

Design and Simulation on Chassis of ATV

S. Simhachalam¹ P. Saiteja² G. Rajesh³ G. Manikanta⁴ K. N. Muzamil⁵

^{1,2,3,4,5}Student

^{1,2,3,4,5}Department of Mechanical Engineering

^{1,2,3,4,5}KG Reddy College of Engineering & Technology, Moinabad, India

Abstract— ATV stands for All Terrain Vehicle which is specially designed for off-road driving. As these are used for very rough terrain, jumps, small scale forestry, border patrol, deserts and these are fun to drive so chassis has to be designed to withstand static and dynamic loads. Earlier ATV's are built with ladder frame and back bone chassis which were heavier and lack protection. To overcome from this problem we are redesigning the chassis of the ATV in solid works, meshing and simulation of the chassis is done in ANSYS software. Finite Element Analysis (FEA) was utilized to determine the maximum stress and displacement of the frame when a particular load is applied onto it. Structure modifications need to be done if the chassis frame could not sustain the applied load. We began the task by conducting extensive research and hence the type of chassis we used is "TUBULAR SPACE FRAME" due to its advantages and withstanding capabilities. The results are provided in form of graphs by analyzing in ANSYS software. After the design process is completed, the fabrication of the frame is conducted by students of the engineering faculty. The fabricated frame will be used as the main part for a project of which a complete ATV will be developed.

Key words: ATV, Finite Element Analysis (FEA)

I. INTRODUCTION

The vehicle which includes the features like all-terrain moving, sporting and fun driving is known as ATV. The vehicle should contain all the parameters like top-speed, handling, ride, and ruggedness.

As the vehicle has to travel in all road conditions it should have the high safety and smaller in size. Hence the vehicle is designed in such a way that it has high strength and performance. Hence the vehicle is designed in SOLIDWORKS software.

Later on the analysis of the chassis is done by ANSYS software and results are compared with other exiting model. The simulated results and our updated model is shown in the below papers.

II. LITERATURE REVIEW

– Kenji KARITA, Yoichiro KOHIYAMA, Toshihiko KOBIKI, Kiyoshi OOSHIMA, Mamoru HASHIMOTO (2003) had developed a chassis made by Aluminium. The material selected for the frame is 6061-T6. They used the Variable section extrusion method for making the chassis. It's developed with the help of computer Aided Engineering. Aluminium material gives an advantage of weight reduction. From this study authors found that the Aluminium chassis meets the target of weight reduction, strength and rigidity. Also they concluded that the remaining technical issues will be addressed to enable commercial adoption of the aluminum frame.

- M. Ravichandra, S. Srinivasalu, Syed Altaf Hussain (2012) studied the alternate material for chassis. They studied and analyzed Carbon/Epoxy, Eglass/Epoxy and S-glass/Epoxy as chassis material in various cross sections like C, I and Box Section. TATA 2515 EX chassis was taken for study. Pro-E and Ansys software were used for this work. Study reveals that the Carbon/Epoxy I section chassis has superior strength, stiffness and lesser weight compared to other materials and cross section.
- N.V.Dhandapani, G Mohan kumar, K.K.Debnath (2012) have used Finite element methods to study the effect of various stress distribution using Ansys software. To investigate the field failure of 100Ton dumper they introduced gussets in failure area. After modification the chassis structure was validated by linear static analysis and found that the modified chassis was safe.
- Sanjay Sharma stated that the main objective of their research work was to find the mode, shape, and corresponding natural frequency of a roll cage for an ATV. Finite element analysis is used to determine the mode shapes and frequencies. They have designed a roll cage which consists of both structural base and 3-D shell which protects the user in case of impact and roll over incidents. CATIA V5 was been used for modelling the roll cage and ANSYS 14.5 for FEA analysis. The body shape of the roll cage was fixed from the front end. The frame of the roll cage consists of three main parts known as boot space, driver cabin and engine chamber.

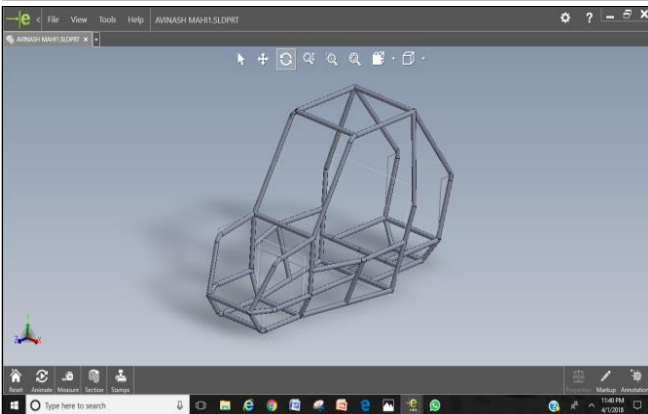
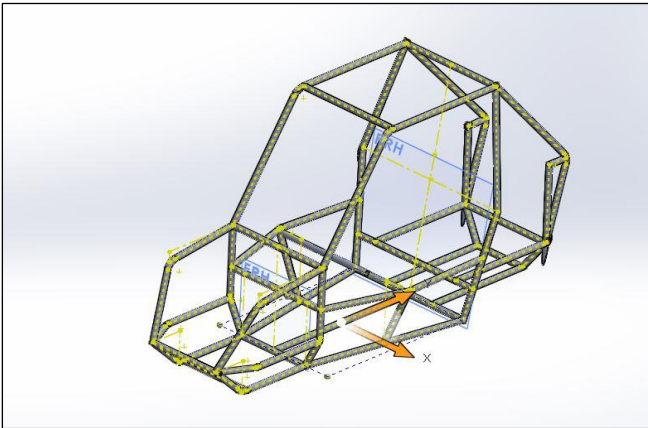
III. SCOPES AND OBJECTIVES

- 1) Provide full safety to the driver, by obtaining required strength and torsional rigidity, while reducing weight through diligent tubing selection.
- 2) Design for manufacturability, as well as cost reduction, to ensure both material and manufacturing costs are competitive with other SAE vehicles.
- 3) Improve driver comfort by providing more lateral space and leg room in the driver compartment.
- 4) Maintain ease of serviceability by ensuring that roll cage members do not interfere with other subsystems.
- 5) Safety and Ergonomics.
- 6) Standardization and Serviceability.
- 7) Maneuverability.
- 8) Maximum control with suspension design.
- 9) Optimize power efficiency.
- 10) Cost of the components.
- 11) Safe engineering practices.

IV. METHODOLOGY

The design of the chassis is done by the SOLIDWORKS software in which we have used different commands such as

BOSS EXTRUDE, BOSS CUT, PIPE STRUCTURE, DIMENSTIONS, ETC.



The above fig shows the design of the chassis which we have updated and designed in the SOLIDWORKS software.

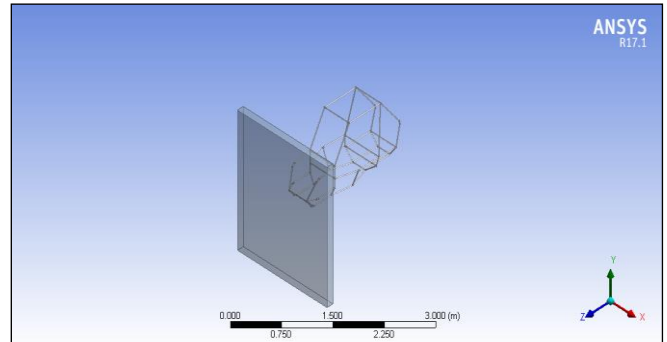
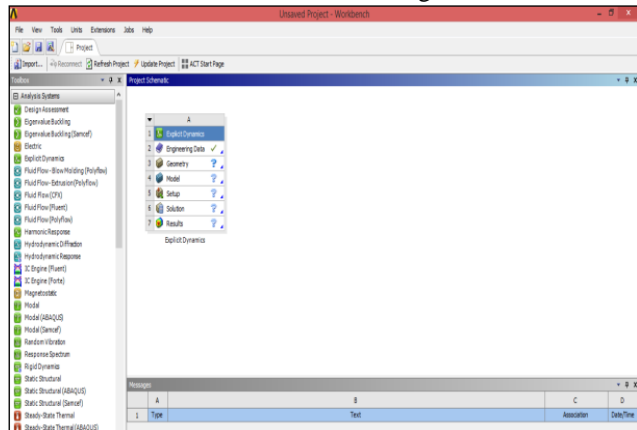
We have also done the report on analysis report on the ANSYS software and redesigned the chassis by resolving the defects found in the analysis. The design development has been increased by 10% to the original exiting design.

We have done the analysis on the chassis under static and dynamic conditions. The result for the chassis has been also obtained by changing the material from exiting material to AISI 1018 monochrome steel.

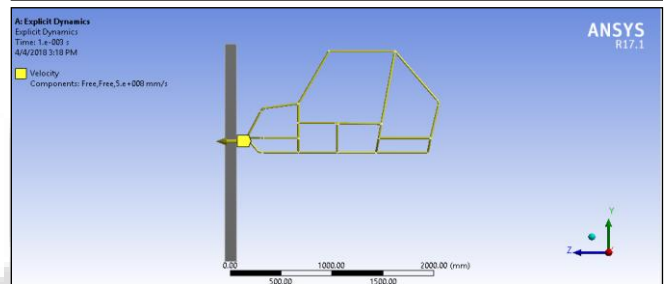
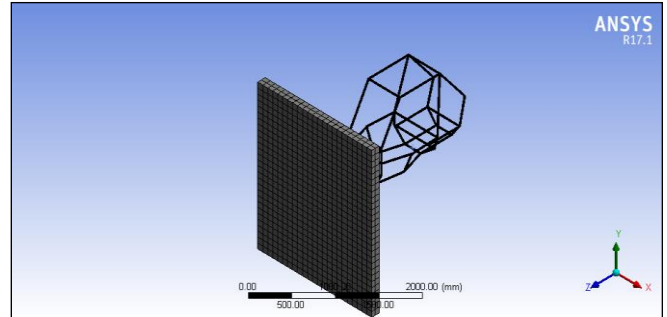
The simulation results of the chassis by step by step procedure are shown below and the inputs which we consider are:-

- Load applied :- 1 ton
- Mesh: fixed size function, medium relevance center
- Boundary conditions:

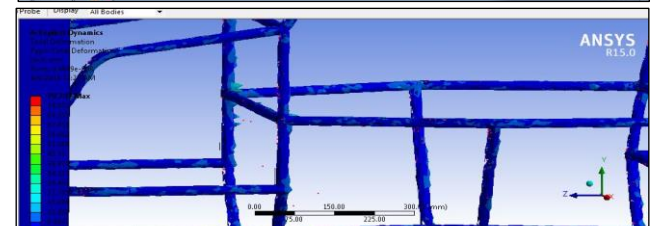
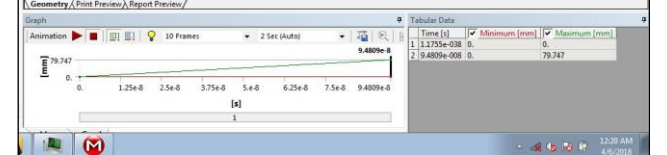
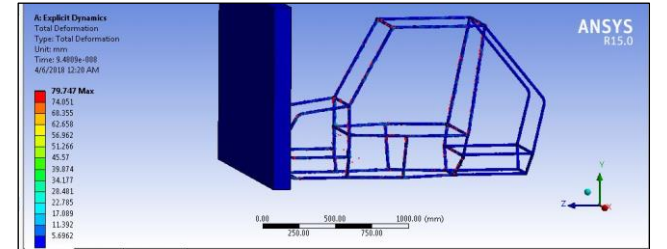
Wall fixid condition and vehicle moving with 50m/sec



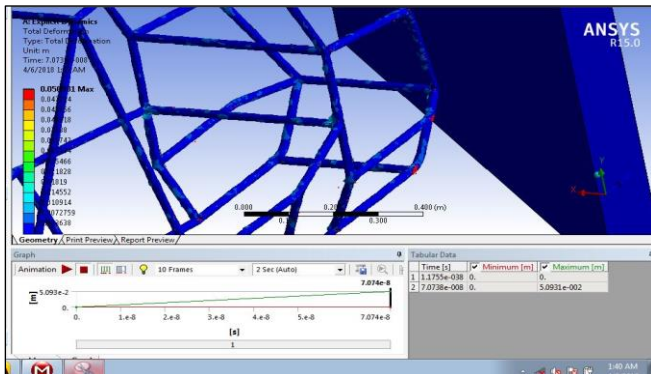
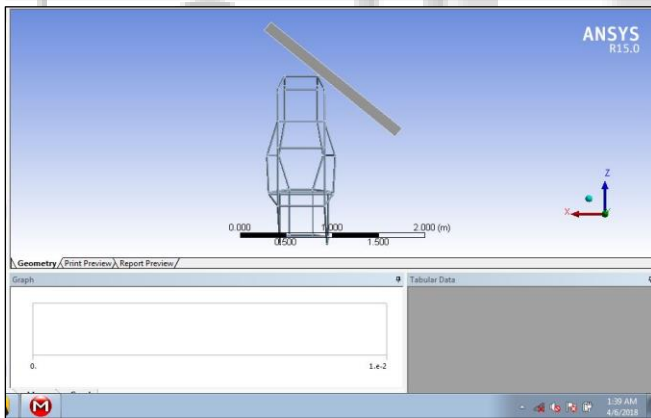
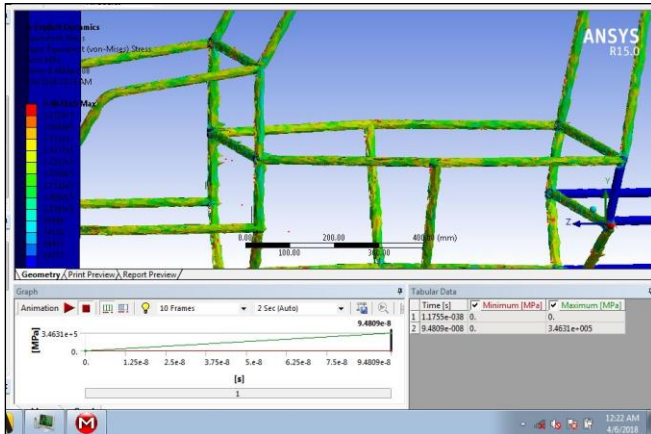
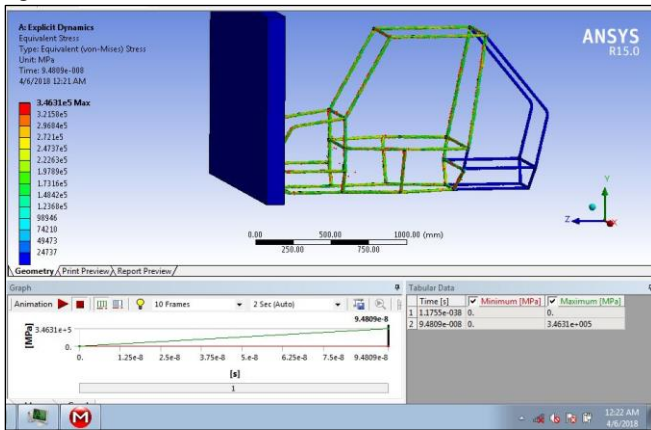
In the above fig we can see the actual arrangement of the impact load at the front.



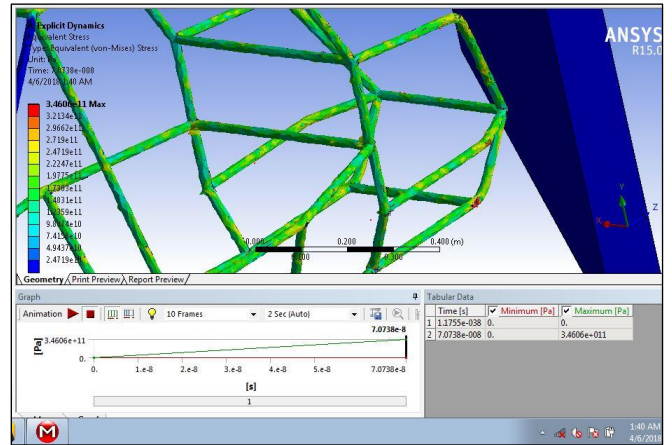
In the above fig we can see the meshing of the components in ANSYS software.



The results of the front impact load is shown in the above figures.



The arrangement of the front side impact arrangement is shown in the above figure.

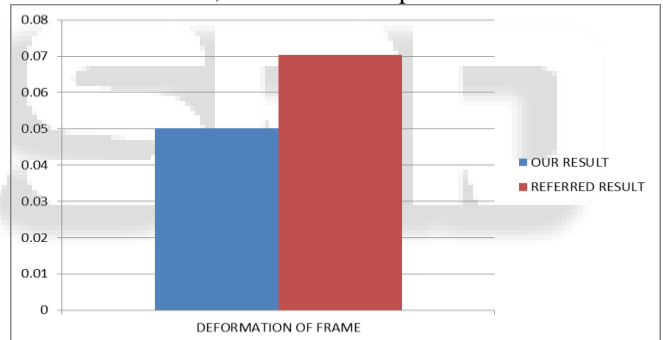


Therefore the result of the updated model is shown in the above figures.

V. RESULT AND DISCUSSION

Test name	Load applied (N)	Resultant Stress (MPa)	Referred stress (MPa)	Factor of safety
Front impact	19839.94	341.73	225.25	1.27
Rear impact	19839.94	302.9	289.34	1.48
Side impact	10491.7285	282.61	250.45	1.53

The above table shows the comparison results of the two models of the front, rear and side impact load.



The above graph shows the clear deformation difference of the redesigned modal and exiting modal. Hence from the above results we can say that our modal is improved upto 10%.

VI. CONCLUSION

- From the above results we can say that through our improvised design we were able to increase overall strength and stability of the chassis frame up to 10%.
- As we have used high grade steel material (AISI 1018 monochrome steel) this can resist both static and dynamic impact loads effectively.
- It is also lighter in weight compared to the other materials and has better strength capabilities which will be helpful in the performance of the vehicle.
- We have used less number of individual components so that there will be the less welded joints. As a single component will have more strength and impact load capabilities than the one which are welded together.
- Our chassis frame design will have more stability and control on both on-road and off-road than the existing design.

REFERENCES

- [1] Vijaykumar V. Patel, and R. I. Patel, "Structural analysis of a ladder chassis frame", World Journal of Science and Technology 2012, 2(4):05- 08 ,ISSN: 2231 – 2587, April 21, 2012. 2. Parkhe Ravindra,*and Sanjay Belkar, "Performance Analysis of Carbon Fiber with Epoxy Resin Based Composite Leaf Spring", International Journal of Current Engineering and Technology, Vol.4, No.2 (April 2014).
- [2] Smita, C. Saddu, Modeling and analysis of composite as an alternative material for leaf spring, volume 11, Issue IV May- June 2014) , PP 39-44.
- [3] J. William Fitch, "Motor Truck Engineering Handbook-Fourth edition" SAE Inc, Warrendale, U.S.A, 1993.
- [4] Nitin S. Gokhale- Practical Finite Element Analysis.
- [5] AutarK.Kaw. Mechanics of Composite Materials. 2e. Taylor and Francis Group, LLc, 2006.
- [6] Cicek Karao a journal on analysis Science and Technology 2012, of a chassis frame with joints was performed by using software
- [7] K. Sowjanya, S. Suresh "Structural Analysis of Disc Brake Rotor" , International Journal of Computer Trends and Technology (IJCTT) – volume 4 Issue 7–July 2013.
- [8] Viraj Parab, Kunal Naik, Prof A. D. Dhale "Structural and Thermal Analysis of Brake Disc" , IJEDR 2014.
- [9] Guru Murthy Nathi, T N Charyulu, K.Gowtham, P Satish Reddy "Coupled Structural Thermal Analysis of Disc Brake", International Journal of Research in Engineering and Technology.
- [10] Swapnil R. Abhang, D. P. Bhaskar "Design and Analysis of Disc Brake" , International Journal of Engineering Trends and Technology Feb 2014.
- [11] www.metweb.com.
- [12] <http://mechanical.srpec.org.in/files/Project/2015/11.pdf>
- [13] Chassis Handbook by Bernd Heibing and Metin Ersoy (Eds.).
- [14] Vehicle Dynamics Theory and Application by Reza N. Jazer.
- [15] CETYS Universidad ATV SAE Design Report by Gustavo Ovies, Andres Magaña, Alejandro Burgas, Iván Pulido, Iván Williams, Bernardo Valadez.
- [16] Race Car Vehicle Dynamics by William F. Milliken and Douglas L. Milliken.
- [17] STATIC ANALYSIS OF ROLL CAGE by Sanatu Chakraborty, Hemant Kumar Neopaney in IJETAE (ISSN 2250-2459, ISO 9001:2008 Certified Journal, Volume 4, Issue 2, February 2014)
- [18] UNDER THE GUIDENCE OF OUR FACULTY GUIDE MR. MANISH VYAS.