ISSN 2395-1621

Material Analysis Of RC Car Chassis

#1Kunal H. Sonawane, #2Sagar S. Mandale, #3Prithviraj D. Mane,
#4Somnath R. Pawar, #5Atul Gowerdipe

[1]ksp6600@gmail.com
[2]mandalesagarr@gmail.com
[3]prithvimane123@gmail.com
[4]somnathpawar5555@gmail.com
[5]atul.gowerdipe@dypic.in

1234Department Of Mechanical Engineering
#5Professor, Mechanical Engineering Department

Dr. D.Y.Patil S.O.E.T., Lohegaon, Savitribai Phule Pune University, Pune,-411047

ABSTRACT

The Radio control dates back to the nineties when it first evolved. It is a hobby for most of the engineering and automobile enthusiasts. It is proved by its rigidity, precision and ability. Not only design of RC car but also maximum stress, maximum equilateral stress and deflection are important criteria for design of the chassis. The chassis of RC car or any vehicle which is the backbone of vehicles that integrates the main component systems. The main objective of this article is to analyse different materials for RC car chassis.

Keywords—RC Car, Chassis, Displacement, FEA Analysis

ARTICLE INFO

Article History

Received: 5th April 2017

Received in revised form : 5th April 2017

Accepted: 7th April 2017

Published online : 8th April 2017

I. INTRODUCTION

The main drawback was the weight of the car. We had found a solution for it. So we decided to reduce the weight of the chassis as this was the only option for reducing the weight of the car without any change in the performance of the car. Radio controlled (or R/C) cars are battery/gas-powered model cars or trucks that can be controlled from a distance using a specialized transmitter or remote. This Radio controlled system is adopted in many vehicles like cars, boats, planes, and even helicopters and scale railway locomotives [1]. The chassis of car is the backbone of RC car or any vehicles. The stress analysis is important in fatigue study and life prediction of components to determine the highest stress point commonly known as critical point which initiates to probable failure, this critical point is one of the factors that may cause the fatigue failure. The magnitude of the stress can be used to predict the life span of the chassis [2]. Not only design of RC car but also maximum stress, maximum equilateral strain and deflection are important criteria for design of the chassis. The greater the energy absorbed by the chassis on impact the lower the energy levels transmitted to a vehicles occupants and surroundings, lowering the chances of failure [3].

Conventionally we use aluminium 7075 chassis for RC nitro car but we have to increase its speed and efficiency for its better performance in race. This can be achieved by reducing car weight. For that, chassis analysis for different materials by using ANSYS 15.0 has to be done. By this analysis will help us for choosing chassis material for our car.

Figure1. RC car

II. LITERATURE REVIEW

Keshav jha, sai phanindra dinesh kakarlamudi, bharath konduru , mukesh reddy vadala, shravani rallapati, sonu

gupth & chanikya virugadinla et al [1] have design a radio controlled car & fabricate a radio controlled cars all parts.

Hemant B.Patil , Sharad D.Kachave , Eknath R.Deore et al [2] Conducted analysis of chassis structures, For determining the strength of the frame, structural analyses were performed for these frames of thicknesses of 4, 5, 6 mm and also changing the position of cross member and change the thickness of cross member near high stress zone. The truck chassis was modelled and the finite element analyses were solved in CATIA V5R10.

Ahmad o. Moaaz, Nouby M. Ghazaly et al [3] done the fatigue analysis of heavy duty truck frames using ANSYS & found that the chassis analysis mainly consists of stress analysis to predict the life of the chassis.

Roslan Abd Rahman et al [4] does stress analysis of heavy duty truck chassis by utilizing a commercial finite element package ABAQUS. To determine critical point so that by design modifications the stresses can be reduces to improve the fatigue life of components. During this he uses ASTM low alloy steel a 710 C (Class 3) with 552 MPa of yield strength and 620 MPa of tensile strength for chassis founds the maximum stress 386.9 MPa at critical point occurred at opening of chassis This critical point is located at element 86104 and node 16045, which is in contacted with the bolt from this he concludes that this critical point is an initial to probable failure.

Cicek Karaoglu and N. Sefa Kuralay [5] did the finite element analysis of a truck chassis. The analysis showed that increasing the side member thickness can reduce stresses on the joint areas, but it is important to realize that the overall weight of the chassis frame increases. Using local plates only in the joint area can also increase side member thickness. Therefore, excessive weight of the chassis frame is prevented.

Karaoglu and Kuralay [6] investigated stress analysis of a truck chassis with riveted joints using FEM. Numerical results showed that stresses on the side member can be reduced by increasing the side member thickness locally.

Kutay Yeilmazcoban, Yasar Kahraman [7], put some works on the chassis optimization by using the finite analysis, his main focus was on the reduced the weight of the chassis for that he used three thickness 4mm, 5mm & 6mm and after analysis he conclude that the 4mm thickness is better because the stress and the displacement in that is better than other two thickness.

O kurdi, R abd-Rahman, M N Tamin [8], works on the stress analysis of heavy duty truck chassis using finite element method , he mainly focus on the important steps in development of a new truck chassis is the prediction of fatigue life span and durability loading of the chassis frame. Fatigue study and life prediction on the chassis is necessary in order to verify the safety of this chassis during its operation. Stress analysis using finite element method can be used to locate the critical point which has the highest stress. This critical point is one of the factors that may cause the fatigue failure.

Haval Kamal Askerl, Thaker salih Dawood and Arkan Fawzi [9],put some works on the stress analysis of standard truck chassis during ramping on block using finite element method and he focused on the intensity and the strength of the frame play a big role in the trucks design. A frame of 6 wheels , Standard dump truck has been studied and analysed using Ansys software.

III. FE ANALYSIS OF CHASSIS

For Analysis of chassis we used different materials.

1] Aluminium 7075
2] Aluminium 6061
3] Carbon fibre
4] Glass fibre

Analysis is done in ANSYS 15.0

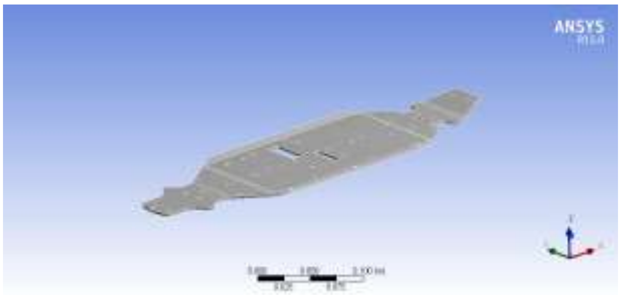


Figure 2. Design of chassis

Dimensions of the chassis are:

Length of the chassis is 400mm and

Width is 122mm

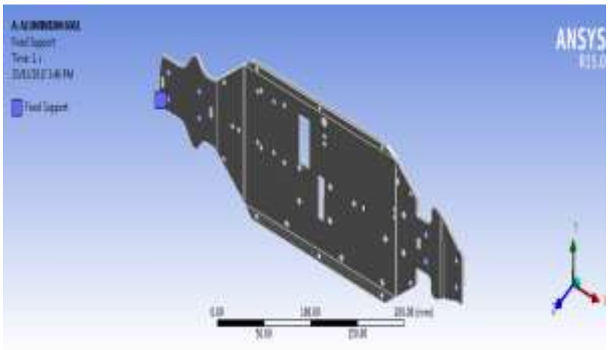


Figure 3. Fixed supports

Fixed supports at Front Axle and Rear Axle

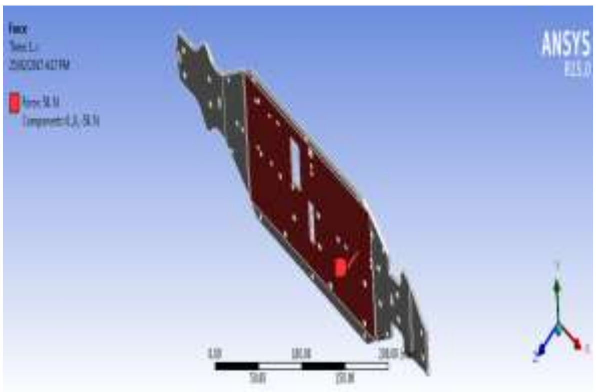


Figure 4. Load acting on chassis

Aluminium 7075

Density	2.81 g/cc
Ultimate Tensile Strength	572mpa
Tensile yield strength	503mpa
Poisson' s ratio	0.33

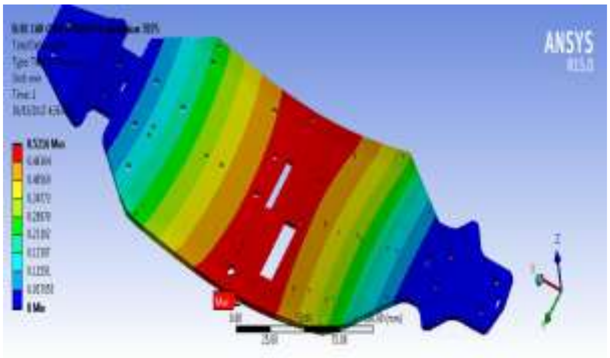


Figure 5 deformation of aluminium 7075

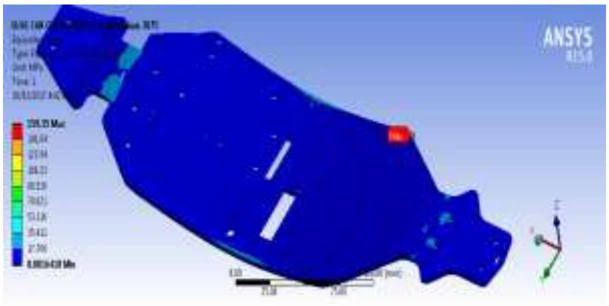


Figure 6 stresses of aluminium 7075

Aluminium 6061

Density	2.7 g/cc
Ultimate Tensile Strength	310mpa
Tensile yield strength	276mpa
Poisson' s ratio	0.33

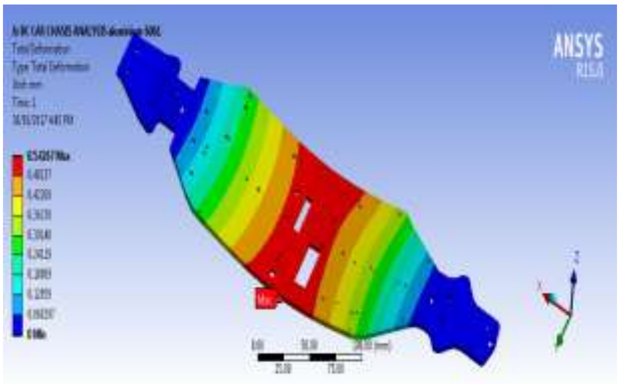


Figure 7 deformation of aluminium 6061

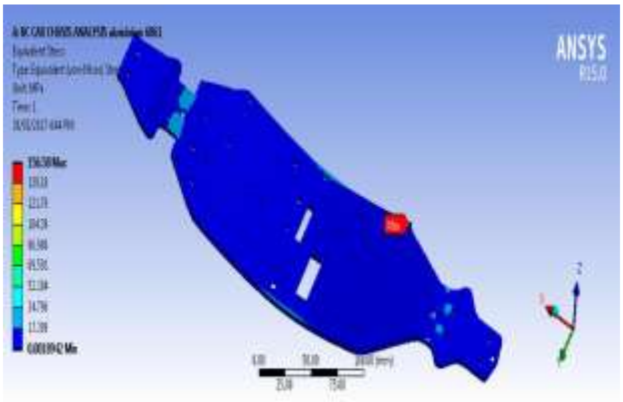


Figure 8 stresses of aluminium 6061

Carbon fibre

Density	1.6 g/cc
Ultimate Tensile Strength	600mpa
Tensile yield strength	540mpa
Poisson' s ratio	0.33

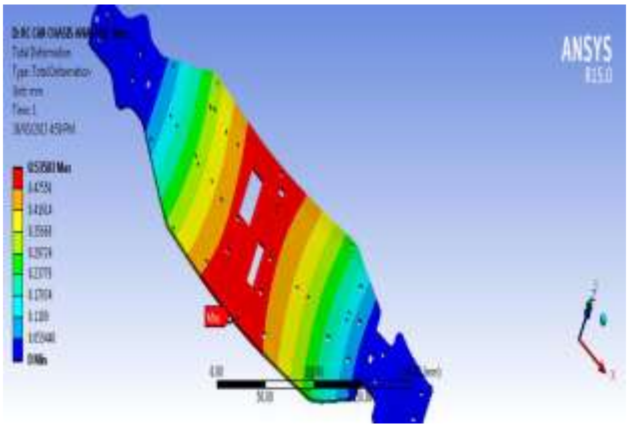


Figure 9 deformation of carbon fibre

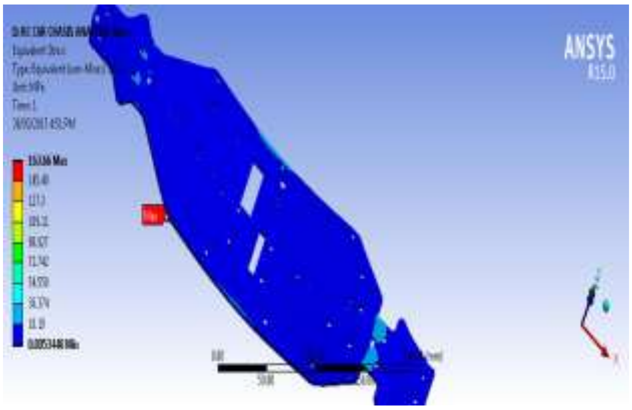


Figure 10 stresses of carbon fibre

Glass Fibre

Density	2.55 g/cc
Ultimate Tensile Strength	600mpa
Tensile yield strength	540mpa
Poisson' s ratio	0.21

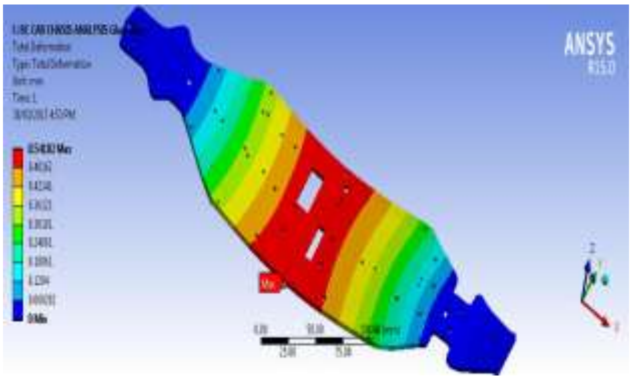


Figure 11 deformation of glass fibre

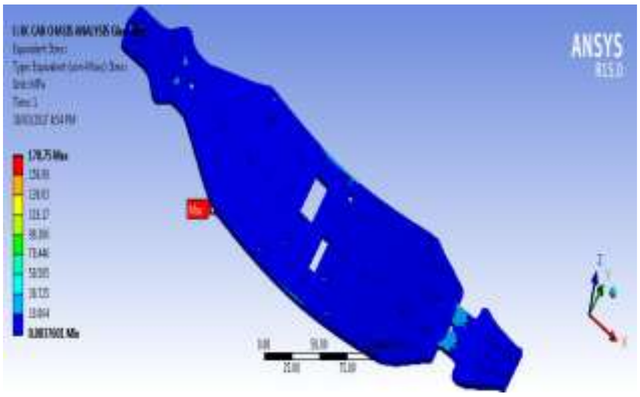


Figure 12 stresses of glass fibre

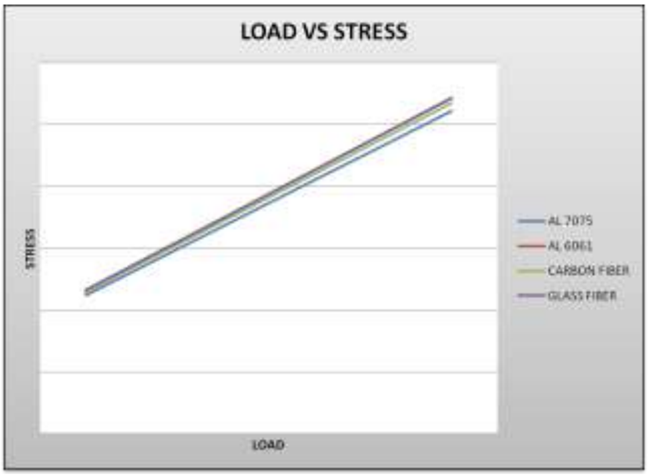
IV. RESULTS

For 70 N load

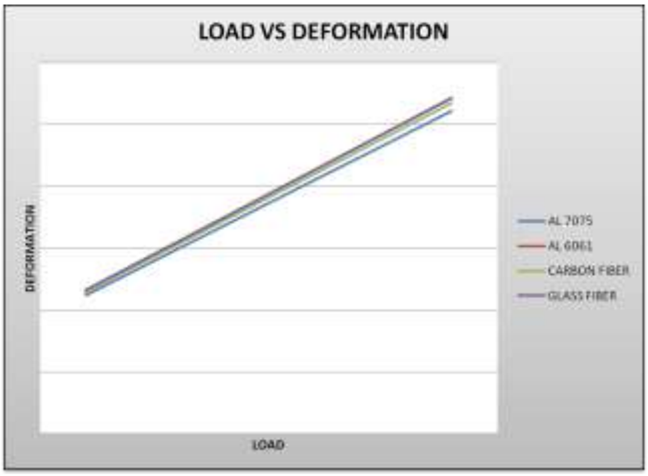
Material	Equivalent strain	Stress (Mpa)	Deformation (mm)	Wt (kg)
Al 7075	0.0028406	159.35	0.5216	0.28448
Al 6061	0.002916	156.58	0.54267	0.27432
carbon fibre	0.0029134	163.66	0.53503	0.16256
Glass fibre	0.0031824	178.75	0.54182	0.25908

V. GRAPHS

LOAD VS STRESSES



LOAD VS DEFORMATION



VI. CONCLUSION

- The analyses are processed in the static and structural conditions. From comparison it is observed that minimum amount of weight is for carbon fibre i.e.0.16256 kg. Thus by using carbon fibre for chassis we can reduce maximum amount of weight of chassis.
- Weight for glass fibre chassis is 0.25908 kg.
- Weight for aluminium 6061 is 0.27432 kg.

REFERENCES

[1] Keshav Jha, Sai Phanindra Dinesh Kakarlamudi, Bharath Konduru, Mukesh Reddy Vadala, Shravani Rallapati, Sonu Gupta and Chanikya Virugadinla, DESIGN AND FABRICATION OF RADIO CONTROLLED CAR" , International Journal of Mechanical Engineering and Technology (IJMET) Volume 6, Issue 8, Aug 2015, pp. 01-11, Article ID: IJMET_06_08_001

[2]Hemant B. Patil, Sharad D. Kachave, Eknath R. Deore," Stress Analysis of Automotive Chassis with Various Thicknesses" , IOSR Journal of Mechanical and Civil Engineering (IOSR-JMCE) Volume 6, Issue 1 (Mar. - Apr. 2013), PP 44-49

[3] Ahmad O. Moaaz1, Nouby M. Ghazaly2," A Review of the Fatigue Analysis of Heavy Duty Truck Frames" American Journal of Engineering Research (AJER) Volume-3, Issue-10, pp-01-06

[4] R. Abd Rahman, Mohd Nasir Tamin, OjoKurdi " Stress analysis of heavy duty truck chassis as a preliminary data for its fatigue life prediction using FEM" JurnalMekanikal December 2008, No. 26, 76 – 85.

[5] Cicek Karaoglu, N. Sefa Kuralay " Stress analysis of a truck chassis with riveted joints" Elsevier Science B.V Finite Elements in Analysis and Design 38 (2002) 1115– 1130.

[6] Karaoglu and Kuralay [6],N.S.,2000," Stress analysis of a truck chassis with Riverted joints" , Elsevier science

Publishers B.V.Amsterdam, the Netherlands, vol.38,1115-1130.

[7] Kutay Yeilmazcoban, Yasar Kahraman ,sakarya university,Mesh. Engg dept.,54187 Serdivan-sakarya Turkey, july2011.

[8] O kurdi, R abd-Rahman, M N Tamin, Faculty of mechanical engg. University Teknologi Malaysia 81310 UTM skudai, johor, stress analysis of heavy duty truck chassis using finite element method.

[9] Haval Kamal Askerl, Thaker salih Dawood and Arkan Fawzi, " Stress analysis of standard truck chassis during ramping on block using finite element method" , June2012.

