

Design & Analysis of Three Wheeled Electric Car Space Frame

Mengu Rohit Ravi¹, Kale Mayur Bapurao², Prateek Mohan Singh³

^{1, 2, 3}, Department Of Mechanical, P.G. Moze College of Engineering, SPPU, (INDIA)

ABSTRACT

The Finite Element Analysis of a chassis space frame has been highlighted in this project. The topic has constrained the study of the chassis space frame. Complex assemblies are to be certainly increasing more triangulization with reducing the Equivalent stresses. The sporty look to the vehicle increases the aesthetics and gives the driving feel. The ergonomics are the main factor with driver safety. The model of the chassis space frame is built using CATIA V5R20 and then imported the .stp file to ANSYS 14.5 to find its finite element module, to perform torsion and bending test on the computational prototype chassis to determine its torsional stiffness. To incorporate a design improvement, optimize and note the effect on the global torsional stiffness of the chassis. The strength, deflection, pressure and torque are calculated the force to be impacted on the chassis. The power train drive should be safe from the foreign and body forces for the continuous drive motion.

KEYWORDS:- CATIA V5R20; ANSYS 14.5; Tadpole Pattern; FEA; CFD; Bending Strength, Bending Stiffness; Structural Analysis; Modal Analysis; Explicit Analysis; CFD (Fluent); Forces.

I.INTRODUCTION

The chassis designed and analyzed is of three wheeled electric car. The need of electric car is increasing to prevent the extinct of non-renewable and conventional energy sources. Secondly to prevent the increase in pollution the electric vehicles are used. But the major drawbacks of electric vehicles are battery life, run-out time and cost. Three wheeler vehicles is a prominent, cost reduced, ergonomic and comfort drive. As the vehicle is in TADPOLE PATTERN (2F 1R) - the weight distribution and C.G. balancing are the main task to run vehicle efficiently with no breakdowns. The analysis done below is Static Structural (Front), Modal (Roll Over), Explicit Dynamics (Front Crash) and Fluid Flow (Fluent/CFD). By doing all these analysis the confirmed results are the vehicle is safe and ready to manufacture. All the design aspects like Strength, Stiffness, C.G., Suspension Hard Points, Torque and Safety were considered while designing the chassis. The chassis is thus optimized to keep the Factor of Safety as low as possible to reduce the cost and life of the vehicle. FEA and CFD analysis are done in ANSYS 14.5 for the better understanding of the deflection and stresses. Turbulence, Velocity Streamline, Pressure Streamline and Volume and Density Rendering are analyzed for the better understanding of the air on the body of the vehicle.

II. IDEALIZATION OF CHASSIS SPACE FRAME

The main aim of chassis is to design and determine the stresses, torsional stiffness, bending stiffness, bending strength, cross sectional area, moment of inertia of the frame. These can be determined by using finite element analysis and thus the final procedure of modal analysis follows at the end. Firstly chassis is designed in CATIA considering the hollow pipe of O.D.-26mm & W.T.-1.65mm. The rough hand sketch was designed considering the suspension hard points. Wheelbase and Track width was considered considering the parameters. Some formulas are mentioned below with the figure given below in Fig.1:-

Bending Strength (X):- $'X' = (S_y I) / C$

S_y = Yield Strength of material; C = Distance from neutral to the extreme fiber

Bending Stiffness (Y):- $'Y' = E_{xx} I_x$

E_{xx} = Modulus of Elasticity; I_x = Second moment of area for structural cross member

Torsional Stiffness (K):- $K = T / \alpha$

T = Torque; α = Angle of deflection

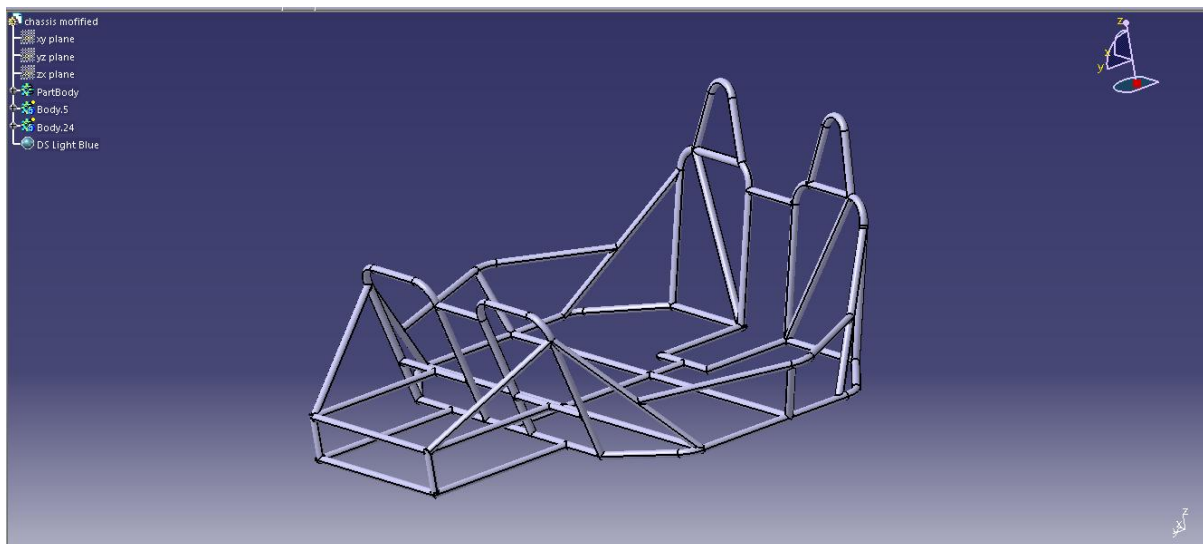


Fig.1:- Chassis Design in CATIA V5R20

After the design the file was converted into the .stp file and imported into ANSYS software. The engineering data and material was selected as structural steel having the density 8.05g/cm^3 . The designed chassis is then mesh for the further analysis step. For the frontal analysis the force calculation as follows and Fig 2 as shown below-

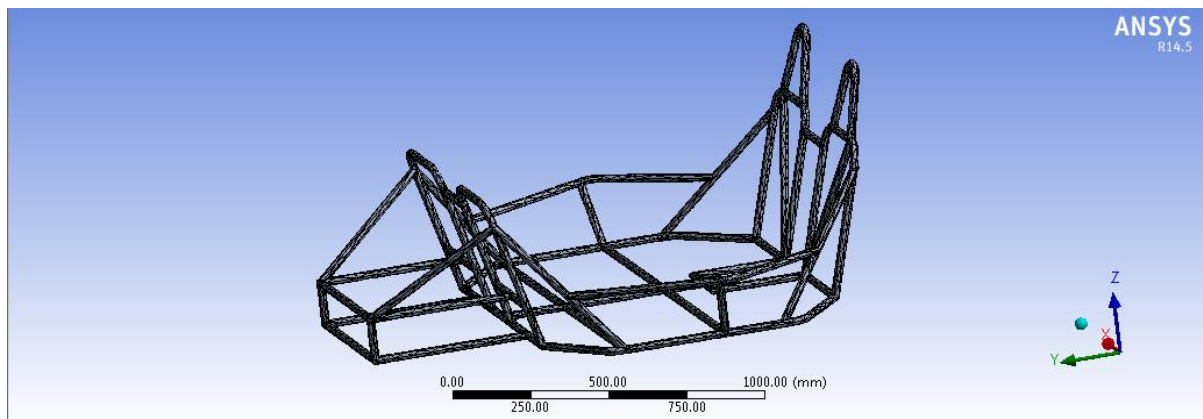


Fig.2:- Meshed Chassis in ANSYS 14.5

$$F = \{M * (V_f - V_i) / T\}$$

M = Mass of the vehicle; V_f = Final Velocity; V_i = Initial Velocity; T = Time of Impact; F = Force

The time of impact considered as 0.1s for large impact force and better results of chassis. The results generated from the impact force for deflection and maximum equivalent stresses are shown below. The results of chassis design are optimized and analyzed. Considering the less factor of safety the whole chassis is designed. Vehicle speed was considered at 60kmph. By the analytical results the maximum total deflection is of 3.78mm and maximum equivalent stress 332.21Mpa. Factor of safety is of 1.023. The Fig 3 & 4 gives the clear view below.

Factor of Safety (F.O.S) = Ultimate Stress / Actual Stress

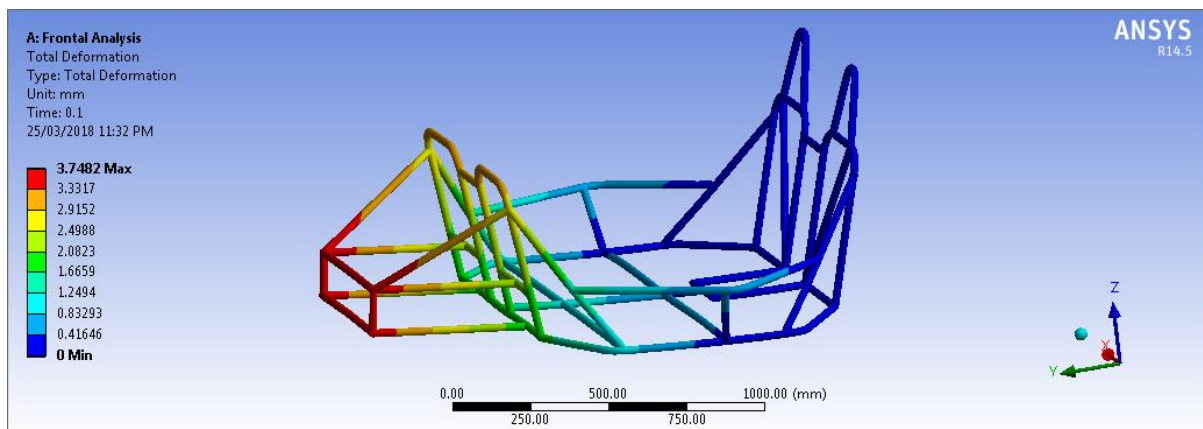


Fig.3:- Total Deformation of Chassis in ANSYS 14.5

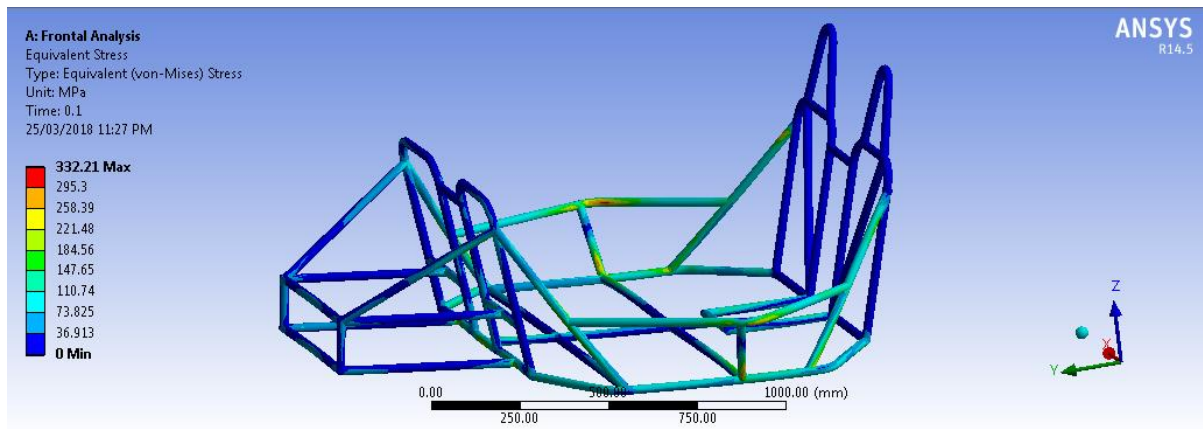


Fig.4:- Maximum Equivalent Stress in ANSYS 14.5

Modal analysis is also done for roll over and resonance. The fundamental frequency is 43.488 Hz. The total deflection of rollover of chassis is 8.9mm. The fixed supports are the base members of the chassis. The fundamental analysis is shown in below diagram Fig.5. The modal analysis gives the better understanding of the resonance, the analytical way of calculating mechanical resonance and the formula is as follows-

$$\text{Frequency} = \left\{ \left[\frac{1}{2\pi} \right] \left(\frac{3.5156}{L^2} \right) \sqrt{\frac{EI}{\rho}} \right\} \quad \dots \quad (\rho = \text{mass} / \text{length})$$

L = Length of the system; E = Modulus of Elasticity; I = Moment of Inertia;

The maximum resonant frequency of the chassis is 2303.13Hz.

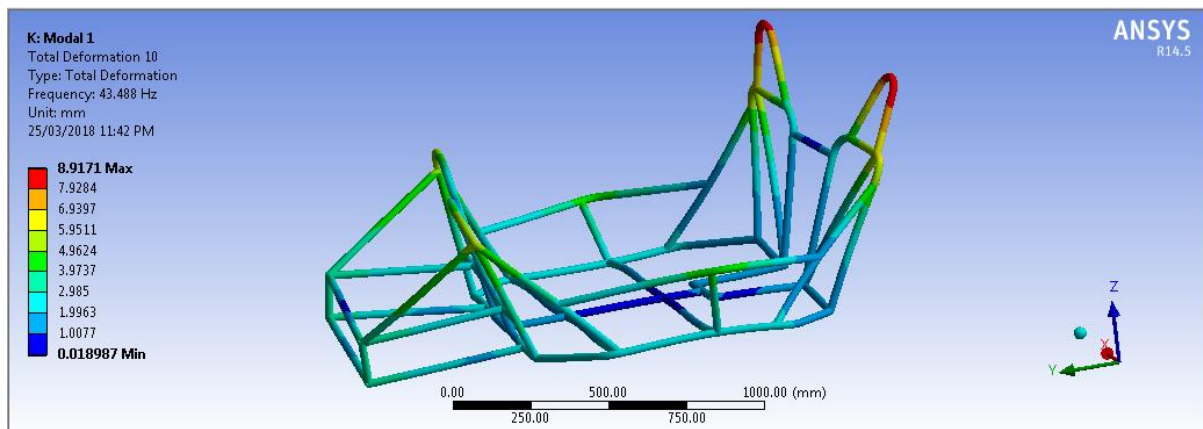


Fig.5:- Modal Analysis in ANSYS 14.5

Explicit Analysis-crash analysis of the front is done considering the velocity of 50kmph (13.85m/s) with the time of impact of 0.1s. The meshed figure is shown below in Fig. 6. The time 0.1s is considered for the maximum force exerted in less time. The block is placed in front of structural steel which is fixed body and frame is given the motion. By meshing both the body and analyzing the system the maximum deflection in the system is of 11.11mm is shown in Fig.7.

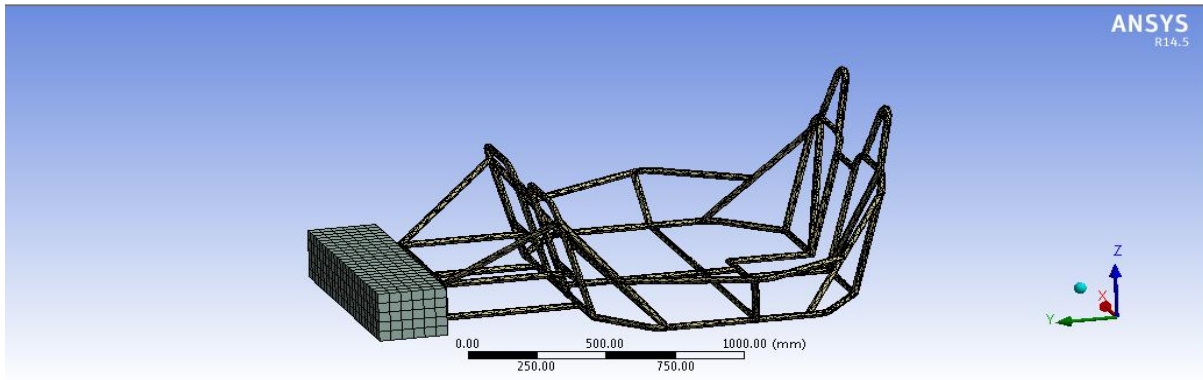


Fig.6:- Meshed Chassis with Block in ANSYS 14.5

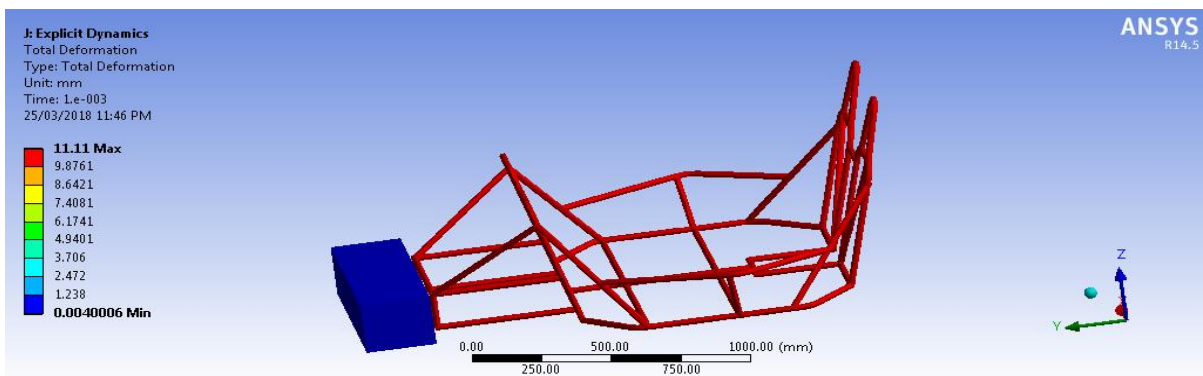


Fig.7:- Total Deformation-Explicit Dynamics in ANSYS 14.5

The directional deformation considering the centre of gravity is maximum up to 0.001mm. The deformation is majorly in the X-direction is shown in Fig.8. In crash the vehicle is destroyed and notes the destruction considering the crumple zone and impact attenuator.

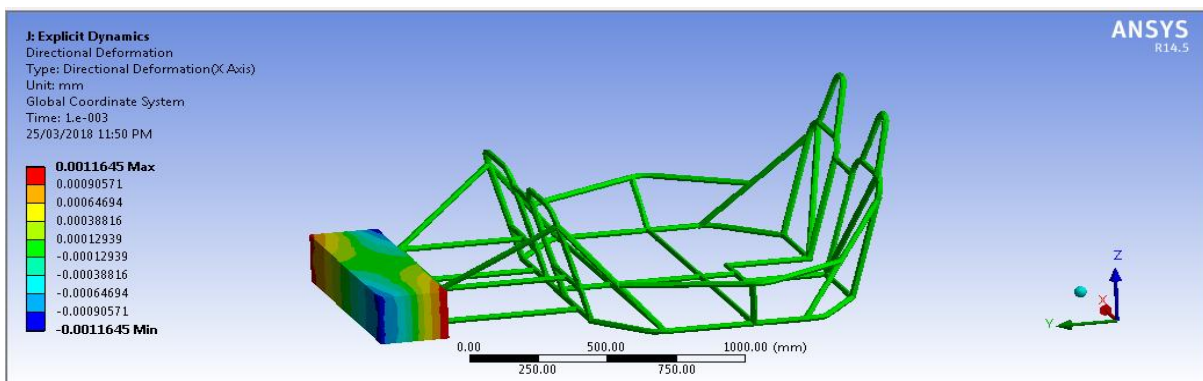


Fig.8:- Directional Deformation in ANSYS 14.5

The computational fluid dynamics of the system is also done. Considering the front nose, the nose is the primary body and boundary walls are the secondary body. The inlet and outlet fluid is considered as air. The both the

bodies are meshed creating a same body part. The meshed images are shown below in Fig. 9 & 10. The results are derived solving 200 iterations. The graphs are generated of turbulence and momentum in u, v & w directions.

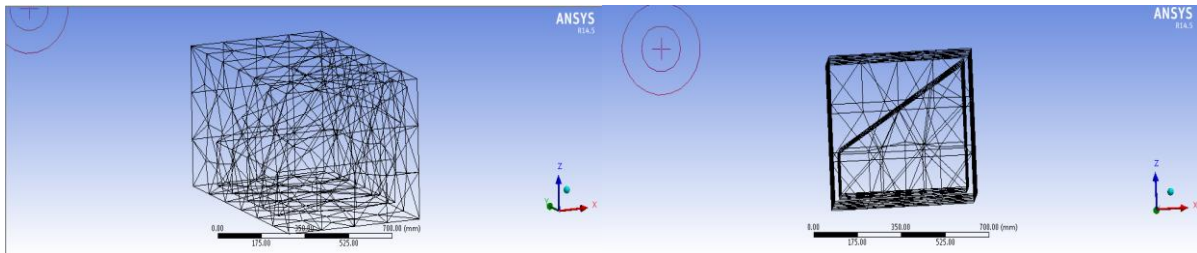


Fig.9 & 10:- Meshed Front Nose & Boundary Wall in ANSYS 14.5

The first result of front nose is related to the velocity streamline. The velocity of the air is maximum up to $5.626 \times 10^1 \text{ m/s}$ (56.26 m/s). As the vehicle velocity is much less than the air velocity is because the passing beneath the triangle section there the pressure decreases and velocity of air increases as shown in Fig 11. The space acts somewhat like venture. The spacing given is 1000. Due to maximum velocity the turbulence is not created near front tires and very less air resistance is created and henceforth less drag.

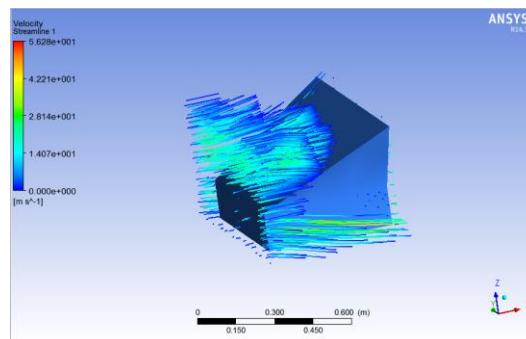


Fig.11:- Velocity Streamline in ANSYS 14.5

The below diagram is related to the density diagram. The density diagram showed the density of air in kg/m^4 . The density of air varies from above surface to the bottom surface. The density of air is more on the top surface as the direct contact with the air from the front as shown in Fig. 12. Whereas in the beneath the triangular section the density is less resulting no whirls loops.

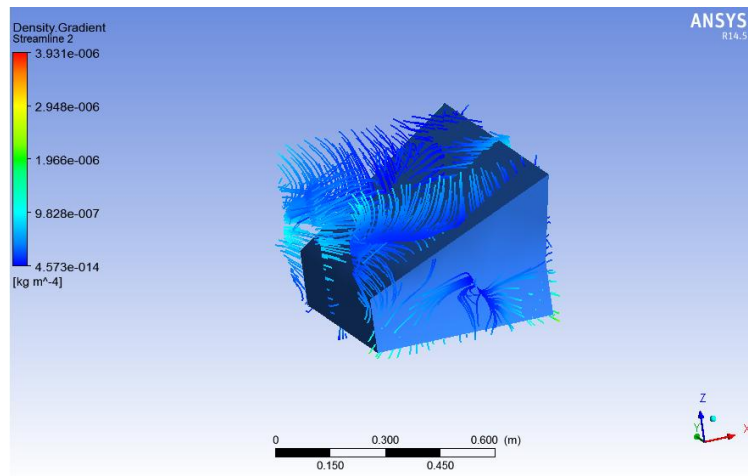


Fig.12:- Density Gradient Streamline in ANSYS 14.5

The density volume rendering is the result corresponding to the volume of air related to the Iso-surfaces. The maximum density volume rendering is 1.2256e+00 Kg/m³ (1.25 Kg/m³). Volume beneath the triangular section area is more as compared to the frontal surface as shown in Fig.13.

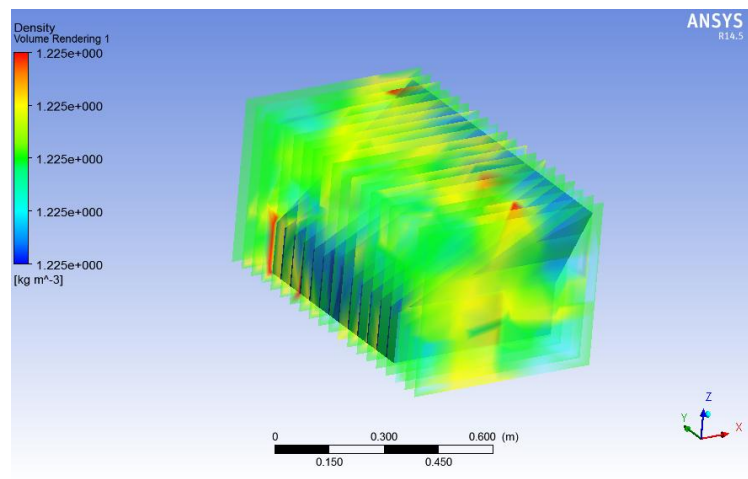


Fig.13:- Density Volume Rendering in ANSYS 14.5

III.CONCLUSION

The designed and analyzed chassis for the three wheeled car is well designed and optimized. The analysis consideration and aspects are the area according to the worst case scenario. Results generated by this analysis are well determined by the analytical method for better results. Very crucial factor ergonomics and driver safety is very well analyzed for the effective feel and drive. Further the material consideration can be changed

accordingly to the requirement of strength in the conditions of use and effectiveness. This project is the small change from our side towards the green technology and electricity as the alternative fuel as a renewable and non-conventional source of energy. The initiative towards the alternative fuel will give a major change to the automobile other sectors of industry.

IV.ACKNOWLEDGE

We acknowledge our main sponsor **HINDUSTAN BLOWERS** for sponsoring us on this project. The trustworthiness of the company on our group makes us proud. They relied on the work capability of our group. We also acknowledge our Faculty Guide- **Prof. Pratik K. Satav** for their valuable support and time every time we needed. Without his support and help this project would be not able to complete. Our Project Faculty Co-ordinators-**Prof. Abhijeet Kadam; Prof. Manjunath Pisotre and Prof. Akshay Nighot**-their valuable inputs and guidance timely made our project plan runs smoothly without any time lag. Their consistent enforcement and support we every time completed the project tasks within time. Their valuable inputs in analysis consideration made the FEA an easy task. The whole mechanical department backed us in the times of need. They supported during the seminars and reviews.

REFERENCES

- [1] Nagarjuna Reddy.Y, Vijaya Kumar S, Study of Different Parameters on the Chassis Space Frame for the Sport Car by using FEA, ISOR Journal of Mechanical & Civil Engineering, Volume 9, Issue 1 (Sept – Oct 2013), PP 01-09.
- [2] W. B. Riley, A. R. George, “Design, Analysis and Testing of a Formula SAE Car Chassis”, USA: SAE International, 2002, 2002-01-3300.
- [3] S. Kumar, “Optimisation of BIW, Chassis and Casting at Mahindra & Mahindra”, India: A proceeding of Altair CAE Users Technology Conference, August 11 -13, Taj Westend, Bangalore, 2005.
- [4] M. Roots, J. Brown, N. Anderson, T. Wanke and M. Gadola, “The Contribution of Passenger Safety Measures to the Structural Performance in Sports Racing Cars”, U.K: Cranfield University, 1995.
- [5] E. Torricelli, L. D’Agostino, A. Baldini, P. Moruzzi, “From Beam to Chassis: How to Increase NVH Performances with an Optimized Moment of Inertia Distribution”, U.K: Proceedings of the World Congress on Engineering 2011 Vol. III WCE 2011, July 6 – 8, 2011.
- [6] Y. Xiang, Q. Wang, Z. Fan, H. Fang, “Optimal crashworthiness design of a spot-welded thin-walled hat section”, USA: University of Iowa, 2006.