Static Structural, Modal and Harmonic Analysis of Alloy Car Wheel Rim using ANSYS Workbench

Dr. M. S. Hebbal[1]
Professor and PG Co-Ordinator, Machine Design
Department of Mechanical Engineering
Basaveshwar Engineering College, Bagalkot

Mukunda Dabair[2]
M Tech (Machine Design)
Department of Mechanical Engineering
Basaveshwara Engineering Collage, Bagalkot

***Abstract*:- The purpose of the car wheel rim is to provide a firm base on which tire could be fitted. The spoke wheel was invented more recently, and allowed the construction of lighter and swifter vehicles. Alloy wheels are automobile wheels which are made from an alloy of aluminum or magnesium metals. The model is created in Catia and then it is imported in Ansys workbench through IGES file. The finite element idealization of this model was then produced using the tetrahedron solid element.**

The given car rim is made up of Al-6061 which is having precipitation aluminum alloy containing magnesium and silicon as its major alloying elements and this is subjected to different boundary conditions. In this project our aim is to carry out the different analysis such as static structural analysis, modal analysis and harmonic analysis. In static structural analysis we found out equivalent von mises stresses, deformation and maximum principal stresses and in the modal analysis we checked the modes shapes of the car rim and further we found out the harmonics stresses, acceleration response, deformation response etc. Later we compared the induced stresses are less than the allowable stresses of the material.

Keywords- Finite element analysis, Ansys Workbench, Modal, Harmonic

1.INTRODUCTION

Wheel is generally composed of rim and disc. Rim is a part where the tire is installed. Disc is a part of the rim where it is fixed to the axle hub. The rim of a wheel is the outer circular design of the metal on which the inside edge of the tire is mounted on vehicles such as automobiles. Wheels rim should not fail during its working. From design point of view, the strength and fatigue life of rim are critical issues. In order to reduce costs, design for light-weight, wheel rim made with materials like,

- aluminium alloy
- steel alloy
- forged steel
- magnesium alloy

The given car rim is made up of aluminum alloy and it is subjected to forces like the pressure of air inside the tire, weight of the car acting through the center. here project aim is to carry out the different analysis such as static structural analysis, modal analysis and harmonic analysis. static structural analysis to find equivalent von mises stresses, deformation and maximum principal stresses and in the modal analysis to check the modes shapes of the car rim and the natural frequencies, further harmonic stresses, acceleration response, deformation response etc. are to analyze. The whole project formed by using design and simulation software's along with finite element analysis.

Properties of Material

Car rim volume is 0.003789m^3

Car rim mass is 10.765 kg

A6061	
Physical properties	
Density (ρ)	2.70 g/cm^3[1]
Mechanical properties	
Young's modulus (E)	68.9 GPa (9,990 ksi)
Tensile strength (σ_t)	124–290 MPa (18.0–42.1 ksi)
Elongation (ε) at break	12–25%
Poisson's ratio (ν)	0.33
Thermal properties	
Melting temperature (T_m)	585 °C (1,085 °F)
Thermal conductivity (k)	151–202 W/(m·K)
Linear thermal expansion coefficient (α)	2.32×10^{-5} K^{-1}
Specific heat capacity (c)	897 J/(kg·K)

Table 1: Material Properties

MatWebis a online material information resource Over 1700 ***Aluminum*** alloy entries are listed in ***MatWeb***

2.LITERATURE SURVEY

K. Venkateswara Rao, Dr. T. Dharmaraju [1]

The author carried out the analysis on wheel rim for different alloys such as steel alloy, aluminum alloy, magnesium alloy and forged steels and found out steel alloy is having higher strength and sustaining more number of cycle and Further he spokes about the optimization of wheel rim by reducing the thickness of a wheel rim.

the stress distribution and fatigue wheel life of wheel rim was summarized on performing more number of iterations on all four different materials. Observed that the aluminium alloy shows the better result in case of number of cycles to failure, compared to all other three materials and the steel alloy is the poor material respectively. Concluded that on optimizing the material thickness, by reducing the material consumption the material life to be extended.

Karan Valetava, Param jethava [2]

Here the author focused on structural behavior and the fatigue life of the rim. The analysis is made for two different materials that is aluminium alloy (A356.2) and carbon fiber. The model is created in CATIA and analysis is carried out in ANSYS. Observed the material behavior in static condition, on application of the load, the deformation, alternative stresses and principal stresses were noted.in fatigue analysis based on the life, safety factor and damage of spoke wheel were analyzed. Concluded with comparative statement that carbon fiber is suggested for the better design in both the cases.

N. Satyanarayana [3]

The author has carried out the analysis on aluminum wheel under the radial load conditions and found out the total deformation of wheel is 0.2833mm. The stresses developed due to the different boundary conditions are 163MPaand 0.038MPa. which is lower than the yield strength of aluminium alloy, the maximum and minimum life of the wheel to be observed at the cross sectional area of the wheel by means of fatigue test that is 1.7667×106 cycles. Concluded with the safe design of rim on comparing with the strength of material considered.

Siva Prasad [4]

Siva Prasad does stress and harmonic analysis of car wheel rim by using ANSYS. This paper explains the structural behavior of the car rim under the several operating loading conditions. 21.3KPa is applied on the circumference of the wheel rim, harmonic response to be observed in between 170Hz to 420Hz.

Concluded with the better results as the design is safe with all aspects and suggested the aluminium alloy as the suitable material for the rim for good results.

Sourav das [5]

This paper is carried out by giving more importance to the optimization of the rim mass. The detail dimensional parameters are to be considered for the better results, on varying the mass of rim, conducted so many iterations to obtain the better life to the rim. Observed with two different loads along the axis of the bolt, 1) radial load at 0^0 from the bolt axis and 2) radial load at 36^0 from the bolt. Concluded that on application of load inclined to the rim the deformation will be reduced.

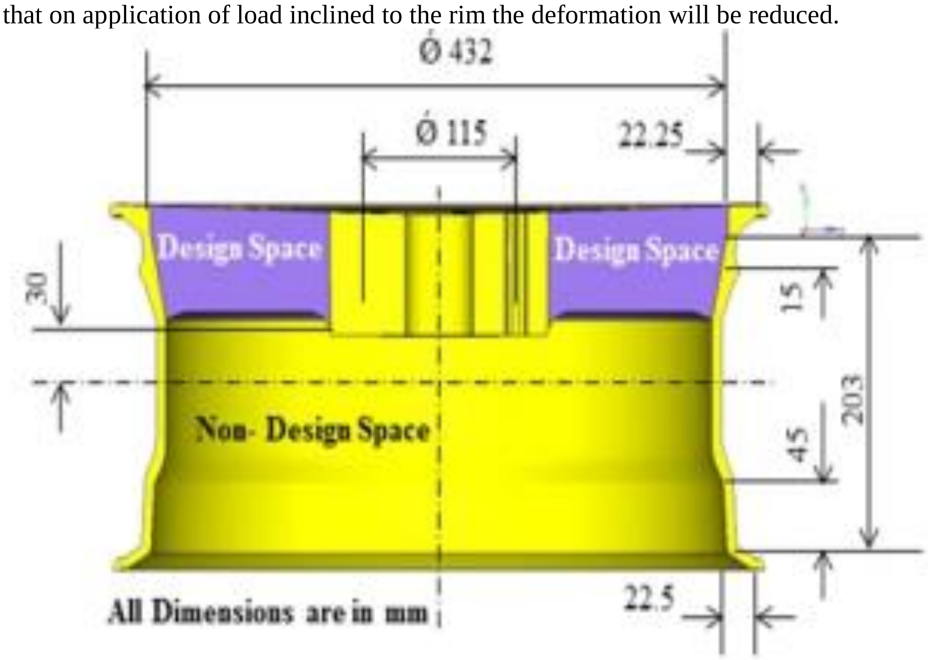


Fig.1: Wheel Rim Dimensions

International Journal of Engineering Research & Technology (IJERT)
ISSN: 2278-0181
Vol. 8 Issue 07, July-2019

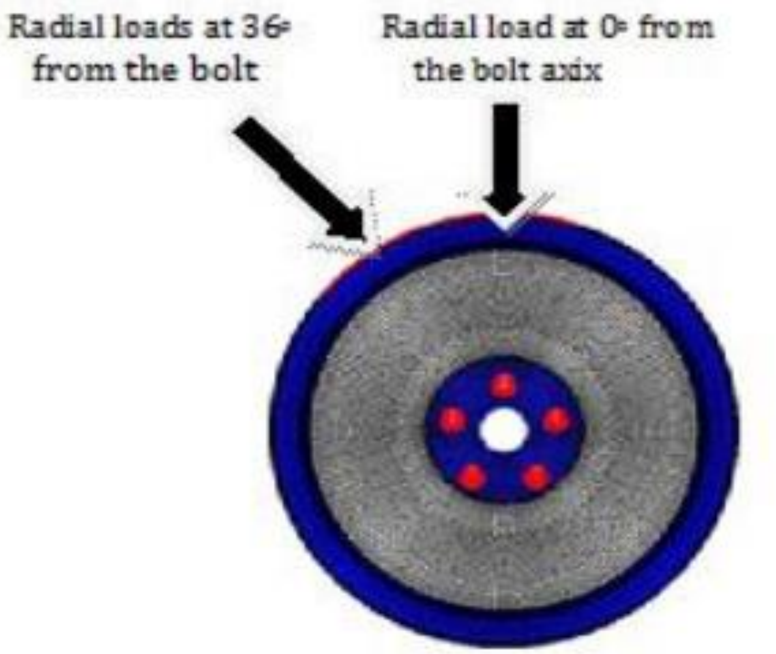


Fig.2: Finite Element representation

3. CAD MODEL OF A CAR RIM

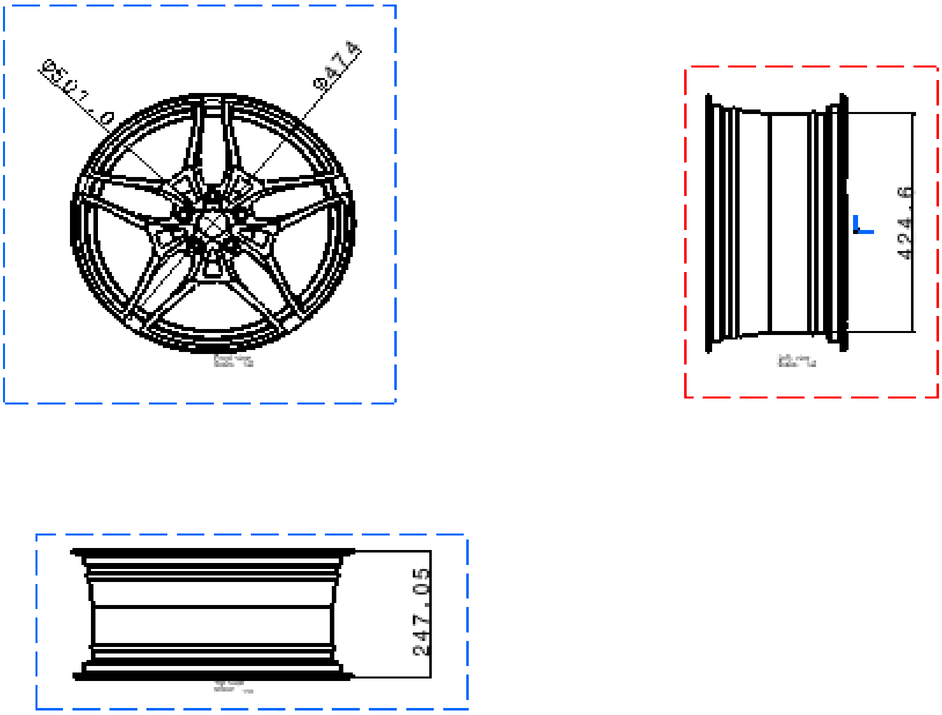


Fig.3: The detailed 2D drawing of a car wheel rim

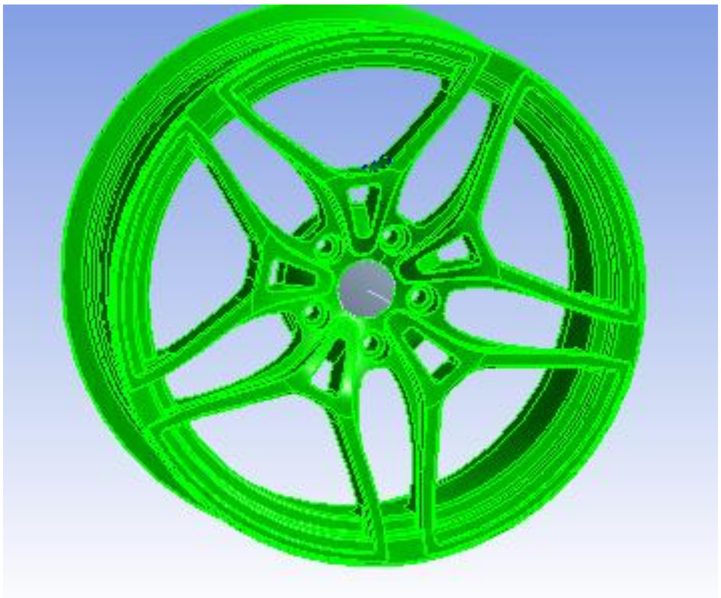


Fig.4: 3D car rim model

Mesh model of a car rim

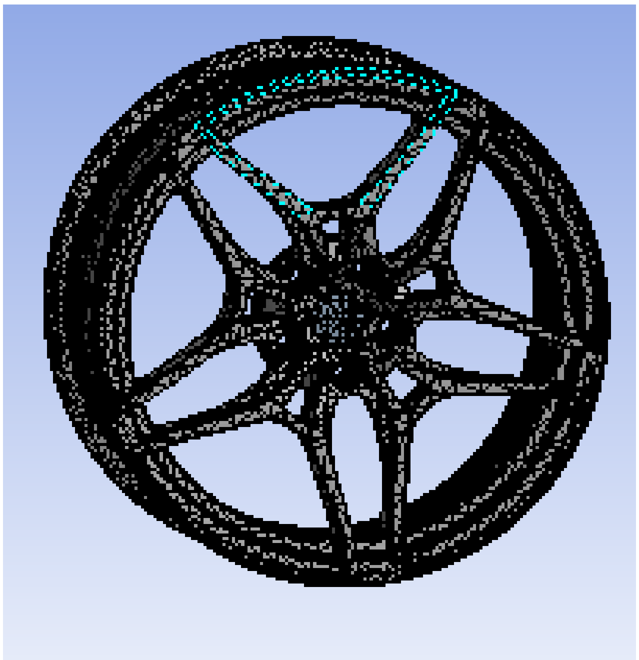


Fig.5: Mesh model of a car rim

The given model is discredited into 127004 nodes and 68880 elements in ANSYS workbench. The tetrahedron and hex elements are used in a meshing. To get results at any point in the model meshing is required. If the number of elements increases the time required for solving will also increases.

BOUNDARY CONDITIONS

- The total mass of vehicle on a rim is 2000kg which is distributed over four wheels
- Each wheel carrying the load = Total mass /4 =500kg.
- Total force on each wheel= 500*9.81=4905N.
- The maximum pressure allowed in a car rim=40psi= 40*.00689=.275MPa

International Journal of Engineering Research & Technology (IJERT)
ISSN: 2278-0181
Vol. 8 Issue 07, July-2019

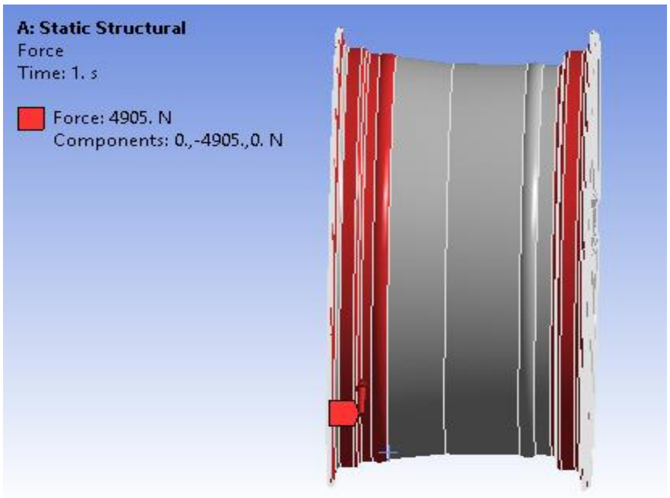


Fig.6: Radial load of 4905N is applied in X-direction

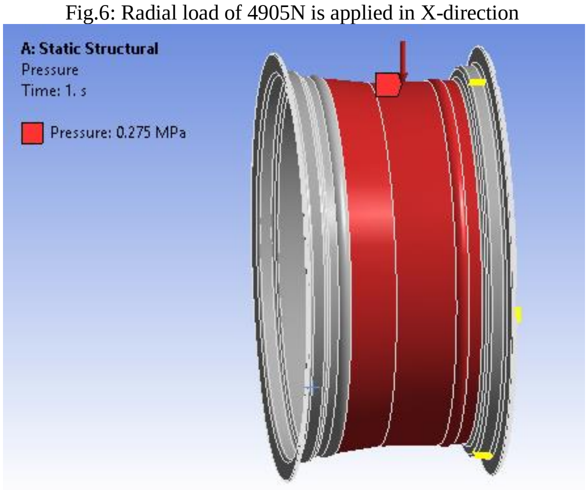


Fig.7: Pressure=0.275MPa is applied on surface of Rim

4.RESULTS AND DISCUSSIONS

Static structural analysis

a) Equivalence von mises stress

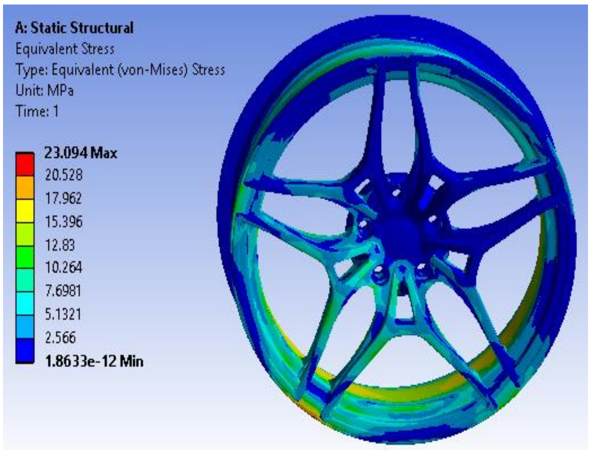


Fig.8: Max Equivalent stress of 23.094MPa is observed

The maximum equivalent stress induced at the outer circumference of the rim with the magnitude of23.094MPa for the given boundary condition. The yield stress of Al 6061 is 240MPa.The induced stress is very less as compared to yield stress of the material. The model can withstand the given load condition.

International Journal of Engineering Research & Technology (IJERT)
ISSN: 2278-0181
Vol. 8 Issue 07, July-2019

b) Maximum Principal Stress

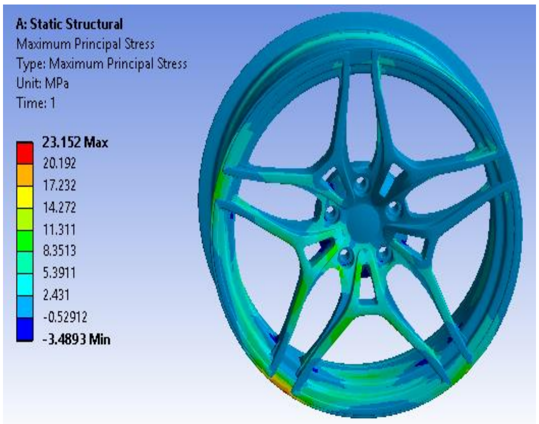


Fig.9: Max principal stress of 23.152MPa is observed which is a singular stress @ Bolt Location

The maximum principal stress induced at the outer circumference of the rim with the magnitude of 23.152MPa for the given boundary condition. The yield stress of Al 6061 is 240MPa.The induced stress is very less as compared to yield stress of the material. The model can withstand the given load condition.

c) Minimum Principal Stress

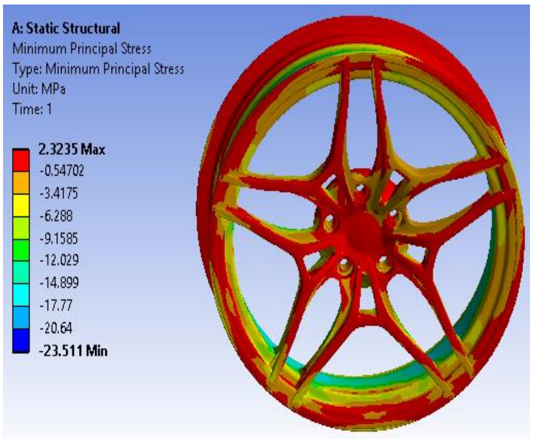


Fig10: Minimum principal stress

The minimum principal stress induced at the outer circumference of the rim with the magnitude of -23.511MPa for the given boundary condition. The yield stress of Al 6061 is 240MPa.The induced stress is less as compared to yield stress of the material. The model can withstand the given load condition.

d) Displacement

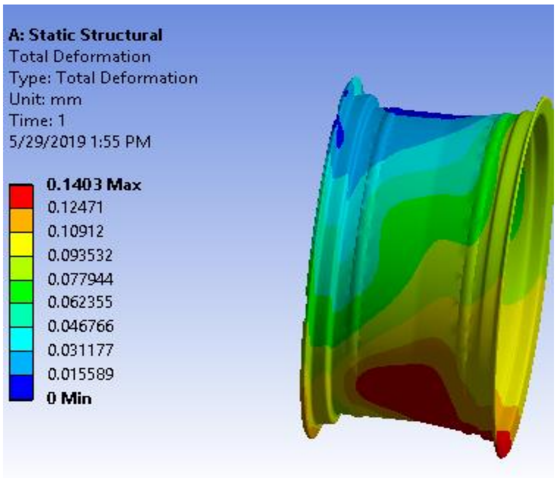


Fig.11: Total deformation observed in the rim is 0.14mm

International Journal of Engineering Research & Technology (IJERT)
ISSN: 2278-0181
Vol. 8 Issue 07, July-2019

The maximum deformation induced at the outer circumference of a rim is with the magnitude of 0.143mm. The percentage of elongation is 12-25% which is very much greater than the induced percentage of elongation. So the model will remain safe with induced deformation of the rim

e) Directional deformation

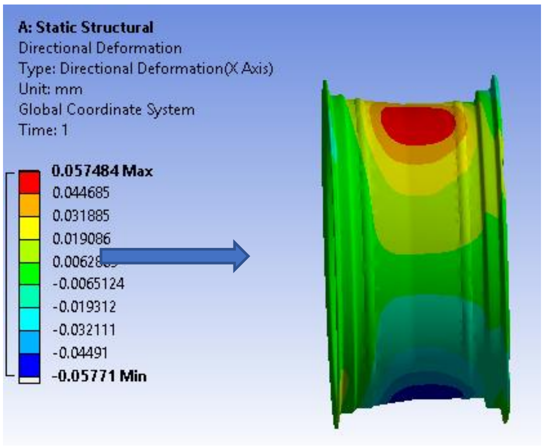


Fig.12: Directional deformation graph of a wheel rim

The maximum deformation induced in the x direction is with the magnitude of 0.05748mm. The percentage of elongation is 12-25% which is very much greater than the induced percentage of elongation. So the model will remain safe with induced deformation of the rim.

f) Equivalent elastic strain

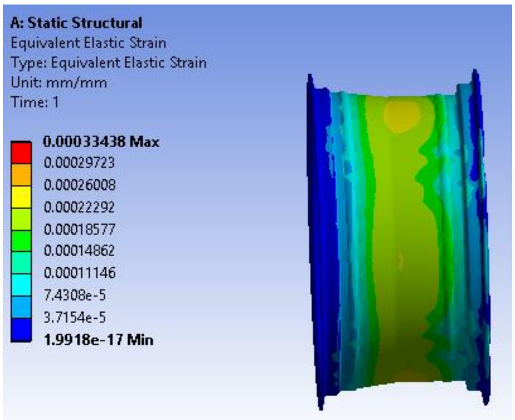


Fig.13: Elastic strain of .00033 was observed

The equivalent elastic strain induced in the given model is $33.48X10^{-5}$ for the given loading condition. It is observed that elastic strain is more at the center of an outer circumferences compared to the both sides.

g) Shear stress

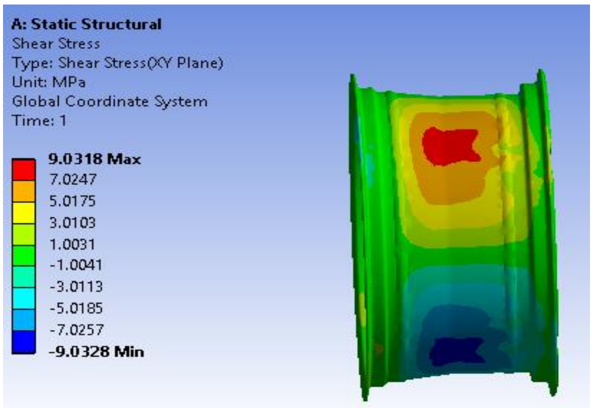


Fig.14: Maximum shear stress of 9.0318MPa was observed

The maximum shear stress induced at the middle part of outer circumference of the rim with the magnitude of 9.03MPa for the given boundary condition. The model will remain safe under the given loading condition.

International Journal of Engineering Research & Technology (IJERT)
ISSN: 2278-0181
Vol. 8 Issue 07, July-2019

Modal analysis

1) *Mode-1 System level Twisting mode along Z axis @291.59 Hz*

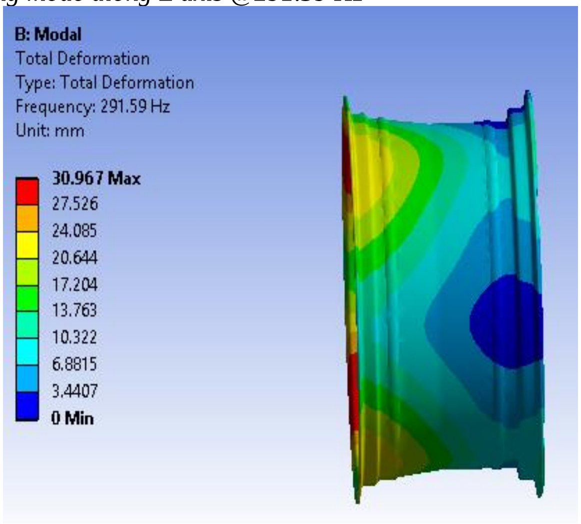


Fig.15: System level Twisting mode along Z axis @79.6Hz

The maximum deformation produced at the outer circumference of the rim with the magnitude of 30.967 for the given boundary condition at 291.59Hz frequency.

2) *Mode 2 System level Bending mode along Z axis @291.78 Hz*

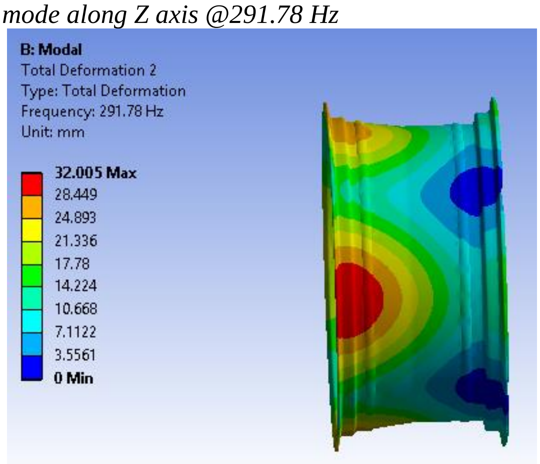


Fig.16: System level Bending mode along Z axis @291.78Hz

The maximum deformation produced at the outer circumference of the rim with the magnitude of 32.005mm for the given boundary condition at 291.78Hz frequency.

3) *Mode-3 System level Bending mode along Z axis @ 442.15 Hz*

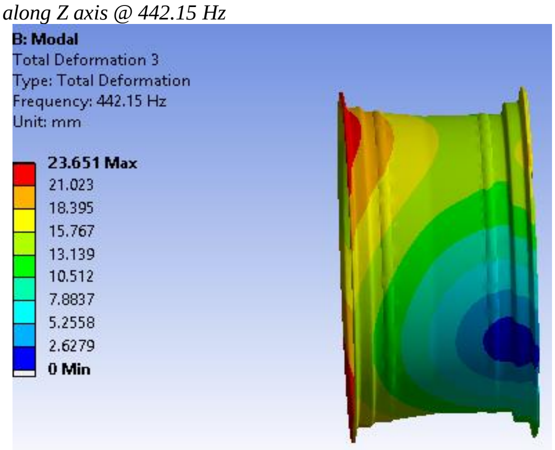


Fig.17: System level Bending mode along Z axis @ 442.15 Hz

International Journal of Engineering Research & Technology (IJERT)
ISSN: 2278-0181
Vol. 8 Issue 07, July-2019

The maximum deformation produced at the outer circumference of the rim with the magnitude of 23.651mm for the given boundary condition at 442.15Hz frequency.

4) Mode -4 System level Bending mode along Z axis @ 442.72 Hz

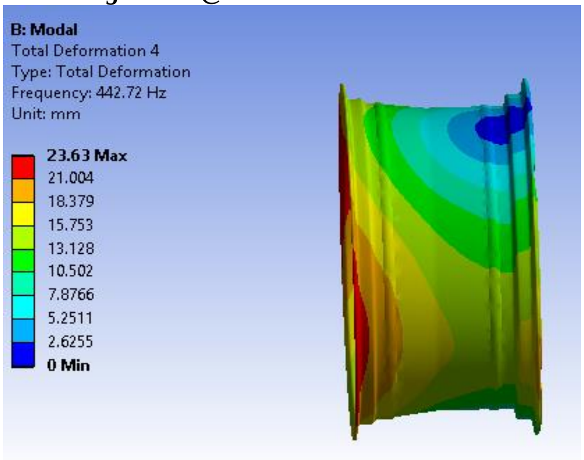


Fig.18: System level Bending mode along Z axis @ 442.72 Hz

The maximum deformation produced at the outer circumference of the rim with the magnitude of 23.63mm for the given boundary condition at 442.7Hz frequency.

Harmonic Response Analysis

- Load is applied using Base Excitation method
- 3G load (29410mm/s^2) is applied at fixed support
- Structural damping of 2% is applied
- Analysis sweeps between 0-150Hz

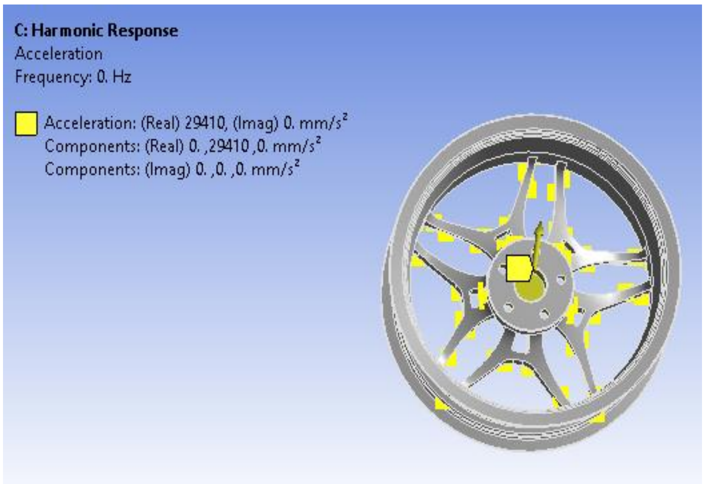


Fig.19: Bolts location is constrained in all DOF 3G load is applied at constrained location.

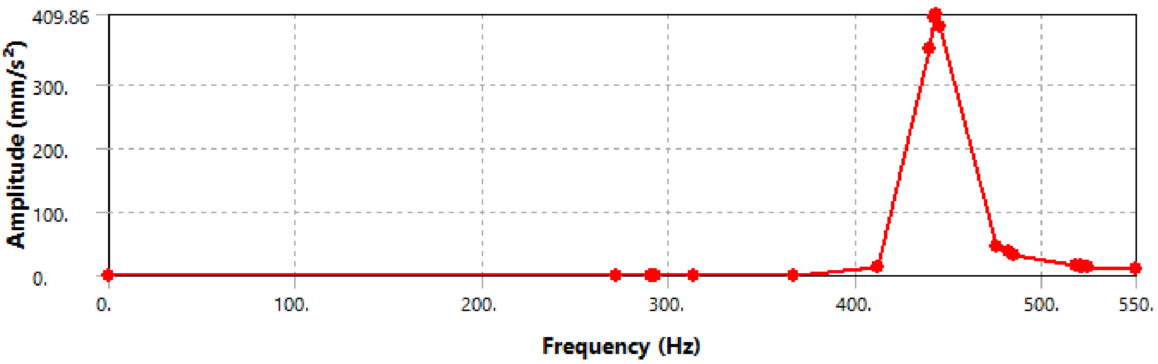
- Acceleration levels at a different frequency

International Journal of Engineering Research & Technology (IJERT)
ISSN: 2278-0181
Vol. 8 Issue 07, July-2019

Tabular Data

	Frequency [Hz]	Amplitude [mm/s²]
1	0.	0.
2	271.55	0.87096
3	289.99	1.0156
4	291.44	1.0156
5	291.56	1.0155
6	291.68	1.0154
7	291.69	1.0153
8	291.69	1.0153
9	291.81	1.0152
10	291.93	1.0151
11	293.39	1.0141
12	313.31	1.1594
13	366.98	0.9898
14	411.81	14.655
15	439.76	356.48
16	441.97	403.41
17	442.15	405.32
18	442.33	406.92
19	442.44	407.74
20	442.54	408.44
21	442.72	409.31
22	442.9	409.86
23	445.12	390.88
24	475.34	46.901

Table 2: Frequency vs Acceleration Response



The Acceleration for initial certain iteration was nearby linear for the step of frequency from 0 to 400mm/s^2 and suddenly raised to 409.86 Hz between the range of 400 to 500 Hz and later on it got reduced to low acceleration.

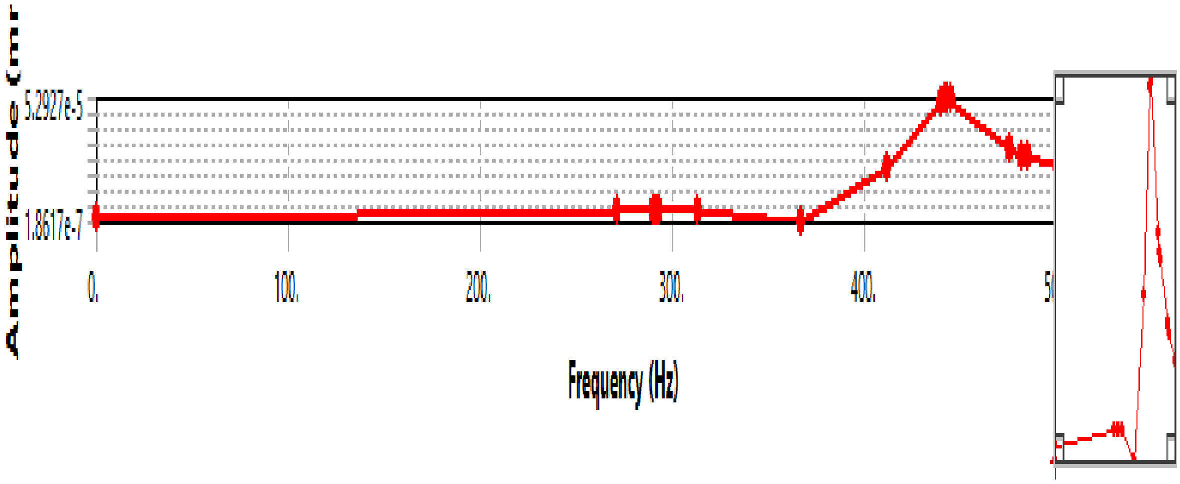
- **deformation response**

International Journal of Engineering Research & Technology (IJERT)
ISSN: 2278-0181
Vol. 8 Issue 07, July-2019

Tabular Data

	Frequency [Hz]	Amplitude [mm]
1	0.	2.347e-007
2	271.55	2.9918e-007
3	289.99	3.059e-007
4	291.44	3.0287e-007
5	291.56	3.0259e-007
6	291.68	3.0231e-007
7	291.69	3.0229e-007
8	291.69	3.0227e-007
9	291.81	3.0199e-007
10	291.93	3.0171e-007
11	293.39	2.9841e-007
12	313.31	2.9918e-007
13	366.98	1.8617e-007
14	411.81	2.1889e-006
15	439.76	4.6692e-005
16	441.97	5.2311e-005
17	442.15	5.2517e-005
18	442.33	5.2682e-005
19	442.44	5.2762e-005
20	442.54	5.2827e-005
21	442.72	5.2898e-005
22	442.9	5.2927e-005
23	445.12	4.9971e-005
24	475.34	5.2579e-006
25	481.57	4.0276e-006
26	484.71	3.5791e-006
27	517.61	1.4947e-006

Table 3: Frequency vs Deformation Response



The Deformation for initial certain iteration was nearby linear for the step of frequency from 0 to 400mm/s^{2} and suddenly raised to 442.9 Hz between the range of 400 to 500 Hz and later on it got reduced to low deformation.

International Journal of Engineering Research & Technology (IJERT)
ISSN: 2278-0181
Vol. 8 Issue 07, July-2019

HARMONIC STRESSES

a) Von-Mises Stress

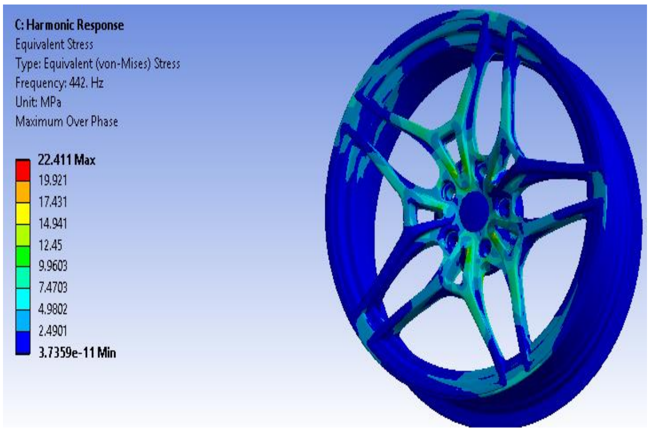


Fig.20: Max equivalent stress of 22.411 MPa is observed

The maximum harmonic stresses induced in a given car rim is 22.41 MPa. The location of a maximum equivalent stress is near the center of a rim. The material yield stress limit is very much greater than induced stresses. The stresses we found out at primary mode frequency@ 442Hz

b) Maximum Principal Stress

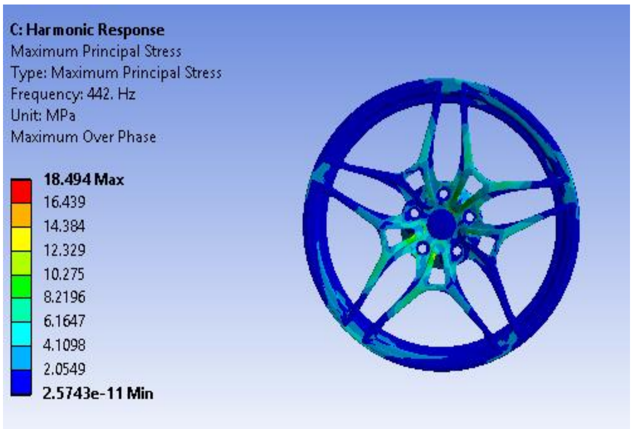


Fig.21: Max principal stress of 18.494 MPa is observed

The maximum harmonic stresses induced in a given car rim is 18.494 MPa. The location of a maximum equivalent stress is near the center of a rim. The material yield stress limit is very much greater than induced stresses. The stresses we found out at primary mode frequency@ 442Hz.

c) Maximum shear stress

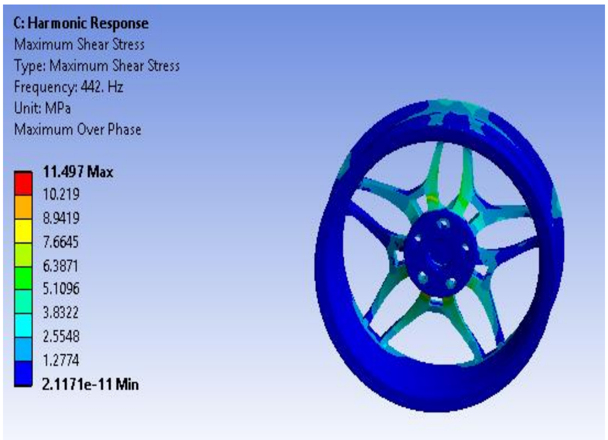


Fig.22: Max shear stress of 11.497MPa is observed

The maximum harmonic stresses induced in a given car rim is 11.497 MPa. The location of a maximum equivalent stress is near the center of a rim. The material yield stress limit is very much greater than induced stresses. The stresses we found out at primary mode frequency@ 442Hz.

RESULT SUMMARY

Sl.No	Type of stress	Structural	Harmonic
1	Equivalent von mises stresses	23.094MPa	22.411MPa
2.	Maximum principal stress	23.069MPa	18.494MPa
3.	Minimum shear stresses	9.03MPa	11.49MPa

It is clearly observed from the above results all the induced stresses in static structural and harmonic analysis in given loading conditions are within the yield stress of Al6061 (car rim material). The yield stress of the Al6061 is 240MPa.

Table 3: static and harmonic results

Mode	Natural frequency(Hz)	Max. deformation(mm)
1	291.59	30.96
2	291.78	32.005
3	442.15	23.65
4	442.78	23.63

Table 4: Modal analysis results

5. CONCLUSION

In static structural analysis we found out the equivalent von mises stresses, maximum principal stresses and maximum shear stresses and those all stresses are well beyond the yield stress of the material. In modal analysis we found out the all modes shape of the given model. In the harmonic analysis we studied the dynamic characteristics like acceleration response and deformation response and found out the harmonic stresses which are well beyond the strength of the material. From above all results it is clear that the given car rim is safe under the loading conditions.

REFERENCES

[1] K. Venkateswara Rao and Dr. T. Dharmaraju, “Analysis of Wheel Rim Using Finite Element Method”, IRJET, Vol 3, Issue 1, 2014.
[2] Karan Valetava, Param jethava, “Fatigue and Static Structural Analysis of Car Wheel using Finite Element Method”, IJTIMES, Vol 2, Issue 2, 2014.
[3] N. Satyanarayana , “Fatigue Analysis of Aluminum Alloy Wheel under Radial Load”, IJMIE, Vol 2, Issue 1, 2012.
[4] T. Siva Prasad,T. Krishnaiah, J. Md. Iliyas, M.Jayapal Reddy, "A Review on Modeling and Analysis of Car Wheel Rim using CATIA & ANSYS", International Journal of Innovative Science and Modern Engineering (IJISME), ISSN: 2319- 6386, Volume-2, Issue-6, May 2014.
[5] Sourav Das, "Design and Weight Optimization of Aluminum Alloy Wheel", International Journal of Scientific and Research Publications, Volume 4, Issue 6, June 2014, ISSN 2250-3153.
[6] Rajarethinam P., Periasamy K., "Modification of Design and Analysis of Motor Cycle Wheel Spokes", International Journal Of Modern Engineering Research (IJMER), PP. 123-127.
[7] Liangmo Wang - Yufa Chen - Chenzhi Wang - Qingzheng Wang, "Fatigue Life Analysis of Aluminum Wheels by Simulation of Rotary Fatigue Test", Journal of Mechanical Engineering 57(2011)1, PP. 31-39.
[8] SunilN. Yadav, N. S. Hanamapure, “Simulating the test conditions to analyze stress distribution and fatigue life of the steel wheel rim of passenger car”.IJER, Vol 6, pp 17-25, 2014.
[9] Liangmo Wang, “Static load finite element model of aluminum wheels for rotary fatigue test”. Journal of Mechanical engineering,57, pp 45-49, 2011
[10] Siva Prasad, “Reassessment of Modeling and Analysis of Commercial Car Wheel Rim using ANSYS”, IJISME, Vol 2, Issue5, 2014.