DESIGN AND ANALYSIS OF WHEEL RIM

Premkumar.D[1] , Dr.V.Balaji[2], Raghul.P[3], Vignesh. P[4] & Vimlesh kumar.S[5]
[1]Assistant Professor,[2]Associate professor
Department of Mechanical Engineering
Prathyusha Engineering College, Thiruvallur, Chennai, India

Abstract: Design has constantly played a key role in automobile engineering. This often leads to multifaceted car designs which need to be fashioned and proof tested with least lead time and expenses. Latest designs and industrialized technologies must be constant .Thus the automobile producer is increasingly more investigating and evolving new design tools to help proceed the excellence of their products. CAE helps to diminish the time required to build a fresh design. It also improves the quality of design. In this study fatigue life calculation, computer aided design, and finite element analysis are the tools which have been used.

***Index Terms*- alloy wheel, Finite element Analysis, Static structural analysis**

1. INTRODUCTION

In an auto each segment is vital yet some are more basic than others Parts which may bomb however won't cause lethal mischances are not cataclysmic in disappointment the auto electrics are a case of this Wheel, breaks, directing or tire disappointment, then again, can cause catastrophic .Road wheels are a standout amongst the most critical wellbeing segments from an auxiliary perspective .They are required to be lighter and more appealing to the client constantly. This implies it has turned out to be important to perform more thorough quality assessments on new wheel plans.

1.1 WHEEL RIM

The edge is the external edge of a wheel, holding the tire. It makes up the external roundabout outline of the wheel on which within edge of the tire is mounted on vehicles. The term edge is additionally utilized non-in fact to allude to the whole wheel, or even to a tire.

Fig-1: Isometric view of alloy wheel

2. DESIGN OF ALLOY WHEEL

2.1. DESIGN PARAMETERS

The design parameters were selected from reference papers and OEM's. Some modifications are carried out by varying the thickness and geometrical dimensions. The existing design is modified by increasing the thickness of the sections used and also evaluating the model using three materials namely grey cast iron, forged steel, aluminum alloy.

Table-1 Specification of materials

Property	Aluminum alloy	Forged steel	Grey cast iron
Poisson's Ratio	0.3	0.31	0.29
Tensile Yield Stress(MPa)	$2.8e^8$	$2.3e^8$	$2.4e^8$
Compressive Ultimate Stress (MPa)	$2.5e^8$	$5.2e^8$	$8.2e^8$
Mass Density(kg/m^3)	2770kg/m^3	7810kg/m^3	7200kg/m^3

2.2. DESIGN OF ALLOY WHEEL USING CATIA

The geometric model of the alloy wheel (Ladder Frame) is done using CATIA Software. The Three Dimensional model of the Ladder Frame is shown in Fig. 2.

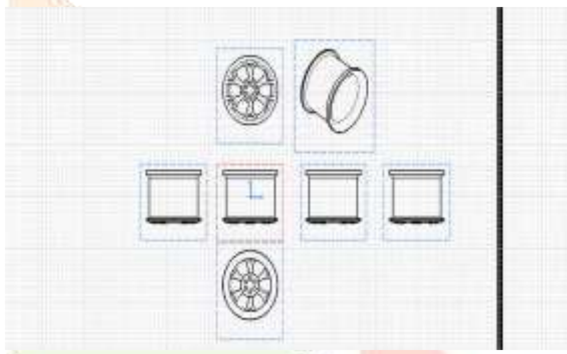

Fig-2: CATIA Model of alloy wheel

The radius of the alloy wheel is 180mm and the width of the wheel rim is 167mm.

Let the following conditions be utilized for analysis of the alloy wheel
Geometry of alloy wheel

- radius= 180 mm
- width= 180 mm
- Weight of the car used in analysis = 725Kg
- Total Load at each wheel= 1618.65N (Weight of the Driver and approximation of 800cc car)

3. ANALYSIS OF ALLOY WHEEL

After the creation of the model using the design software, the geometric model is converted into STEP format in order to avoid data losses occurring due to importation of the geometric design file. There are three main steps involved in the analysis software, namely: pre-processing, solution and post processing. In the pre-processing stage, the geometric domain of the model is imported along with its material properties and boundary constraints. Further the geometric is meshed in several nodes and elements for accurate evaluation of the problem.

3.1. FINITE ELEMENT ANALYSIS OF ALLOY WHEEL USING ANSYS WORKBENCH

The model of the rim is saved in STEP format which is imported into ANSYS Workbench. The imported model is shown in Fig-3

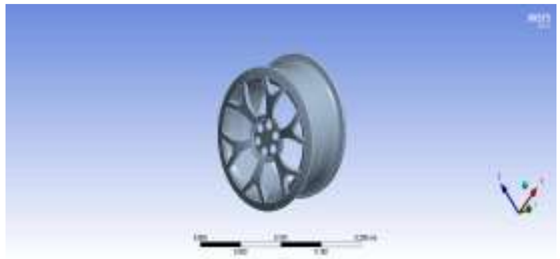

Fig-3: Imported model in ANSYS Workbench

3.2. MESHING AND BOUNDARY CONDITIONS

The model is meshed initially with 218447 nodes and 126360 tetrahedral elements. The meshed model is represented in the Fig-4.

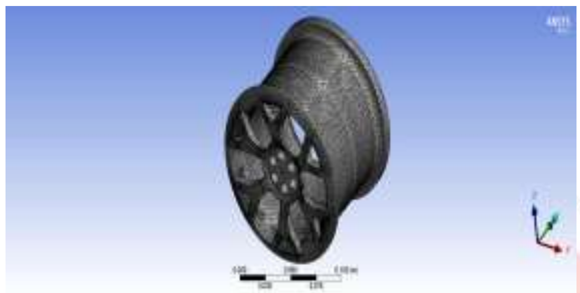

Fig-4: Meshed model of Alloy wheel

The alloy is provided with necessary working loading which is considered to be uniformly distributed throughout the alloy. The maximum weight of the structure carried by the alloy is considered to be 1616.65N.

3.3. STATIC STRUCTURAL ANALYSIS OF ALLOY WHEEL

The Finite element static structural analysis of ansys model is experimented using three different materials –aluminum alloy, forged steel and grey cast iron. The contour plots of all the three materials comprising the Deformation are shown in Fig-5 to Fig 7.

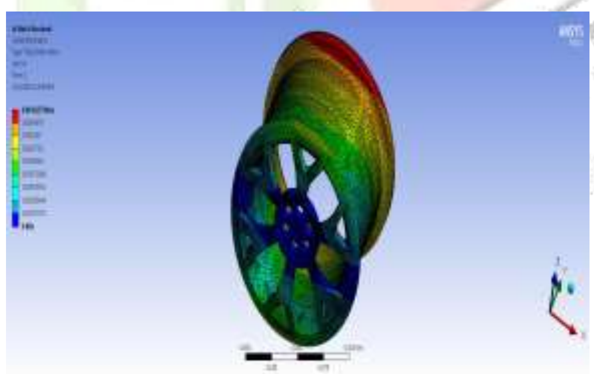

Fig-5 Total Deformation of aluminum alloy

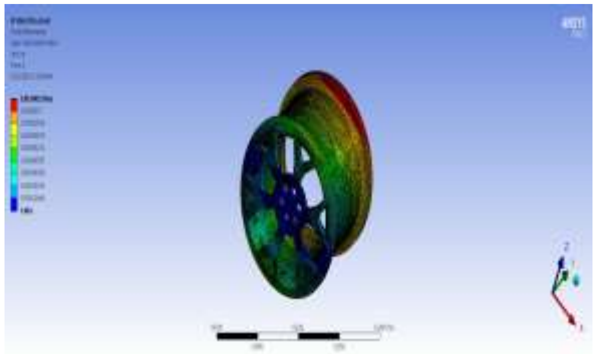

Fig-6 Total Deformation of grey cast iron

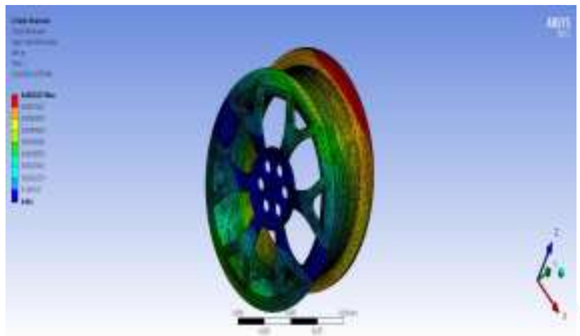

Fig-7 Total Deformation of forged steel

4. RESULTS

The problem given is a complex structure, the theoretical method cannot be used to determine accurate values and hence FEA results are taken into consideration. By using ANSYS, the results are tabulated in the Table-4.

Table-4 Results

Material	Aluminum alloy	Forged steel	Grey cast iron
Von Mises Stress (MPa)	8.3359e8	8.4888e8	8.5884e8
Deformation (mm)	0.0016175	0.00082257	0.0010482
Max. Stress Intensity (MPa)	5.9177e7	7.0253e7	7.0141e7
Equivalent Strain (mm/mm)	0.01359	0.0070331	0.0090661

5. CONCLUSION

This paper focuses on improving the strength and aerodynamics of alloy. After the analysis of the alloy wheel, it is found that the Equivalent Stress value is Highest for grey cast iron and Lowest for aluminum and the deformation is highest for aluminum and lowest for forged steel. All the three materials have the working stress value less than their Tensile Yield Strength and hence the design is safe for all the three materials.

6.REFERENCES

[1] Cook, R D., Malleus, D. S and Plesha M. E *'Concepts and Applications of Finite Element Analysis'*, (1989), Published by John Wiley& Sons, Inc., New York.

[2] *NAFEMS -A Finite Element Primer,* (1992)

[3] Segerling, L J., *Applied Finite Element analysis,* (1984), Published by John Wiley and Sons, In c , New York.

[4] Davies, A J., *The Finite Element, a First Approach,* (1980), Published by Clarendon Press, Oxford.

[5] Strang, G., Fix, G. J- *An Analysis of the Finite Element Method,*, Published by Prentice-Hall, In c , Englewood cliffs, N. J.

[6] Turner, M. J, Clough, R W, Martin, H C and Topp, L J , *'Stiffness and Deflection Analysis of Complex Structures',* (1956), Journal of Aeronautical Science, Vol 23, pp 805-832

[7] Argyris, J. H., and Kelsey, S , *'Energy Theorems and structural analysis Aircraft Engineering',* (1955), Vols 26 & 27.

[8] Clough, R W. *'The Finite Element in Plane Stress Analysis',* (1960),Proc. 2nd A S.C.E Conf. on Electronic Computation, Pittsburgh94

[9] Robinson, J, *'The Lives and Work of Early FEM Pioneers',* (1985),Robinson and Associates, Horton Road Woodlands, Wimbome, Dorset,England