Structural Analysis of Automotive Chassis Frame and Design Modification for Weight Reduction of Solar Vehicle

Dr. Deepak Paliwal[1], Md Zuber Khan[2], Vivek Mochi[3], Vaibhav Lohar[4], Pavan Lohar[5], Yogendra Menaria[6], Avish Sahu[7], Shubham Ojha[8]

[1]Head of Mechanical, Department of Mechanical Engineering, Geetanjali Institute of Technical Studies, Dabok, India

[2]Assistant Professor, Department of Mechanical Engineering, Geetanjali Institute of Technical Studies, Dabok, India

[3,4,5,6,7,8] Student, Department of Mechanical Engineering, Geetanjali Institute of Technical Studies, Dabok, India

***Abstract*- Automotive chassis is an important part of an automobile. The chassis serves as a frame work for supporting the body and different parts of the automobile. Also, it should be rigid enough to withstand the shock, twist, vibration and other stresses. Along with strength, an important consideration in chassis design is to have adequate bending stiffness for better handling characteristics. So, maximum stress, maximum equilateral stress and deflection are important criteria for the design of the chassis. This report is the work performed towards the optimization of the automotive chassis with constraints of maximum shear stress, equivalent stress and deflection of chassis under maximum load.**

Structural systems like the chassis can be easily analyzed using the finite element techniques. A sensitivity analysis is carried out for weight reduction. So a proper finite element model of the chassis is to be developed. The chassis is modeled in PRO-E. FEA is done on the modeled chassis using the ANSYS Workbench

I. INTRODUCTION

Automobile chassis usually refers to the lower body of the vehicle including the tires, engine, frame, driveline and suspension. Out of these, the frame provides necessary support to the vehicle components placed on it. Also the frame should be strong enough to withstand shock, twist, vibrations and other stresses. The chassis frame consists of side members attached with a series of cross members Stress analysis using Finite Element Method (FEM) 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. The magnitude of the stress can be used to predict the life span of the truck chassis.

2. BASIC CALCULATION FOR CHASSIS FRAME

The bending moment diagram for vertical loading is shown in fig.(D is the point where is maximum bending and shear)

Moment in vertical direction =

M_{DV} = 50.4Nm.

Moment in horizontal direction = M_{DH} = 0.

Resultant Bending moment at point D =

= $M_D = \sqrt{(M_{DV})^2 + (M_{DH})^2}$

= $\sqrt{(50.4)^2 + (0)^2}$

= 50.4 Nm.

Twisting moment =T_{DV} = 135.89 Nm.

Resultant twisting moment at point D =

= $T_D = \sqrt{(M_{DV})^2 + (M_{DH})^2}$

= $\sqrt{(135.89)^2 + (0)^2}$

= 135.89 Nm.

Equivalent Twisting moment =

= $T_e = \sqrt{M_D^2 + T_D^2}$

= $\sqrt{(50.4)^2 + (135.89)^2}$

= 144.93 Nm.

Equivalent Bending moment =

= $M_e = {}^1/_2(M + \sqrt{M^2 + T^2})$

= ${}^1/_2 (50.4 + \sqrt{(50.4)^2 + (135.89)^2})$

= 97.66 Nm.

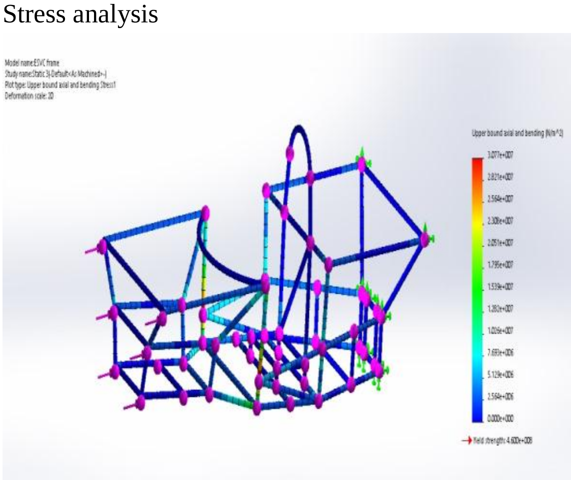
$T_{e=}\pi/16 * T * D^3$(where, $T = 200*10^5$)
$144.93 = \pi/16 * 200 * 10^5 * D^3$
$M_e = \pi/32 * S * D^3$(where $S= 190^*10^5$)
$97.66 = \pi/32 * 190 * 10^5 * D^3$
D = 0.031m = 1.25in.
$T_e = \pi/16 * T* (D^3 – d^3)$
$144.93 = \pi/16 * 200* 10^5 * (0.029^3 – d^3)$
d = 0.029m = 1.18in.
$M_e = \pi/32 * S * (D^3 – d^3)$
$97.66 = \pi/32 * 190* 10^5 * (0.028^3 – d^3)$
d = 0.028m = 1.14in.
Thickness t = D – d
= 2 mm.
material used was AISI 4130 stainless steel

Outerdiameter	1.5in
Wall thickness	2mm

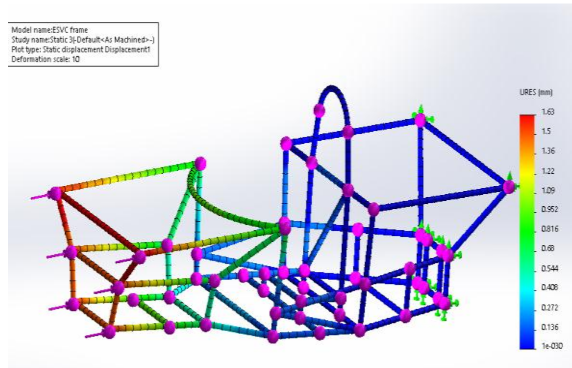
3. CHASSIS ANALYSIS

Force calculation as follows-
For ,static condition the force is applied is 2g frce and vehicle collision speed assumed to be 60 km/h, collision time be 0.1 sec, mass for the vehicle is 200 kg and frame mass is 25 kg.
Calculation-
F=25*(60*1000/3600-0)/0.1
=4165 N *2
= 8330N

A) Front impact
Stress analysis

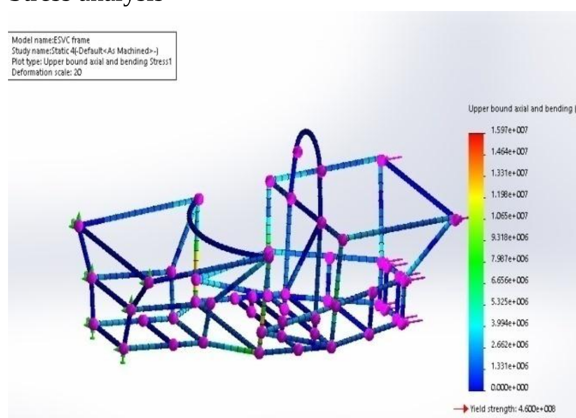


Deformation on front impact-

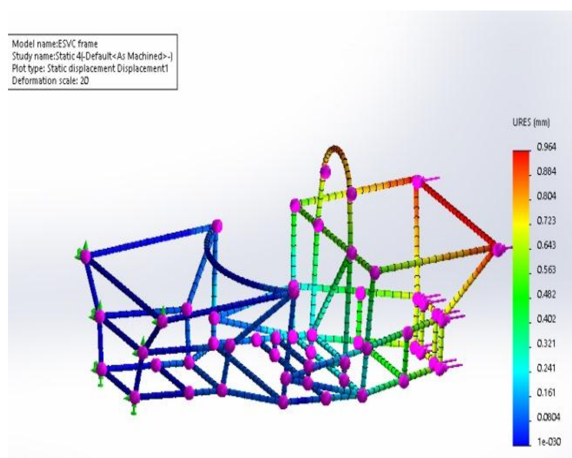


Max stress	307Mpa
deformation	1.63mm
F O S	2.36

B) Rear impact
Stress analysis

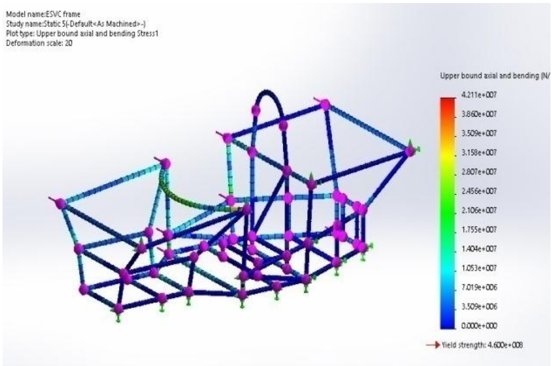


Deformation at rear-

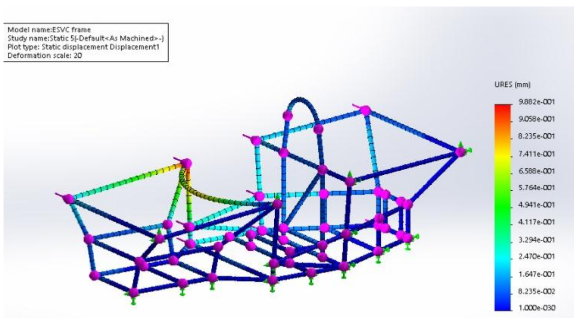


Maximum stress	159 Mpa
Deformation	0.904mm
F O S	2.85

C) Roll Over Impact-



Deformation



Maximum Stress	421Mpa
Deformation	9.88mm
F O S	3.2

4. CONCLUSION

The chassis frame component has been modelled using Creo (Pro-E) 2.0 and analyzed in ANSYS WORKBENCH 15.0. The various parameters such as Nodal displacement, stress distribution are completely analyzed and studied. The study shows that the area where the stress concentration is maximum due to applied load and the portions that has to be considered in the design of chassis frame in order to avoid frequent failures to improve its reliability.

Stress analysis of chassis has been done to predict the weak points. Several state of the art papers and even books on chassis stress analysis have been presented in the recent years. This study makes a case for further investigation on the design of car chassis using Ansys software.

REFERENCE

[1] Stress analysis of a car chassis with riveted joints by Cicek Karaoglu*, N. Sefa Kuralay, Department of Mechanical Engineering, DEU Faculty of Engineering, 35100 Bornova, Izmir, Turkey ,Finite Elements in Analysis and Design 38 (2002) 1115–1130

[2] CAR CHASSIS STRUCTURAL THICKNESS OPTIMIZATION WITH THE HELP OF FINITE ELEMENT TECHNIQUE I. Kutay YILMAZÇOBAN*, Yaşar KAHRAMAN, TOJSAT : The Online Journal of Science and Technology - July 2011, Volume 1, Issue 3

[3] Stress analysis of Car chassis as a preliminary data for its fatigue life prediction using FEM. Roslan Abd Rahman, Mohd Nasir Tamin, Ojo Kurdi* Jurnal MekanikalDecember 2008, No. 26, 76 – 85

[4] Finite element analysis" by – Chandra Patla.

[5] "Finite element analysis" by – Robert D. cook [9] A text book of Finite element analysis By Seshu P.

[6] PSG Design Data Book for Standard Data-M/s Kalaikathir Achchagam, Coimbatore2004 Machine design by R S Khurmi, S chanda and Co. Ltd., New delhi-2001.

[7] Beam formula with shear and moment diagram, American forest and paper associatiom, Inc, American Wood Council, 1111 19th St., NW. Suite 800, Washington. DC 20036, 202 – 463.