International Journal of Applied Engineering Research ISSN 0973-4562 Volume 13, Number 11 (2018) pp. 9887-9892

Structural Analysis of Automotive Chassis, Design Modification and Optimization

Rohan Y Garud

M.Tech. CAD-CAM Department, Symbiosis Institute of Technology, Symbiosis International University, Lavale, Pune – 412115, India.

Shahid C Tamboli

Assistant Professor, Department of Mechanical Engineering, Symbiosis Institute of Technology, Symbiosis International University, Lavale, Pune – 412115, India.

Dr. Anand Pandey

Professor, Department of Mechanical Engineering, Symbiosis Institute of Technology, Symbiosis International University, Lavale, Pune – 412115, India.

Abstract

Chassis is a part of automobile vehicle which is used to support various components such as engine, gearbox, clutch, propeller shaft, brakes, fuel tank etc. It is used to propel the vehicle. It carries all the necessary load and weight of the components; hence it should be strong enough to uphold the shock, twist, vibration and other stresses. This paper reviews the research work carried out, in the area of chassis with the constraints of maximum stress, deflection and critical regions under loading conditions. Simulation technique was adopted in design optimization for weight reduction, better material utilization and suitable cross section. Web height and thickness was changed to check their effect on weight.

Keywords: Chassis; Optimization; Stress, deflection and vibration Analysis, FEA.

INTRODUCTION

Chassis is considered as a central frame of the vehicle which has to carry all the components and support all the loads. These loads include the weight of each component and the forces which occur during acceleration, deceleration, and cornering. Chassis should be rigid enough to absorb the shock, twist, vibration and other stresses. The critical consideration for chassis design is resistance to bending and torsional stiffness apart from strength for better handling characteristics. The behavior and stress distribution of medium duty truck frame under dynamic conditions was studied[1]. In another study analysis of heavy duty truck chassis working in mining region was carried out[2].

Chassis structure is mainly divided into three types as Conventional frame, Integral frame and Semi Integral frame. In conventional frame, it consists of two long side members and 5 to 6 cross members joined together with the help of rivets and joints. In Integral frame, instead of conventional chassis all the units of vehicle are attached to body, which are mostly used in all forms of cars and light commercial vehicles. In Semi Integral frame, half part of frame is fixed on the front side to which engine gearbox and front suspension is mounted. Sub-frame chassis of a dump truck which is used in mining operation was analyzed for fatigue crack occurrence[3]. In conventional frame, the cross members consist of I-section or C-section which have good resistance towards bending stiffness. A tubular section which offers better torsional stiffness whereas box section is better in bending as well as torsional stiffness. This cross section plays an important role in carrying load of the vehicle. Basically a C-section should consist of at least 4mm thick and a ratio of 3:1 for web depth to flange width. This arrangement provides better bending stiffness as compare to solid square section. For heavy duty purpose, I section should be used which is formed by placing two C section back to back providing better rigid support. This was proved by a study in which structural design of I-beam was investigated[4]. Apart from cross section rivets stress analysis on riveted joints was investigated[5].

Automobile frames are manufactured from aluminum and steel which make it heavy and directly affects the efficiency of the vehicle. So now a day the lightweight composite material like carbon fiber, epoxy glasses etc. are used as a manufacturing material for chassis. The composite materials have a specialty that they are lighter in weight and good in strength as compared to the conventional steel used in chassis frame. Investigators carried out FEA analysis to determine the stress analysis of chassis to determine occurrence of failure during the testing process. HyperMesh and OptiStruct 8.0 simulation software was used to determine the cause of failure[6]. In another work static analysis for the chassis considering 8 ton capacity was carried out[7].

It was felt that chassis need to be analyzed against different design parameters such as materials, topology, cross sections from static, dynamic and weight reduction point of view. For this purpose modal, static and optimization was carried out in ANSYS. A work on vibration and dynamic analysis in

International Journal of Applied Engineering Research ISSN 0973-4562 Volume 13, Number 11 (2018) pp. 9887-9892

ABAQUS presented the strength of a frame while in operating conditions[8]. In another study of stress analysis, the fatigue life of the chassis components was presented by determining the high critical stress value[9].

ANALYSIS OF EXISTING MODEL OF CHASSIS

The model of chassis is as shown in Figure 1 whose length is 3352.8 mm, front and rear width is 1050.8 mm and 760 mm respectively. Material used for chassis is Steel. Static analysis on various cross sections by considering structural steel as material was carried out to check deformation and stress[10].

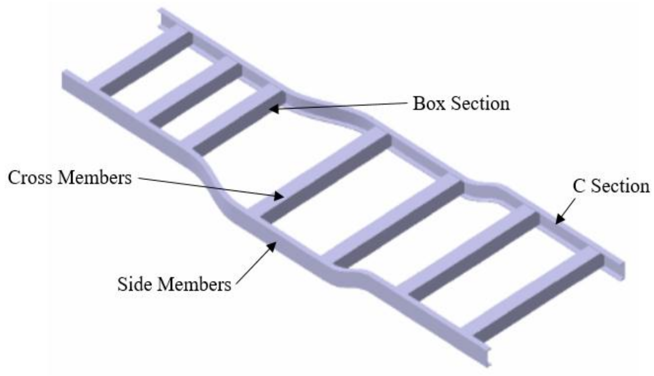

Figure 1: Model of Chassis

CAD model of chassis was designed in CATIA and analysis was carried out in ANSYS WORKBENCH. Meshing was done in Hypermesh software by using Hexa and Tetra elements. 978684 number of elements were formed as fine meshing was used. Meshed file was imported in Ansys. The results of static structural and modal analysis were compared with original and modified model of chassis.

Static structural: A static structural analysis was used to calculate displacements, stresses, strains and forces in a structure due to the application of load. Here loading condition was assumed to take place in an equilibrium condition whereas load and structures response was varied with respect to time. Figure 2 shows the result of static analysis with maximum deformation value of 1.8285 mm and Figure 3 shows the maximum stress value of 128.85 MPa.

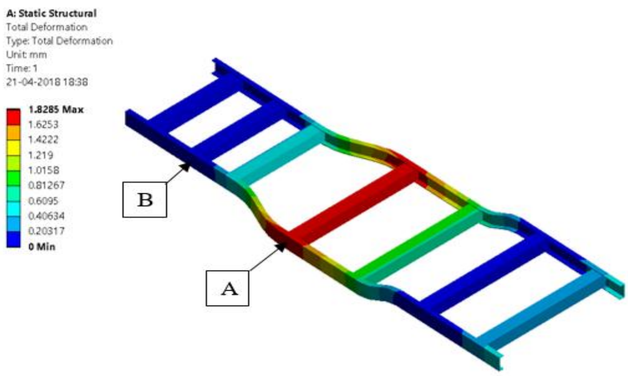

Figure 2: Total Deformation-Point A and B are located atthe region joining the cross members and side members.

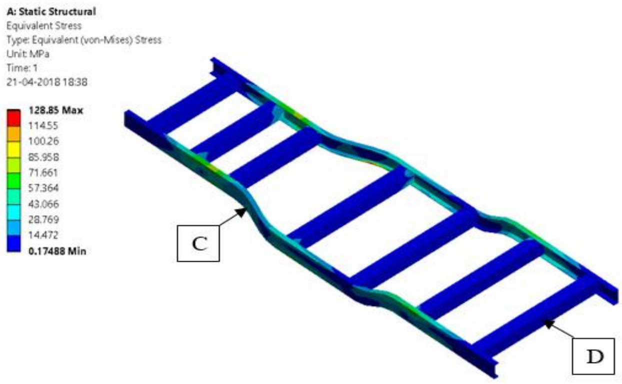

Figure 3: Equivalent Stress-Maximum stress was observed at point C, that is on lower side of side member and minimum was observed at point D that is on cross members.

Modal Analysis: Modal analysis was used to determine the vibrational characteristics of a structure. It calculates natural frequencies along with its mode shape. Natural frequencies and modes shape obtained in modal analysis were provided as input parameters for a transient and harmonic analysis. Static as well as dynamic analysis of frame was carried out by considering factor such as weight, velocity and road profile11. Figure 4 shows the result of modal analysis.

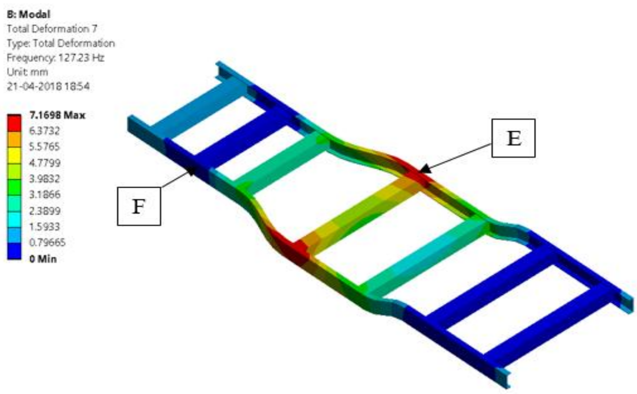

Figure 4: Modal Analysis-Maximum deformation was observed at point E, that is on upper side of side member and minimum was observed at point F.

CHASSIS OPTIMIZATION

To check the sensitivity between material, thickness and number of cross section following combinations were simulated. The nomenclature system for a particular case is as follows -

"Material – Thickness-Cross members". Where, material was designated as St, Al, Mg, Ti, CFRP and AHSS which means Steel, Aluminum, Titanium, Carbon Fiber Reinforced Polymer and Advanced High Strength Steel respectively. Thickness was designated as "T" which was varied from 3 to 12 mm whereas cross members which were designated as

International Journal of Applied Engineering Research ISSN 0973-4562 Volume 13, Number 11 (2018) pp. 9887-9892

"CM" were 5 CM indicates five cross members and 4 CM indicates 4 cross members.

By permutation and combinations different cases were finalized in two parts. First part is the optimization of chassis with 5 cross members along with the material and second part is the optimization of chassis with 4 cross members along with the material. Table 1 shows the deflection and stress values for 5 cross members. Figures 5-9 represents the graphical information mentioned in Table 1

Table 1. Deflection, stress and frequency values for 5 cross members

Designation	Deformation (mm)	Stress (MPa)	Frequency(Hz)	Weight (kg)
Al-T3-5CM	6.5956	239.98	144.7	28.729
Mg-T3-CM	10.414	235.83	143.38	18.688
CFRP-T3-5CM	2.4607	236.07	187.67	16.283
Ti-T3-5CM	4.8827	235.71	130	47.916
AHSS-T3-5CM	2.807	236.07	145.42	81.263
Al-T8-5CM	2.8822	74.96	134.98	66.088
Mg-T8-CM	4.5485	74.963	133.49	42.945
CFRP-T8-5CM	1.075	74.844	148.36	37.458
Ti-T8-5CM	2.132	74.936	138.11	110.23
AHSS-T8-5CM	0.99708	74.844	135.94	187.77
Al-T12-5CM	2.1328	66.552	129.21	86.707
Mg-T12-CM	3.3683	66.821	127.92	49.145
CFRP-T12-5CM	0.7955	66.233	184.95	49.145
Ti-T12-5CM	1.5795	66.953	148.12	144.62
AHSS-T12-5CM	0.73279	66.233	130.08	245.72

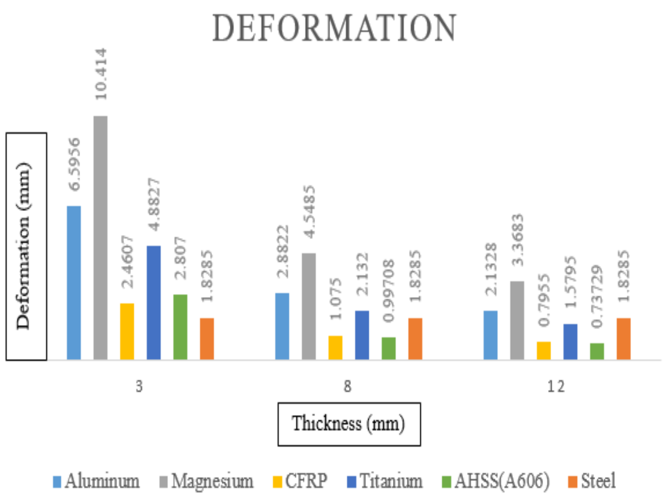

Figure 5: Comparison between original and modified chassis deflection.

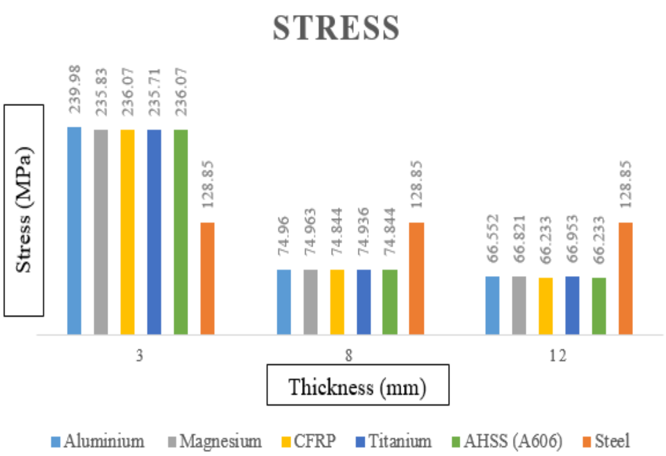

Figure 6: Comparison between original and modified chassis stress

The results of different materials are compared with original chassis

1. The deformation and stress values decrease with increase in material thickness.
2. Chassis of material AHSS, 8 mm & 12 mm thickness shows 45% & 59% decrease in deformation respectively.
3. Material CRPF & AHSS, 12mm thickness shows 48% decrease in stress.
4. Stress & deformation values of 3 mm thickness are greater than the original chassis.

Based on the above discussion for 8 mm thickness with Advanced High Strength Steel Figures 7-9 shows the deflection and stress contour results.

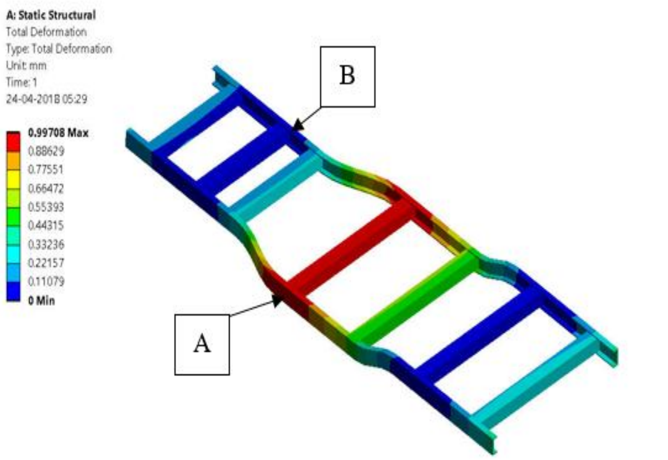

Figure 7: Total Deformation-Maximum deflection was observed at point A, that is region joining the cross members and side members and minimum was observed at point B.

International Journal of Applied Engineering Research ISSN 0973-4562 Volume 13, Number 11 (2018) pp. 9887-9892

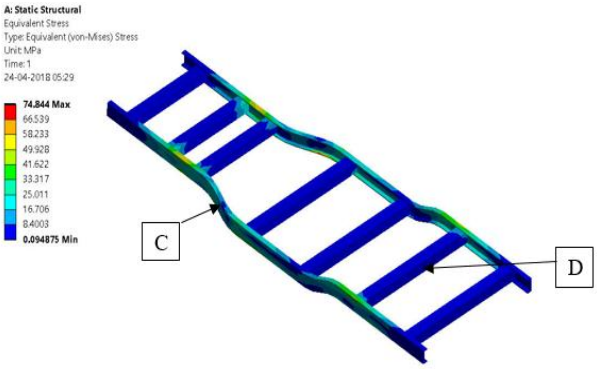

Figure 8: Equivalent Stress-Maximum stress was observed at point C, that is on lower side of side member and minimum was observed at point D that is on cross members.

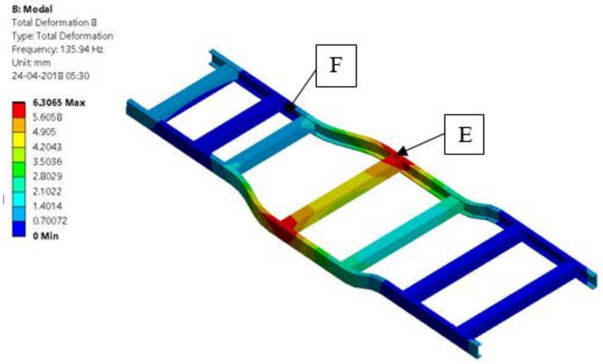

Figure 9: Modal Analysis-Maximum deformation was observed at point E, that is on upper side of side member and minimum was observed at point F.

Table 2. shows the deflection and stress values for 4 cross members. Figures 10-11 represents the graphical information mentioned in Table 2

Table 2. Deflection, stress and frequency values for 4 cross members

Designation	Deformation (mm)	Stress (MPa)	Frequency (Hz)	Weight (kg)
Al-T3-4CM	8.21	239.89	140.04	26.488
Mg-T3-4CM	10.49	211.24	129.43	17.186
CFRP-T3-4CM	2.4811	211.26	108.52	14.99
Ti-T3-4CM	4.9174	211.19	117.28	44.112
AHSS-T3-4CM	2.83	241.31	141.63	75.143
Al-T8-4CM	2.8374	88.985	141.29	61.076
Mg-T8-4CM	3.9253	63.242	117.87	39.689
CFRP-T8-4CM	0.92802	63.407	115.73	34.617
Ti-T8-4CM	1.8403	63.205	137.59	101.87
AHSS-T8-4CM	0.86012	63.407	120.033	173.53
Al-T12-4CM	2.5	68.114	113.89	75.545
Mg-T12-4CM	3.9472	68.138	112.53	49.091
CFRP-T12-4CM	0.93291	68.055	105.65	42.818
Ti-T12-4CM	1.85	68.145	149.76	126
AHSS-T12-4CM	0.8646	68.055	147.27	214.09

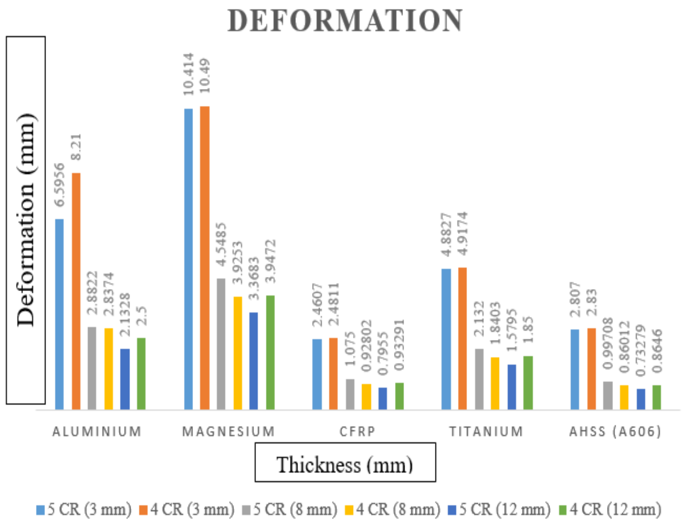

Figure 10: Comparison between original and modified chassis deflection.

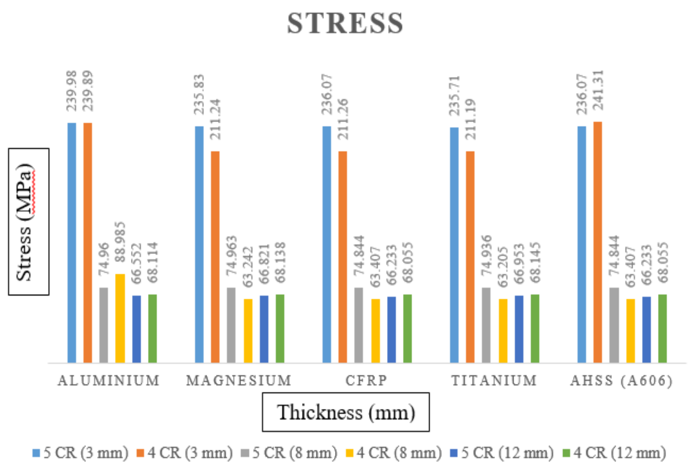

Figure 11: Comparison between original and modified chassis stress.

The deformation and stress results of 5 cross members are compared with 4 cross members.

1. The deformation values decreases with increase in material thickness for 4 cross members.
2. AHSS material Chassis with 8 mm & 12 mm thickness shows 52.7% decrease in deformation.
3. Deformation values of 3 mm thickness for 4 cross members are greater than the 5 cross members.
4. The Stress values decreases with increase in material thickness for 4 cross members.
5. Stress values of 8 mm thickness are less than 12 mm thickness.
6. Stress values for 8, 12mm thickness are less than 3mm thickness.

Figures 12 - 14 shows the stress and deformation contours for Advanced High Strength Steel material with 8 mm thickness and 4 cross members.

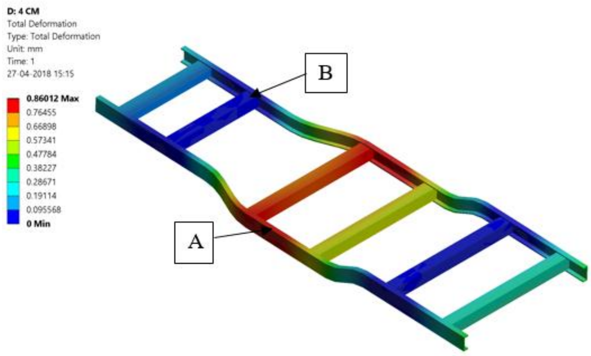

Figure 12: Total Deformation-Maximum deflection was observed at point A, which is region at the top ofthe cross members and minimum was observed at point B.

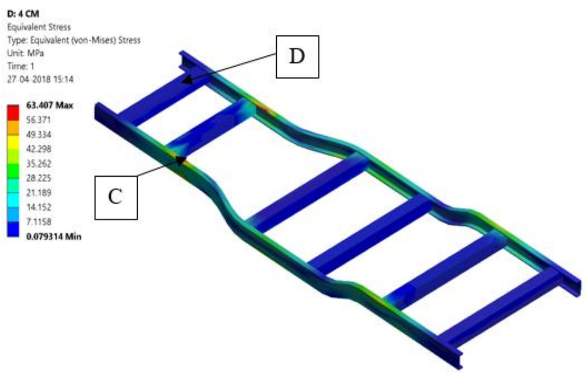

Figure 13: Equivalent Stress-Maximum stress was observed at point C, which is on top side of side member and minimum was observed at point D which is on cross members.

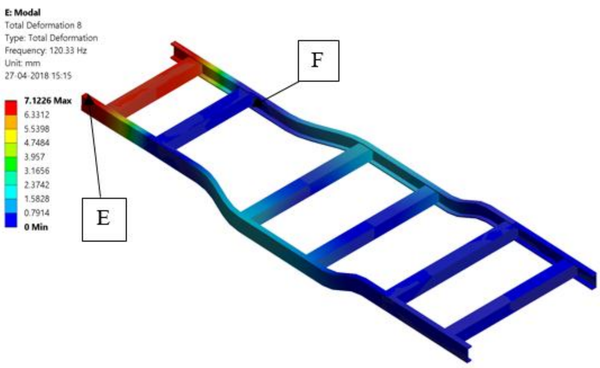

Figure 14: Modal Analysis-Maximum deformation was observed at point E, that is on upper side of side member and minimum was observed at point F.

RESULTS AND DISCUSSION

Based on the above result obtained, following summary is drawn.

1. CFRP and AHSS were showing better results as compared to the original chassis.
2. The deformation and stress values decrease with increase in material thickness, so 8 and 12mm thick chassis showed better results.
3. With decreasing number of cross members & increasing thickness, values of stress and deformation decreased.
4.

Above results were evaluated on four parameters – deflection, stress, modal frequency and weight. In the best case of AHSS the weight obtained was 173.53 Kg. Though CFRP is giving highest performance values for above parameters its manufacturing technique is not fully established. Because of this AHSS was chosen. Still the weight is at higher side. From weight reduction point of view, change in geometry topology was being carried out by changing the C section to I section for side members whereas box section of cross members was replaced with T and I section. With this change the weight obtained was 162.5 kg. Following are the contour results of the best chassis from weight reduction point of view. The combination adopted was C section for side member and T section for cross member. Figures 15-16 shows the deformation and stress result for the T section.

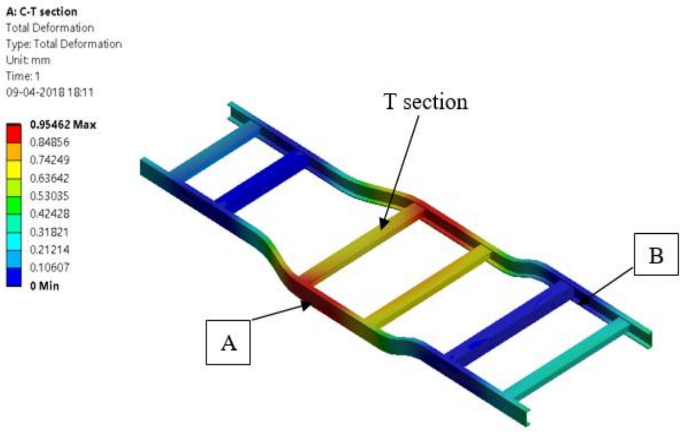

Figure 15: Total Deformation-Maximum deflection was observed at point A, which is region at the side of the cross members and minimum was observed at point B below the side members.

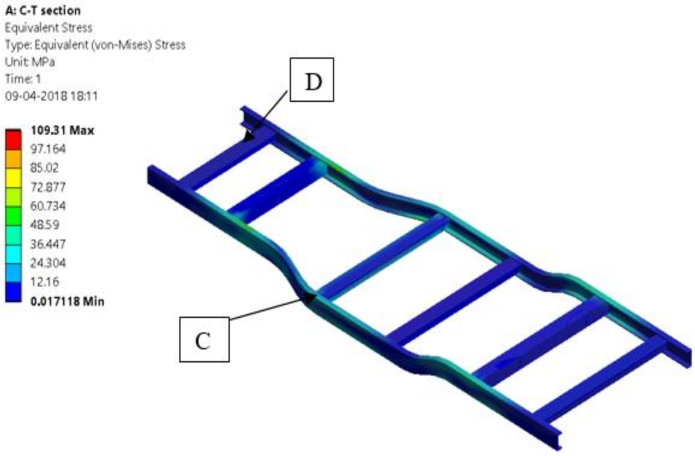

Figure 16: Equivalent Stress-Maximum stress was observed at point C, which is on top side of cross members and minimum was observed at point D which is on cross members.

CONCLUSION

From the optimization carried out on the chassis, it can be concluded that 8 mm thick Advanced High Strength Steel chassis shows better result as compared to original 5 mm thick steel chassis. Also from weight reduction point of view, change in geometry topology was carried by changing Box section to T section for cross members.

FUTURE SCOPE

As a future scope of study, manufacturing techniques for CFRP chassis could be explored for better quality with less cost.

REFERENCE

[1] M. Zehsaz, F. VakiliTahami and F. Esmaeili, “THE EFFECT OF CONNECTION PLATE THICKNESS ON STRESS OF TRUCK CHASSIS EITH RIVETED AND WELDED JOINTS UNDER DYNAMIC LOADS”, Asian Journal of Applied Sciences, ISSN 1996-3343, Volume No. 2, Issue No. 1, 22-35,2009.

[2] SenZheng , Kai Cheng , Jixin Wang, Qingde Liao, Xiaoguang Liu, Weiwei Liu, “FAILURE ANALYSIS OF FRAME CRACK ON A WIDE-BODY MINING DUMP TRUCK”, Engineering Failure Analysis, Science Direct, Elsevier, Engineering Failure Analysis 48 (2015) 153–165.

[3] Chen Yanhong, Zhu Feng, “THE FINITE ELEMENT ANALYSIS AND THE OPTIMIZATION DESIGN OF THEYJ3128-TYPE DUMP TRUCK’S SUB-FRAMES BASED ON ANSYS”, SciVerse Science Direct, Elsevier, Procedia Earth and Planetary Science 2 (2011) 133 – 138.

[4] A.N. Savkin, A.S. Gorobtsov, K.A. Badikov, “ESTIMATION OF TRUCK FRAME FATIGUE LIFE UNDER SERVICE LOADING”, ScienceDirect, Elsevier, Procedia Engineering 150 (2016) 318 – 323.

[5] MohdAzizi Muhammad Nor, Helmi Rashid, Wan MohdFaizul Wan Mahyuddin,MohdAzuanMohdAzlan, Jamaluddin Mahmud, “STRESS ANALYSIS OF A LOW LOADER CHASSIS”, Elsevier Ltd. Sci Verse Science Direct Procedia Engineering 41 (2012) 995 – 1001.

[6] CicekKaraoglu, N. SefaKuralay, “STRESS ANALYSIS OF A TRUCK CHASSIS WITH RIVETED JOINTS”, Finite Elements in Analysis and Design 38 (2002) 1115–1130, 2002 Elsevier Science B.V.

[7] Yuan Ren, Yongchang Yu, Binbin Zhao, Chuanhui Fan, He Li, “FINITE ELEMENT ANALYSIS AND OPTIMAL DESIGN FOR THE FRAME OF SX360 DUMP TRUCKS”, ScienceDirect, Elsevier, Procedia Engineering 174 (2017) 638 – 647.

[8] RoslanAbd 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

[9] S.S.Sane, GhanashyamJadhav, Anandraj .H, “STRESS ANALYSISOF A LIGHT COMMERCIAL VEHICLE CHASSIS BY FEM”.

[10] D.V. Bhope, N.K.Ingole, “STRESS ANALYSIS OF TRACTOR TRAILER CHASSIS FOR SELF WEIGHT REDUCTION,” International Journal of Engineering Science and Technology (IJEST), Vol.3 No. 9, 2011.

[11] Akash Singh Patel, JaideepChitransh, “DESIGN AND ANALYSIS OF TATA 2518TC TRUCK CHASSIS FRAME WITH VARIOUS CROSS SECTIONS USING CAE TOOLS”, International Journal Of Engineering Sciences & Research Technology, ISSN: 2277-9655, Patel et al., 5(9): September, 2016.