYS

The existence of two zones with high stresses, disposed in the central area of the disk. These stresses are responsible for the fatigue breaks of the rim. The finite element analyses have identified the likely failure locations in the wheel bolt holes, ventilation holes and welding seam. Stress distribution on rims varies from one region to another. Based on this type of FEM analysis, one can decide as to which parts are critical, then, can strengthen those zones. On the other hand, stress distribution may not be that high on some other parts, hence, excess can be removed from these regions to prevent material extravagance. Furthermore, if the residual stresses remain in the critical zones of the rim, it should be taken into consideration that these parts will be more enduring, hence safer. For FEA of wheel rim we considered following case of dynamic load or moment applied: Moment applied on mounting holes and disc. During experimentation of strain gauges were used to know the strain developed at various positions. These strain gauge readings were compared with FEA results of both cases. The scheme with minimum error will be considered for study of stresses developed in rim and future failure analysis.

8. Radial Fatigue test

8.1The Experimentation for Radial Fatigue Test is as follows:

Sr. No.	Spot No.	Spot Dia.
1	S1=21	D1=8.4 mm
2	S2=18	D1=8.4 mm

Table 4. RFT Experiment Rim Specifications

8.2 Boundary Conditions

A radial load of 250 N is applied on Rim, Along with maximum pressure.
Pressure is applied along the profile of rim and is equally applied over periphery of Rim.
Radial Road is applied on rim, which is a type of bearing load in Finite Element Analysis. From the plots of vonmises stresses maximum stress is 1.9741X107 N/mm2.Which is below maximum limit given so it Rim safe\for this combination.

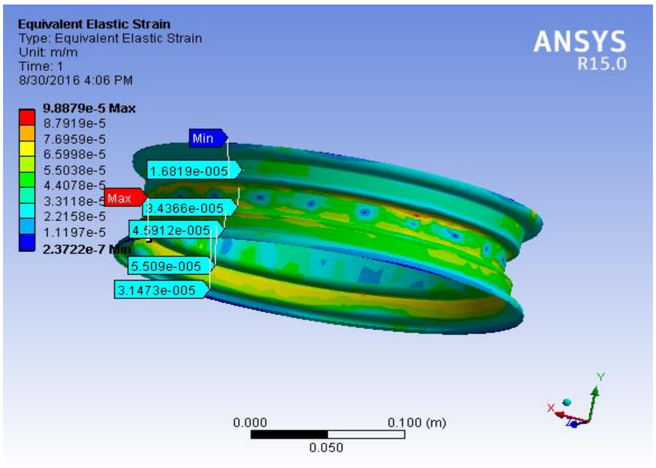

Fig.5.Equivalent Elastic Strain 1

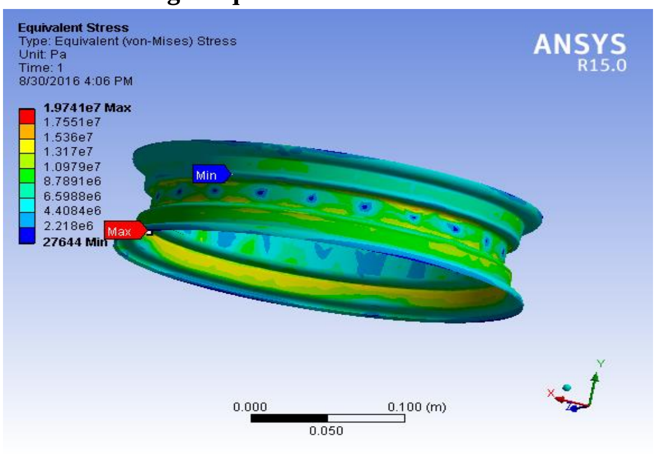

Fig.6.Equivalent Stress 1

8.3 Outcome of FEA of RFT:

Sr. No.	Spot No.	Spot Dia.	Maximum Stress ($X10^7$ N/mm^2)	Result
1	S1=21	D1=8.4 mm	1.9741	**PASS**
2	S2=18	D1=8.4 mm	2.1384	**PASS**

Table.5. Outcome of FEA of RFT

8.4 Optimized Result Level

Expt No.	Spot No.	Spot Dia.	Rim Thk	Result of			
				Weld Strength Test	Dynamic Cornering Fatigue Test	Radial Fatigue Test	Optimum
1.1	S1	D1	T1	Fail	---	---	
2.1	S1	D1		**Pass**	**Pass**	**Pass**	
2.2	S1	D2		**Pass**	Fail	---	
2.3	S1	D3		Fail	---	---	
2.4	S2	D1	T2	**Pass**	**Pass**	**Pass**	**Optimum Level**
2.5	S2	D2		**Pass**	Fail	---	
2.7	S3	D1		Fail	---	---	

Table. 6. Optimize Result after Finite Element Analysis

So From Experimental and FEA results an Optimized level is Spot Numbers 18 and Spot Diameter 8.0 mm

Conclusions

A Multi-objective analysis concept is carried out to optimize the number of spot weld on the Rim. is carried out in steps by step manner. We tried to minimize the number of Experiments and levels of Experiments. All experiments were considered at First Test, and then proper Finite Element Analysis is done. Then Experimentation for the same test is done and compared. In this way a filter is applied to extensive Experimentation. For the safe combinations we carried DCFT with FEA as well as Experimentation. Then the best combinations were tested for RFT. Here we got the final optimize result. Experimental results were compared to finite element results for validating the methods adopted. The experimental results and the modifications and identification of the proper methods for applying the radial load, Comparisons of uniquely identified rim geometry's and the effects of inflation pressure on the state of stress, effect of moment on Rim, effect of change in spot diameter and its number on Rim is verified.

References

1. Li, Z., DiCecco, S., Altenhof, W., Thomas, M., Banting, R., & Hu, H. (2014). Stress and fatigue life analyses of a five-piece rim and the proposed optimization with a two-piece rim. *Journal of Terramechanics*, *52*, 31-45.
2. Xiao, D., Zhang, H., Liu, X., He, T., & Shan, Y. (2014). Novel steel wheel design based on multi-objective topology optimization. *Journal of Mechanical Science and Technology*, *28*(3), 1007-1016.
3. Pletz, M., Daves, W., Yao, W., Kubin, W., & Scheriau, S. (2014). Multi-scale finite element modeling to describe rolling contact fatigue in a wheel–rail test rig. *Tribology International*, *80*, 147-155
4. Zheng, Z. G., Sun, T., Xu, X. Y., Pan, S. Q., & Yuan, S. (2014). Numerical simulation of steel wheel dynamic cornering fatigue test. *Engineering Failure Analysis*, *39*, 124-134.
5. Ezrin, M. (2013). *Plastics Failure: Cause and Prevention*. Carl Hanser Verlag GmbH Co KG.
6. D'Andrea, A., & Tozzo, C. (2016). Interface stress state in the most common shear tests. *Construction and Building Materials*, *107*, 341-355.
7. Irastorza-Landa, A., Van Swygenhoven, H., Van Petegem, S., Grilli, N., Bollhalder, A., Brandstetter, S., & Grolimund, D. (2016). Following dislocation patterning during fatigue. *Acta Materialia*, *112*, 184-193.