ISSN: 2277-9655
Impact Factor: 4.116
CODEN: IJESS7

IJESRT
INTERNATIONAL JOURNAL OF ENGINEERING SCIENCES & RESEARCH TECHNOLOGY

LINEAR STATIC ANALYSIS OF A CAR ALLOY WHEEL DESIGN USING FINITE ELEMENT ANALYSIS

Venkatesh.K*, Manjunatha Babu. N.S, Mohan Kumar. K
[*] Department of Mechanical Engineering,
Dr. T. Thimmaiah Institute of Technology, K.G.F – 563120, India

DOI: 10.5281/zenodo.245612

ABSTRACT

The requirements for improved stiffness, reliability, fatigue life and increased efficiency involves challenges of developing innovative design solutions. The present work mainly focus on the design of car alloy wheel, where the analytical and FEM analysis approach was implemented to analyze baseline design. Initially static analysis was performed to obtain total deformation, strain and the stress of car alloy wheel. Three Dimensional model was created using CATIA while it was discretized using ANSYS to perform post processing analysis for obtaining expected solution. The results were obtained through linear static analysis in terms of Total deformation, Minimum principal stress, Max Principal stress on 4 arms wheel and later dynamic analysis (modal analysis) was been done to obtain different modes with different frequency for 4 arms wheel.

KEYWORDS: Alloy wheel, Principal stress, Deformation & dynamic analysis.

INTRODUCTION

Wheels0 have been made, using various casting0 techniques such as sand0 casting, gravity die0 casting, centrifugal0, squeeze and low0 pressure die0 casting. Sand and gravity0 castings are less controllable0operations and have0problem with blow0holes and0shrinkages. Aluminum0wheels should not fail during0service as their0strength and fatigue0life are critical. In order0 to reduce0costs, design0for light-weight, and limited-life is increasing0 being used for all0vehicle component. In the actual,product development automotive0wheels have complicated0geometry and must satisfy0manifold design criteria, such as0style, weight, 0manufacturability, and performance. In addition to a 0fascinating wheel0style, wheel0design also needs to accomplish a lot of engineering0objectives including some required performance and durability0requirements. The present0research work0 is focused0 on the automobile sector, specifically on the car alloy wheel rim design to improve the quality of the wheel by evaluating the fatigue life, structural integrity, over speed & burst speed margin and to reduce cost and weight reduction & ease of replacing. Design modifications of the existing alloy wheel rim include conversion of elliptical cross section in to an rectangular cross section for a good overall outlook and style. The linear and bilinear static structural analysis is also imparted for higher efficiency and longer life. The pressure distribution about the rim surface is to be maintained at 32psi and load consideration on the rim when this pressure decreases below 32psi will be more on the bolt and bolt holes. Over speed & burst speed margin is the limit where in the rim should withstand the stress & strain on the rim which is under operational condition and tough road condition. The tendency0 of a 0material is to break, under0repeated cyclic0loading at a stress considered0less than the 0tensile strength in a 0static test. Fatigue0cracks can terminate the0usefulness of0a structure or component by more ways than just fracture.

GEOMETRY OF 4 ARM ALLOY WHEEL

2D Drawing Front View **Side view**

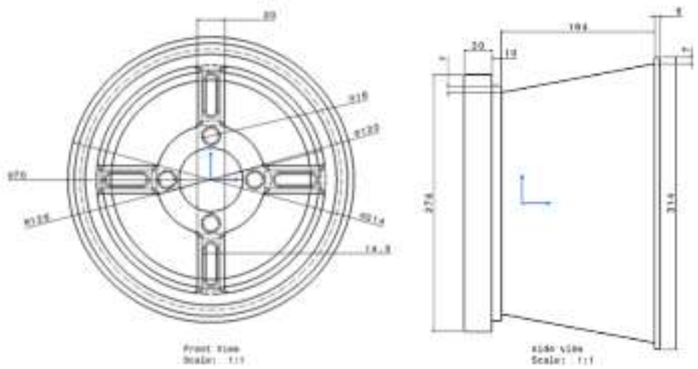


Figure 1: 2D Diagram of the 4 arms wheel.

3D MODEL **MODEL WITH MESH**

Figure 2: 3D Diagram of the 4 arms wheel and with mesh.

The model of 4 arms wheel as shown in fig 1 was designed in unigraphics and later loaded in to ansys 14 to perform linear static structural analysis. The meshing was been done for the 4 arms wheel.

Linear stress, strain and deformation of a 4 arms wheel by linear static structural analysis

Equivalent stress **Maximum principal stress**

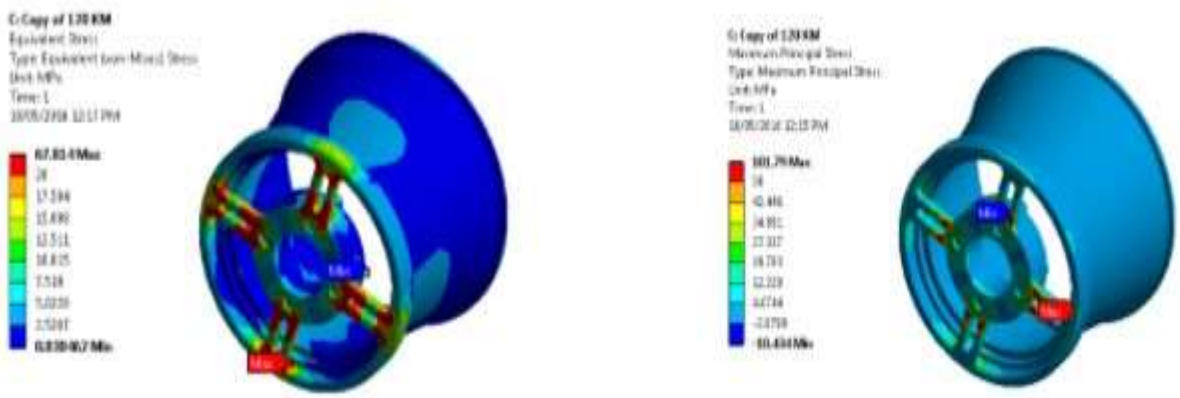


Figure 3: Equivalent stress and Maximum principal stress for 4 arms wheel in linear stress analysis

ISSN: 2277-9655
Impact Factor: 4.116
CODEN: IJESS7

Minimum principal stress **Total deformation**

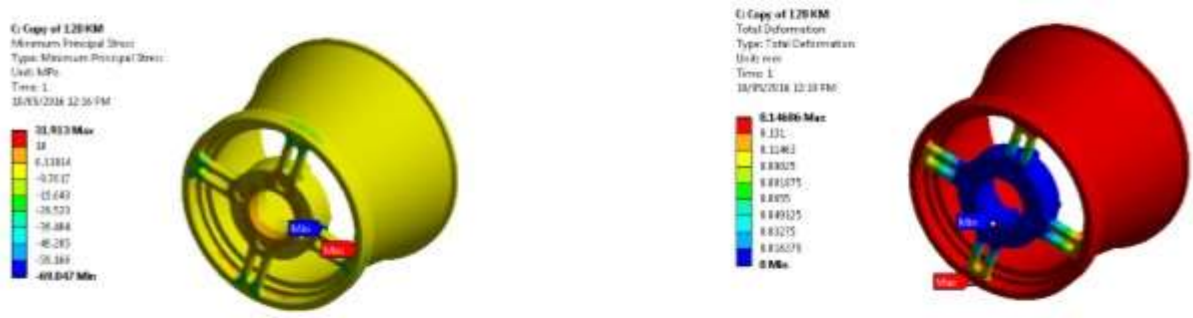


Figure 4: minimum principal stress and Total deformation for 4 arms wheel in linear stress analysis.

LINEAR STATIC STRUCTURAL ANALYSIS FOR 4 ARMS WHEEL

The model of 4 arms wheel as shown in fig 1 was done in unigraphics and later loaded in to ansys 14 to perform linear static structural analysis. The meshing was done for the 4 arms wheel as shown in figure 2, The boundary conditions were applied on the rim surface of the aluminium wheel with pressure of 0.25 Mpa, rotational velocity of 222.2 rds or 2122 rpm and bolt pretension of 23340 N. After applying the boundary conditions following results were obtained:

1. Von-mises stress= 67.814 Mpa
2. Maximum principal stress= 101.79 Mpa
3. Minimum principal stress=31.913 Mpa
4. Total Deformation = 0.146868 mm

Bi-linear results of stress, strain and deformation of a 4 arms alloy wheel by linear static structural analysis.

Equivalent stress **Maximum principal stress**

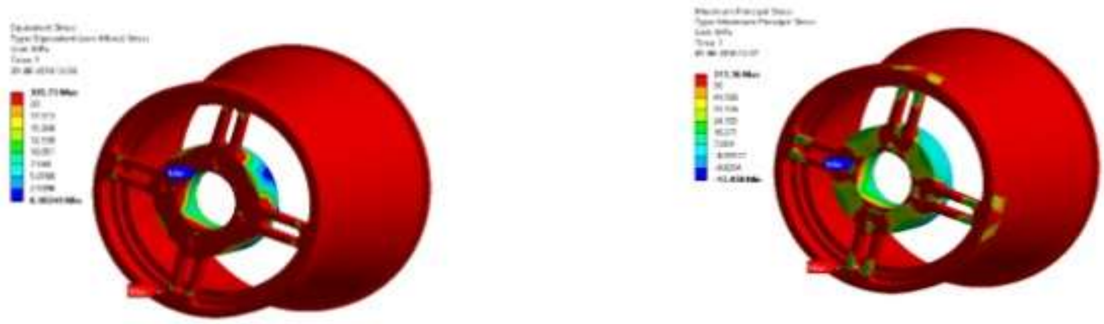
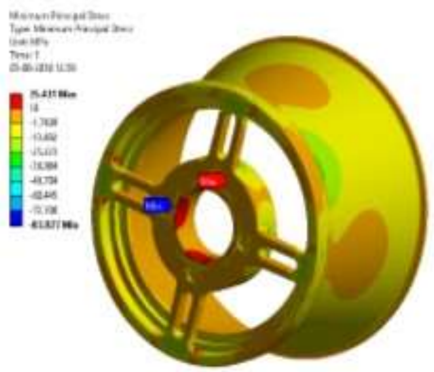


Figure 5: Equivalent stress and Maximum principal stress for 4 arms wheel in linear stress analysis

Minimum principal stress **Total deformation**



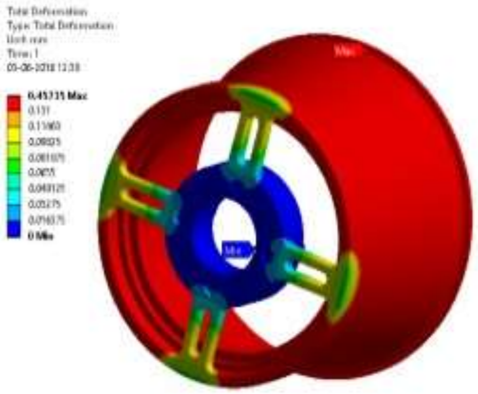


Figure 6: minimum principal stress and Total deformation for 4 arms wheel in linear stress analysis.

BI-LINEAR STATIC STRUCTURAL ANALYSIS FOR 4 ARMS WHEEL

Nonlinear stress analysis calculates the stresses and deformations of products under the most general loading and material conditions for:

1. Dynamic (time dependent) loads.
2. Large component deformations.
3. Nonlinear materials, such as rubber or metals, beyond their yield point.

Nonlinear analysis is a more complex approach, but results in a more accurate solution than linear analysis, if the basic assumptions of a linear analysis are violated. If the linear analysis assumptions are not violated, then the results of a linear and nonlinear analysis will be the same.

While performing the nonlinear analysis, the time component is important both in controlling the loading (individual load components can be active at different times) and in capturing the response to an impulse load of impact. SOLIDWORKS Simulation provides either an automatic or a manual time control method with a force, displacement, or arc length convergence control. You get power and flexibility to solve challenging and complex simulation problems simply in a straightforward manner.

The model of 4 arms wheel as shown in fig 6 was done in unigraphics and later loaded in to ansys 14 for Bi-linear static structural analysis for further process. The boundary conditions were applied on the rim surface of the aluminium wheel with pressure of 0.25 Mpa, rotational velocity of 222.2 rds or 2122 rpm and bolt pretension of 23340 N. After applying the boundary conditions following results were obtained:

1. Von-mises stress= 305.73 Mpa
2. Maximum principal stress= 317.36 Mpa
3. Minimum principal stress=35.415 Mpa
4. Total Deformation = 0.45735 mm

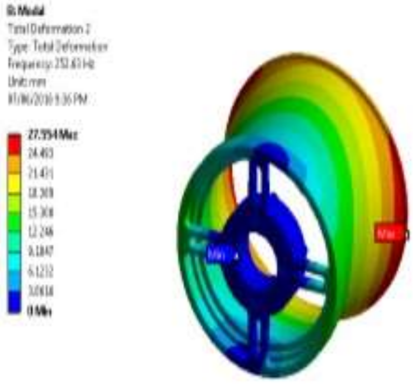
DYNAMIC ANALYSIS OR MODEL ANALYSIS

Table 1: Based on frequency tabular data

SL No	mode	Frequency (Hz)
1	1.	252.4
2	2.	252.63
3	3.	394.5
4	4.	415.87
5	5.	449.42
6	6.	522.66

Mode 1 **Mode 2**

ISSN: 2277-9655
Impact Factor: 4.116
CODEN: IJESS7



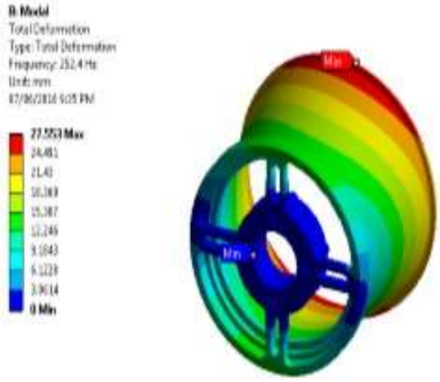
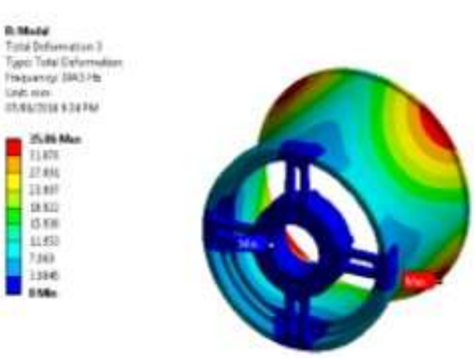


Figure 7: Total deformation in 1st mode with frequency 252.4 Hz. Figure 8: Total deformation in 2nd modewith frequency 252.63 Hz.

Mode 3



Mode 4

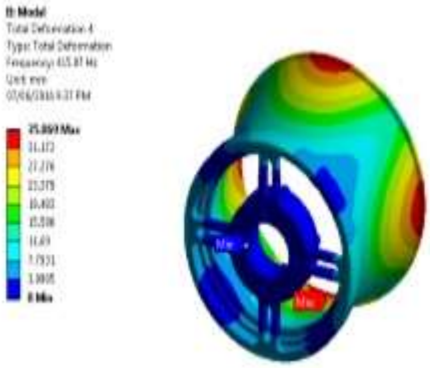
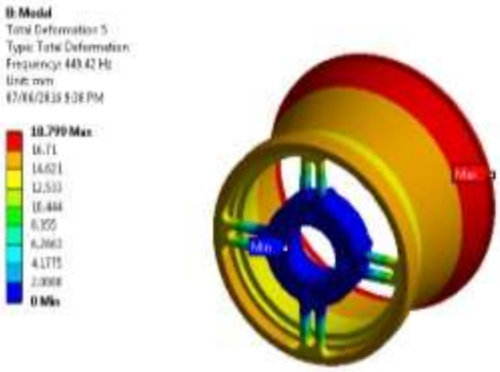


Figure 9: Total deformation in 3rd mode with frequency 394.5 Hz. Figure 10 : Total deformation in 4th mode with frequency 415.87 Hz.

Mode 5



Mode 6

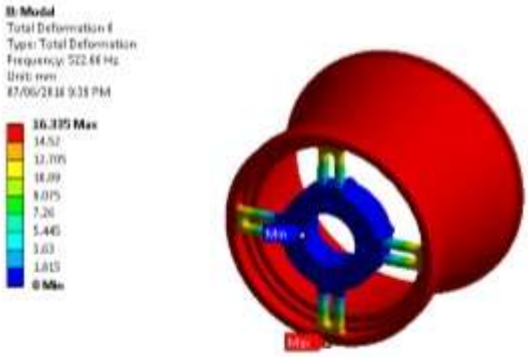


Figure 11: Total deformation in 5th mode with frequency 449.42 Hz. Figure 12: Total deformation in 6th mode with frequency 522.66 Hz.

DYNAMIC ANALYSIS/MODAL ANALYSIS

Dynamic analysis is the testing and evaluation of a program by executing data in real time as its main objective is to find errors in a program while it is running , rather than by repeatedly examining the code offline.

The modal analysis was carried out on the 4 arm rim wheel under different modes and frequency as shown in above figures. The maximum deformation was observed to be 27.553 mm for mode 1 with frequency 252.4 Hz. Similarly for mode 2 deformation 27.554 mm with frequency 252.63 Hz, for mode 3 deformation 35.86 mm with

ISSN: 2277-9655
Impact Factor: 4.116
CODEN: IJESS7

frequency 394.5 Hz, for mode 4 deformation 35.069mm with frequency 415.87 Hz, for deformation 18.799mm with frequency 449.42 Hz were obtained.

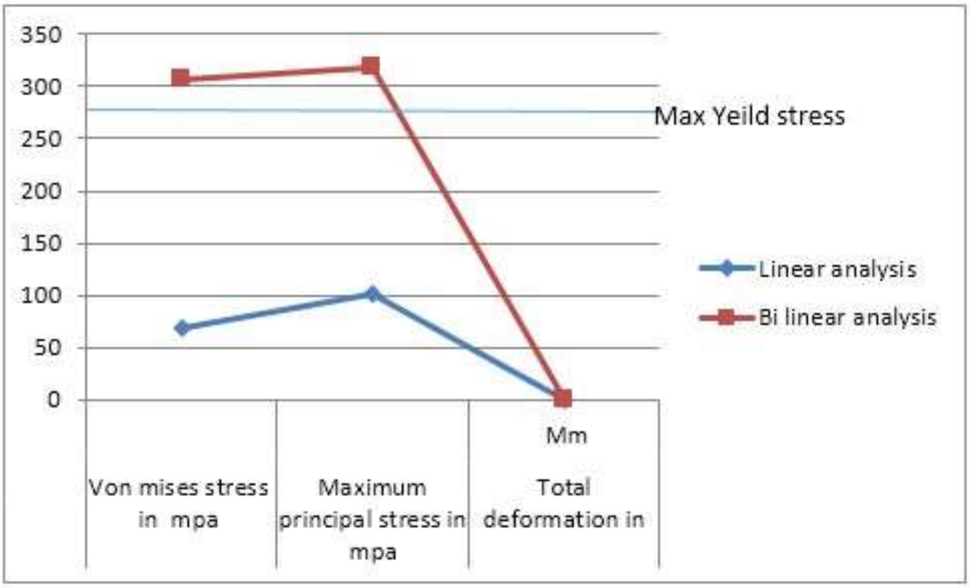
RESULET AND DISCUSSION

COMPARISON BETWEEN LINEAR AND BI LINEAR VALUES OF THE 4 ARMS WHEEL

Table 2: comparison between linear and bi linear values of the 4 arms wheel.

4 arms wheel	Von mises stress in Mpa	Maximum principal stress in Mpa	Total deformation in mm
Linear analysis	67.814	101.79	0.14686
Bi linear analysis	305.73	317.36	0.45735

It was observed that the von mises stress obtained from linear analysis for 4 arms wheel was less compared with the results obtained from bilinear analysis. The reason for high von mises stress in bi linear is because of high pressure and high RPM. It was also observed that the stress obtained in linear analysis were within the yield stress for aluminum, Hence the design is safe.



Graph 1 : linear and nonlinear stress values

CONCLUSION

1. Static linear analysis was carried out and maximum equivalent stress is 67.814 Mpa which was less than the yield stress of a given material and for applied load, hence the Design is safe.
2. From the above graph it was observed that the max yield stress obtained was within the limit 260 Mpa above which it will yield for failure.
3. Dynamic model analysis was carried out to find six initial modes and natural frequency of the car alloy wheel.

REFERENCES

[1] Wang Qiang, Zhang Zhi-min, Zhang Xing, Li Guo-jun, Trans. Nonferrous Met.Soc.China 20 (2010)599-603
[2] Wang Jian-hong, Long Si-Yuan, Cao Han-xue, Special Casting & Nonferrous alloys, 2004(5) 21-23
[3] Peng Ying-hong, Wang Ying-chun, Li Da-yong, J China Mechanical Engineering 2006 17(19) 2034-2037.
[4] Wu Zeng-chen, Long Si-yuan, Xu Shao-yong, j.Foundry 2005 54(9), 878- 880
[5] Cai Suo-qi, Cui Er-xin, J Foundry Technology, 2001 (5) 8-10
[6] JIS D 4103, Japanese Industrial Standard, Disc wheel for Automobiles, 1989.
[7] Grubisic V, Fischer G, SAE Technical Paper Series 830135; 1984: 1.508- 1.525
[8] Hsu YL, Wang SG, Liu TC, J Chin Inst Industrial Eng 2004; 21 (6), 551- 558

[9] Sunil N Yadav, NS Hanamapure, Int Jr. of Engg Sci, and Innovative Tech. Vol 2 Issue 5, Sept (2013) 213-239

[10] P. Ramamurty Raju, B. Satyanarayana, K. Ramji, K. Suresh Babu, Engineering Failure Analysis, 14 (2007)791-800.