



# DESIGN AND STRUCTURAL ANALYSIS OF COMPLETE CAR USING CAE TOOLS

Mrs. B. Sandhya Rani <sup>1</sup>, Mr. Jogipet Saicharan Goud <sup>2</sup>

<sup>1</sup> Assistant Professor, <sup>2</sup> M.Tech Student, Dept. of Mechanical Engineering,  
Ellenki College of Engineering and Technology (Autonomous), Patalguda(V), Ameenpur (M),  
Sanga Reddy, Telangana, India

**Abstract:** A very nonlinear transient dynamics phenomenon is a vehicle crash. One common application of Finite Element Analysis (FEA) is to simulate a simplified load analysis in order to determine how the car and its components will react. Impacts of this study include models that were created with Catia and examined with Ansys software. The body and chassis parts should be supported by the vehicle. Additionally, it must sustain static stresses without experiencing excessive distortion or deflection. The provided model is put through a frontal collision test, and software is used to calculate the stresses and deformation that affect a stiff load. The following non-linearities are included in the process's entire development in 3D CAD and FE models: touch interaction, plasticity in vehicle components, static impact on the vehicle, and non-linearities in material behavior. The FE model of the vehicle uses a variety of materials. To acquire a rough resemblance to the automotive components utilized in the studies, the design was altered.

## I- INTRODUCTION

An illustration of a highly nonlinear transient dynamics phenomenon is an automobile collision. Finding out how the car would respond in a frontal or sideways collision is the aim of a crash study. Crashworthiness simulation frequently uses finite-element analysis (FEA). In this area, non-linear Finite Element simulations are quite helpful. In this study, a vehicle frame model is used to simulate and assess impacts and collisions using ANSYS software. The chassis, the backbone of a large vehicle, is primarily made to safely support the maximum weight under all anticipated operating circumstances. The body and the components of the chassis should be supported by the frame. Additionally, it must not deform or deflect excessively when subjected to static or dynamic stresses. The given model is assessed in frontal collision scenarios using ANSYS software, and the resulting stresses and deformation are computed in relation to a ramp loading time of 80 ms. The existing frame's crashworthiness may be assessed and design improvements can be explored using the crash analysis simulation findings.

To determine each spring component's mechanical crushing behaviour in the simulated system, the findings of physically hazardous laboratory trials have to be presented. In order to simulate auto crash situations, non-linear spring-mass systems that had been modified were tried to be used in the 1970s. "First principle" simulations involve determining the basic material properties and the structural geometry (theology of automotive body steel, glass, plastic components, etc.), much like more intricate finite element models.

### Benefits

A crash simulation yields data without requiring a new automobile model to undergo destructive testing. In this manner, tests may be carried out in a computer rapidly and affordably, allowing for design optimization prior to the actual car prototype being produced. Before investing time and resources in a real crash test, issues can be resolved with a simulation. Designers can tackle some difficulties that would have been practically impossible without the aid of a computer thanks to the enormous flexibility of printed output and graphical presentation.

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

### Frame structure

Vehicle body structures are designed and constructed by safety engineers to endure both static and dynamic service loads over the course of the vehicle's life cycle. The body of the vehicle provides most of the rigidity in torsion and bending. Additionally, it features a specially designed occupant compartment to lessen injuries in the event of an accident.

### Body Cars from Steel Materials

The development of low alloy steels with great strength has received a lot of interest worldwide. The necessity to minimize driving fuel consumption, save materials and energy during production, raise the strength and dependability of parts at existing levels, reduce their maximum total mass, and extend the working life of cars all sparked development. Different kinds of steel are used in the building of automobile bodies. Dual phase steels make up the largest portion of all steels used in automobile building.

## 2 - LITERATURE REVIEW

Cars, which are wheeled, self-powered motor vehicles principally used for transportation, are produced by the automotive industry. The primary purpose of cars, which usually include seats for one to eight people, is to transport people, not goods. The main purpose of cars, which typically have four wheels and tires, is to operate on public roads. Vehicles can be grouped according on their size or weight. The size of a car is determined by its wheelbase, while its weight is determined by its curb weight.

Andrew Hickey et al. quasi-statically simulated a car accident using the finite element method (FEM). Passenger safety is one of the most important design considerations in the automotive industry. Therefore, a crash test is a crucial component in verifying the car's design.

Beyond Sam Kim and colleagues looked at an accident that included a neighborhood electric vehicles (NEV) subframe and upper body. LS-DYNA was used to model the behavior of the NEV's front platform assembly, and the outcomes of a frontal collision were captured. Research on the safety of low-cost passengers has been carried out.

The capacity of a vehicle's structure to absorb energy in crashes is known as crashworthiness. There should be no net deceleration for the occupants, and the vehicle should be constructed to withstand greater speed.

Lin et al. conducted a computer simulation of an examination of a vehicle collision. A high-velocity car colliding with a stationary vehicle and a higher-velocity vehicle striking with a wall are the two collision situations they have studied. The study's objectives were to build a bumper model, evaluate its resistance to impact loads, and pinpoint the factors that could result in injuries to drivers and passengers in auto accidents. Simulations are performed on the bumper to ensure that its design meets safety requirements.

Teja Sagar et al. have investigated several automotive collision simulations utilizing computer software in order to shorten the time needed for automobile development and lower manufacturing costs. Research on a car's frontal impact crash study affected the vehicle's crashworthiness. Crash tests are one of the destructive tests done on cars to determine their crash compatibility and to ensure safer designs in terms of crashworthiness. To make sure the cars are safe in a variety of scenarios, including different kinds of collisions, from different perspectives, and with different objects, including other automobiles, the automakers conduct a variety of crash tests.

Praveen et al. performed an examination of an automobile collision in on-linear transient dynamics. In the crash test, the car's deformations are ascertained by frontal and sideways impact analysis. The carsimulations' crashworthiness is assessed using Finite Element Analysis (FEA).

A. R. Mortazavi Moghaddam et al. improved the bumpers' characteristics by using glass materials. Here, LS-DYNA and CATIA were used for structural analysis and conceptual design, respectively. The standard data on bumpers, element type, procedure, boundary conditions, and material mechanical characteristics are all collected from the previous study.

Lin and associates developed a numerical approach to simulate and analyse auto accidents on a computer. Two collision scenarios—a high-speed test vehicle hitting a wall and a high-speed vehicle hitting a stationary vehicle—were investigated.

## 3 -PROJECT METHODOLOGY

From the first assignment to the chassis design task until the commencement of production, the chassis design process involved numerous steps. These include identifying the limitations, figuring out the necessary performance standards, researching design methods

and approaches, designing the chassis using CAD and modeling tools, and finally beginning building.

A different method for simulating crashes is to use the Macro Element Method. The Macro Element Method differs from the two previously stated approaches in that it has a fewer number of elements in its structure. Instead of using partial differential equations to compute structure deformation, the technique is based on experimental data. The software package Pam-Crash, which began crash simulation, is frequently used for applying the Finite Element Method. Although this approach enables precise structural modeling, its drawbacks include lengthy computation times and high processing unit requirements. The Macro Element Methodology is used in Visual Crash Studio. Although it has some modeling and boundary condition restrictions when compared to FEM, it may be used without sophisticated computers and takes significantly less time to calculate. There are two approaches that are complementary to one another. The Finite Element Method works well in the last stages of the structure design process, whereas the Macro Element Method is helpful in the early stages.

### Design Criteria

The design standards have been impacted by decisions made at every step of the design process. Using a space frame design that complies with the SAE 2014 rules book's standard standards is the first choice. Second, it has been shown that the required performance requirements are weight and torsional stiffness, two properties that are frequently inversely connected. When wheels are subjected to vertical acceleration (bumps), the increased unsprung mass caused by hub motors increases the forces acting on the chassis, which is a disadvantage (Smith, 1984). Consequently, the FSAE will require more torsional rigidity than other FSAE cars.

### Torsional Stiffness

To build an automobile with the highest torsional stiffness, one must study the fundamental or generalized equation for torsion. This expression shows that torsional stiffness is inversely proportional to length and proportional to both the material shear modulus and the polar moment inertia. By applying these crucial relationships and contextualizing them inside the chassis, it is possible to optimize the stiffness of the chassis. Length is related to the car's wheelbase, and since the minimum wheelbase permitted by the regulations was chosen, this relationship has already been used.

### Improvements

In order to boost speed, this straightforward flow chart had major changes along the way. Ansys modelling was not required at every iteration thanks to the

adoption of Pro-e Simulations FEA. Setting up and running the FEA took no time at all because Pro-e Simulation was already interfaced with Pro-e; however, analyzing a new model in Ansys would require many hours of setup and input. Nevertheless, the program's lack of mesh refinement, type selection, element selection, and complex loading conditions—a limitation that will be discussed later—did limit the options for performing the FEA.

## 4 - DESIGNING OF CAR COMPONENTS

### Chassis design

After using the FEA to compare the two chassis, a new iteration was produced, as seen in the figure. To achieve the performance technique, the two figures below use somewhat different design techniques. Numerous triangular construction sections are incorporated into the chassis to boost its strength. Before the final design was created, numerous revisions were done.

### Structural analysis

In a typical accident simulation, the structure of an automobile's body is examined using spatial discretisation, which breaks down the body's continuous movement in real time into smaller positional changes over tiny, discrete time increments. Discretization is the process by which the surface of thin sheet metal components is separated into countless triangle or quadrilateral pieces, each of which is as long as the distance between the "nodes" to which its corners are attached (almost a million in 2006). All elements have mass, which is distributed towards their connecting nodes as mass moments of inertia and concentrated masses. With six kinematic degrees of freedom, each node can rotate around three different axes and move in three linear directions under translation.

Sometimes, external structural loads are applied, like extra weights from outside masses or gravity loads from the self-weight of the parts. The time-dependent equations of motion (in dynamic equilibrium) can be expressed as follows using a column vector (or column matrix) that contains the forces and moments of each node.

### Solution

The accelerations, velocities, and displacements are calculated by solving the previously provided system of linear equations for each discrete time,  $t$ , during the crash period. The mass matrix's diagonal structure simplifies this solution. How long the computer takes depends on the quantity of finite components and solution time steps. Therefore, what is the largest stable time step for a specific material in the numerical crash simulation model if  $d_{min}$  is the smallest distance between any two nodes?

This distance may change during a simulation; thus, it is necessary to update the stable time step continuously as the solution changes.

## BILL OF MATERIAL

S. No	Component	Material	Quantity	Volume (m <sup>3</sup> )	Area (m <sup>2</sup> )	Mass (Kg)	Density (Kg m <sup>3</sup> )	Weight (Kg)
1	Tire	Rubber	4	0.003	0.712	2.744	910	26.91
2	Rim	Steel	4	0.024	0.775	185.036	7860	1815.2
3	Radiator Frame	Steel	1	0.02	1.215	158.15	7860	1551.45
4	Chassis	Steel	1	0.098	3.859	773.746	7860	7590.44
5	Bumper	Steel	1	0.013	0.752	98.61	7860	967.36
6	Axle	Steel	2	0.006	0.364	60.652	7860	594.99

## 5 - DESIGN METHODOLOGY

### Introduction to CATIA

The French company Dassault Systems created the multi-platform commercial CAD/CAM/CAE software suite known as CATIA (Computer Aided Three-dimensional Interactive Application). Dassault Systems' product lifecycle management software package relies heavily on CATIA, which is integrated into the C++ programming language. Competitors in the high-end CAD/CAM/CAE market include Cero Elements/Pro, NX (Unigraphics), and CATIA. Dassault Systems released CATIA V5, a 3D CAD program, in 1999. CATIA V4 was replaced by an entirely new and quite different design tool. Depending on the IT environment, the user interface, which currently has an MS Windows layout, can be readily integrated with standard software packages like SAPR3 products, Microsoft Office, and other graphic applications.

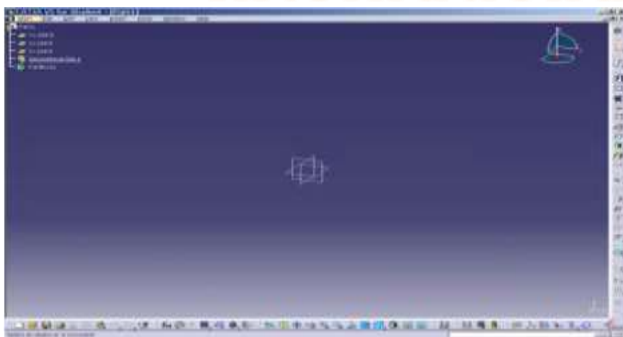
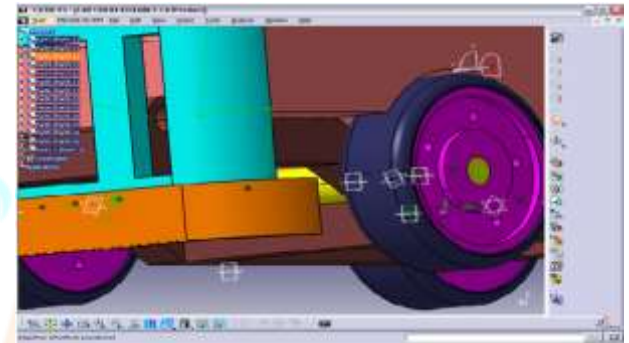


Fig: 5.1: Home Page of CatiaV5

### 5.4 Multitasking Industrial Robot Modeling in CATIA V5

CATIA V5 software was used in the design of this multitasking industrial robot. This software is utilized in heavy engineering, consumer goods, automotive, and aerospace, among other industries. It is an extremely capable program for creating intricate 3D models and CATIA Version 5 applications, such as part and assembly design.



### 5.4.3 Assembly Modeling of CAR

Using restrictions, coincidence, contact, offset, angle, fix component, flexible, manipulation, etc., all of the components are put together in this modeling.

**Multi View:** With this command, all of the component or model's views can be edited under the workbench and seen on the screen simultaneously.

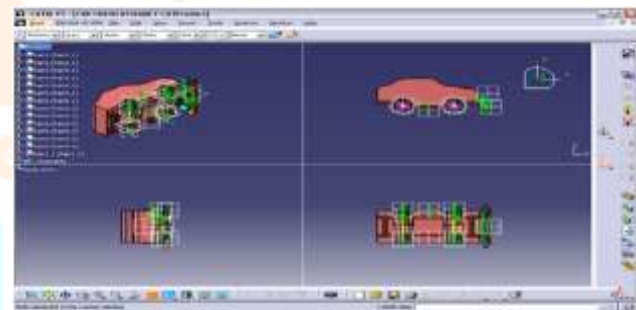


Fig: 5.17: Using Multi View Command

## 6 - ANALYSIS OF COMPLETE CAR

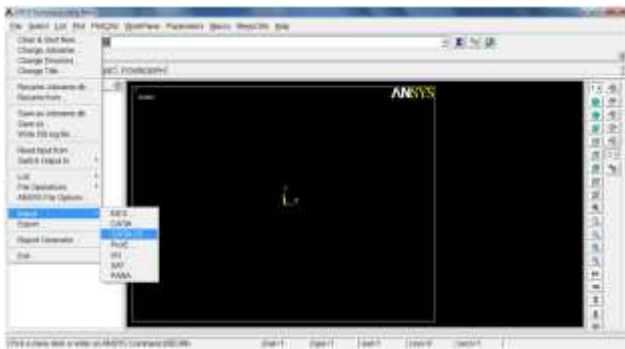
### 6.1 Procedure for FE Analysis Using ANSYS:

ANSYS is used for the car's analysis. In order to compete, assembly is not necessary; instead, moments must be applied at the specified axis. Establishing the assembly site.

### 6.2 Preprocessor

The following actions were taken at this stage:

- Use the ANSYS window to import the file.



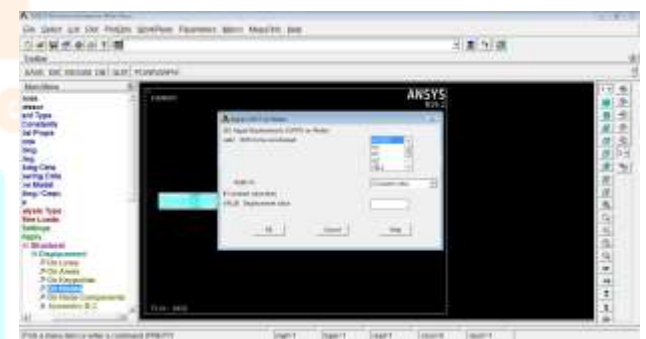
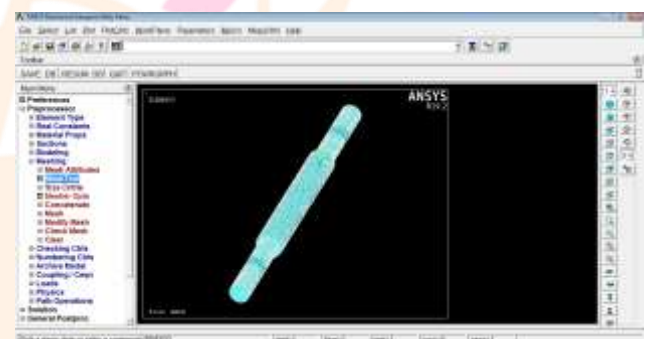
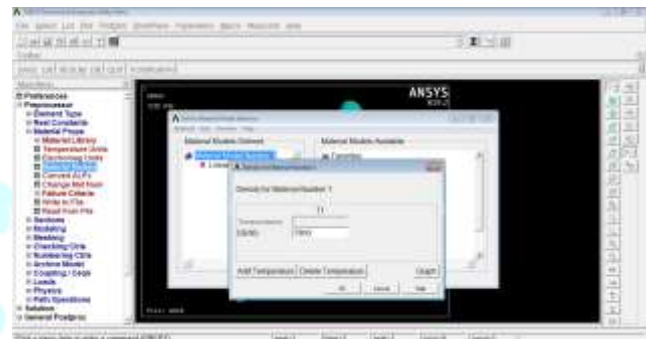
**Fig.6.1: Import panel in Ansys.**

**6.2.1 Meshing:**

A polygonal or polyhedral mesh that approximates a geometric domain is created through the process of mesh generation. Many times, the terms "grid generation" are used interchangeably. Examples of applications that are commonly seen on a computer screen are computational fluid dynamics and finite element analysis. Common sources include CAD, NURBS, B-rep, and STL (file format), but the input model form can take many different forms. With contributions from computer science, engineering, and mathematics, the field is very interdisciplinary. Prisms, hexahedra, pyramids, or tetrahedra must be present in three-dimensional meshes for finite element analysis. Any polyhedral can be used for the finite volume approach. Multi-block structured meshes—piecewise organised arrays of hexahedra—are typically needed for finite difference techniques.

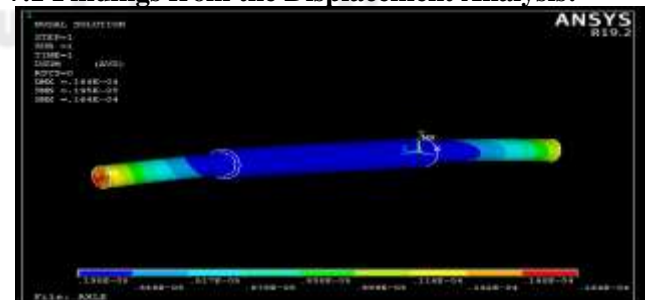
**6.3 Analysis Procedure of Car:**

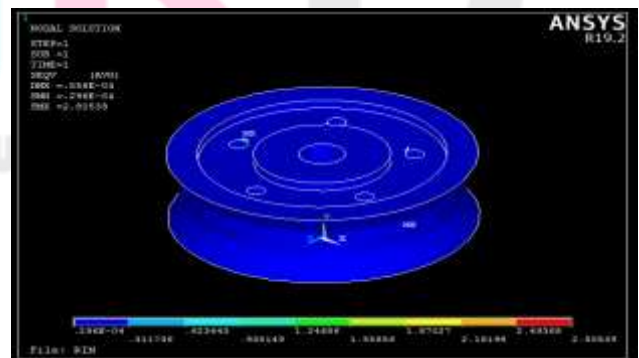
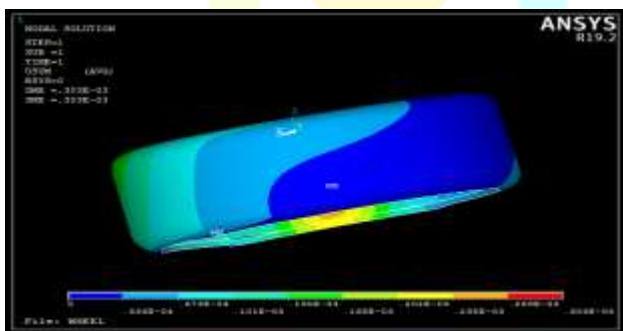
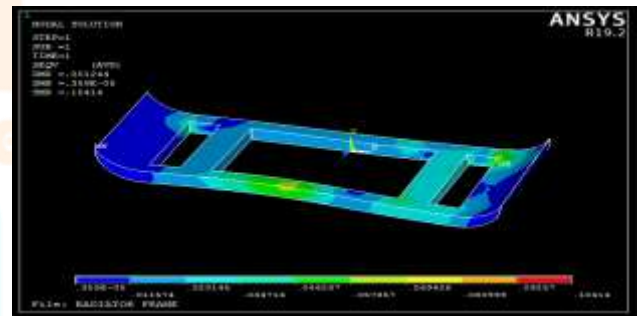
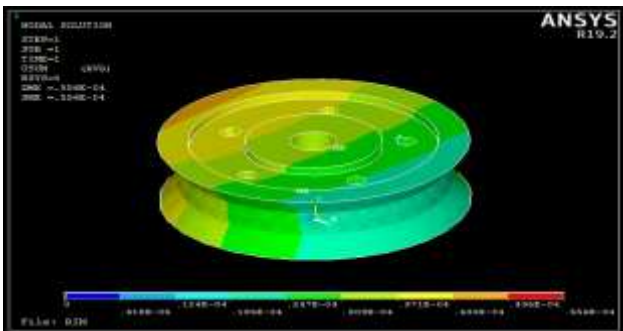
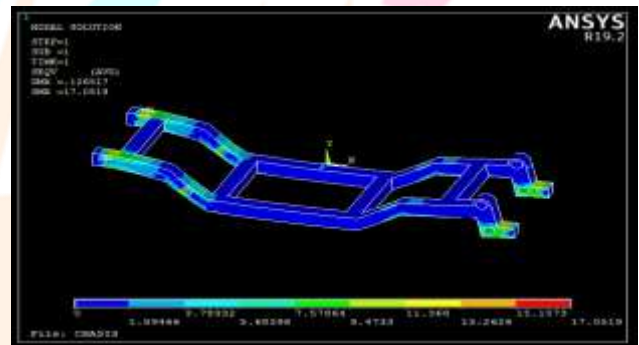
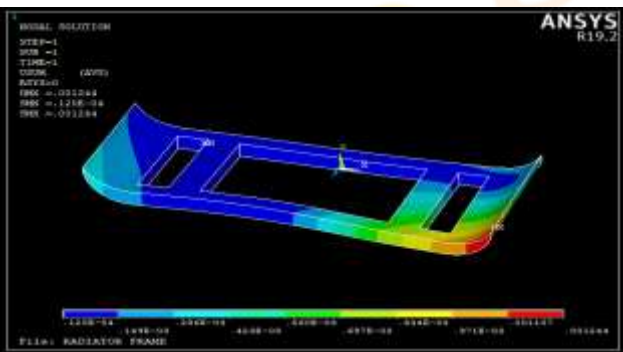
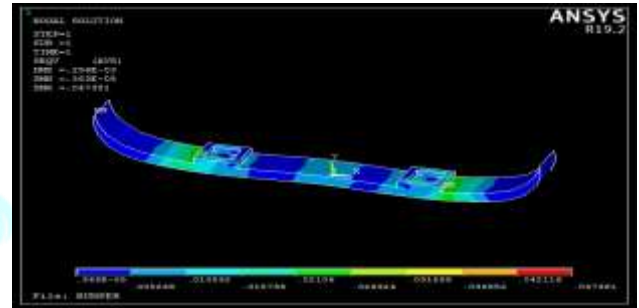
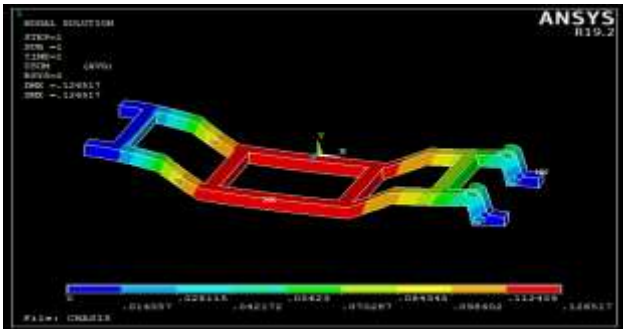
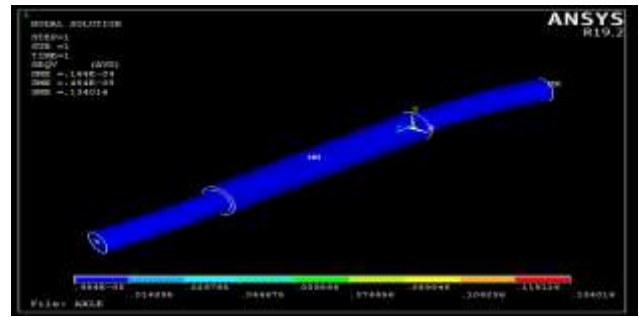
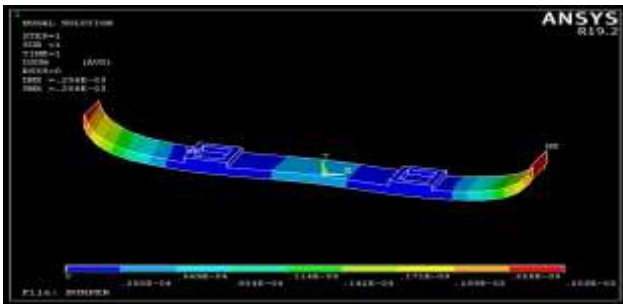
A tetrahedral element that exhibits quadratic displacement behaviour is ideal for modeling irregular meshes, as those generated by different CAD/CAM systems. Each of the ten nodes that make up the element has three degrees of freedom and can translate in the nodal x, y, and z dimensions. Additionally, the component may experience excessive strain, deflection, swelling, creep, stress stiffening, and plasticity.



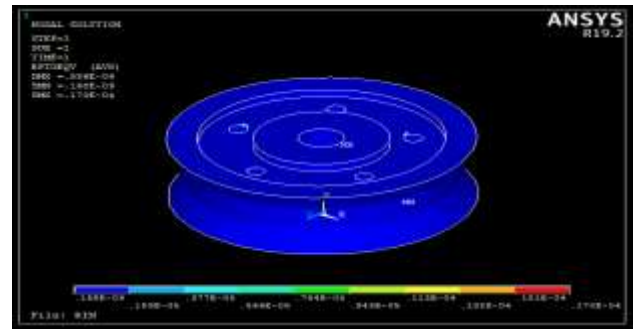
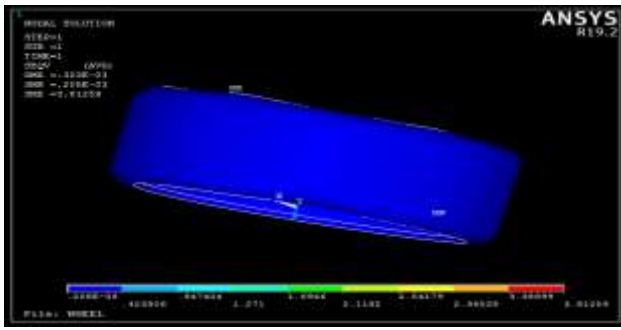
**7 - DISCUSSION ON ANALYSIS RESULT**

**7.1 Findings from the Displacement Analysis:**

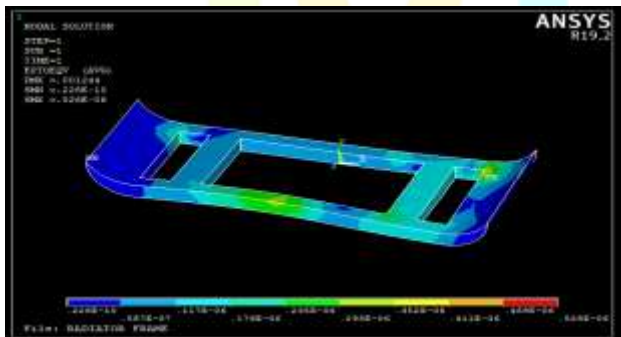
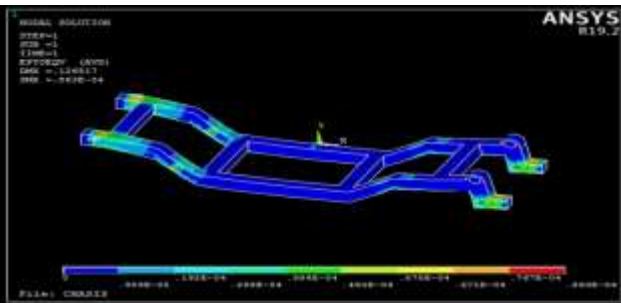
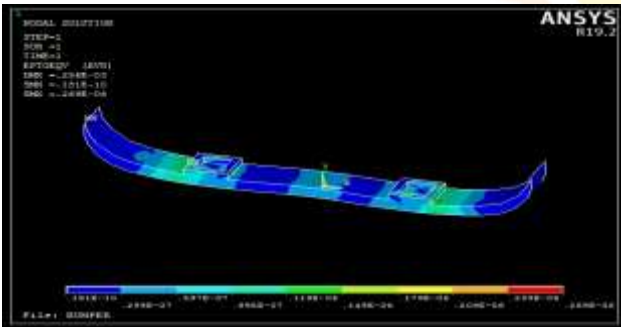
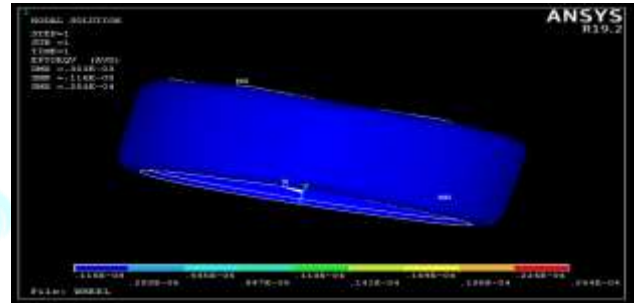
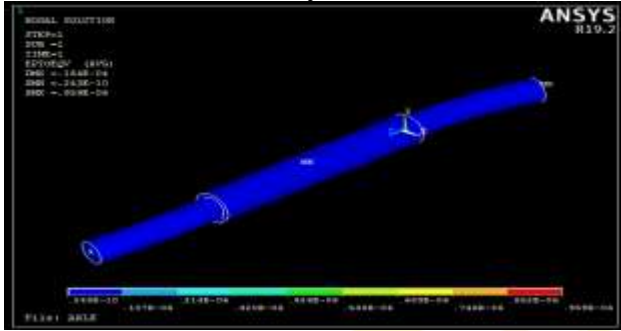




7.2 Results of Stress analysis:



**7.3 Results of Strain analysis:**



**8 - CONCLUSIONS & FUTURE SCOPE**

**Conclusions**

The work of this report, which examined the viability (strength, ability to sustain the signs, performance in the event) of the design structure, has met the project's goals.

The designs used in the creation of this project allow for operational flexibility. Because of this innovation, it is now more appealing. The statistics above demonstrate that our goal was to create a system that was both safe and secure. The dynamic behavior of the compressed air-driven system is modeled using a highly nonlinear approach. To find out how the control settings affect the dynamic response, parametric research is carried out.

Annoys is used to mesh and solve them, as seen in the above figures, and the displacement for the car chassis is extremely little. stresses created at the fastening place (minimum stress that is permissible); the strain value is below the yield point and extremely low in relation to the yield value of the materials provided. Thus, we may say that our design parameters are roughly right. As a result, the mechanism's design also operated perfectly. All of these facts strongly suggest that we have achieved our goal.

**Future Scope**

- The car is spared from actual destruction.
- Simulated analysis becomes cost-effective.
- The same FEA model can be evaluated with different models.
- The findings are used to evaluate the design's crashworthiness and look into potential design enhancements.
- It is clear from this study that the yield strength and yield material qualities, in addition to rail thickness, have a major impact on the vehicle's load behavior.

- For members with the same thickness and material yield, additional research can be conducted using various geometrical shapes with variable moments of inertia.
- Additionally, by making the bumper lighter, these designs may improve fuel efficiency.

## 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.
- P. Ravinder Reddy, and Thota Harish, "Design and crash analysis of passenger car frontal bumper beam using Hypermesh and Radioss," International Journal of Emerging Research in Management & Technology, ISSN: 2278-9359, Volume-4, Issue-12.
- Chunke Liu, Xinning Song, and Jiao Wang, "Simulation analysis of car front collision based on ls-dyna and hyper works," Journal of Transportation Technologies, Vol. 4, Issue 3, June 2014, pp. 337-342.
- R. Ranjithkumar, J.P. Ramesh, Modeling and Analysis of a Car Bumper Using Various Materials by FEA Software, Journal of Chemical and Pharmaceutical Sciences ISSN: 0974-2115, Special Issue 9: April 2015, pp 294 - 298.
- A.R. Mortazavi Moghaddam, M. T. Ahmadian, Design and Analysis of an Automobile Bumper with the Capacity of Energy Release Using GMT Materials, World Academy of Science, International Journal of Mechanical and Mechatronics Engineering, Vol:5, No:4, 2011, pp 865 - 872.
- Lin C. S., Chou K. D. and Yu C. C., Numerical simulation of vehicle crashes, Appl. Mech. Mat. 590, 135 (2014).
- Chunk Liu, Xinning Song, Jiao Wang, Simulation Analysis of Car Front Collision Based on LS-DYNA and Hyper Works, Journal of Transportation Technologies, 2014, 4,337- 342
- Yoshiaki Nakazawa, Kenji Tamura, Michitaka Yoshida Katsutoshi Takagi and Mitsukoshi Kano Development of Crash-Box for Passenger Car with High Capability for Energy Absorption.
- Gangadhar Biradar, Anjan Babu V. An Automotive Crash Box Performance Analysis and Simulation during Frontal Rigid Barrier Crash.