DESIGN OF WHEEL RIM BY USING DESIGN OF EXPRIMENTS

MINAJ IQBAL ATAR.

ME Student, Department of Mechanical Engineering (Design), TSSM's Bhivrabai Sawant Collage of Engineering and Research, Narhe, Pune,

A. K. MAHALE

Professor, Department of Mechanical Engineering (Design), TSSM's Bhivrabai Sawant Collage of Engineering and Research, Narhe, Pune,

Abstract- Main objective is to analyze the reason of failures of the rim. Mainly the cracks on the surface, bending due to impact loading. Vibration and the hold pressure of the tire can damage the rim. The damage such as rust, dents, etc. which results in increased vibration while running, loss of air pressure and even sometimes the complete structural failure. This can damage the rims which could result in failure of the Rim during running conditions. Changes can be made to a rim and visible damage could lead to greater damage which can’t be seen by naked eye, so a repaired rim will never be structurally sound as original rim. There are some more causes of failure, this project will discuss about these failures which can arise in rim. This project is all about the design, analysis and calculation of von-mises stresses and deflections with the help of ANSYS. The part which is under maximum stress as well as respective deformation value can be easily detected.

Keywords: Cracks, Vibration, Air pressure, von-mises stresses, ANSYS.

1. INTRODUCTION

It started several thousands of years ago; wheel is used when the human race began to travel the heavy objects from one place to another. Original wheel were sliced and used in this form for centuries on the carts and wagons. Wheel rim is an inseparable part of an automobile mounted on the axle hub of a vehicle. Its main functions are to rotate over the axle of an automobile so as to use power from engine to take automobile in motion, provide support for braking system over its body, dissipate heat generated in the body of wheel rim to surrounding environment, support whole body weight as well as withstand against impact load due to pot holes and road irregularities. There are many different types of wheel rims and they can be divided into many types depending on the manufacturing processes material used etc. This is an exciting time to be a part of the automobile industry (wheel industry) which is witnessing major structural changes. Even though the demands on the industry have never been greater. Customer expectations of wheel quality, safety, reliability and utility are at an all-time high. There is a call for sustainable designs with higher life expectancy thus the wheel manufacturers are investigating and developing more design tools to improve the quality of their products. CAE helps them to reduce the time to produce that design which improves the quality of design.

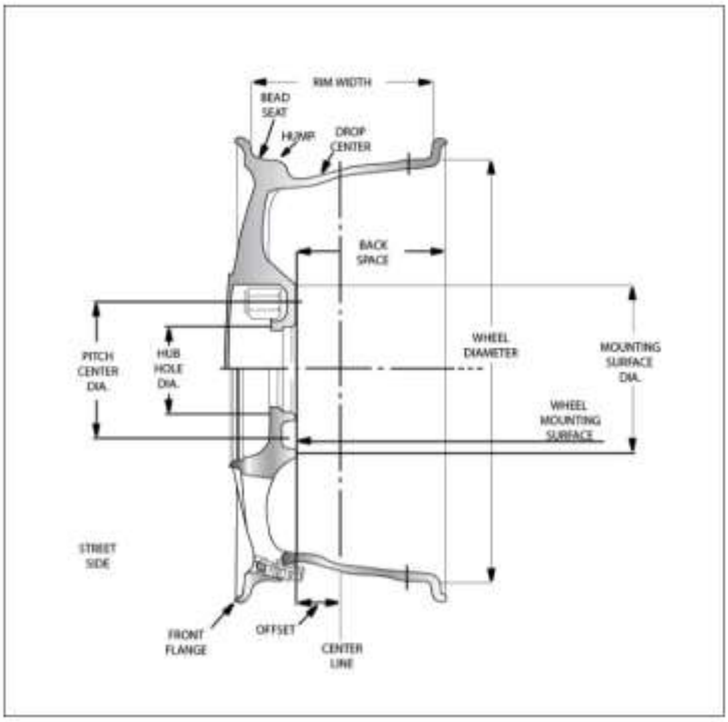

Figure 1: Wheel Rim Terminology

1.1 PROBLEM STATMENT

Wheel Rim must be strong enough to sustain load and components with lesser weight and Cost. Weight reduction is the main factor in automobile industries, because if weight of car increases the fuel consumption as well as the cost required to run automobile also increases. Keeping this factor in mind Wheel Rim should be design with high strength by analyzing different factors like stress-strain values, deformation etc. Optimizing the Wheel Rim by varying parameters, no of spokes present, geometry of spokes in the Wheel Rim for modelling and analysis. This project involves design, analysis and optimization of Wheel rim with constraints of equivalent stress and deflection of wheel rim under maximum load.

1.2 OBJECTIVE

1. To perform analysis of base Wheel Rim model to identify critical stress location and deflection at maximum load
2. Selection of Material and propose revised model of wheel rim.
3. To perform analysis on revised wheel rim model to identify critical stress location and deflection at maximum load.

2. LITERATURE SURVEY

Mangire et al. [2] did comparison study of original rim and with the new rim which is modeled by him. Rims are crucial components of vehicle. The wheel is a component which movements the object on the surface when the force pressing the object to the surface efficiently. In the present every vehicle was designed with alloy wheels which are more efficient than spokes wheels. In this paper rim modelled from the existing dimensions by using the SOLIDWORKS. One model is used in day to day life, second one is changed which can be used in new vehicles and the last one is the modified version of new rim.

Patel et al. [5] Evaluation of the fatigue life of steel rim is proposed in paper. The ANSYS software is used to obtain the static results on steel wheel. The two types of material used which are Aluminum and structural steel. The prediction for fatigue life for both the wheels are made using this. The analysis is showing that the baseline wheel failed the test and initiation of the crack was around the hub.

Dr. Torgal et al. [6] the main objective in this paper is to analyze the failure causes of rim. The rim undergoes with surface cracks, bents. Damage to the rims will cause increase in vibration. Damage, such as rust, cracks, dents, etc. which could result in excessive vibration, pressure, loss and even structural failure of the rim. The paper tell us about various failures which can arise in the rim. This paper tells about the calculation of von-misses stresses and deflections with the help of analysis software.

Das et al. [4] The paper tells us the design of aluminum alloy wheel for automobile application which can be carried out paying more attention on the optimization of the mass of the wheel. The FEA it shows that the optimized mass of the wheel rim could be reduced to around 30%. The FEA shows that the stress generated in the optimized component which is below the actual yield stress of the Al alloy. S-N curve of the component describes that the endurance limit is 90 MPa which is below the yield stress of the material.

Chaitanya et al.[7] Automotive organizations are paying their major interest in the weight reduction of components to minimize fuel cost. This weight can be reduced by introducing new materials and manufacturing processes with optimization of design. Minimizing the weight in the wheel is more effective than minimizing the weight in other components because of its rotational moment of inertia effect during its motion and also the tyre take the overall vehicle load and provides cushioning effect. By reducing the weight we can achieve the objective the reducing of unsprung mass, by which the inertia loads and overall weight are reduced with improvement of performance and fuel economy. In this paper an attempt is made to minimize the weight of the wheel by replacing the aluminium alloy with composites. From the finite element calculations it is found that the mass of the wheel rim can be reduced to 50% from the existing alloy wheels. The analysis also shows that after the optimization the stresses generated from the wheel rim will be below the yield stress. This gave a new approach in the field of optimization of passenger car wheel rim. In this work the modelling is done by using CATIA and analysis is made by using ANSYS.

Parmar et al. [8] the paper gives us the idea about modelling chemical and physical processes of Wheels. They also have importance for the safety of the vehicle and care is needed to check their durability. Growth of the vehicle industry has impact on the design, material selection for rim and manufacturing methods of the rims. The rims loading is a complex; improvement in the design of wheel will be possible when their loading is understood. In this paper, the review about various papers which is based on analyzed with the FEM, using different boundary conditions. Stresses are studied to find the region with maximum stress and fatigue life prediction are studied in order to improve life cycle of rim and also to present the modified design solution.

Kale et al. [9] Wheel rim is an inseparable part of an automobile. Its main functions are to help propulsion of a vehicle on the road as well as withstand against the vehicle weight and practical conditions such as turning and braking, environmental conditions. In automobile wheels rotates continuously during propulsion therefore total load on a vehicle induces alternating fatigue stress in a wheel rim. In order to sustain all loads and practical conditions wheel materials must be of good quality. This paper presents various materials used for wheel rims and their properties such as thermal conductivity, corrosion resistance, price, availability, maintenance, rim manufacturability from that material, availability, manufacturing processes as well as advantageous and disadvantageous of materials.

Bawne et al. [10] The main function of car wheel rim is to provide a firm base. Their dimensions, shape should be suitable to adequately accommodate the particular tyre required for the vehicle. In this project a wheel rim belonging to the disc wheel category is considered. The rim is modeled by using software. So the modelling of the wheel rim is made by using CATIA, model is imported to ANSYS for analysis work.

By using ANSYS, it will reduce the time compared with the computational time taken by a human. Static analysis work is carried out by using different materials such as aluminum alloy, magnesium alloy and structural steel and their results are observed. Now wheel is subjected to modal analysis and their results are discussed. In this analysis the results for static and dynamic analysis are observed and they have concluded that magnesium alloy can be a replace for aluminium alloy.

3. ORIGINAL RIM RESULTS

Figure 2: Front view of wheel rim

3.1 Analysis done on the Original Rim:

We have applied the 8000 N Radial Load and 3000N. As shown in the figure below.

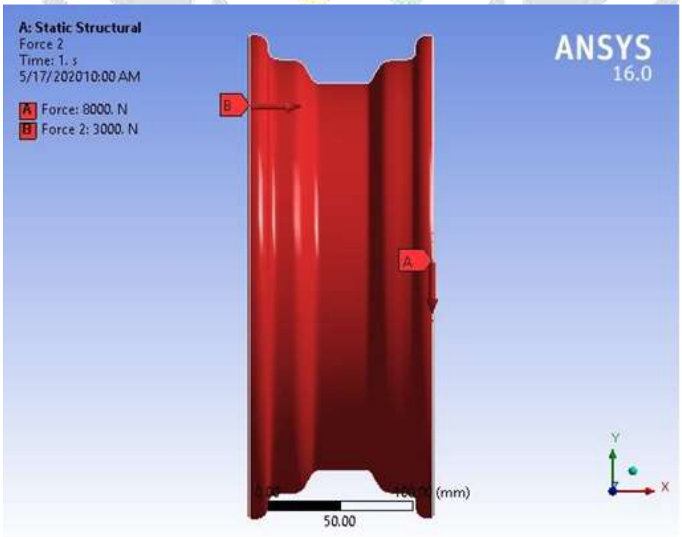

Figure 3: Loading Conditions

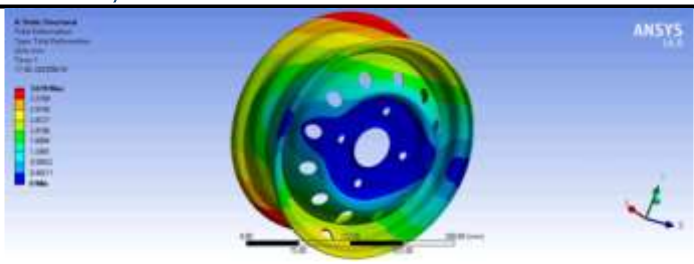

Figure 4: Static Analysis

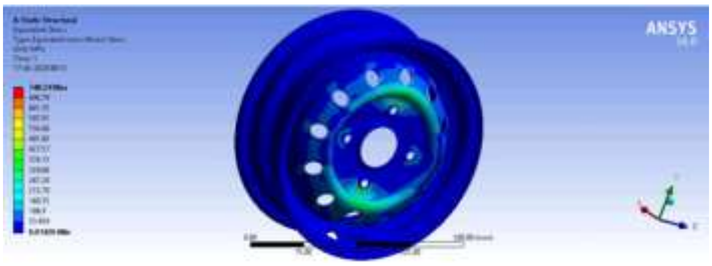

Figure 5: Stress Analysis

Maximum deformation for the static analysis for the original Wheel rim is 3.619 mm with the stress value of 748.24 MPa.

As of now our main aim is to reduce the deformation of the rim as well as the stress induced in the rim.

4. CAD MODELLING AND RESULTS

Figure 6: CAD Model 1

As the Fig. Is the first model we have changed the shape of spokes and we have kept the number of spokes as 6. We have also changed the number of bolts required to mount them on the hub as compared to the original one

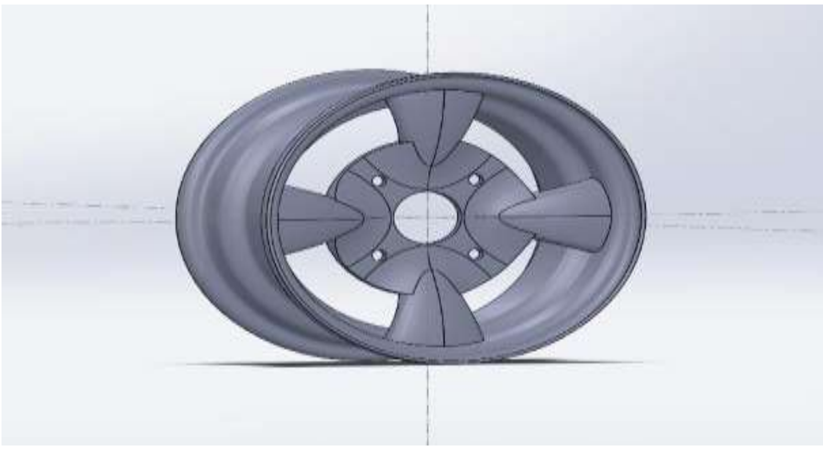

Figure 7: CAD Model 2

Fig. is the second model we have created using the SOILDWORKS. In this we have kept changed the shape of the spokes and the number of spokes used are 4.

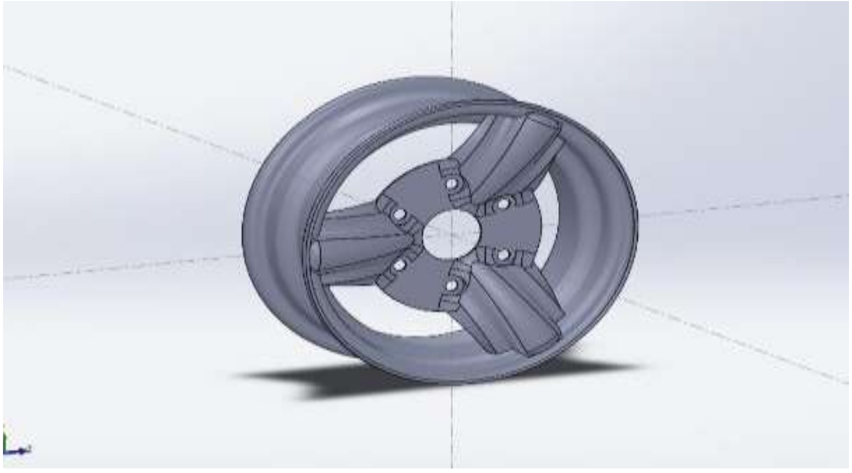

Figure 8: CAD Model 3

Fig. is the updated model of the model 1. In this we have reduced the number of the spokes as well as the number of bolts required to mount them on the hub.

Figure 9: CAD Model 4

Fig. shows the newer model of the rim which are used in the modern cars. In this we have changed the thickness of the spokes as well as shape. Here we have kept the number of bolts used to mount on the hub as 4

Figure 10: CAD Model 5

Fig. Shows the updated model of the model 4. In this we have changed the shape of the spokes as well as thickness of the spokes.

4.1 Model 5 : Static Analysis (Aluminium Alloy)

We have applied the 8000 N Radial Load and 3000 N. As shown in the figure below.

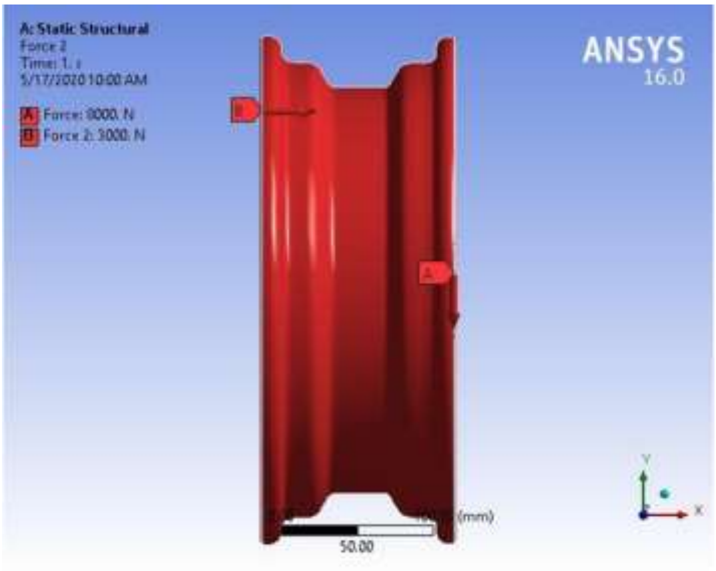

Figure 35 : Loading Conditions

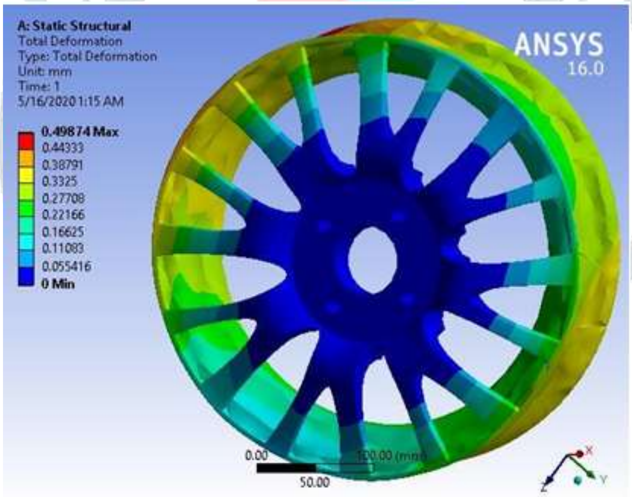

Figure 36 Total Deformation

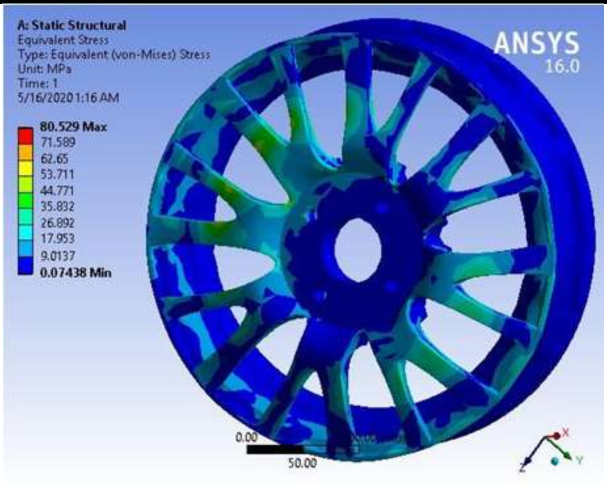

Figure 37 Equivalent Stress

4.2 Model 5: Static Analysis (Magnesium Alloy

Figure 47 Loading Conditions

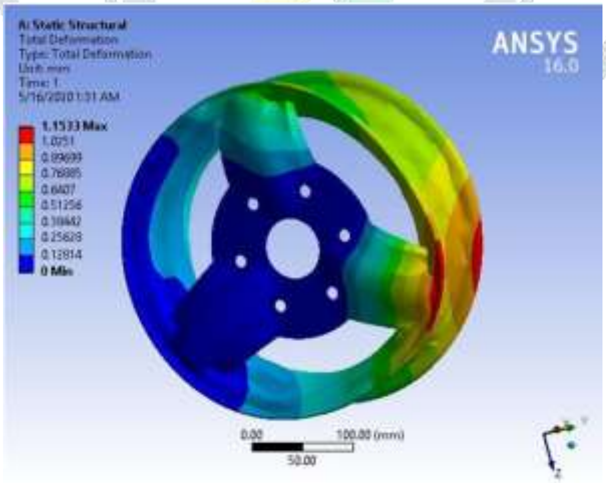

Figure 48 Total Deformation

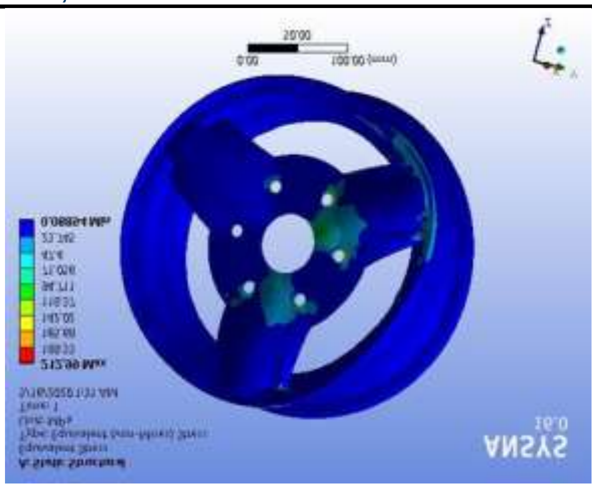

Figure 49 Equivalent Stress

5. DESIGN OF EXPERIMENTS

5.1 Model 5: DOE (Aluminum Alloy)

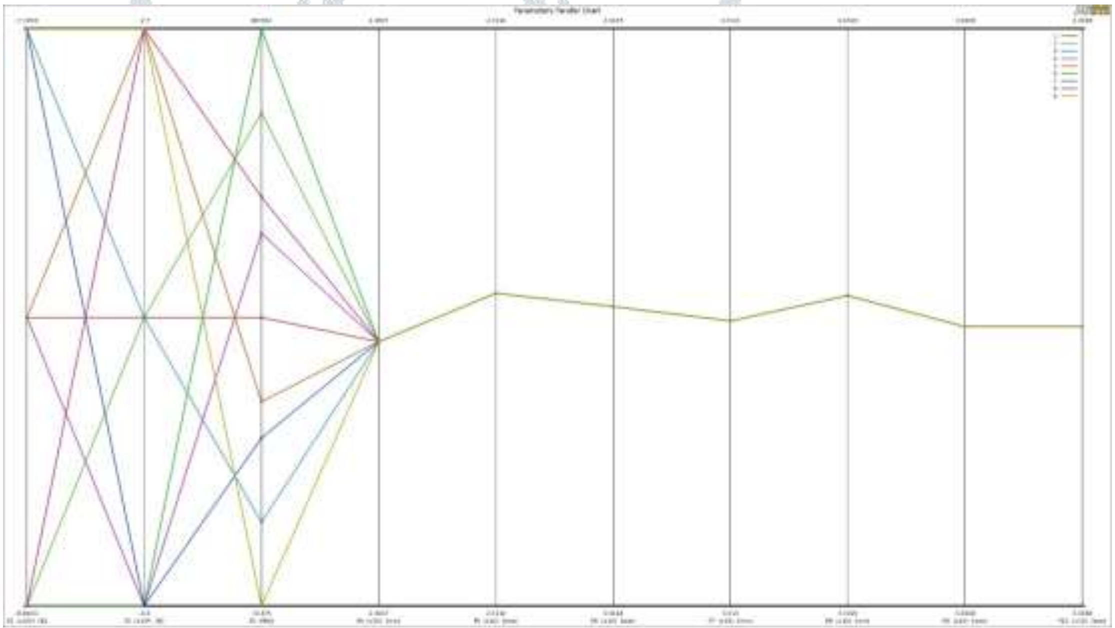

Figure 69: Design point vs parameters

The above chart gives us the information about the inputs and outputs achieves for the design points. If we choose any of the line given in the chart we are able to find the values achieved for that design point.

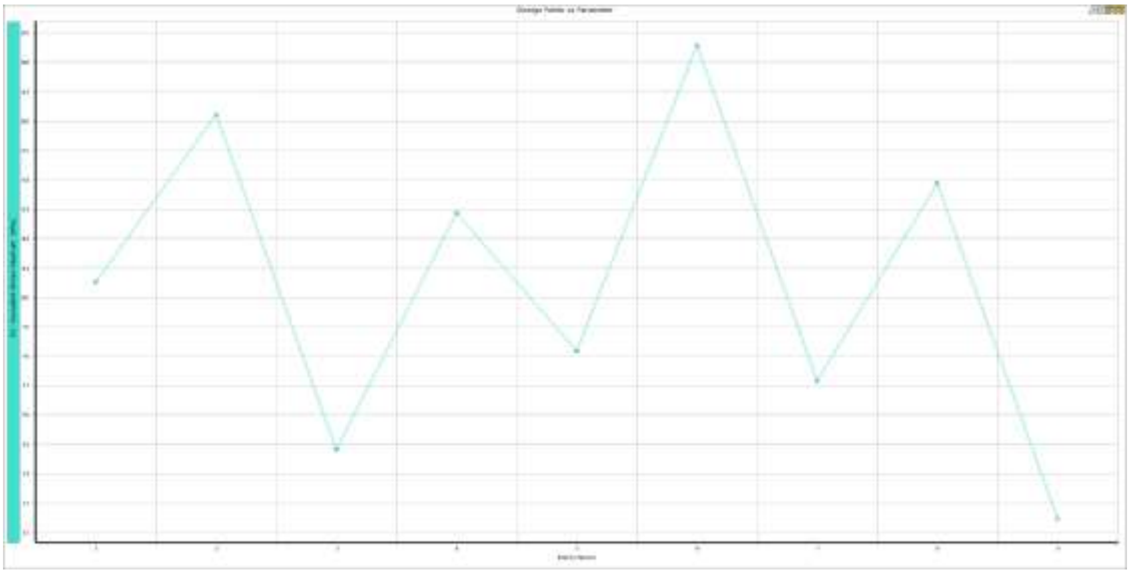

Figure 70 : Design point vs Equivalent maximum stress

The above graph shows Design point vs Equivalent Maximum Stress. From this graph, we have got the values for the stress achieved for respective design point.

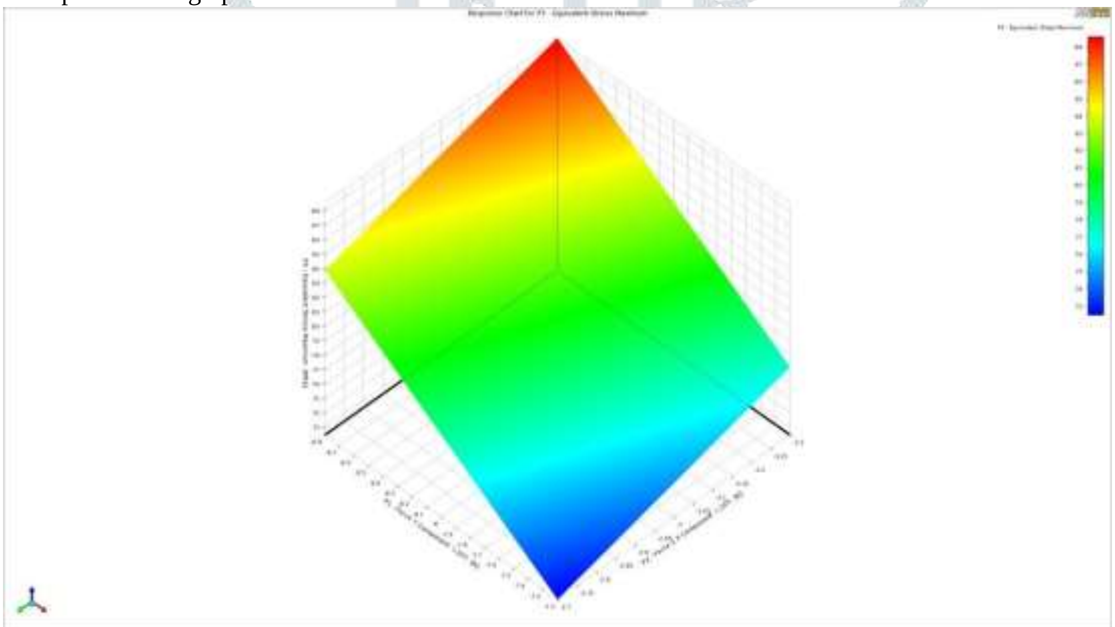

Figure 71 : Response Surface

The figure shows the response surface for the response surface achieved by the Equivalent Maximum Stress for the rim model under the radial and lateral loading conditions.

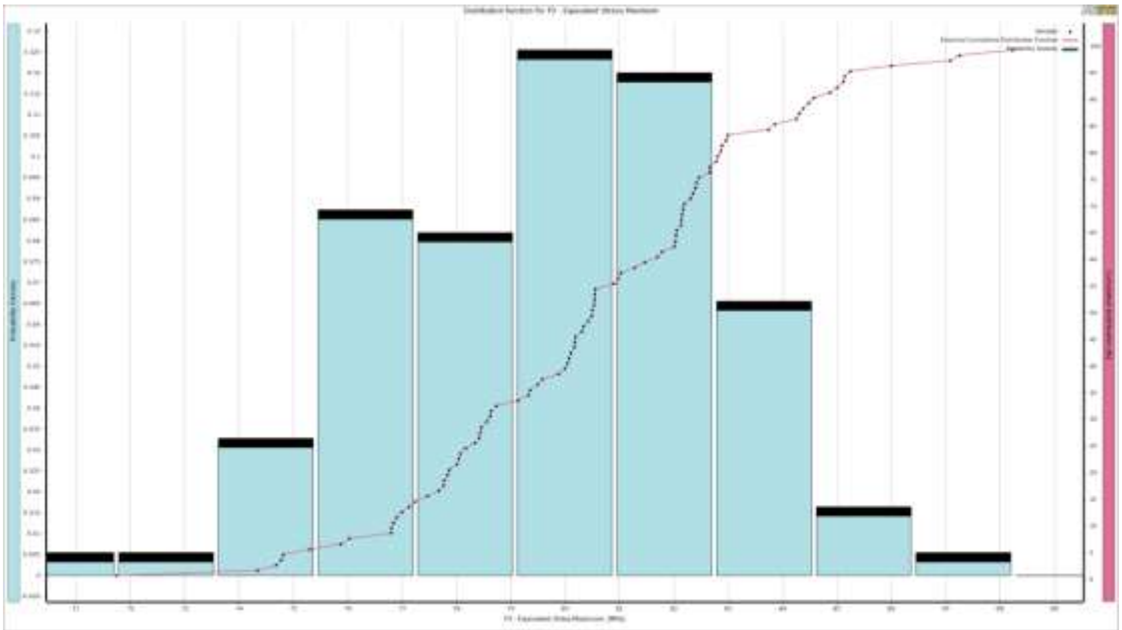

Figure 72 : Six-Sigma Analysis

5.2 Model 5: DOE (Magnesium Alloy)

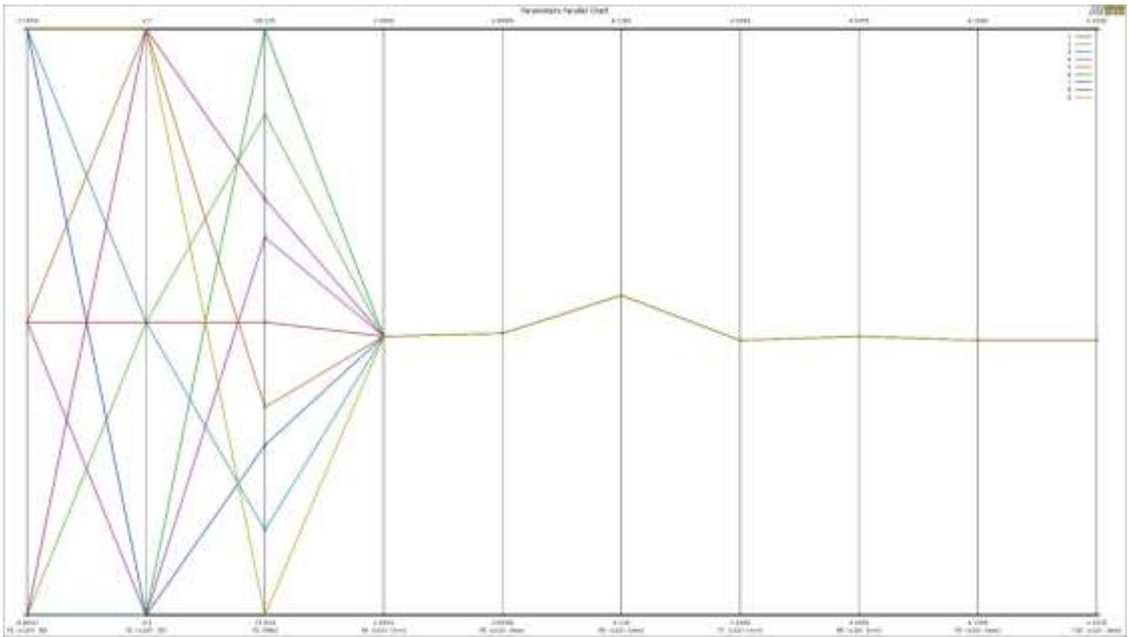

Figure 89 : Design point vs parameters

The above chart gives us the information about the inputs and outputs achieves for the design points. If we choose any of the line given in the chart we are able to find the values achieved for that design point.

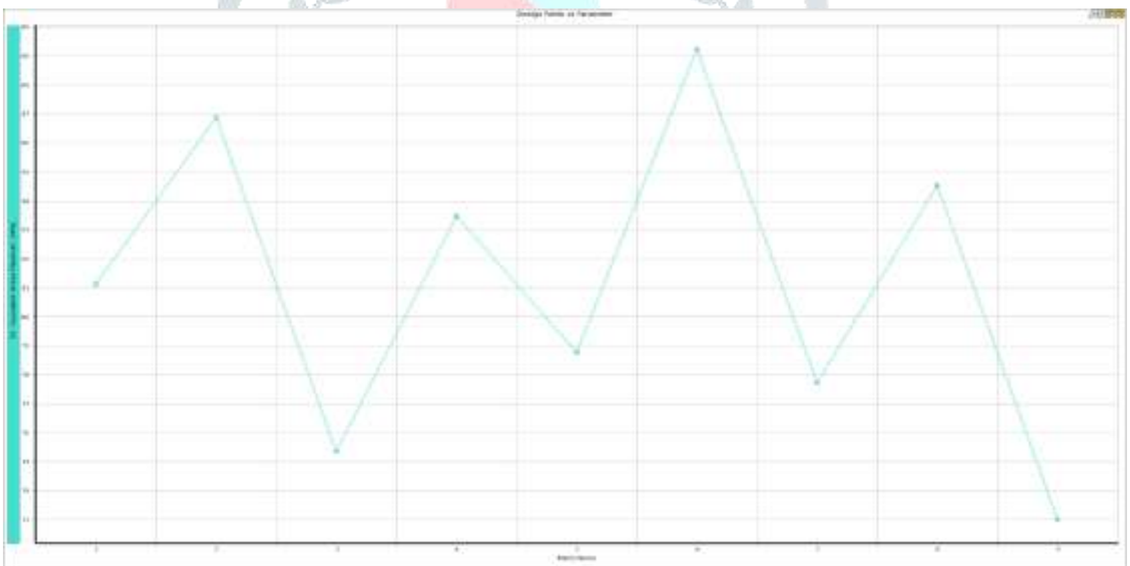

Figure 90 : Design point vs Equivalent maximum stress

The above graph shows Design point vs Equivalent Maximum Stress. From this graph, we have got the values for the stress achieved for respective design point.

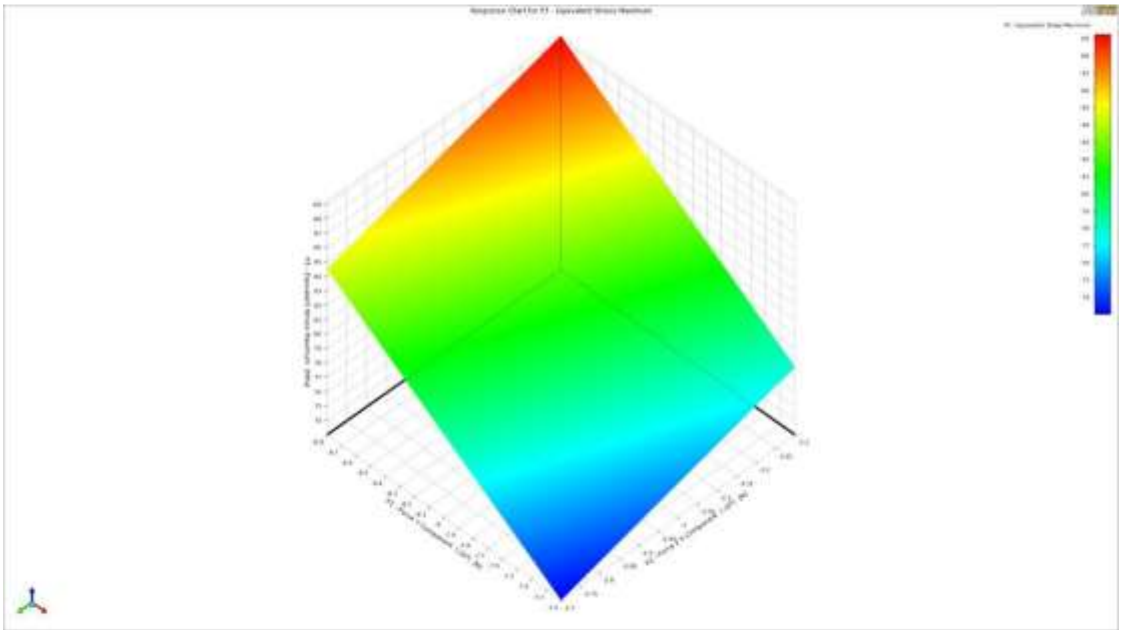

Figure 91 : Design point vs parameters

The figure shows the response surface for the response surface achieved by the Equivalent Maximum Stress for the rim model under the radial and lateral loading conditions.

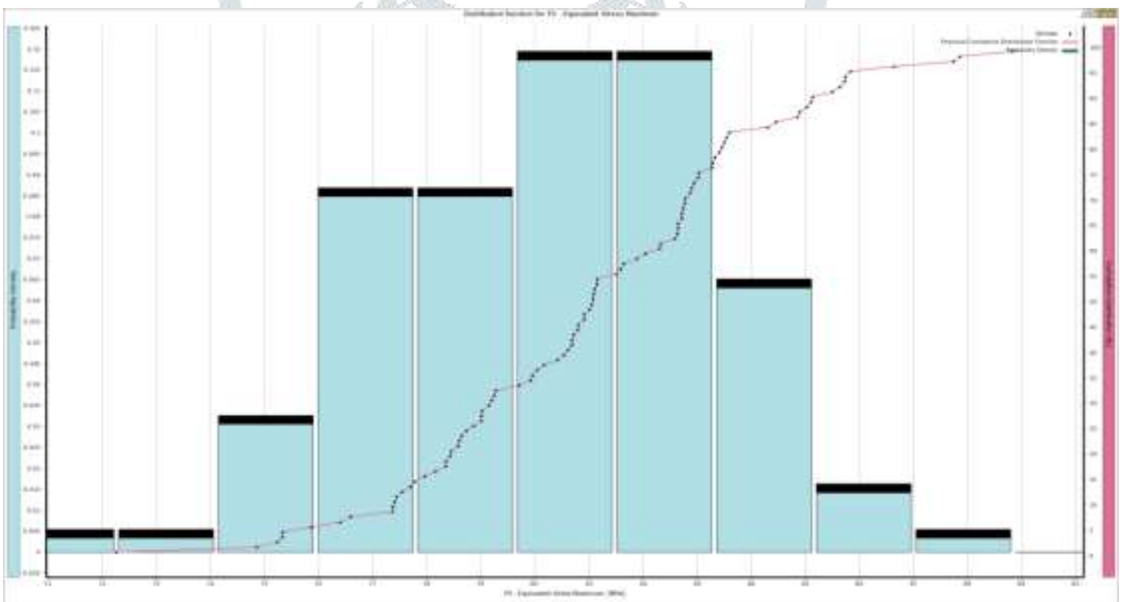

Figure 92 Six-Sigma Analysis

CONCLUSION

1. Design 5 and Design 3 has the least deformation value respected the minimum stress on the wheel rim.
2. Design 5 has least deformation value as compared to its natural frequency.
3. Maximum stress is 80.52 MPa which less as compared to yield strength of the respected material i.e. Aluminium Alloy 280MPa.
4. Design 5 and Design 3 has the least deformation value respected the minimum stress on the wheel rim.
5. Design 5 has least deformation value as compared to its natural frequency
6. Maximum stress is 81.12 MPa which less as compared to yield strength of the respected material i.e. Magnesium Alloy 193 MPa.
7. From This we have concluded that Design 5 is the better among all models.

FUTURE SCOPE

1. Using CFD analysis we can determine the drag force exerted on the wheel rim while in running conditions.
2. Using CFD analysis we can determine the Heat dissipation from the wheel rim while in running conditions.

REFERENCE

[1] ”The Aluminium Automotive Manual”, Version 2011 European aluminium association.

[2] "Deformation Behaviour Analysis Of Car Wheel Rim Under Different Loading Using Finite Element Method" International Journal of Engineering and Technology Volume 5 M. Sabri, M. Rezal, A. Mu’az, K. Shahril, J.Ihsan ,

[3] "Static and fatigue analysis of automotive wheel rim", sachin s .mangire , prof Sayed l. K , prof . Sayyad l. B

[4] “Design and Weight Optimization of Aluminum Alloy Wheel” by Sourav Das, (CAE Analyst) altairengineering India Pvt Ltd, Bangalore at International Journal of Scientific and Research Publications, Volume 4

[5] "Failure analysis of steel wheel by using finite element method", Hement patel, Rajeshkumar Satankar

[6] "STRESS ANALYSIS OF WHEEL RIM", Dr. Suwarna Torgal, Swati Mishra

[7] Mass Optimization Of Automobile Wheel Rim”, S. Chaitanya, B.V.Ramana Murty.

[8] “Fatigue And Static Structural Analysis Of Car Wheel Using Finite Element Method - A Review”, Hitesh N. Parmar, Ashish Jashvantlal Modi

[9] “A Review On Materials Used For Wheel Rims”, H. N. Kale, Dr. C. L. Dhame Jani, Prof. D. S. Galhe.

[10] “Optimization Of Car Rim”, Mr. Sushant K. Bawne, Prof. Y. L. Yenarkar.Bureau of Indian Standards :-Performance Requirements And Methods Of Test For Wheels/Rims For Trucks, Buses Trailers, Multi Utility Vehicles And Multi-Purpose Passenger Vehicles.