IJMTARC – VOLUME – IX – ISSUE – 38, JUL - SEP, 2022 ISSN: 2320-1363

STRUCTURAL DESIGN AND CRASH ANALYSIS OF COMPLETE CAR USING CAE TOOLS

Kolli Avinash, Theppela Naveen Kumar, Shaik Sameer Basha, K Md Suhaib, Garji Hanumanthu, Mr.D.Ibrahim Kalil Basha

Dept. of Mechanical Engineering, Brindavan Institute of Technology and Science, Kurnool, Andhra Pradesh, India

ABSTRACT:

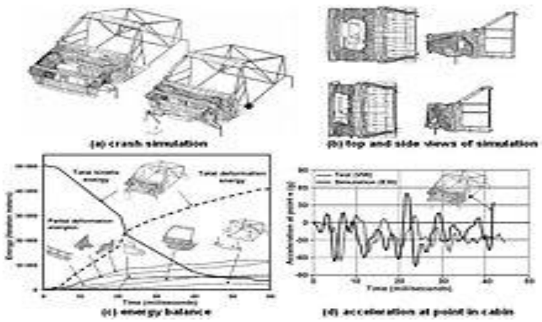
Vehicle crash is a highly nonlinear transient dynamics phenomenon. The purpose of a crash analysis is to see how the car will behave in a frontal pole test with a simplified full car. Crashworthiness simulation is one typical area of application of Finite Element Analysis (FEA). In this project impacts and collisions involving a car model are designed using Catia and analyzed using Ansys software. The car should support the chassis components and the body. It must also withstand static and dynamic loads without undue deflection or distortion. The given model is tested under frontal collision conditions and the resultant deformation and stresses are determined on the car model to impact a rigid column or pole using software. The process is developed fully in 3D CAD model and FE models of different column types featured of the following non linearties: Static impact at different vehicle speeds, plasticity in column and vehicle parts, contact interaction and non linearties material behavior. Design was modified to get approximate similarity with the car used in the experiments. All together different material are used in the FE model of the car.

I.INTRODUCTION

Vehicle crash is a highly nonlinear transient dynamics phenomenon. The purpose of a crashanalysis is to see how the car will behave in a frontal or sideways collision. Crashworthiness simulation is one typical area of application of Finite-Element Analysis (FEA). This is an area in which non-linear Finite Element simulations are particularly effective. In this project impacts and collisions involving a car frame model are simulated and analyzed using ANSYS software. The frame should support the chassis components and the body. It must also withstand static and dynamic loads without undue deflection or distortion. The given model is tested under frontal collision conditions and the resultant deformation and stresses are determined with respect to a time of 80 Mille sec for ramp loading using ANSYS software. The crash analysis simulation and results can be used to assess both the crashworthiness of current frame and to investigate ways to improve the design.

Origin

In the years 1970 attempts were made to simulate car crash events with non-linear spring-mass systems after calibration, which require as an input the results of physical destructive laboratory tests, needed to determine the mechanical crushing behavior of each spring component of the modeled system. "First principle" simulations like more elaborate finite element models, however, need only the definition of the structural geometry and the basic material properties (theology of car body steel, glass, plastic parts, etc.) as an input to generate the numerical model.



The first successful frontal full car crash simulation: a Volkswagen Polo collided with a rigid concrete barrier at 50 km/h (ESI 1986).

In the following years, German car makers produced more complex crash simulation studies, simulating the crash behavior of individual car body components, component assemblies, and quarter and half car bodies in white (BIW). These experiments culminated in a joint project by the Forschungsgemeinschaft Automobil-Technik (FAT), a conglomeration of all seven German car makers (Audi, BMW, Ford, Mercedes-Benz, Opel, Porsche, and Volkswagen), which tested the applicability of two emerging commercial crash simulation codes. These simulation codes recreated a frontal impact of a full passenger car structure (Haug 1986) and they ran to completion on a computer overnight. Now that turn-around time between two consecutive job-submissions (computer runs) did not exceed one day, engineers were able to make efficient and progressive improvements of the crash behavior of the analyzed car body structure.

Application

Crash simulations are used to investigate the safety of the car occupants during impacts on the front end structure of the car in a "head-on collision" or "frontal impact", the lateral structure of the car in a “side collision” or “side impact”, the rear end structure of a car in a "rear-end collision" or “rear impact”, and the roof structure of the car when it overturns during a "rollover". Crash simulations can also be used to assess injury to pedestrians hit by a car.

Material properties of steel

Property	Nomenclature	Value
Young's modulus	EX	200000 N/mm^2
Density	DENS	7800 kg/mm^3
Poisson’s ratio	NUXY	0.3
Ultimate strength	UT	340-2100 MPa

II. LITERATURE REVIEW

A car is a wheeled, self-powered motor vehicle and aproduct of the automotive industry which is mainly used astransportation. Cars are constructed primarily fortransportation of people rather than goods, which generally have seating for one to eight people. Cars are mainly designed to run on roads, which typically have four wheels with tires. Automobiles can be classified by size or weight .Size classification of an automobile is based on wheel base and weight classification of an automobile is based on curbweight.

Andrew Hickey et al. have performed a quasi-static simulation, to simulate the car crash by using finite element method (FEM). Safety of passengers is one of the most important design considerations in the automobile community. Therefore, a crash test is a crucial step to validate the car design. Experimental crash tests results in higher cost, and acquired data might not be correct. Therefore, a numerical modeling and the simulations are used for studying a car crash than to perform experimental testing. Hence, a powerful numerical tool, FEM plays acrucial role in crash test simulations.

C. Sadhasivam et al. has performed a detail car body mode analysis and stress analysis based on 3D LS-DYNA in ANSYS. Modal analysis has been performed to know the natural frequencies and mode shapes of a car body structure. Vibration and crash analysis of the car body is performed, which includes dynamic, static and crash analysis. Most of the automobile manufacturers generally prefer lightweight materials to reduce weight and these include composites, aluminum, magnesium or new types of high strength steels. These materials have a limited strength or ductility, in case of rupture which is a most common seen phenomenon during a crash accident. Material joining failure is also one of the consequences on the vehicle crashworthiness. In a car crash, front-part of the automobile structure absorbs a lot of impact and undergoes plastic deformation. Most of the vehicles are designed to increase the absorption efficiency, to enhance the safety of passengers and reliability of the vehicle. Crashworthiness of different parts of a vehicle needs to be evaluated at the initial stage of the vehicle design only. The dynamic behavior of a structural memberis always different from the static behavior, there forecrash worthiness of the vehicle can be known by impact analysis. Hence, it is necessary to check the crash ability of car structure for both safety and fuel economy. The two ways of ensuring safety are by performing a crash test of a car or by simulating the crash analysis of the modeled car structure in analysis software.

Byeong Sam Kim et al. have performed a crash analysis of sub frame and upper body for neighborhood electric vehicle (NEV). NEV's front platform assembly behavior was simulated in LS-DYNA and results were observed when it is subjected to a frontal car crash. The safety of passengers at low cost reduction has been researched. When a vehicle crashes, the passengers inside the vehicle must be free from injury and the vehicle must be able to withstand impact loads. In crash accidents, capability of the vehicle structure to absorb the energy can be defined as crashworthiness. The vehicle structure should be designed to withstand higher speed and the passengers should not experience a net deceleration.

Lin et al. had performed the computer simulation of a car crash analysis. They have analyzed two crash situations: a higher speed car crashing into a wall and a high velocity car crashing into a static car. The objective of the research was to know the sources which can harm the driver and the passengers when car accidents occur and to create a model of a bumper for knowing its potentiality to withstand impactloads on it. The Simulations on bumper are performed to assure that the bumper design meets the safety requirements.

Tejasagar et al. has studied different car crash simulations by using computer software's because to reduce automobile developing time and to reduce the cost of manufacturing. A frontal impact crash analysis of a car was researched, which made an impact on car crashworthiness. Crash test is one of the destructive tests performed on thecar for ensuring safer design in crashworthiness and to knowthe crash compatibility of

ISSN: 2320-1363

automobiles. The vehicle manufacturers perform different crash tests to ensure safety of the cars under various conditions such as various types of crashes, from different sides, different angles and with different objects, including other vehicles.

Praveen et al. has performed a car crash analysis innon-linear transient dynamics. In the crash test, frontalcollision and sideways collision analysis is performed to know deformations of the car. Crashworthiness of the carsimulations is performed in Finite Element Analysis (FEA).

The chassis frame takes the loads of a heavy vehicle; its function is to carry the loads on the vehicle safely for each operating condition. The frame of chassis should be able to support different chassis components and vehicle structure. Chassis frame should withstand both static loads and dynamic loads without any distortion or deflection in the vehicle. The frontal collision and side collision conditions are tested on the generated model, the total deformations and stresses developed are determined.

Saeed Barbat et al. have modeled a car to evaluate the effects of design variables on dummy responses for front-to side vehicle crash analysis. The striking or hitting vehicle was selected to be a SUV while the struck vehicle was a small size passenger car which consists of four seats. Adeterministic approach that allowed analytical prediction equations for dummy responses was generated. A baseline front crash vehicle to the side crash was modeled in FE and correlated to a physical front crash to the side crash.

III. PROJECT METHODOLOGY

The design process of the chassis consisted of many steps, from the initial assignment to the task of chassis design to the start of construction. These steps are; to identify the restriction, determine the required performance criteria, research design techniques and methodology, use CAD and modeling software to design chassis and lastly start construction.

Throughout these steps, choices must be based on achieving the targets set down to met performance requirement for condition the car will and can be reasonably expected to be subjected to under racing conditions. For example designing a rear air foil on a FSAE car that can only produce more down force than it own weight at speed greater than the car is expected to travel in competition is a futile exercise.

A subdivision of the metal exterior of the car, called a finite element, is connected to nodes at each vertex. Large number of crash simulations uses a method of analysis called the Finite Element Method. The complex problems are solved by dividing a surface into a large but still finite number of elements and determining the motion of these elements over very small periods of time.

Design Criteria

The decisions made in the various design steps so far that have shaped the design criteria. Firstly, the choice to use a space frame design that abides by the standard rules outlined in the SAE 2014 rules book. Secondly, the required performance criteria have been identified as weight and Torsional stiffness, two properties that are generally inversely proportional to one another. –

IV.DESIGNING OF CAR COMPONENTS

Chassis design

We have compare the two chassis using the FEA a new iteration was created as shown in the figure. The below two figure have the slightly different design technique to achieve the performance technique .the strength of the chassis is increased by incorporate many triangular structure section there where many different iteration which was made before the final design was made. The below figure show the frame before the iteration is done. The FEA is done on the old frame by using the Ansys as a tool and the result which was obtained is shown below. As the method of analysis was done in the three stages such as the front impact test, side impact test., rear impact test. The below figure provide the detail information of the changes which are done in the old frame and the new frame, the various view of the frame is provided.

Structural analysis

In a typical crash simulation, the car body structure is analyzed using spatial discretization, that is, breaking up the continuous movement of the body in real time into smaller changes in position over small, discrete time steps. The discretization involves subdividing the surface of the constituent, thin, sheet metal parts into a large number (approaching one million in 2006) of quadrilateral or triangular regions, each of which spans the area between "nodes" to which its corners are fixed. Each element has mass, which is distributed as concentrated masses and as mass moments of inertia to its connecting nodes. Each node has 6 kinematic degrees of freedom, that is, one node can move in three linear directions under translation and can rotate about three independent axes.

If the nodes move during a crash simulation, the connected elements move, stretch, and bend with their nodes, which cause them to impart forces and moments to their nodal connections. The forces and moments at the nodes correspond to the inertia forces and moments, caused by their translational (linear) and angular accelerations and to the forces and moments transmitted by the resistance of the structural material of the connected elements as they deform. Sometimes, additional external structural loads are applied, like gravity loads from the self weight of the parts, or added loads from external masses.The forces and moments of all nodes are collected into a column vector (or column matrix), and the time dependent equations of motion (in dynamic equilibrium) can be written as follows.

$$\mathbf{Ma} = \mathbf{F}_{ext} - \mathbf{F}_{int}$$

where vector $\mathbf{Ma}$ (mass times acceleration vector) collects the inertia forces at the nodes, $\mathbf{F}_{ext}$ collects the external nodal loads, and $\mathbf{F}_{int}$ collects the internal resisting forces from the deformation of the material. *M* is a diagonal matrix of the nodal masses. Each vector (*u*, *v*, *a*, *F*, etc.) has dimension 6 times the total number of nodes in the crash model (about 6 million “degrees of freedom” for every 1 million "nodes" in 3-D thin shell finite element models).

V. DESIGN METHODLOGY

5.1 Introduction to CATIA

CATIA (Computer Aided Three-dimensional Interactive Application) is a multi-platform CAD/CAM/CAE commercial software

IJMTARC – VOLUME – IX – ISSUE – 38, JUL - SEP, 2022

ISSN: 2320-1363

suite developed by the French company Dassault Systems. Written in the C++ programming language, CATIA is the cornerstone of the Dassault Systems product lifecycle management software suite. CATIA competes in the high-end CAD/CAM/CAE market with Cero Elements/Pro and NX (Unigraphics).

The concept of CATIA V5 is to digitally include the complete process of product development, comprising the first draft, the Design, the layout and at last the production and the assembly. The workbench Mechanical Designis to be addressed in the Context of this CAE training course.

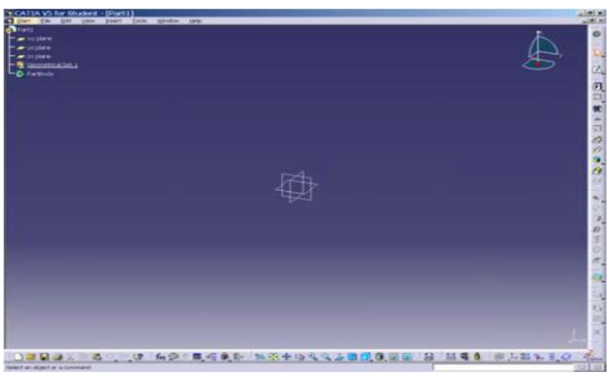


Fig: 5.1: Home Page of CatiaV5

Modeling of Complete Car in CATIA V5

This Complete Caris designed using CATIA V5 software. This software used in automobile, aerospace, consumer goods, heavy engineering etc. it is very powerful software for designing complicated 3d models, applications of CATIA Version 5 like part design, assembly design.

The same CATIA V5 R20 3d model and 2d drawing model is shown below for reference. Dimensions are taken from. The design of 3d model is done in CATIA V5 software, and then to do test we are using below mentioned software's.

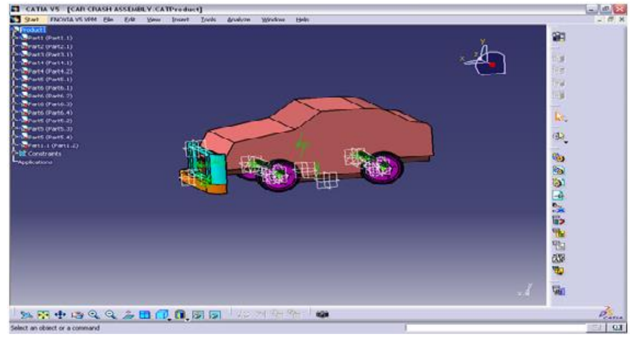


Fig: 5.2: Model design of Car in CATIA-V5

VI. ANALYSIS OF COMPLETE CAR

6.1 Procedure for FE Analysis Using ANSYS:

The analysis of the Car is done using ANSYS. For compete assembly is not required, is to carried out by applying moments at the location along which axis we need to mention. Fixing location for assembly.

6.2 Preprocessor

In this stage the following steps were executed:

- **Import file in ANSYS window**

File Menu > Import> STEP > Click ok for the popped up dialog box > Click

Browse" and choose the file saved from CATIAV5R20> Click ok to import the file

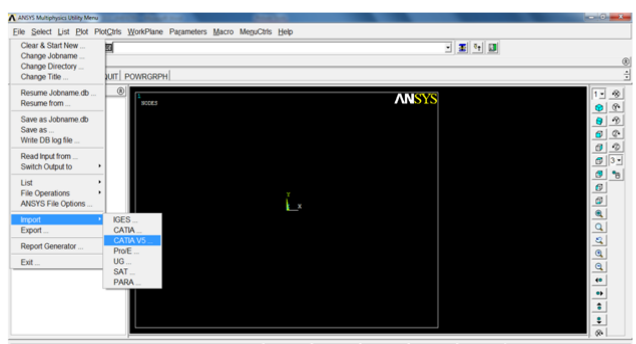


Fig.6.1: Import panel in Ansys.

6.2.1 Meshing:

Mesh generation is the practice of generating a polygonal or polyhedral mesh that approximates a

IJMTARC – VOLUME – IX – ISSUE – 38, JUL - SEP, 2022 ISSN: 2320-1363

geometric domain. The term "grid generation" is often used interchangeably. Typical uses are for rendering to a computer screen as finite element analysis or computational fluid dynamics. The input model form can vary greatly but common sources are CAD, NURBS, B-rep and STL (file format). The field is highly interdisciplinary, with contributions found in mathematics, computer science, and engineering.

Three-dimensional meshes created for finite element analysis need to consist of tetrahedral, pyramids, prisms or hexahedra. Those used for the finite volume method can consist of arbitrary polyhedral. Those used for finite difference methods usually need to consist of piecewise structured arrays of hexahedra known as multi-block structured meshes.

Meshing is an integral part of the computer-aided engineering (CAE) simulation process. The mesh influences the accuracy, convergence and speed of the solution. Furthermore, the time it takes to create a mesh model is often a significant portion of the time it takes to get results from a CAE solution. Therefore, the better and more automated the meshing tools, the better the solution. From easy, automatic meshing to a highly crafted mesh, ANSYS provides the ultimate solution. Powerful automation capabilities ease the initial meshing of a new geometry by keying off physics preferences and using smart defaults so that a mesh can be obtained upon first try. Additionally, users are able to update immediately to a parameter change, making the handoff from CAD to CAE seamless and aiding in up-front design. Once the best design is found, meshing technologies from, ANSYS provide the flexibility to produce meshes that range in complexity from pure hex meshes to highly detailed Hybrid meshes. It has a range of meshing tools that cater to nearly all physics. While the meshing technologies were developed to meet specific needs in the areas of solid, fluid, electromagnetic, shell, 2-D and beam models, access to these technologies is available across all physics.

VII. DISCUSSION ON ANALYSIS RESULT

7.1 Results of Displacement analysis:

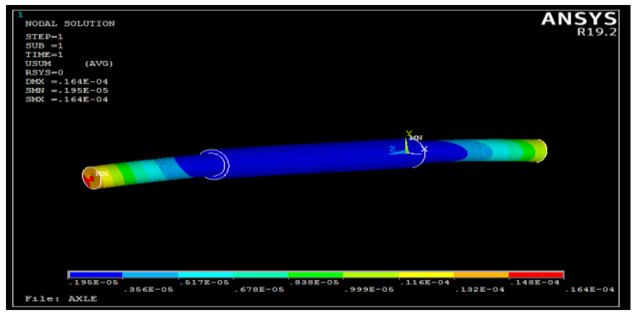


Fig.7.1 Axle Displacement

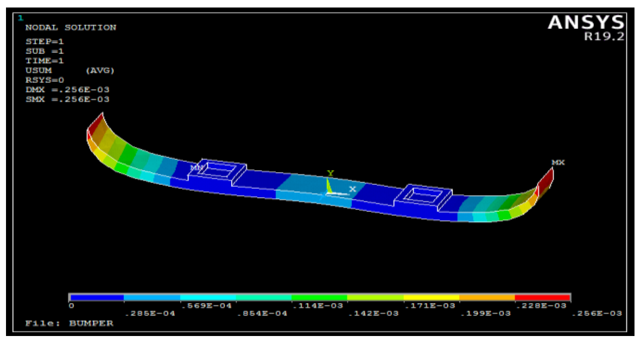


Fig.7.2Bumper Displacement

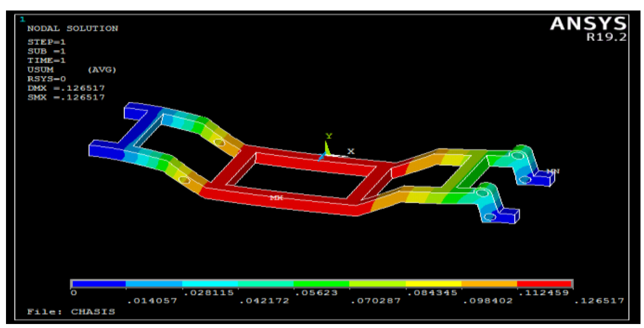


Fig.7.3Chassis Displacement

IJMTARC – VOLUME – IX – ISSUE – 38, JUL - SEP, 2022

ISSN: 2320-1363

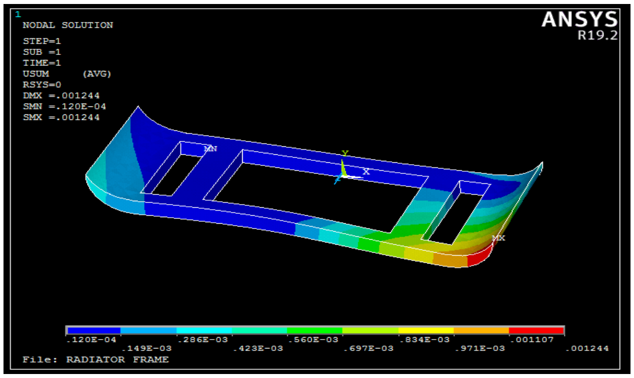


Fig.7.4Radiator Frame Displacement

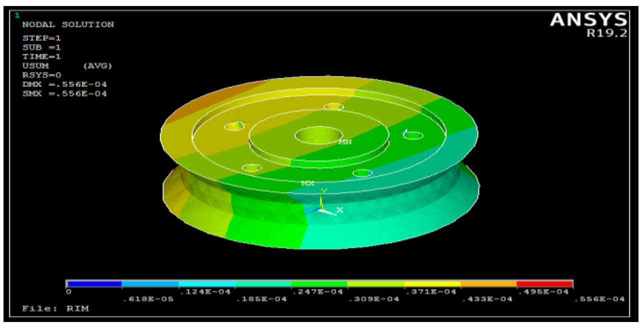


Fig.7.5Rim Displacement

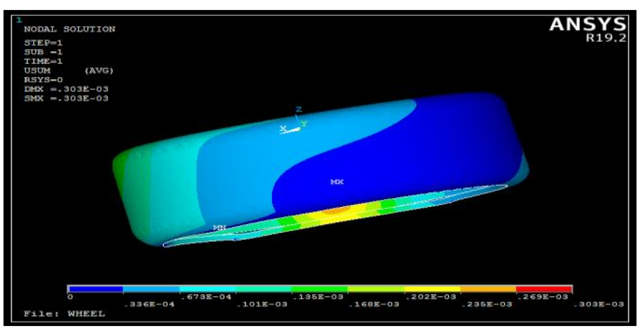


Fig.7.6Wheel Displacement

7.2 Results of Stress analysis:

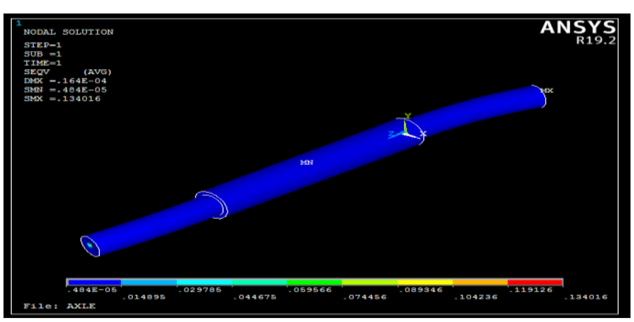


Fig.7.7AxleStress

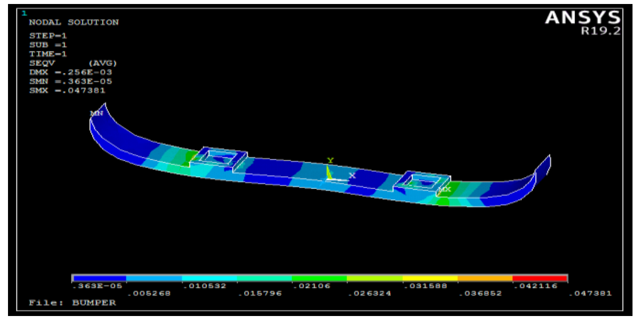


Fig.7.8Bumper Stress

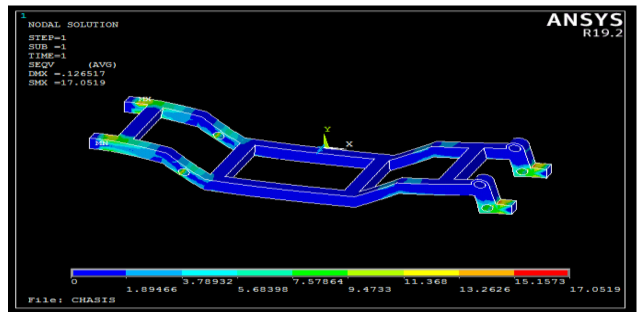


Fig.7.9ChassisStress

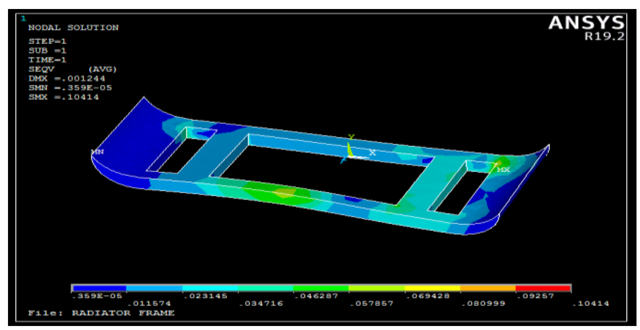


Fig.7.10Radiator Frame Stress

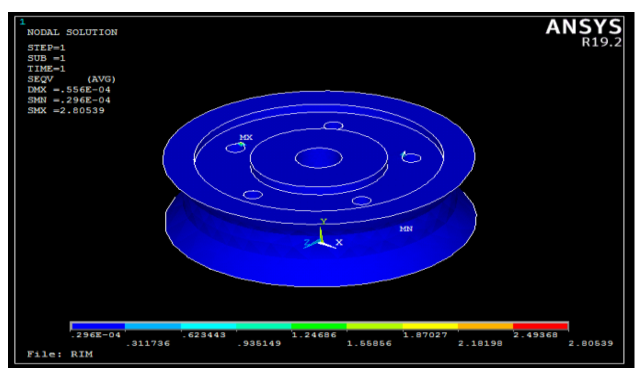


Fig.7.11Rim Stress

IJMTARC – VOLUME – IX – ISSUE – 38, JUL - SEP, 2022 ISSN: 2320-1363

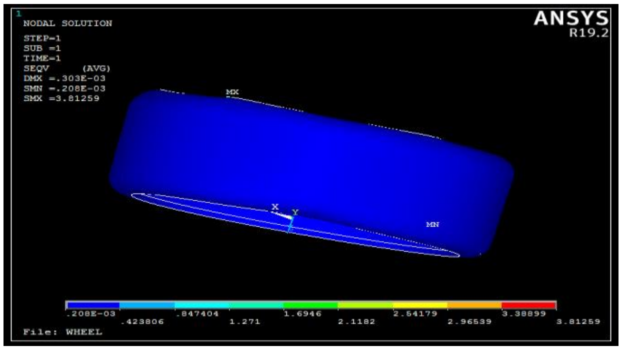


Fig.7.12Wheel Stress

7.3 Results of Strain analysis:

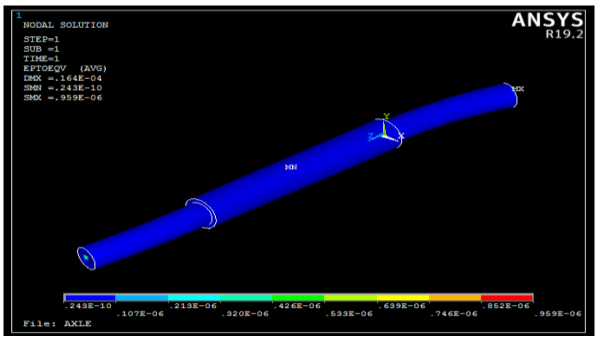


Fig.7.13AxleStrain

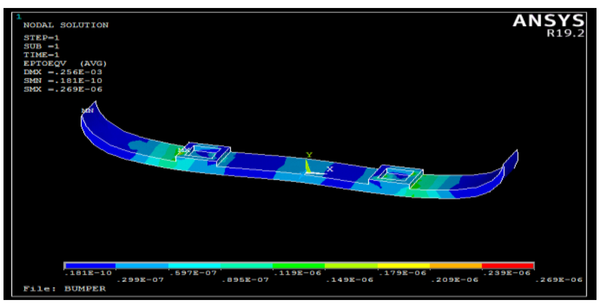


Fig.7.14BumperStrain

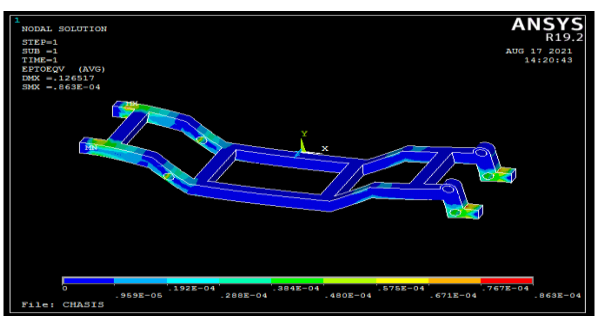


Fig.7.15ChassisStrain

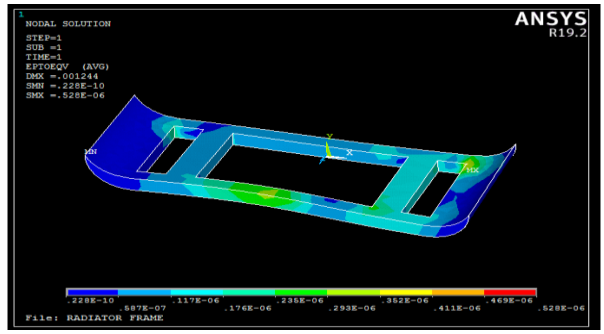


Fig.7.16Radiator Frame Strain

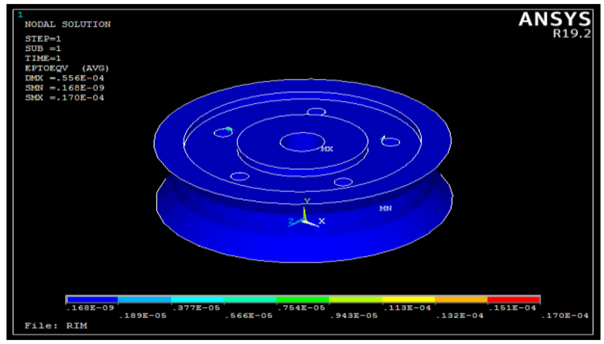


Fig.7.17RimStrain

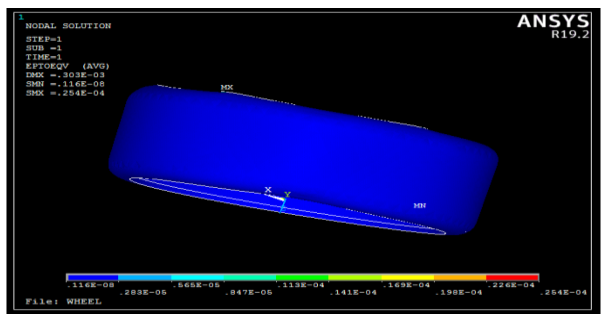


Fig.7.18WheelStrain

VIII. CONCLUSIONS & FUTURE SCOPE

CONCLUSIONS

The objectives of the project have been achieved by the work of this report, which is to investigate the viability (strength, ability to support the signs, performance in the event of crash) of the thin walled collapsible sign post as a replacement to the existing rigid structure. A range of collapsible post designs has been analyzed and evaluated to investigate the viability of the thin walled collapsible sign post as are placement to the existing rigid structure.

The finite element analysis software was used to predict the behavior of the alternative sign posts. Design optimization has been carried out to achieve better performance under both load and impact. The work done in this project also demonstrated that could be a very useful tool for design and optimizing the roadside furniture such assign post. It could save many expansive physical crash tests and speed up the product development process.

This project is made with the designs, that it provides flexibility in operation. This innovation has made the more desirable. It can be seen from the above results that, our objective to design safe and secured for this system. A highly nonlinear model for the dynamic behavior of the compressed air driven system is considered. A parametric study to investigate the influence of the control parameters on the dynamic response is conducted.

As shown in above figures they are meshed and solved using Ansys and displacement obtained, which is very less. Stresses developed at the fixing location (Minimum Stress which is acceptable); and the strain value, is very less compared to yield value of given materials; this is below the yield point. So we can conclude our design parameters are approximately correct. Therefore, the design of the mechanism worked flawlessly as well. All these facts point to the completion of our objective in high esteem.

FUTURE SCOPE

- The actual destruction of the vehicle is avoided.
- The crash test using simulation becomes economical.
- A variety of model can be tested for the same FEA model.
- The results are used to assess both the crashworthiness and to investigate the ways to improve the design.
- This study is evident that along with thickness of rail the yields strength and yield material properties have significant bearing on the load behavior of vehicle.
- Further study can be taken with different geometrical shapes with varying moment of inertia for members with the same thickness and yield of material.
- The bumper of a car can be made as springs can be placed in the bumper in order to reduce the impact that is transferred to the passenger compartment. These designs may also reduce the weight of the bumper leading to increase fuel efficiency.

IX. REFERENCES

- Andrew Hickey, and Shaoping Xiao, “Finite element modeling and simulation of car crash,” International Journal of Modern Studies in Mechanical Engineering, ISSN 2454-9711, Volume 3, Issue 1, 2017, pp. 1-5.
- C. Sadhasivam, and S. Jayalakshmi, “Simulation of car body by development of static and dynamic analysis,” International Journal of Automobile Engineering Research and Development, ISSN (P): 2277-4785; ISSN: 2278–9413 Vol. 4, Issue 3, June 2014, pp. 1-6.
- Byeong Sam Kim, Kyoungwoo Park, and Youn-Min Song, “Finite element frontal crash analysis

of vehicle's platform with upper and sub frame body," International Association for Automation and Robotics in Construction, June 2011, pp. 1326-1329.

- Lin C. S, Chou K.D. and Yu C.C, "Numerical simulation of vehicle crashes," Applied Mechanics and Materials, 590, December 2014, pp. 135.
- Tejasagar ambati, K.V.N.S. Srikanth and P. Veeraraju, "Simulation of vehicular frontal crash-test," International Journal of Applied Research in Mechanical Engineering, ISSN: 2231 –5950, Volume-2, Issue-1, 2012.
- T. Ananda Babu, D. Vijay Praveen and Dr. M. Venkateswarao, "Crash analysis of car chassis frame using finite element method," International Journal of Engineering Research & Technology (IJERT), ISSN: 2278-0181 Vol. 1 Issue 8, October - 2012.
- Saeed Barbat, Xiaowei Li and Priya Prasad, "Vehicle-to vehicle front-to-side crash analysis using a CAE based methodology," in Passive Safety Research and Advanced Engineering, Ford Motor Company, United States, Paper Number 07-0347.