ARPN Journal of Engineering and Applied Sciences

www.arpnjournals.com

DESIGN AND THEORETICAL ANALYSIS OF SUB-FRAME STRUCTURE ON DUMP TRUCK AND VALIDATION THROUGH FEM APPROACH

Rajesh M.[1], Suresh P.[2], Prithuvi Rajan S.[1], Rahulson M.[1], Sathish T.[1] and Srireguram[1]
[1]Department of Mechanical Engineering, Karpagam College of Engineering, Coimbatore, India
[2]Department of Mechanical Engineering, Muthayammal Engineering College, Rasipuram, Namakkal District, India
E-Mail: suresh.me2004@gmail.com

ABSTRACT

This study is made to enhance the performance of the sub frame in YJ3128 type Dump truck by reducing the vibration, weight and provide uniform load in static conditions. Methods/statistical Analysis: A three different case studies has been made by modeling the sub frame with three different cross section namely ‘C’, ‘I’ and Rectangular sections. The work is performed by varying the position of the cross members in the sub frame and validated through FEM approach. The dimension and material for this study is taken from the existing model of YJ3128 type dump truck. Findings: Based on the comparison of theoretical and analytical results of max principal stress and deflection, the optimized sub frame has been identified. Application: It is mainly utilized in carrying the power train unit of the heavy duty vehicle that is used in minings, construction sites and highways for transporting sands and gravels.

Keywords: power train, sub frame, static analysis, finite element method, ansys.

INTRODUCTION

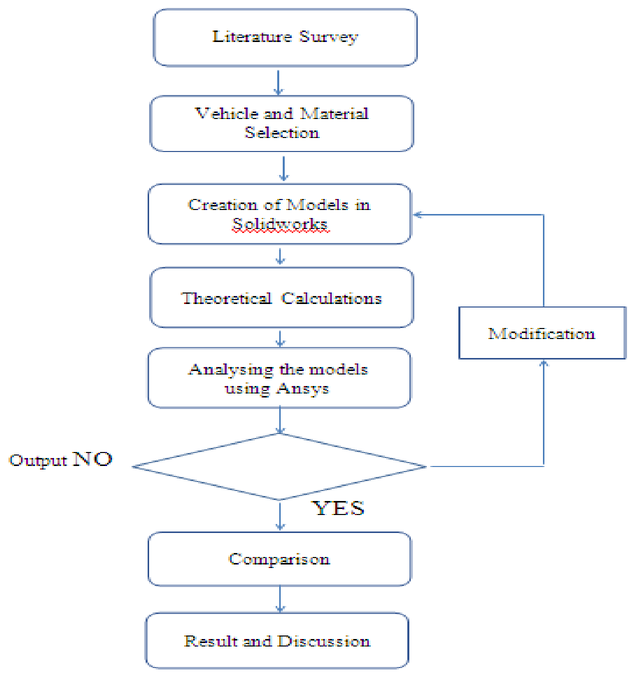
The Dump truck is a truck with an open container at the back that can be raised at an angle, so that its load can fall out. It has the major application in construction of large industrial structures, construction of highways and as well as in mining constructions. A dump truck is mounted with hydraulic system which can lift its back container. The chassis is considered to be one of the significant structures of an automobile. The chassis provides the strength needed for supporting the different components present in the vehicle. Another supporting member of the chassis is sub frame. The entire power train of the vehicle is hold by sub frame mounted with the chassis by bolts, clamps and welds. One of the major problem faced by the power train (Engine, Gear box, Propellar shaft, Differential gear) is higher vibration. Due to the continuous act of vibration, the resonance will occur and tends to induce noise in the vehicle. So the main purpose of installing sub frame is to isolate and decay the oscillation of vibration. Hence if the design of the sub frame is invalid, then there may be a chance of inducing secondary vibration and create new problem in the sub frame. So our objective is to change the existing model sub frame into a new model sub frame. Therefore, by changing the design of the sub frame, life of the hydraulic system can be increased. Furthermost, it also ensures low noise level and less vibrations throughout the vehicle. The researchers worked in this field provided the valuable resource of the information which make interests us to work in this paper. [1] Chen Yanhong *et al*., explained about the performance and working condition of the dump truck and analyzed its stress concentration by using ANSYS. By providing an improvement and optimization, there is no excessive stress concentration found, the loads are equally distributed and the torsional stiffness of the chassis is enhanced. [2] Dharmendrasinh Parmar *et al*., analyzed with the weight optimization and the sub frame design for the dump truck with different materials like 16Mnl and SAE-AISI-1080. Then they finally concluded that the 16Mnl is found to be having more strength and stiffer with the result of ‘C’ cross section with less deflection and increased in fatigue life period. [3] Katamaraju Ediga Madhu Latha *et al.*, worked instructural and dynamic analysis of a car chassis using FEA and tried to compare two materials i.e. steel st37 and Carbon fiber epoxy through ANSYS and MATLAB. They found that Carbon Fiber have less shear stress, strain and deformation when compare to steel. On Comparing density, then steel is denser than carbon fiber. [4] Sairam Kotari *et al*., researched with the concept of finite element analysis for his TATRA chassis. They concentrated in installing the Antenna and Electronic components in their chassis. They found that stress levels attain maximum value. After certain modifications, the TATRA Chassis with suitable reinforcement, increase in thickness and addition of stiffeners the levels of yield stress are found to be greatly reduced. From the obtained values, it is concluded that the modified TATRA chassis is capable to withstand the previous load up to 15 tones. [5] Feng Guo-yu *et al*., their analysis is based on the vibration control. Before optimization, the vibration on the floor near driver’s seat is so strong, but the phenomenon disappears through the optimization of sub frame. Hence the frequency of vibration is significantly decreased. So the optimization of the sub frame is very beneficial to vibration control. [6] Hirak Patel *et al*., discussed about the weight optimization of the truck chassis by sensitivity analysis. In this analysis, they used different cross section for stress analysis and they bring upto 20% weight optimization in the truck chassis. They also compared the values of Maximum Principal Stress and deflection for the further cross section. [7] Sandip Godse *et al*., examined the static load analysis of the TATA ace chassis by using ANSYS workbench and stress optimization by using different reinforcement technique. They did their project by making necessary design changes in the existing design of TATA chassis and also adding stiffeners. This supports the increased load carrying capacity of the vehicle and thus

VOL. 16, NO. 24, DECEMBER 2021 ISSN 1819-6608
ARPN Journal of Engineering and Applied Sciences

www.arpnjournals.com

provides equal load distribution. [8] Mohd Azizi Muhammad Nora *et al.*, prepared a model of an actual low loader structure and simulated its stress analysis consisting of I-beam design by using FEA. They tried with the material of Low Alloy Steel with yield strength of 552 MPa and tensile strength with 620 MPa. They finally notified that the location of the maximum deflection and stress concentration coincide with theoretical location under uniformly distributed loading condition. [9] Dhandapani N.V *et al.*, employed a technique of an analytic hierarchy process for analyzing stress concentration in automotive chassis. They found that Max principal stress is acting along the horse collar part and further concluded that the stress can be eliminated by increasing the cross section of the horse collar area. [10] Jadav Chetan *et al.*, has reviewed the investigations that made on the different fatigue analysis techniques of automobiles frames. [11] Ahmad *et al.*, has reviewed the investigations on different fatigue analysis techniques of heavy duty truck frames. He stated lot of analytical and numerical techniques are available for the fatigue analysis of heavy truck. [12] Rajappan has analyzed the static and dynamic load characteristics by FE models. He has identified the location of high stress area; analyzing vibration, natural frequency and mode shape using FEM. [13] Purendu has optimized the sub frame for commercial vehicle application. It has carried out in material selection, no. of parts selection, manufacturing process of individual parts and assembly to reduce the overall weight. [14] Bhaskar *et al* has examined the static analysis of chassis model using ANSYS 7.1 on plain road and bump conditions. [15] Shubham *et al.*, has studied the characteristics of mode responses of heavy vehicle chassis under vibrations. The natural frequencies and mode shapes of a structure are determined by modal analysis.

METHODOLOGY



MODELING OF SUB FRAME

Material Properties:

The dump truck's frame is equipped by two components, the main-frame and the sub one and they are all provided with various cross sectional beams. The Horizontal frame, vertical frame and cross members are mounted with the main frame. These frames are connected together by clamps and bolts.

The Dimensions of our sub-frame consists,

Total length of the frame	4700 mm
Forth width of the frame	901 mm
Back width of the frame	761 mm
The thickness of the frame	20 mm
The Cross member of the frame	180 x 180 (l*b)

The geometry of sub frame is modeled by selecting the availability of different material. The Existing materials of the sub frame are high carbon steel, composite materials and different types of alloys. In the present work, 16Mnl is chosen and its properties are shown in the Table-1,

Table-1. Properties of material.

Material	16Mnl
Poisson ratio (μ)	0.3
Young's Modulus (E)	2.e+005 MPa
Ultimate Strength	630 MPa
Yield Strength	530 MPa
Density	7.85e-006 kg mm^-3

We have made case studies by changing the cross section and changing the position of the cross member of the sub frame approximately based on the distribution of load. The Case studies are following as,

Table-2. Properties of sub frame.

	CROSS SECTIONS	CROSS MEMBERS
Case Study I	Rectangular	7
	'C' Section	
	'I' Section	
Case Study II	Rectangular	4
	'C' Section	
	'I' Section	
Case Study III	Rectangular	5
	'C' Section	
	'I' Section	

Table-2 shows the properties of the sub frame modeled with three types of cross sections namely

VOL. 16, NO. 24, DECEMBER 2021 ISSN 1819-6608
ARPN Journal of Engineering and Applied Sciences

www.arpnjournals.com

Rectangle, C and I sections with various number of cross members.

Case Study I:

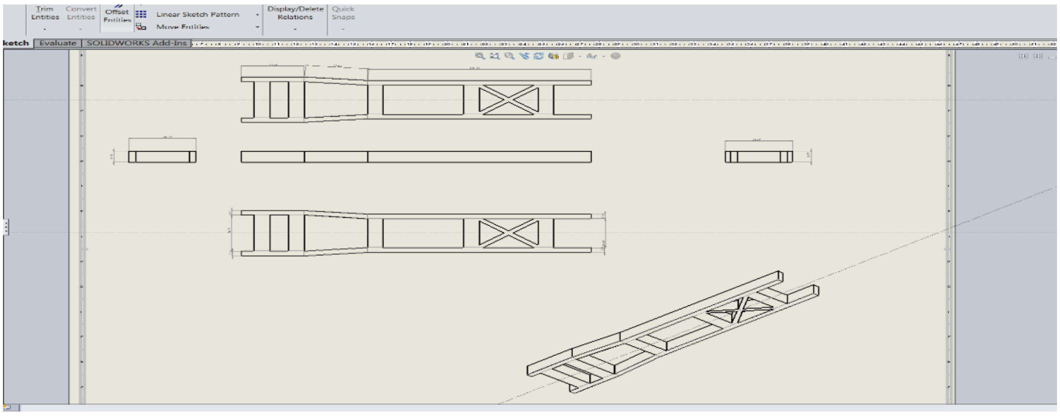


Figure 3. Solid Rectangular section model.

Figure 3 is a 2D diagram modeled in the form of solid rectangular cross section with the 'X' type cross member provided with seven cross members.

Case Study II:

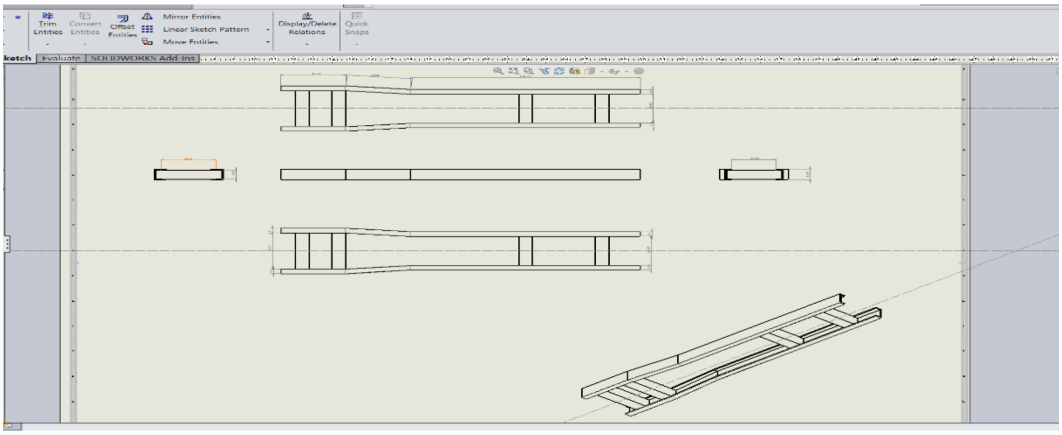


Figure 4. Solid 'C' section Model.

Figure-4.4 states is a 2D diagram modeled in the form of 'C' cross section with four cross members by eliminating 'X' type cross member.

Case Study III:

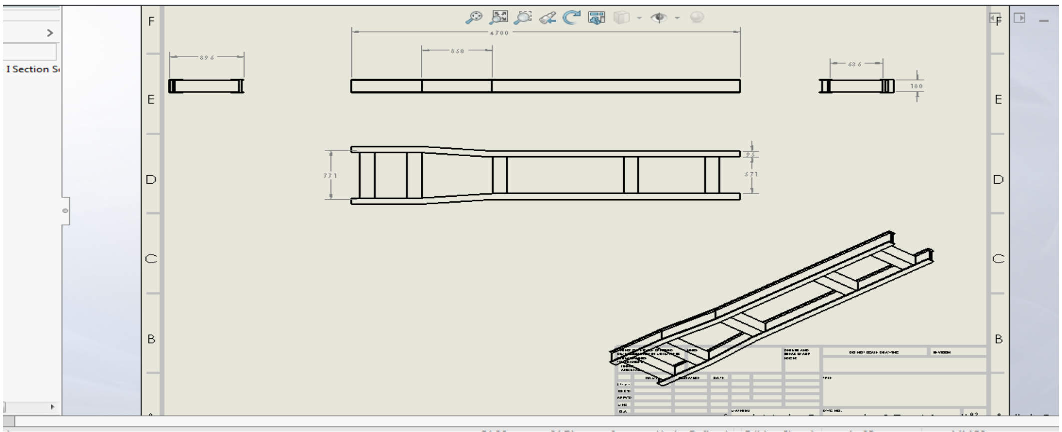


Figure 5. Solid 'I' section model.

VOL. 16, NO. 24, DECEMBER 2021 ISSN 1819-6608
ARPN Journal of Engineering and Applied Sciences

www.arpnjournals.com

Figure 5 is a 2D diagram modeled in the form of 'I' cross section with five cross members by eliminating 'X' type cross member.

THEORETICAL CALCULATIONS

To find Maximum principle stress:-

We know that,

$$\sigma_{max} = \frac{\sigma x - \sigma y}{2} + \sqrt{\left(\frac{\sigma x - \sigma y}{2}\right)^2 + \tau xy^2}$$

$$\sigma_{min} = \frac{\sigma x - \sigma y}{2} - \sqrt{\left(\frac{\sigma x - \sigma y}{2}\right)^2 + \tau xy^2}$$

$$\tan 2\theta = \left(\frac{2\tau xy}{\sigma x - \sigma y}\right)$$

Here,

$\sigma x = 121266.3 N/mm^2$

$\sigma y = 0$

For Rectangular, 'C', 'I' Section

$\sigma_{max} = 0.856 MPa$

$\sigma_{min} = 0.218 MPa$

To find Deflection:-

We Know that,

$$y_{max} = \frac{WL^3}{48EI}$$

Therefore,

Deflection for Rectangular Section Ymax = 0.017mm

Deflection for 'C' Section Ymax = 0.049mm

Deflection for 'I' Section Ymax = 0.061mm

STRUCTURAL ANALYSIS OF SUBFRAME

Loads Acting on the Sub Frame:

Table 3 shows that various loads act on the front and rear side of the dump truck sub frame.

Table 3 Loads acting on the sub frame.

NAME	WEIGHT (kg)	DISTANCE (mm)
Front shock absorber (left side)	250	280
Front brake System	750	850
Engine Assembly	1100	1700
Front shock absorber (Right side)	200	280
Differential Unit	1100	3200

Ansys Analysis:

Case Study I:

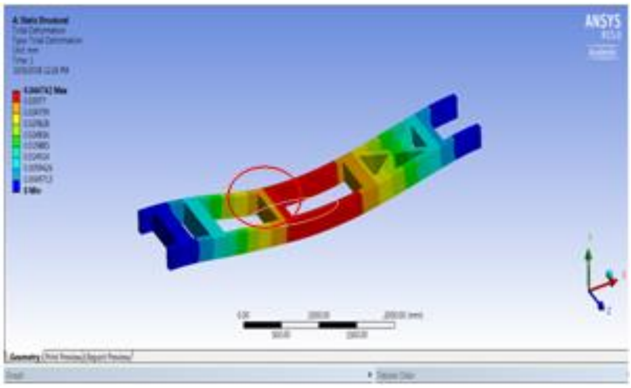


Figure 6 (a).

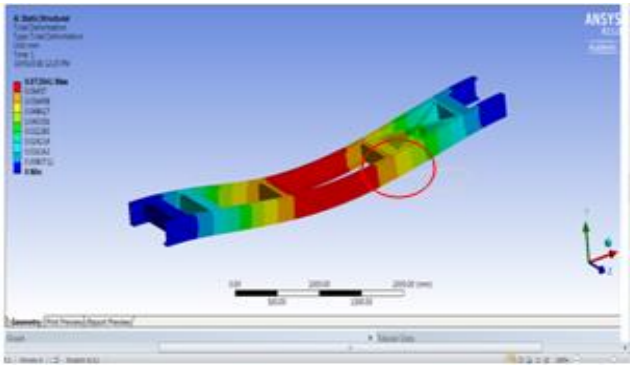


Figure 6 (b).

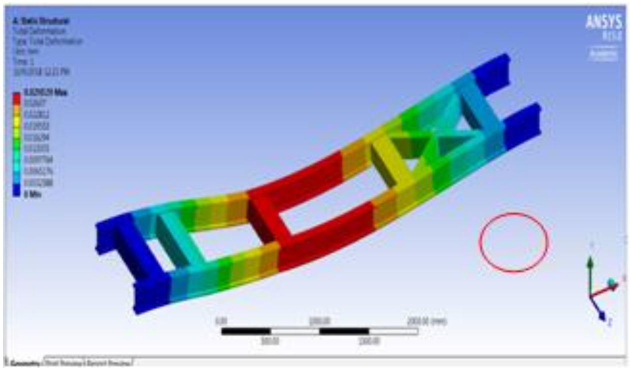


Figure 6 (c).

Figure 6 (a) shows solid rectangular section with seven cross member undergoes major deformation in the right side cross member joint. In Figures 6 (b, c) show the solid C & I sub frame with seven cross members gets deflected in the third and fourth cross member on its left joint.

VOL. 16, NO. 24, DECEMBER 2021 ISSN 1819-6608
ARPN Journal of Engineering and Applied Sciences

www.arpnjournals.com

Case Study II:

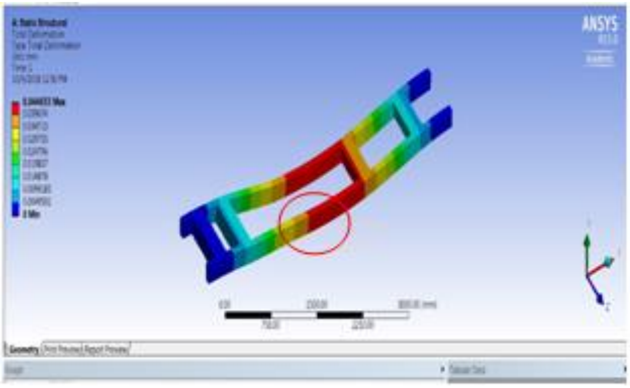


Figure 7 (a).

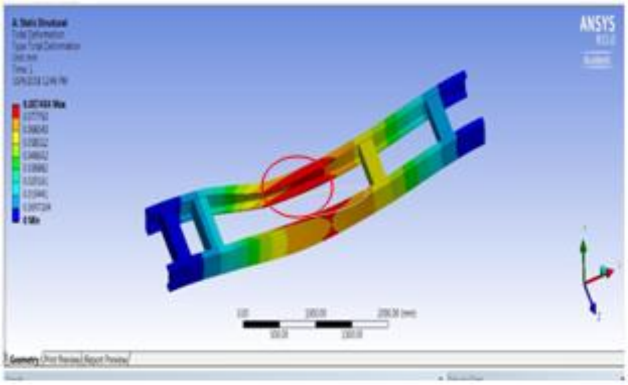


Figure 7 (b).

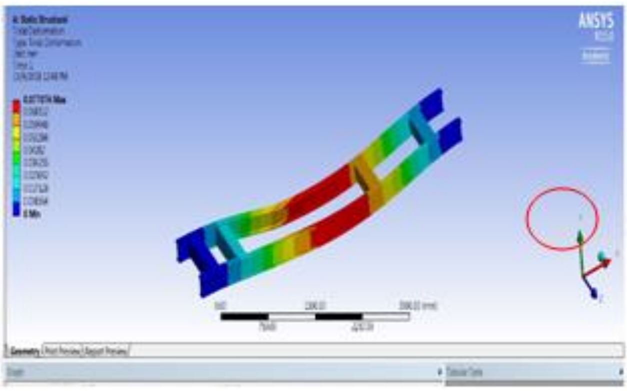


Figure 7 (c).

From the above Figures 7 (a, b, c) show the deformation occurred in solid rectangular, C & I sub frame with four cross members. In Figure 7 (a) reveals that deformation occurs in the center part of the sub frame where else in Figures 7 (b, c) show crack propagation between second and third cross member on both the joints.

Case Study III:

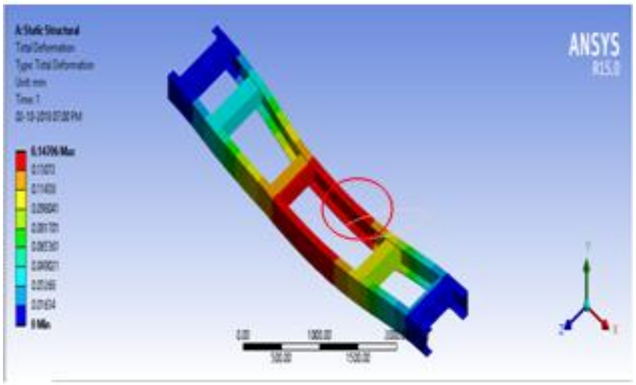


Figure 8 (a).

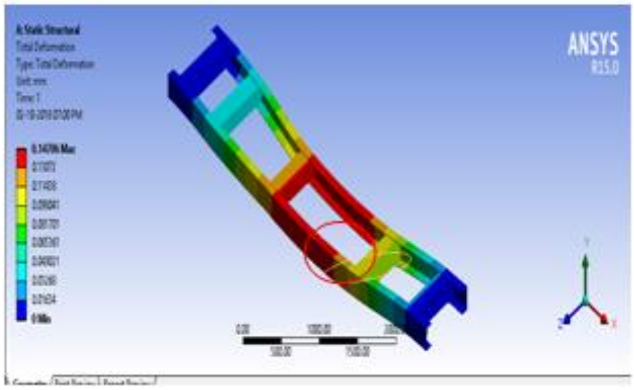


Figure 8 (b).

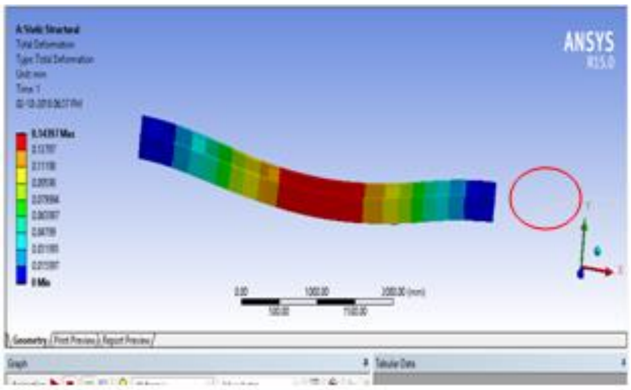


Figure 8 (c).

Figures 8 (a, b, c) shows the deformation of the Solid rectangular, C & I sub frame with five cross members. In Figure 8(a) reveals that deformation occurs in between third and fourth cross member of the sub frame where else in Figures 8 (b, c) show crack propagation from the third cross member till fourth cross member joints.

COMPARISON

The Table 4 represents the comparison between values of Max principal stress, Min principal stress and Von Misses stress for different case studies with different cross members. The maximum and minimum values are occurred in all cases studies.

Table 4. Comparison of principal stress.

Case Studies	Cross Sections	Maximum Principal Stress (MPa)	Minimum Principal Stress (MPa)	Von Misses Stress (MPa)
I	Rectangle	0.752	0.159	0.570
	C	1.285	0.342	1.062
	I	1.230	0.369	1.096
II	Rectangle	9.765	2.814	6.532
	C	9.945	2.866	6.652
	I	3.898	1.147	2.963
III	Rectangle	1.787	0.387	1.387
	C	3.196	0.934	2.452
	I	3.034	0.896	2.313

Graph for Principal stress:

Case study 1:

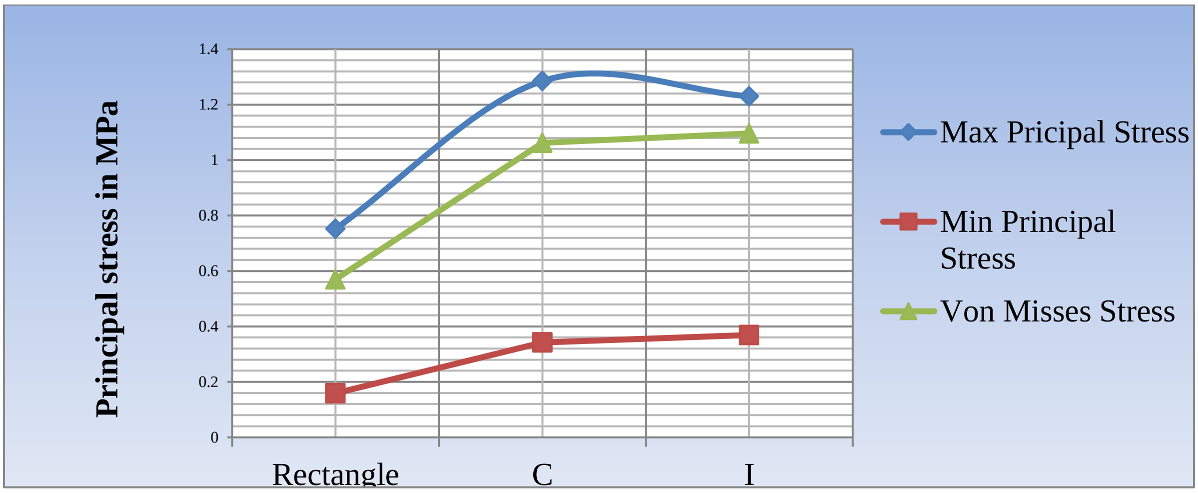


Figure 9 (a). Principal stress for sub frame in MPa.

Case study 2:

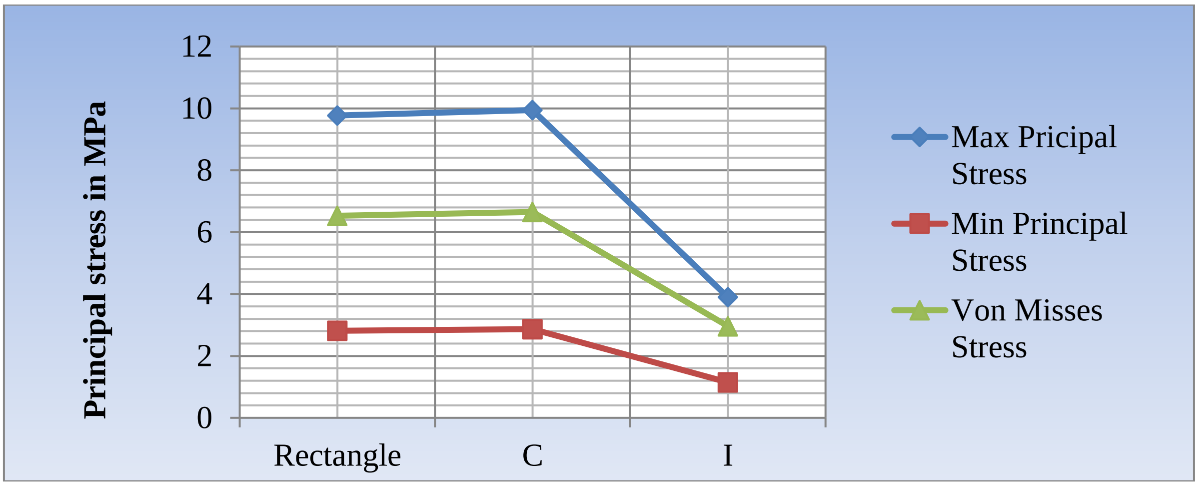


Figure 9 (b). Principal stress for sub frame in MPa.

Case study 3:

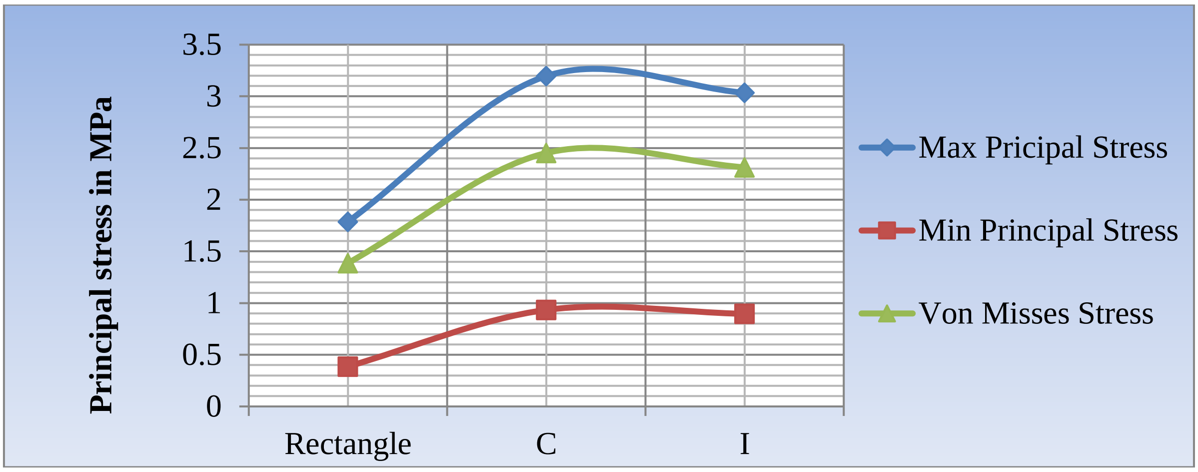


Figure 9 (c). Principal stress for sub frame in MPa.

From the above Figures 9 (a, b, c) reveal the graphical view for all the maximum principle stress, minimum principle stress and von misses stress values for all case studies.

Graph for Deflection of sub frame:

Case study 1:

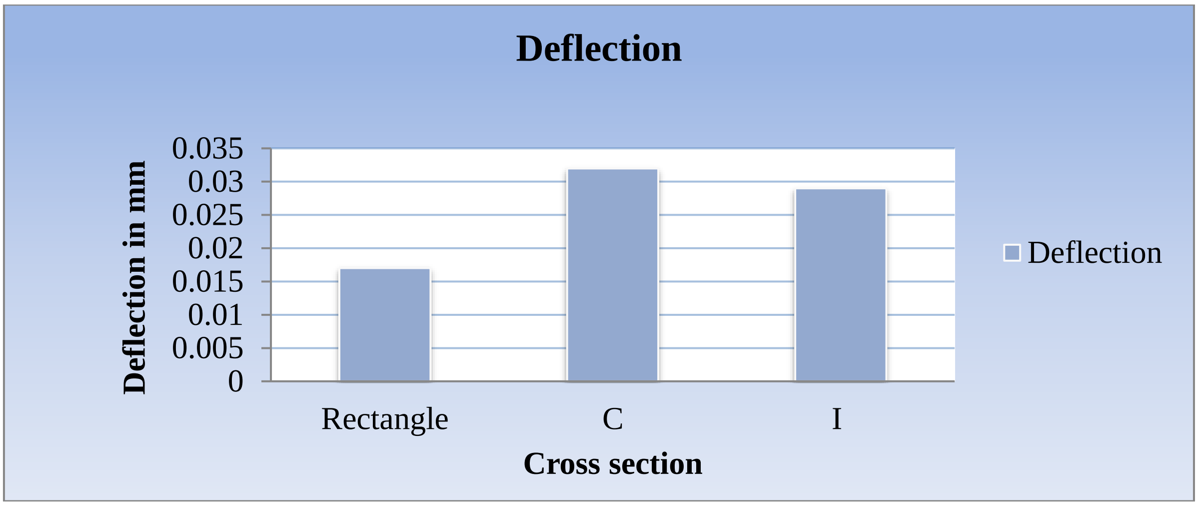


Figure 10 (a). Deflection of sub frame.

VOL. 16, NO. 24, DECEMBER 2021 ISSN 1819-6608
ARPN Journal of Engineering and Applied Sciences

www.arpnjournals.com

Case study 2:

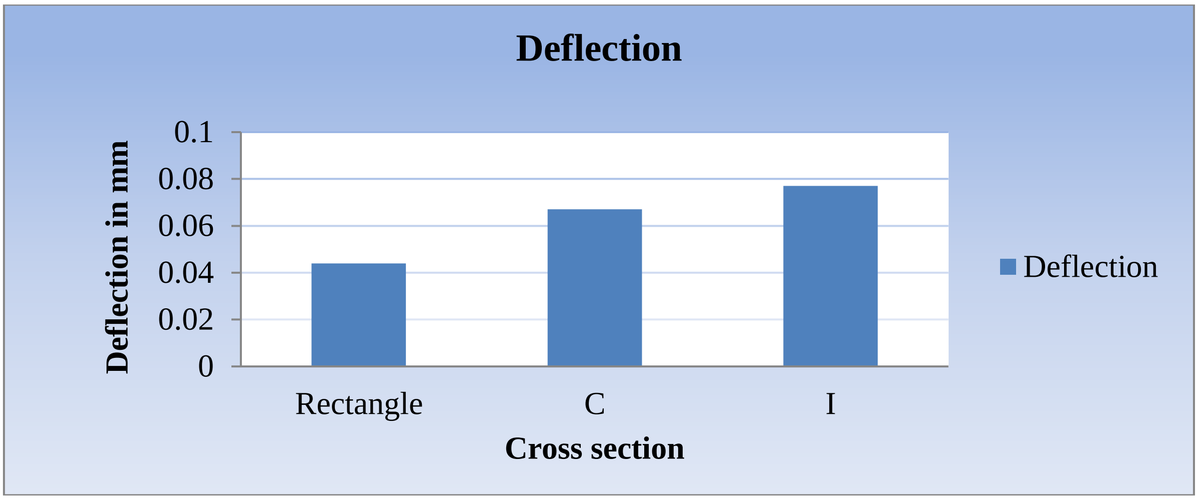


Figure 10 (b). Deflection of sub frame.

Case study 3:

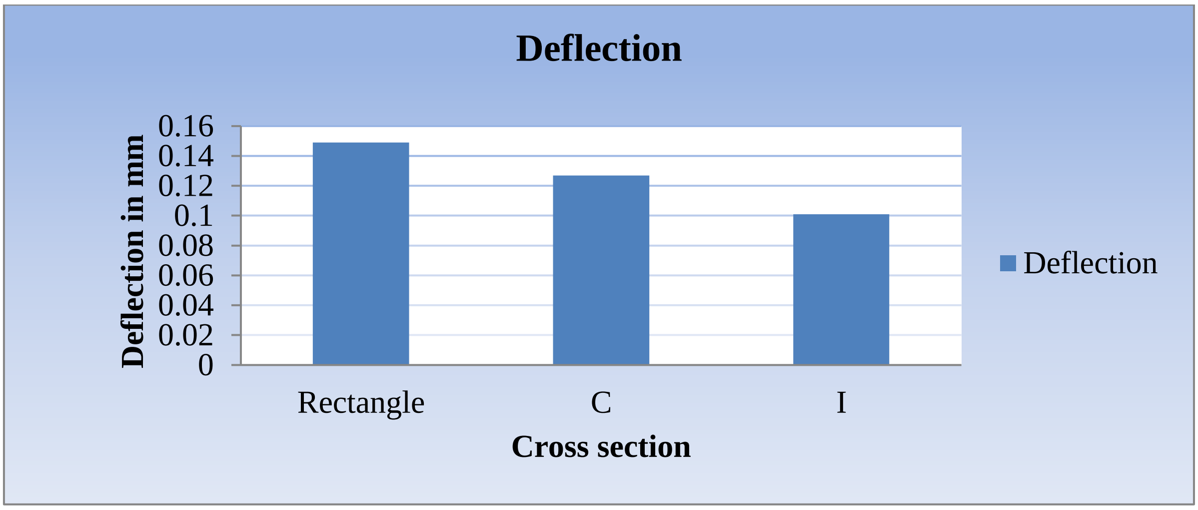


Figure 10 (c). Deflection of sub frame.

From the above Figures 10 (a, b, c) reveals the graphical view for maximum and minimum deflection values for all case studies.

RESULTS AND DISCUSSIONS

Case Study 1:

From the Table 4 and Figure 9 (a) and 10 (a), the results of the 'C' section provides the Max Principal stress wherease its deflection is high but in the case of solid rectangular section Max Principal stress is less and its deflection is less about 0.017mm. In the case of 'I' section both are greater. Therefore, solid rectangular section is better than 'C' section based on deflection.

Case Study 2:

By taking the values of Table 4 and considering the Figure 9 (b) and 10 (b), the result of 'C' section having the more Principal stress whereas its deflection is less but in the case of solid rectangular section Max Principal stress is about 9.765MPa and its deflection is less. Hence in the case of 'I' section the deflection is less when compared with the both sections. Therefore 'I' section is better than 'C' section.

Case Study 3:

On comparison with Table 4 and Figure 9 (c) and 10 (c), the results of the 'C' section provides the Max Principal stress with 3.196MPa and its deflection is less. In the case of 'I' section Max Principal stress is more rather than its deflection meanwhile in the case of solid rectangular section the Max Principal stress is minimum about 1.787MPa. Therefore 'C' section better than 'I' section on deflection.

CONCLUSIONS

Finite element analysis is effectively utilized for addressing the concepts and formulas for the various design stages. Thus the various cross sections of the sub frames for C, I and Rectangle section are analysed. Based on the analysis results of the present work, the following conclusions can be drawn.

The above three case study reveals that, the 'C' section in the case study III provided with five cross members having less deformation and providing uniform load distribution as expected. So this sub frame model is suitable for dump truck and further analysis.

Meanwhile in case study II, 'I' section has lesser deflection than both of the cross sections but the clamping work for this type of cross section is very much difficult. Hence this type is not recommended for the practical utilization.

In case study I, solid rectangular section is better than other sections but on considering weight it is having more cross member when compared to other models of sub frame. So it is neglected.

FUTURE WORK

In future work, the above selected sub frame model is dynamically analysed by using Ansys Workbench. Then it is fabricated and the results are compared through the static and dynamic load conditions. Further the loads acting externally such as resistance caused by air, tyre friction, Front and rear suspension control effect, braking effect and some other above loads are taken into account for analysis to meet actual life requirements.

REFERENCES

[1] Dharmendrasinh Parmar *et al*. 2015. Design And Weight Optimization of YJ3128 Type Dump Truck's Frame. IJAERD, 2(5), C.U. Shah University, Wadhwan 363030, India.

[2] Chen Yanhong *et al*. 2011. The Finite Element Analysis and the Optimization Design of the Yj3128-Type Dump Truck's Sub-Frames Based on ANSYS. Elsevier Institute of Kaifeng University, Kai Feng, China.

[3] Dhandapani N.V *et al*. 2017. Stress Analysis in Automotive Chassis Using Analytic Hierarchy Process. IJPAM ISSN: 1311-8080, 117(20): 569-578 Karpagam College of Engineering, Coimbatore, India.

[4] Katamaraju Ediga Madhu Latha *et al*. 2017. Static and Dynamic Analysis of a Car Chassis using FEA. IJIRSET, 6(8), G.P.R College of Engineering, Andhra Pradesh, India.

[5] Ganga Rao G *et al*. 2015. Design, static and dynamic analysis of automobile chassis. IJRAET, 1(3): 07-11, Chirala Engineering College, Chirala.

[6] Jadav Chetan S *et al*. 2012. A Review of the Fatigue Analysis of an Automobile Frames. IJACR, 2(4, 6), Vidya Mandal Institute of Technology Bharuch, Gujarat, India.

[7] Ahmad O. Moaaz *et al*. 2014. A Review of the Fatigue Analysis of Heavy Duty Truck Frames. AJER, 3(10), Beni Suef University, Beni Suef-62511, Egypt.

[8] Dr.R. Rajappan *et al*. 2013. Static and Modal Analysis of Chassis by Using Fea. IJES, 2(2), Mailam Engineering College, Mailam.

[9] Purnendu K Dash *et al*. 2015. Stress Level & Risk of Failure Improvement Study in Subframe for Commercial Vehicle. IJERGS ISSN 2091-2730, 3(6), Alard College of Engineering & Management, Pune India.

[10] E.Bhaskar *et al*. 2014. Static and Dynamic Analysis of Chassis. IJR ISSN 2348-6848, Priyadarshini College of Engineering & Technology, Kanuparthipadu, SPSR Nellore-4.

[11] Dhandapani N.V *et al*. 2016. Effective design and development of automotive interior door analysis. 2nd International Conference on Advanced Engineering and Technology for Sustainable Development (ICAETSD-2016 at Karpagam College of Engineering, Coimbatore, India.

[12] Shubham Bhise *et al*. 2017. Modal analysis of Truck Chassis Frame. IJSER, 8(3), ISSN 2229-5518, Saraswati College of Engineering, Navi Mumbai, India.

[13] Wang Hai-fei *et al*. 2014. Random vibration analysis for the chassis frame of hydraulic truck based on ANSYS. ICPR J. Chem. Pharm. Res., 6(3): 849-852, Chang'an University, Xi'an, Shaanxi, China.

[14] Roslan Abd Rahman *et al*. 2008. Stress analysis of the heavy duty truck chassis as a preliminary data for its fatigue life prediction using FEM. Jurnal Mekanikal.

[15] Vinay N.Rao *et al*. 2016. Failure analysis of mixed mode crack growth in heavy duty truck frame rail. North Carolina State University.

[16] Punarselvam E and Suresh P. 2019. Non-Linear Filtering Technique Used for Testing the Human Lumbar Spine FEA Model. Journal of Medical Systems, Springer.

[17] R. Meenakshi and P. Suresh. 2019. WEDM of Cu/WC/SiC composites: development and machining parameters using artificial immune system. Journal of Experimental Nanoscience, Taylor and Francis.

[18] Natarajan N and Suresh P. 2015. Experimental investigations on the microhole machining of 304 stainless steel by micro-EDM process using RC-type pulse generator. International Journal of Advanced Manufacturing Technology, Springer. 77: 1741-1750.

[19] Punarselvam E and Suresh P. 2019. Investigation on human lumbar spine MRI image using finite element method and soft computing techniques. Cluster Computing, Springer.

[20] Senthilkumar R and Suresh P. 2019. Experimental study on electrical discharge machining of Inconel using RSM and NSGA optimization technique. Journal of the Brazilian Society of Mechanical Sciences and Engineering, Springer. 41: 35.

[21] Gopinath P and Suresh P. 2018. Effect of Jute Reinforcement Parameters on Mechanical Properties of Composite Structures. ASTM International Journal of Testing and Evaluation.

[22] Suresh P. and Kesavan R. 2009. Ergonomic analysis of spine during lifting of loads by Finite Element Modeling - A Case Study. Industrial Engineering Journal. 1(7): 7-10.

[23] Suresh P., Kesavan R. and M. Madheswaran. 2008. Finite Element Modeling and Analysis of Magnetic Resonance Imaging of Lumbar Spine Using Pro/E Software. International Journal of computational Intelligence and health Care Informatics (IJCIHCI). I(1): 50-56.

[24] S. Balasubramani, M. Suganthi and P. Suresh. 2013. An Empirical Study on Consumer Preference towards Hyundai Cars in Salem City. Paripex - Indian Journal of Research. 2(9): 20-22.

[25] Suresh P, L. Francis Xavier, J. Saravanakumar, Yuvaraj Yeswanth. 2015. Drilling of Aluminium Metal Matrix Composite using Automatic Coolant System. International Journal of Applied Engineering Research, ISSN 0973-4562 10(85).

[26] Suresh P, L. Francis Xavier. 2016. Wear behavior of Aluminum Metal Matrix Composite prepared from Industrial waste. The Scientific World Journal. Vol. 2016, Article ID 6538345.

[27] Suresh P, L. Francis Xavier. Studies on dry sliding wear behavior of aluminium metal matrix composite prepared from discarded waste particles. International Journal of Advanced Engineering Technology, Article in press.

[28] Suresh P, L. Francis Xavier, R. Balaragavendheran, P. Yeshwanth Kumar, S. Deepak. 2016. Studies on the influence ofdrilling cycle on the surface roughness of the drilled holes. ARPN Journal of Engineering and Applied Sciences. 11(2).

[29] Suresh P, N. Subramani Dr. J. Ganeshmurali, Dr. R. Kesavan. 2015. Enhancement of Aluminium 2024 Properties by Reinforcing With Alumina and Graphite. International Journal of Applied Engineering Research. 10(85).