A PROJECT REPORT

ON

"CAR CRASH ANALYSIS AND OPTIMIZATION OF VEHICLE FRONT"

Submitted by

ANSARI AZIM MD SHARIF

KHAN MOHD ASIF SAYEEDUDDIN

ANSARI AMIR MOHD RASHID

PATEL AKRAM MASTAN

In partial fulfillment for the award of the Degree

Of

BACHELOR OF ENGINEERING

IN

MECHANICAL ENGINEERING

UNDER THE GUIDANCE

Of

Prof. SHAKIL TADAVI

DEPARTMENT OF MECHANICAL ENGINEERING

ANJUMAN-I-ISLAM

KALSEKAR TECHNICAL CAMPUS NEW PANVEL,

NAVI MUMBAI – 410206

UNIVERSITY OF MUMBAI

ACADEMIC YEAR 2016-2017

ANJUMAN-I-ISLAM

KALSEKAR TECHNICAL CAMPUS NEW PANVEL

(Approved by AICTE, recg. By Maharashtra Govt. DTE,

Affiliated to Mumbai University)

PLOT #2&3, SECTOR 16, NEAR THANA NAKA, KHANDAGAON, NEW PANVEL,NAVI MUMBAI-410206, Tel.: +91 22 27481247/48 * Website: www.aiktc.org

CERTIFICATE

This is to certify that the project entitled

"CAR CRASH ANALYSIS AND OPTIMIZATION

OF

VEHICLE FRONT"

Submitted by

ANSARI AZIM MD SHARIF

KHAN MOHD ASIF SAYEEDUDDIN

ANSARI AMIR MOHD RASHID

PATEL AKRAM MASTAN

To the Kalsekar Technical Campus, New Panvel is a record of bonafide work carried out by him under our supervision and guidance, for partial fulfillment of the requirements for the award of the Degree of Bachelor of Engineering in Mechanical Engineering as prescribed by **University Of Mumbai**, is approved.

Project co-guide **Internal Examiner** **External Examiner**

(Prof. Shakil Tadavi)

Head of Department **Director**

(Prof.Zakir Ansari) (Dr.Abdul Razzak Honnatugi)

ANJUMAN-I-ISLAM

KALSEKAR TECHNICAL CAMPUS NEW PANVEL

(Approved by AICTE, recg. By Maharashtra Govt. DTE,

Affiliated to Mumbai University)

PLOT #2&3, SECTOR 16, NEAR THANA NAKA, KHANDAGAON, NEW PANVEL,NAVI MUMBAI-410206, Tel.: +91 22 27481247/48 * Website: www.aiktc.org

APPROVAL OF DISSERTATION

This is to certify that the thesis entitled

"CAR CRASH ANALYSIS AND OPTIMIZATION

OF

VEHICLE FRONT"

Submitted by

ANSARI AZIM MD SHARIF

KHAN MOHD ASIF SAYEEDUDDIN

ANSARI AMIR MOHD RASHID

PATEL AKRAM MASTAN

In partial fulfillment of the requirements for the award of the Degree of Bachelor of Engineering in Mechanical Engineering, as prescribed by University of Mumbai approved.

(Internal Examiner) **(External Examiner)**

___________________ ___________________

Date: ___________

ACKNOWLEDGEMENT

After the completion of this work, we would like to give our sincere thanks to all those who helped us to reach our goal. It's a great pleasure and moment of immense satisfaction for us to express my profound gratitude to our guide **Prof. SHAKIL TADAVI** whose constant encouragement enabled us to work enthusiastically. His perpetual motivation, patience and excellent expertise in discussion during progress of the project work have benefited us to an extent, which is beyond expression.

We would also like to give our sincere thanks to **Prof. ZAKIR ANSARI,** Head Of Department, from Department of Mechanical Engineering, Kalsekar Technical Campus, New Panvel, for their guidance, encouragement and support during a project.

I am thankful to **Dr. ABDULRAZZAK HONUTAGI,** Kalsekar Technical Campus New Panvel, for providing an outstanding academic environment, also for providing the adequate facilities.

Last but not the least I would also like to thank all the staffs of Kalsekar Technical Campus (Mechanical Engineering Department) for their valuable guidance with their interest and valuable suggestions brightened us.

ANSARI AZIM MD SHARIF
KHAN MOHD ASIF SAYEEDUDDIN
ANSARI AMIR MOHD RASHID
PATEL AKRAM MASTAN

ABSTRACT

An automotive bumper is an important member for safety point of view. Bumper with Fascia in front most part, are design such that, after an impact car can sustain without damage to vehicle safety systems. The intention of the project is to study the most important variable like material of the fascia and bumper, structure of the bumper and shape of the bumper beam in order to improve the crashworthiness during the collision and also study the behavior of the bumper system at different frequency by using CAD and CAE Software. More accents are given on selection of bumper material.

TABLE OF CONTENTS

LIST OF TABLE

LIST OF FIGURE

Chapter 1

INTRODUCTION

1.1 Introduction

Car accidents are happening every day. Most drivers are convinced that they can avoid such troublesome situations. Nevertheless, we must take into account the statistics – ten thousand dead and hundreds of thousands to million wounded each year. These numbers call for the necessity to improve the safety of automobiles during accidents. Automotive bumper system is one of the key systems in passenger cars. Bumper systems are designed to prevent or reduce physical damage to the front or rear ends of passenger motor vehicles in collision condition. They protect the hood, trunk, grill, fuel, exhaust and cooling system as well as safety related equipment such as parking lights, headlamps and taillights, etc. A good design of car bumper must provide safety for passengers and should have low weight. Different countries have different performance standards for bumpers. Under the International safety regulations originally developed in North America, a car's safety systems must still function normally after a straight-on pendulum or moving-barrier impact of 4.4 km/h to the front and to the front corners of 2.8 km/h at 20” above the ground with the vehicle loaded or unloaded. In North America (FMSS: Federal Motor Vehicle Safety Standards) Today, it is common to produce structural body parts via stamping or hydroforming. The structural and complexity potential of stamped and hydro formed steel parts is very high however, machine investment costs for these manufacturing processes are significant. In addition, the tools can be complicated and costly. A process offering an economic alternative to the above mentioned is roll forming.

1.1.1 Overview and role of bumper system:

Bumper has been an important feature in protecting the vehicle from serious damage to the car component in a low speed collision. Especially when the collision causing damage to the expensive-to-repair part likes fender, hood and intercooler. Bumper is also involves in improving the performance of the car. Bumper size and the aerodynamic feature of the bumper are the important aspects in lowering the coefficient of drag, CD. The efficient bumper design will also increase the down force of the car when it accelerates to give more grips to the tire and the road. This will give a good handling to the driver ever in high speed driving.

The car bumper is designed to prevent or reduce physical damage to the front and rear ends of passenger motor vehicles in low-speed collisions. Automobile bumpers are not typically designed to be structural components that would significantly contribute to vehicle crashworthiness or occupant protection during front or rear collisions. It is not a safety feature intended to prevent or mitigate injury severity to occupants in the passenger cars. Bumpers are designed to protect the hood, trunk, grille, fuel, exhaust and cooling system as well as safety related equipment such as parking lights, headlamps and taillights in low speed collisions.

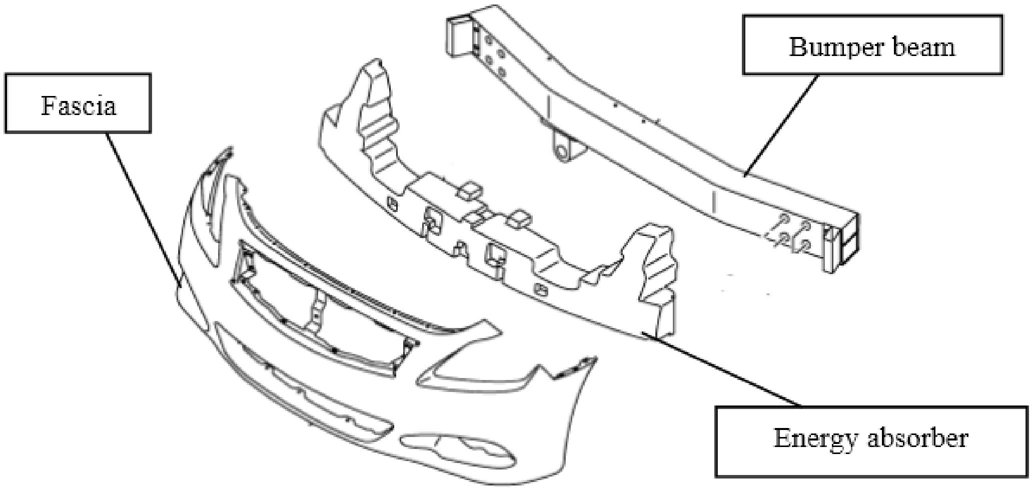


Fig 1.1 Common Bumper System

Bumpers beam are made of heavy sheet metal and are mounted on the front and rear of the car. Bumpers are bent and formed into specific shapes in order to absorb and deliver momentum during a collision. In the event of a collision, the bumper absorbs some of the impact, which decreases damage to the car and its occupants. It also protects the front of the car by diverting all of the car's momentum to the object with which it has collided. The bumper beam is mounted to the car's chassis with special impact absorbers. These shock absorbers are often spring loaded. In slow speed collisions, this allows the bumper to compress, and then extend back to its original position. All bumpers are designed to absorb the energy of the impact. They do this through a series of valves and air chambers. Some car bumpers have hydraulic chambers. In the event of a collision, the absorption unit allows air and/or hydraulic fluid to pass through small openings. Forcing the air/fluid through the valve openings absorbs the energy from the collision. The bumper's job is to minimize damage, primarily to the occupants of the vehicle and to the vehicle itself.

1.1.2 Crash simulation of bumper for bumper design

A crash simulation is a virtual recreation of a destructive crash test of a car or a highway guard rail system using a computer simulation in order to examine the level of safety of the car and its occupants. Crash simulations are used by automakers during computer-aided engineering (CAE) analysis for crashworthiness in the computer-aided design (CAD) process of modelling new cars. During a crash simulation, the kinetic energy, or energy of motion, that a vehicle has before the impact is transformed into deformation energy, mostly by plastic deformation (plasticity) of the car body material (Body in White), at the end of the impact.

Data obtained from a crash simulation indicate the capability of the car body or guard rail structure to protect the vehicle occupants during a collision (and also pedestrians hit by a car) against injury. Important results are the deformations (for example, steering wheel intrusions) of the occupant space (driver, passengers) and the decelerations (for example, head acceleration) felt by them, which must fall below threshold values fixed in legal car safety regulations. To model real crash tests, today's crash simulations include virtual models of crash test dummies and of passive safety devices (seat belts, airbags, shock absorbing dash boards, etc.). Guide rail tests evaluate vehicle deceleration and rollover potential, as well as penetration of the barrier by vehicles.

To design a bumper system it has to be tested for different criterion as given below.Analysis performed on bumper system is

- Modal Analysis
- Explicit Dynamic Analysis
- Fatigue Analysis

1.1.3 STATIC 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 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 six kinematic degrees of freedom that are one node can move in three linear directions under translation and can rotate three independent axes.

If the nodes move during a crash simulation, the connected elements move, stretch, and bend with their node, which causes 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.

where vector (mass times acceleration vector) collects the inertia forces at the nodes, collects the external nodal loads, and 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).

1.1.4 TIME ANALYSIS

A crash simulation uses time discretization as well to separate the continuous changes in time into very small, usable segments. The dynamic equations of motion hold at all times during a crash simulation and must be integrated in time, t, starting from an initial condition at time zero, which is just prior to the crash. According to the explicit finite difference time integration method used by most crash codes, the accelerations, velocities, and displacements of the body are related by the following equations.

In these equations the subscripts n±1/2, n, n+1 denote past, present, and future times, t, at half and full-time intervals with time steps and, respectively.

1.2 Problem Statement

Bumper plays a big role to prevent more critical damage of the car. This problem based on the material has been used to make this bumper. The types of material play are a big role to influence the front car bumper condition after crashing.

This study deals with material type that best suit to be a composite car bumper in order to energy absorption using supplementary attachment to optimize its performance in terms of energy and fulfill the aspect of strength, lightweight, and impact absorption along with the fatigue life of bumper.

1.3 Objectives

The objectives of this project are:

- The objective of this project is to estimate the expected fatigue lifetime on the basis of degree of damage and time observations.
- To design a composite car bumper by using CAD software that is DS Solidworks.
- To perform an impact analysis of composite car bumper by using ANSYS software.
- To propose a suitable composite material for the car bumper.
- To study the behavior of bumper system under different frequency.

Chapter 2

LITERATURE SURVEY

T. Ambati, S. Srinath, P. Veeraraju [1], presented the dynamic analysis of a vehicle using LS DYNA software, also proposed some alternative materials in order to reduce impact shock and to increase the toughness of the body parts and decrease the weight. The main purpose was the reduction of the weight of the vehicle. The Test model 2 (new material) will experience lower forces as a result of its lower weight than the test model 1(NCAC material). The model used here was that of a Chevrolet C1500 pick-up truck. The results obtained by the simulation were then Validate by comparing it with the test results of the same test performed by the NCAC (National Crash Analysis Center).

Byeong S.K, K. Park, Youn-Min Song [2], present the crash analysis of upper body and sub frame for NEV electric car. NEV Vehicle's front platform assembly behavior when subjected to a frontal crash is described in this article. The frontal crash of the integrated car system is successfully simulated in LS-DYNA. According to the basic principle of the dynamic non-linear finite element method, the basic crash describe equation and FE discretized equation are established. The steel vehicle body frame for EV crash impact analysis performed for the Al body for the upper body of the EV, model is based on the frame model, static, dynamic, impact analysis was carried out over an early model of crash energy absorption and passenger safety and has made good results in terms of content, EV for the upper body, and sub frames crash analysis of the regulations (FMVSS 208) for body-frame through analysis of the conflict for an electric car will be optimized design.

Erika Fechova, Jozef Kmec, Alena Vagaska Drazan Kozak [3], present the calculations of deformation work and biomechanical limits for human organism (especially head injuries) at the frontal crash of the car into the barrier using the material properties of individual types of steel. Based on the experimental and theoretical results deformation member of the most suitable steel is proposed which protects solid parts of the construction at collisions from damaging and at the same time suitable biomechanical criteria are set, K material constant and n strain hardening exponent were obtained from $y = k\emptyset^{n}$ by using Matlab, and if increase of K material constant and n strain hardening exponent were recorded at the higher strain rate. But at TRIP steel and DIN 1.4301 steel these properties occurred at the lower strain rate. The samples were taken 90o angle - wise. Mechanical properties are influenced up to a certain interval of deformation. While n value increases, steel sheet is more suitable for cold forming. It can be seen from these findings that production costs can be saved on steel sheets at production of constructions by reducing their dimensions (particularly thickness) at keeping very good strength and safety properties. Not only these properties are improved, but the mass of a car and fuel consumption are reduced, and control of the car is improved too. The task is to create the lightest car body that can be much stiffer and compared with current car bodies from the point of view of strength.

Saeed Barbat, Xiaowei Li, PriyaPrasad [4], presents finite element models of an "average" SUV and an "average" passenger vehicle were used to explore the effects of geometry, stiffness and mass in front-to-side impact simulations. A design of experiments methodology involving Latin Hypercube sampling was employed to select the appropriate number of simulations and the design levels of each of the design variables that should be incorporated in each simulation. Five design variables, the SUV rail height, rail thickness, mass, bumper width and bumper metal thickness were chosen. Thoracic Trauma Index (TTI) and pelvis acceleration of the SID dummy responses were selected for the system responses. These responses were normalized by their baseline corresponding values. The main effect plots were generated to identify the significance of individual design variable. The responses were characterized by quadratic polynomial surfaces. Pair-wise comparisons of the effects of the design variables were used to assess their individual influence on TTI and pelvis acceleration. The pair-wise comparisons were based on the response surfaces generated from the 21 FE simulations. When a pair of design variables was compared, the remaining design variables were set to their BASELINE levels.

Touraj Gholami, Jurgen Lescheticky, Ralf Pabmann [5], presents Passive safety simulation is a well-established tool in the development process of automobiles at BMW Group. Based on the demands on crash analysis software, which has been defined in a crash vision 2004, ABAQUS/Explicit has been tested for crashworthiness simulation. The recent introductions of a general contact capability and a new spot weld modeling technique have been key to enabling this testing. First experiences of component-based applications and of whole car crash analysis exemplify the strong architecture and existing complement of robust features in ABAQUS/Explicit, and demonstrate its general feasibility for crash simulation.

General feasibility has been proved in many component applications and in whole car crash analysis. Productive usage of ABAQUS/Explicit for head impact analysis has been started. The development of needed features for occupant protection simulation, an efficient parallel version, and a new barrier model are the next development steps before introducing ABAQUS/Explicit into the productive crashworthiness simulation at BMW Group.

Wlodzimierz Bedkowski [6], presents an overview of the assessment of fatigue life of machine parts and structures under service loads. At the same time, the complexity of the problem and its multithreading is shown. In the author's opinion, the main problem is to formulate an appropriate fatigue criterion based on a load parameter (stress, strain, energy or other) which should allow including as many factors determining fatigue life of machine components and structures under operating conditions as it is necessary to determine the extent of applicability to a wide range of different structure materials. Such an accepted hypothesis, as a part of the overall assessment of the fatigue life algorithm, must be verified on the basis of results of fatigue tests carried out, in the first place, on samples subject to random multi axial loadings and then on the real construction components

Jadav Chetan, Panchal Khushbu, Moulvi Nauman [7], presents the investigations that have been made on the different fatigue analysis techniques of a vehicle suspension system. A number of analytical and experimental techniques are available for the fatigue analysis of the vehicle suspension system. Determination of the different analysis around different condition in vehicle suspension system has been reported in literature. An attempt has been made in the article to present an overview of various techniques developed for the fatigue analysis of component of suspension system.

Hariharan Sankarasubramanian Sudipto Mukherjee [8], presents impact on automobile, a pedestrian suffers multiple impacts with the bumper, hood and the windscreen. Optimization of the car front using a scalar injury cost function has been demonstrated. The results for impacts simulated in MADYMO show good co-relation with Euro-NCAP ratings for existing vehicles. Optimization of the car front to minimize the injury cost converges to vehicle profiles with features known from earlier studies to be pedestrian friendly. A method to design car fronts for pedestrian safety is evolved.

C. SADHASIVAM & S. JAYALAKSHMI [9], presents the car body structure fatigue damage is mainly caused by the vehicle dynamic response, which is usually expressed as stress or strain time history. Stress/Strain calculations for fatigue life estimation can be performed in the time domain or Frequency domain. The detail car body stress analyses with ANSYS were performed based on 3Dlsdyna and mode analysis is also performed here. Modal analysis is usually used to determine the natural frequencies and mode shapes of car body structure. Vibration and Crash Analysis of Car Body using Ansys is carried out and including dynamic, static, crash analysis and so on. The main objective of the project is to find fundamental characteristics like frequency, stress and displacement for different material and velocity influence in the car body structure. If the fundamental frequency is increased beyond our designable value, if possible engineering changes can be made in the car body structure. In the car body structure the factor of safety value may be a half of the factor of safety of the existing material. Car body design can be modeled by using Pro-Engineer modeling software. Analysis can be analyzed by using Ansys and 3DLSDYNA.

Manideep Kumar Vandanapu, Ratnakar Pandu [10], Presents the improve crashworthiness of the bumper beam. Crashworthiness is the ability of the bumper beam to prevent occupant injuries in the event of an accident and this is achieved by minimizing the impact force during the collision. This study was investigated the difference of producing bumper beam using roll forming method compare to stamping method. Based on observations design improvements will be made in terms of shape, size and or material based on design modification objectives. The study was focused on existing design performance, advantage and limitations. Modified front bumper design will be tested using FEM software for impact loads as per international standards.

Mr. Siddique Ahmed, Mr. Gajendra Patil,[11], presents the Optimize of Bumper shape to increase pedestrian safety. The main focus will be to design a bumper shape to reduce lower and upper leg injuries. The system will be analyzed using computational codes like LS Dyna and Optimization tools like HyperStudy. This study indicates that the increase in the bumper area to optimize its shape not always a solution. If we just keep on increasing the bumper size then the bumper becomes very weak and fails during the pedestrian collision. The bumper shape optimization is done by a new method called HPERMORPH technique.

Intaek Lee, Katsutoshi Ainaka, Junji Saiki Minoru, Yubuchi,[12], presents the assembling, executing, managing and interpreting the results has prevented this level of parametric study. StudyWizard, developed by Altair Engineering, is a software technology specifically designed to automate and extract meaningful design information from parametric analytical studies. Using StudyWizard, this paper will demonstrate a strong relationship between aggressivity and injury revealed by FARS data set for an oblique offset vehicle to vehicle crash. For this paper, a full-sized sedan compatibility study is performed for impacts with a light truck and van. The aggressivity characteristics; vehicle mass, stiffness and stackup will be discussed. To conclude, compatibility improvements will be examined.

Shokri El housini,[13], presents during a vehicles frontal crash, passenger jeopardize high acceleration and energy opposite to the mass follow direction of their bodies. This fact causes high injuries to the passenger whole body, head, neck, chest, legs. We started thinking of reducing this deceleration effect on passenger during the crash. This is because succeeding in reducing mass deceleration effect on the passenger bodies will lead to save passengers from serious injuries. Increasing the length of front bumper crash boxes was a method to improve the impact energy absorption. However, increasing the geometry (the length of the crush box) of the front bumper assembly will lead to an endless chain of structure sub assembly changes are methodology works on combining different types of materials and design optimization to control the crash deceleration while maintaining the geometry of the front bumper. This methodology works on absorbing or discharging the energy of impact before the energy being transmitted in full to the passenger. On the other work, we protect the passenger from excessive energy which is generated by the crash before its reach them. By having the control over these variables, our vehicle becomes safer to ensure the safety for everyone, this methodology has a high potential to be applied to improve the side impact crash worthiness as well.

Steven Hoffenson, Panos Papalambros, Michael Kokkolaras, Matthew Reed,[14], presents A major challenge in automotive design is the creation of safe vehicles with minimal environmental impact. This study presents a modeling framework for evaluating and optimizing body designs for improved occupant safety and fuel efficiency. Vehicle body mass is considered as the main link between safety and sustainability. The modeling framework includes frontal crash energy absorption, restraint system design and fuel economy. Preliminary results indicate a Pareto-optimal curve between safety and sustainability may exist when other factors are not taken into account. A more complete modeling framework that includes more sophisticated sustainability and safety metrics, cost and other market considerations, is suggested in order to support better design and regulatory decisions.

Sankarasubramanian Hariharan, Sudipto Mukherjee, Anoop Chawla,[15], This paper presents an attempt to optimize the front-end profile of a passenger car based on pedestrian-vehicle multibody crash simulations for pedestrian safety. A single unitary measure for injuries to human body in the form of "injury cost" is used. It is representative of the loss to a human due to a crash including cost for partial impairment and an indicative cost for death. Vehicle front-profile optimization problem is formulated as a single objective minimization problem using by Genetic Algorithms implemented on MATLAB. Car front-end profile is described by 22 variables. A MADYMO ellipsoid based model of a 50th percentile male pedestrian at 50% gait cycle impacted laterally with front of vehicle at 40kmph. Injury cost converged within 10 generations to an optimal value 60% less than the lowest injury cost among ten existing car profiles. Convergence towards one profile indicates that existence of one front-profile which is global optimal and not a local minimum in the range considered. This paper presents an attempt to optimize the front-end profile of a passenger car based on pedestrian-vehicle multibody crash simulations for pedestrian safety. A single unitary measure for injuries to human body in the form of "injury cost" is used. It is representative of the loss to a human due to a crash including cost for partial impairment and an indicative cost for death. Vehicle front-profile optimization problem is formulated as a single objective minimization problem using by Genetic Algorithms implemented on MATLAB. Car front-end profile is described by 22 variables. A MADYMO ellipsoid based model of a 50th percentile male pedestrian at 50% gait cycle impacted laterally with front of vehicle at 40kmph. Injury cost converged within 10 generations to an optimal value 60% less than the lowest injury cost among ten existing car profiles. Convergence towards one profile indicates that existence of one front-profile which is global optimal and not a local minimum in the range considered.

K. Preston White, Jr., W. Thomas HollowelI H. Clay Gabler, Walter D. Pilkey,[16], The purposes of this paper are to (1) outline the current implementation of the SSOM and (2) to report a benchmark application of the simulationoptimization approach to the design of a passenger vehicle for maximum safety performance in frontal collisions. The SSOM was initially developed at the Ford Motor Company, during the period from 1975 through 1978 [l-3]*. Since that time, extensive modifications and refinements to the original program package have been implemented by researchers at the University of Virginia [4-9]. Major modifications of the SSOM include the incorporation of new AISbased biomechanical transforms and new accident data bases using data from the National Crash Severity Study (NCSS) file. Details of the current biomechanical transforms and accident databases are given here. In addition, new features for univariate and multivariate sensitivity analysis and for accidentdistribution report generation are demonstrated.

M. Pohlaka, J. Majak ,M. Eerme,[17], Aim of the current study is to design the extra frontal protection system of a car satisfying the requirements of directive 2005/66/EC of the European Parliament and of the Council. The car frontal protection system is treated as additional energy absorbing element. An analysis of a car-pedestrian crash situation is performed by use of explicit Finite Element Analysis solver and the following stiffness analysis with implicit Finite Element Analysis solver. For modeling response surface and search for optimal solution the LS-OPT software is utilized and some comparative simulations are performed by use of MATLAB. The experimental validation of the models designed is performed.

Hao Chen, Yali Yang and Liangjie Wang,[18], focused on Energy absorption performance is one of the most important indexes in the vehicle safety during impact. Research on the car frontal structure energy performance and structure optimization was conducted in this paper. Whole vehicle model was established by HyperMesh and simulated in LS-DYNA. Simulation results indicated that modification was need for the original structure to meet requirement. Based on simplified whole vehicle model, orthogonal design optimization was implemented, including bumper cross beam material (A), bumper cross beam thickness (B), energy absorber groove distance (C), and front longitudinal beam groove number (D), with 3 levels for each factor. The best option was B3D1A3C3 was gained by using range analysis and integrated balance method. Simulation results showed that both front and total energy absorption were improved. The optimized structure increased front energy absorption to 51.1%, which can meet the industry requirement.

Chapter 3

METHODOLOGY

3.1 Introduction

To achieve the required goal, we have started with idea generation and concept. Different related papers been searched out and the approach towards the analysis for such kind of systems is gathered as well as the different difficulties faced by the different authors in their respective work has been studied. From this literature review the further work possibilities and difficulties to be overcome, and further scope of work has been identified

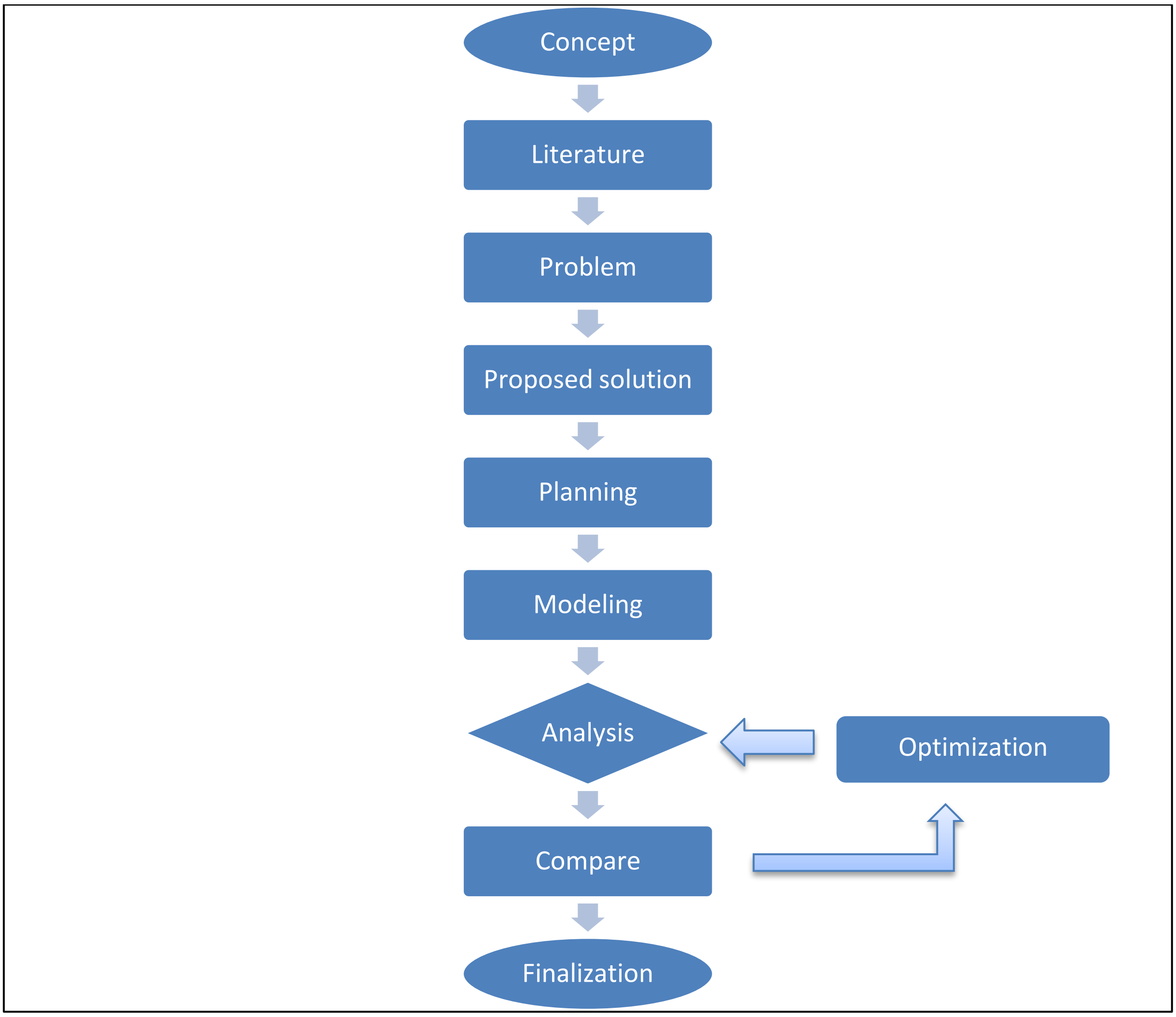


Fig 3.1 Flow chart of methodology

From the available literature and methods used for car crash analysis has been found that Several numerical methods can predict the behavior of material deformation under different load cases and these results can be simulated which can give us an idea of the actual crash and its behavior.

3.2 FEA approach

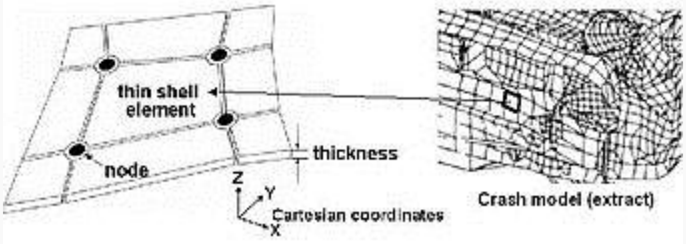


Fig 3.2 Subdivision of metal exterior

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. Another approach to crash simulations is performed by application of Macro Element Method. The difference between two mentioned above methodologies is that the structure in case of Macro Element Method consists of smaller number of elements. The calculation algorithm of structure deformation is based on experimental data rather than calculated from partial differential equations.

Pam-Crash started crash simulation and together with LS-DYNA is a software package which is widely used for application of Finite Element Method. This method allows detailed modeling of a structure, but the disadvantage lies in high processing unit requirements and calculation time. The Visual Crash Studio uses Macro Element Methodology.

In comparison with FEM it has some modeling and boundary condition limitations but its application does not require advanced computers and the calculation time is incomparably smaller. Two presented methods complement each other. Macro Element Method is useful at early stage of the structure design process while Finite Element Method performs well at its final stages. Before starting the modeling analysis for this project several mathematical techniques will be used to predict the behavior under different load cases the method that will be used are as given below.

The mathematical method that can be used for the analysis will be the

1) Numerical methods which involves time step and mass scaling as well as the contact analysis.
2) The connecting technique modelling which is useful for deformable and useful modelling.
3) Material modelling

3.3 Fatigue life

Fatigue life is a function of the magnitude of the fluctuating stress, geometry of the specimen and test conditions. An S-N diagram is a plot of the fatigue life at various levels of fluctuating stress.

ASTM defines fatigue life, as the number of stress cycles of a specified character that a specimen sustains before failure of a specified nature occurs. For some materials, notably steel and titanium, there is a theoretical value for stress amplitude below which the material will not fail for any number of cycles, called a fatigue limit, endurance limit, or fatigue strength.

Engineers have used any of three methods to determine the fatigue life of a material: the stress-life method, the strain-life method, and the linear-elastic fracture mechanics method. One method to predict fatigue life of materials is the Uniform Material Law (UML). UML was developed for fatigue life prediction of aluminum and titanium alloys by the end of 20th century and extended to high-strength steels, and cast iron.

3.4 Stress-cycle (S-N) curve

In high-cycle fatigue situations, materials performance is commonly characterized by an S-N curve, also known as a Wohler curve. This is a graph of the magnitude of a cyclic stress (S) against the logarithmic scale of cycles to failure (N).S-N curves are derived from tests on samples of the material to be characterized (often called coupons) where a regular sinusoidal stress is applied by a testing machine which also counts the number of cycles to failure. This process is sometimes known as coupon testing. Each coupon test generates a point on the plot though in some cases there is a run out where the time to failure exceeds that available for the test (see censoring). Analysis of fatigue data requires techniques from statistics, especially survival analysis and linear regression. The progression of the s-n curve can be influenced by many factors such as corrosion, temperature, residual stresses, and the presence of notches. the good man-line is a method used to estimate the influence of the mean stress on the fatigue strength.

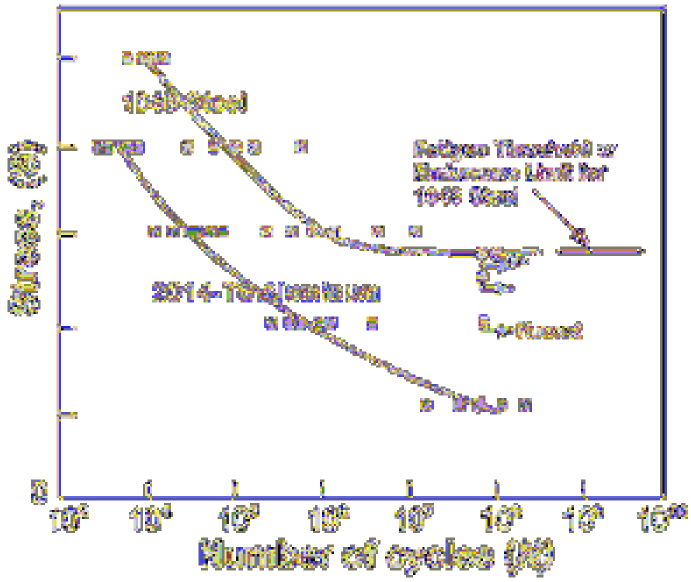


Fig 3.4 Stress cycle (S-N) curve

3.5 Fatigue crack formation analysis

Predicts cycles to failure based purely on material data of fatigue specimens. Even though the total fatigue life includes the growth of cracks, cracks are not explicitly modelled.

Fatigue performance of structures is more accurately described as follows:

1) The presence of stress risers such as holes, manufacturing errors, corrosion pits, and maintenance damage serve as nucleation sites for fatigue cracking.
2) During service, sub-critical nucleate from these sites and grow until catastrophic failure, i.e. unstable crack growth occurs.
3) From an economic point of view, a costly component cannot be retired from service simply on detecting a fatigue crack.
4) Hence, reliable estimation of fatigue crack propagation and residual life prediction, combined with inspections, are essential so that the component can be timely serviced or replaced.

Chapter 4

MODELLING

4.1 Introduction

Modeling is the use of computer systems to aid in the creation, modification, analysis, or optimization of a design.

Modeling and simulation (M&S) refers to using models physical, mathematical, or otherwise logical representation of a system, entity, phenomenon, or process as a basis for simulations methods for implementing a model (either statically or) over time to develop data as a basis for managerial or technical decision making. Modelling & Simulation supports analysis, experimentation, and training. As such, M&S can facilitate understanding a system's behavior without actually testing the system in the real world. For instance, to determine which type of spoiler would improve traction the most while designing a race car, a computer simulation of the car could be used to estimate the effect of different spoiler shapes on the coefficient of friction in a turn. Useful insights about different decisions in the design could be gleaned without actually building the car. In addition, simulation can support experimentation that occurs totally in software, or in human in the loop environments where simulation represents systems or generates data needed to meet experiment objectives. Furthermore, simulation can be used to train persons using a virtual environment that would otherwise be difficult or expensive to produce.

The use of M&S within engineering is well recognized. Simulation technology belongs to the tool set of engineers of all application domains and has been included in the body of knowledge of engineering management. M&S helps to reduce costs, increase the quality of products and systems, and document and archive lessons learned.

4.2 Types of Modeling

There are many types in modeling some of them are listed below which we used.

4.2.1 3D Wire Frame Modeling

A wire-frame model is a visual presentation of a 3-dimensional (3D) or physical object used in 3D computer graphics. It is created by specifying each edge of the physical object where two mathematically continuous smooth surfaces meet, or by connecting an object's constituent vertices using straight lines or curves. The object is projected into screen space by drawing lines at the location of each edge. The term wire frame comes from designers using metal wire to represent the three-dimensional shape of solid objects. 3D wire frame allows constructing and manipulating solids and solid surfaces. The 3D solid modeling technique efficiently draws higher quality representations of solids than the conventional line drawing.

Using a wire-frame model allows visualization of the underlying design structure of a 3D model. Traditional two-dimensional views and drawings can be created by appropriate rotation of the object and selection of hidden line removal via cutting planes.

Since wire-frame renderings are relatively simple and fast to calculate, they are often used in cases where a high screen frame rate is needed (for instance, when working with a particularly complex 3D model, or in real-time systems that model exterior phenomena). When greater graphical detail is desired, surface textures can be added automatically after completion of the initial rendering of the wire frame. This allows the designer to quickly review solids or rotate the object to new desired views without long delays associated with more realistic rendering.

4.2.2 Surface Modeling

A mathematical technique for representing solid appearing objects by stretching a surface over it with 3D curves. It essentially describes external aesthetics and surface boundaries of an object.

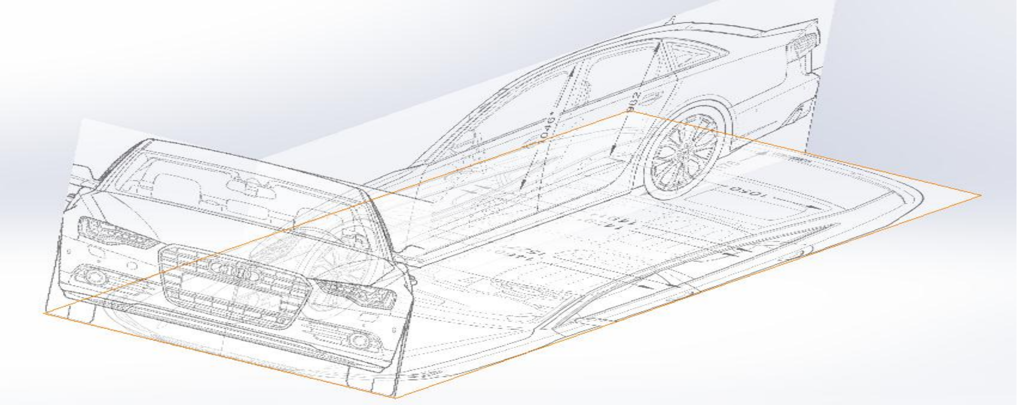


Fig 4.1 Vector image of vehicle

A mathematical technique for representing solid appearing objects. Surface modeling is a more complex method for representing objects than wireframe modeling, but not as sophisticated as solid modeling. Surface modeling is widely used in CAD (computer-aided design) for illustrations and architectural renderings. It is also used in 3D animation for games and other presentations.

Although surface and solid models appear the same on screen, they are quite different. Surface models cannot be sliced open as can solid models. In addition, in surface modeling, the object can be geometrically incorrect; whereas, in solid modeling, it must be correct.

Surface modelling is a technique for engineering Freeform Surfaces with a CAD system. The technology has encompassed two main fields. Either creating aesthetic (class A surfaces) that also perform a function; for example, car bodies and consumer product outer forms, or technical surfaces for components such as gas turbine blades and other fluid dynamic engineering components. CAD software packages use two basic methods for the creation of surfaces. The first begins with construction curves (splines) from which the 3D surface is then swept (section along guide rail) or meshed (lofted) through. The second method is direct creation of the surface with manipulation of the surface poles/control points. From these initially created surfaces, other surfaces are constructed using either derived methods such as offset or angled extensions from surfaces; or via bridging and blending between groups of surfaces.

4.2.3 Solid Modelling

Solid modeling is the most advanced method of geometric modeling in three dimensions. Solid modeling is the representation of the solid parts of the object on your computer. The typical geometric model is made up of wire frames that show the object in the form of wires. This wire frame structure can be two dimensional, two and half dimensional or three dimensional. Providing surface representation to the wire three dimensional views of geometric models makes the object appear solid on the computer screen and this is what is called as solid modeling. Solid modeling (or modelling) is a consistent set of principles for mathematical and computer modeling of three-dimensional solids. Solid modeling is distinguished from related areas of geometric modeling and computer graphics by its emphasis on physical fidelity. Together, the principles of geometric and solid modeling form the foundation of computer-aided design and in general support the creation, exchange, visualization, animation, interrogation, and annotation of digital models of physical objects.

4.3 Various Components of Bumper

4.3.1 Plastic Fascia

Almost 28% of the total accidents are encountered annually in road traffic crashes worldwide are pedestrians. Pedestrian means a person traveling on foot, whether walking or running. So to protect the pedestrians to some extent in low speed car front crash there is a provision on front bumper that is to use the covering made up of softer material like plastic or foam as compared to human bone so that the injury will be reduced considerably. While protecting the human body the plastic fascia also functions as the aerodynamic structure for the front lower part of the vehicle and also it gives a good aesthetic characteristic to the vehicle. Here, the plastic fascia used in the project, is made up of Polyethylene. Also in some cases of low speed front impact, only the plastic fascia gets damaged sustaining all the impact energy by itself and protecting the reinforcing beams

Plastic fascia is designed by using solid works 2014 software which provides great flexibility to design and optimize the model. The designed fascia is shown below. Material used for this fascia is polyethylene.

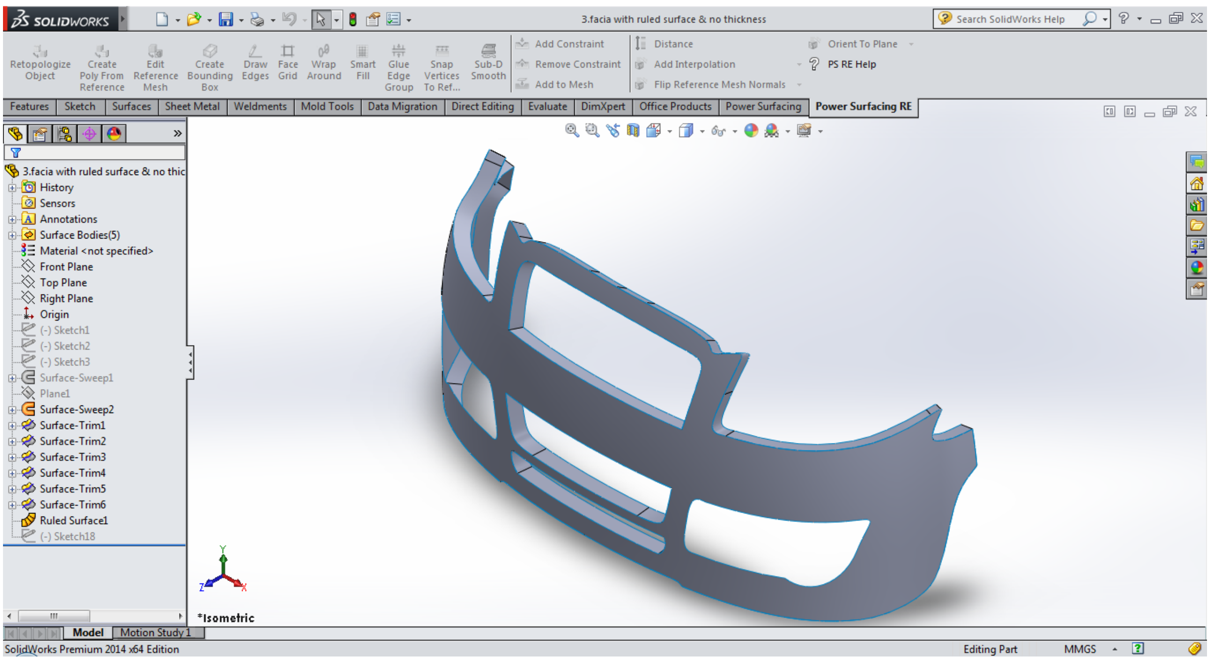
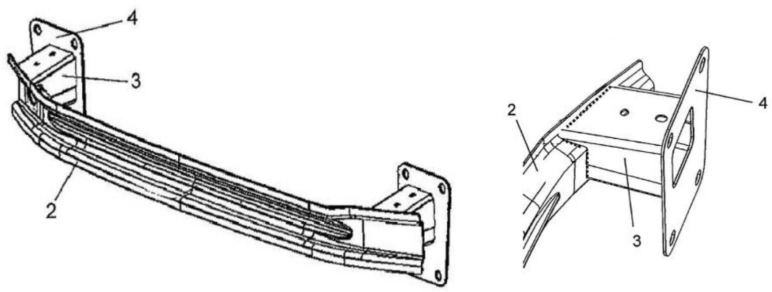


Fig 4.2 Plastic Fascia

4.3.2 Crush box

A crush box includes a tubular body having a polygonal section and disposed between an end portion of a bumper beam and a vehicle body frame as an attachment. In other words, Crush boxes are designed to be sacrificial parts and should be replaced if damaged. Tubular body is crushed into a bellows shape to absorb impact energy as a compressive load is applied to the tubular body from the bumper beam in the axial direction of the tubular body. In low speed dynamic front crash after the absorption of impact energy by plastic fascia, second element to absorb energy is crush boxes. If these two absortions are not enough then the surplus impact energy is absorbed by bumper beam to damage itself protecting the vehicle safety system. Thus crush boxes increase the crashworthiness of the bumper beam.



2-Bumper Beam

3-Crush Box

4-Vehicle Body Frame

Fig 4.3 Crush box

The figure no. 4.3 is of general system and figure no. 4.4 is of designed system, in which the vehicle body frame is not shown.

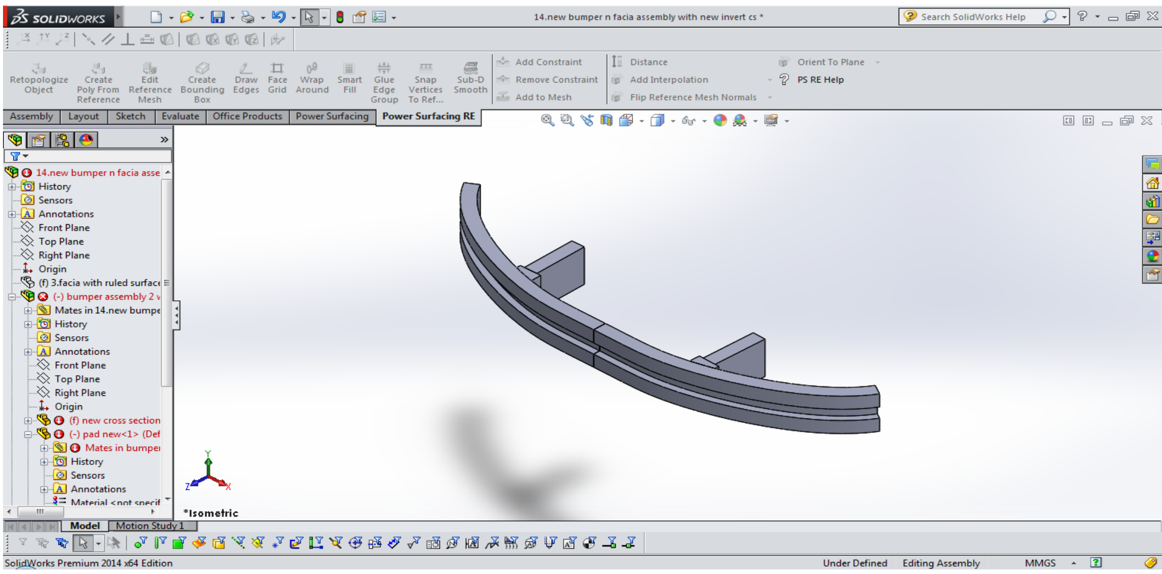


Fig 4.4 Designed bumper

4.3.3 Reinforcing beam

The reinforcing beams are key components of the bumper systems. Reinforcement beams help absorb the kinetic energy from a collision and provide protection to the rest of the vehicle. By staying intact during a collision, beams preserve the frame. Design considerations for reinforcing beams include strength, manufacturability, weight, recyclability and cost. Steel reinforcing beams are usually roll formed or hot stamped using ultra high-strength steel. Typical cross sections, used in the project, are shown in Figure. Roll formed beams are the most common but hot stamped beams have the lowest average mass of all steel bumper systems and are becoming more popular as a result. The most common cross section for roll formed beams is the B-section and the most common sections for hot stamped beams are box and hat sections. Sometimes a stamped or roll formed face or back plate is welded to a roll formed or hot stamped C-section to create a boxed section. All steel reinforcing beams receive corrosion protection. Some beams are made from hot-dip galvanized or electro-galvanized sheet. The zinc coating on these products provides excellent corrosion protection. Other beams are protected after fabrication with a paint system such as E-coat. Since steel reinforcing beams are becoming stronger and lighter with thinner gauges being used, more beams are using both zinc coating and E-coating to meet corrosion protection requirements.

In this project three different reinforcing beams are used by varying the cross-sectional area and material i.e. Steel, Aluminium & Magnesium Out of these three one is shown in fig 4.5

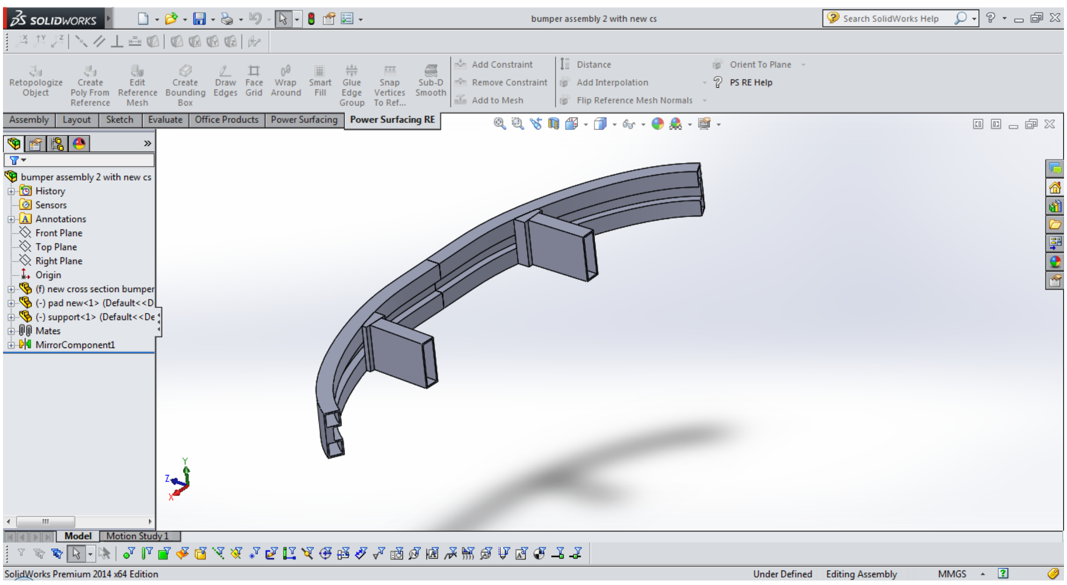


Fig 4.5 B type (c/s-1)

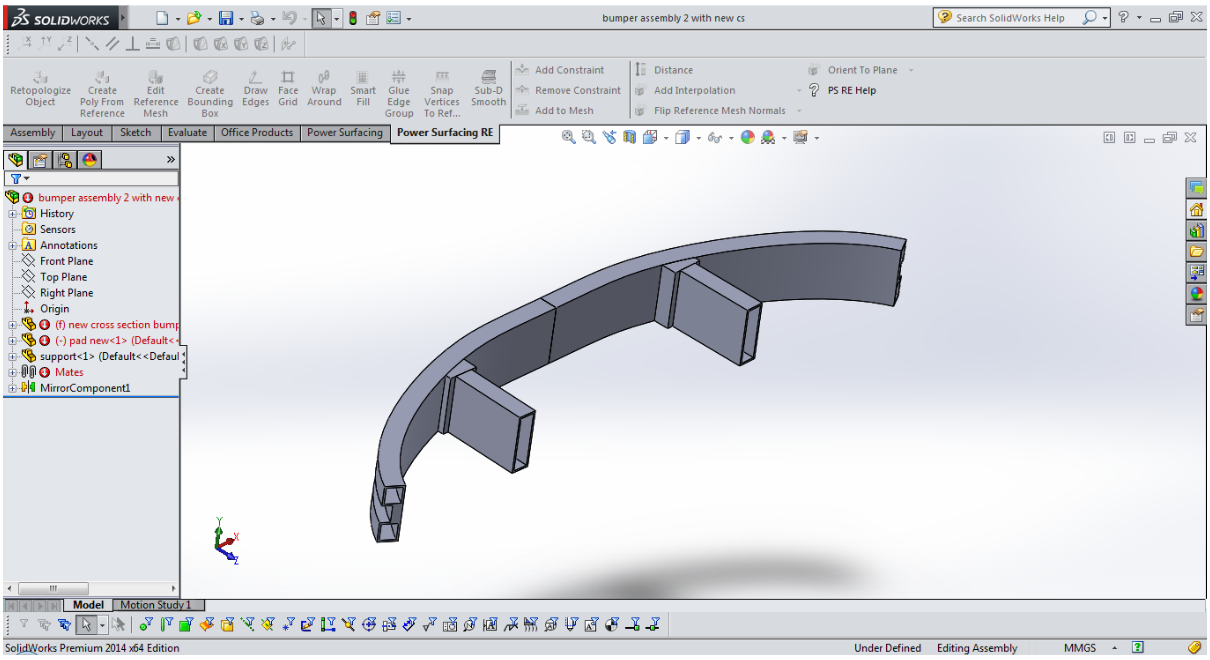


Fig 4.6 Invert B type (c/s-2 2)

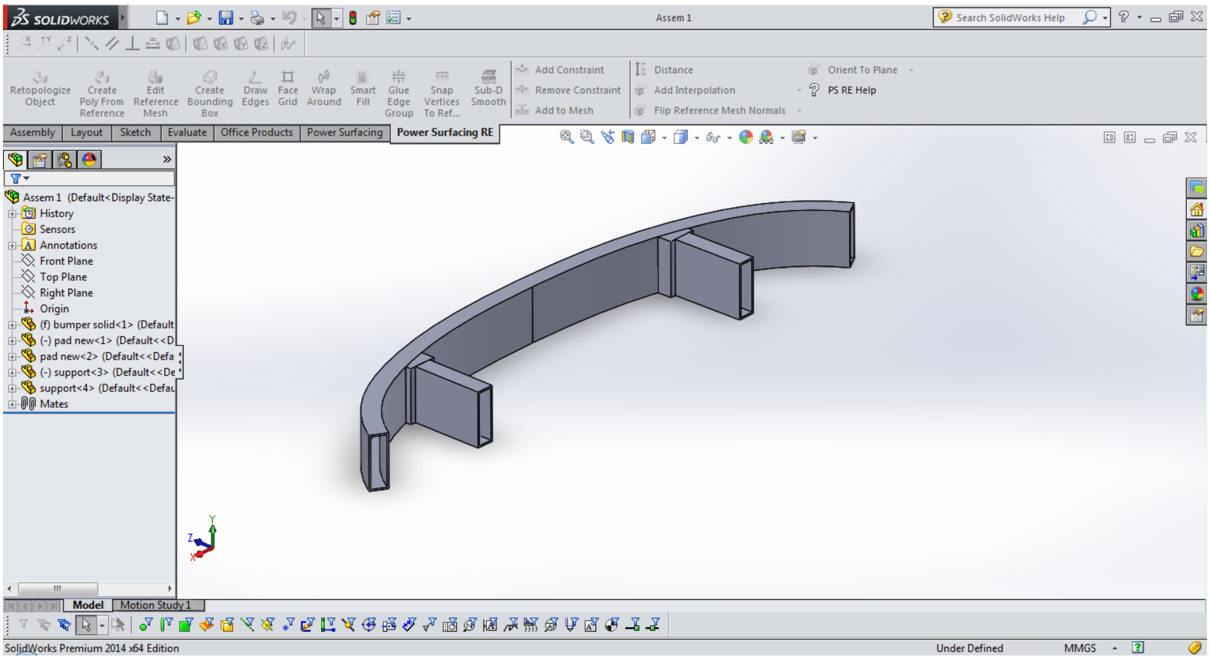


Fig 4.7 Box type (c/s-3)

The final assembled bumper system used in this project is shown in fig 4.8

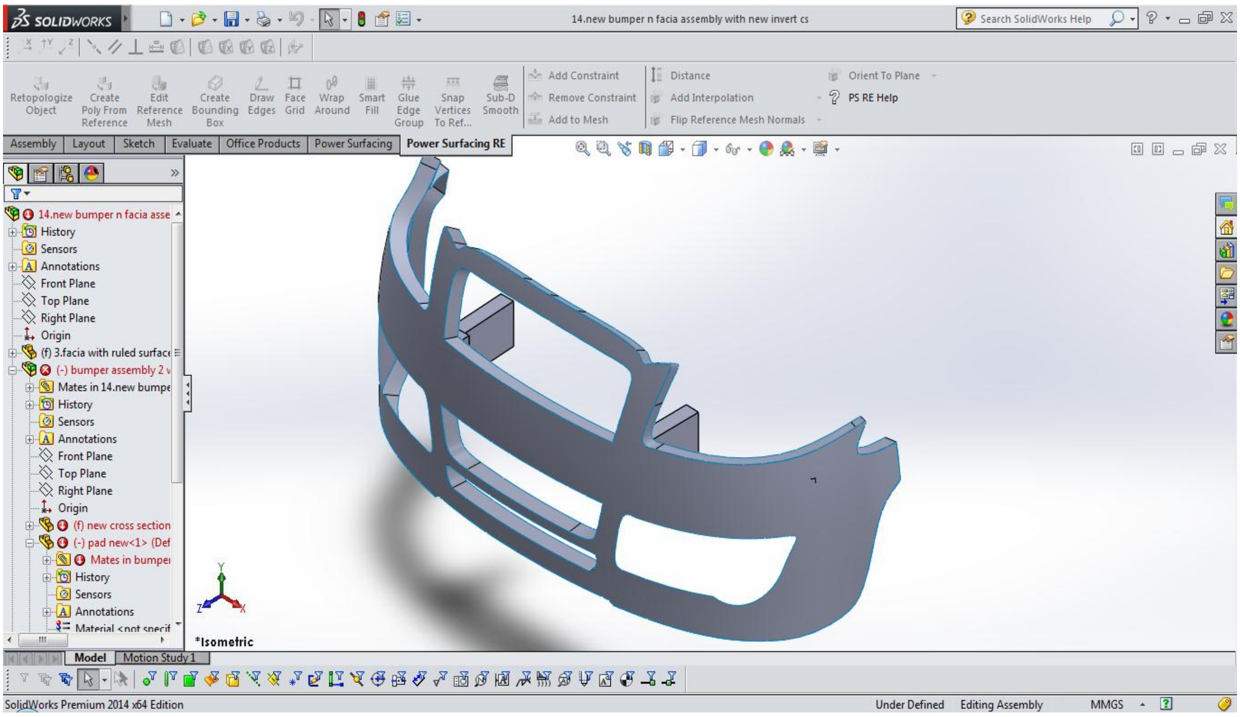


Fig 4.8 Designed bumper system

Chapter 5

ANALYSIS

5.1 Introduction

Analysis is the process of breaking a complex topic or substance into smaller parts in order to gain a better understanding of it. The technique has been applied in the study of mathematics and logic.

Verify operation and performance as you create your design with easy-to-use simulation and design analysis tools that are fully integrated with 3D CAD software, accelerating your design process, reducing the number of prototypes, saving time and development costs.

CAD software integrates easy-to-use analysis tools with design to verify operation and performance during product development. Design analysis increases product innovation by reducing risk in design, significantly reduces the number of physical prototypes needed, and helps lower material and other costs.

5.2 Engineering Data

Engineering data provides an overview of the engineering data manager and demonstrates the procedure for using the engineering data to enter, store, view, organize and import material property data. It is a tool for defining, storing, and organizing material properties. Engineering data manager can use properties stored in material libraries and can be saved for other workbench properties. Engineering data can be added as a standalone or as a part of an analysis system.

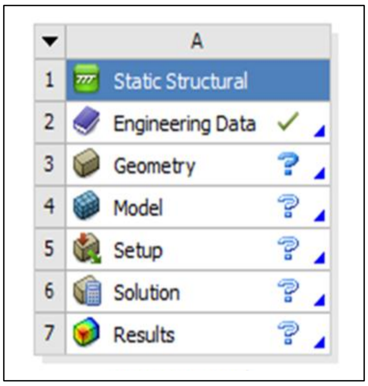


Fig 5.1 Flow of analysis

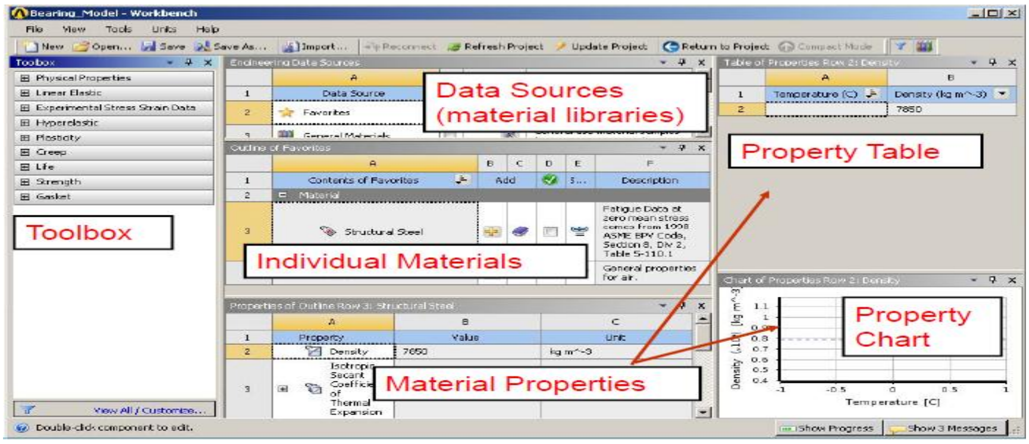


Fig 5.2 Property of material

As three materials along with three different bumper models are used in this project, out of them the engineering data for one bumper model with three materials are shown below, in the table 5.1

Object Name	Wall	Attachment Member	Crushable Boxes	Bumper Beam	Fascia
Thickness	0.1m	0.0047625 m	0.0047625 m	0.0047625 m	3.e-003 m
Bounding Box					
Length X	2.5 m	5.4885e-002 m	8.1221e-002 m	0.83335 m	1.8601 m
Length Y	1. m	0.14603 m	0.14601 m	0.146 m	0.78841 m
Length Z	0.1 m	0.20144 m	4.2498e-002 m	0.3455 m	0.60128 m
Volume	0.25 m³	3.7814e-004 m³	3.3364e-004 m³	2.5097e-003 m³	4.5735e-003 m³
Material					
Assignment	Structural Steel		Mild Steel		Polyethylene
Mass	1962.5 kg	2.9684 kg	2.619 kg	19.702 kg	4.3448 kg
Material					
Assignment	Structural Steel		Aluminium Alloy		Polyethylene
Mass	1962.5 kg	1.0474 kg	0.92417 kg	6.952 kg	4.3448 kg
Material					
Assignment	Structural Steel		Magnesium Alloy		Polyethylene
Mass	1962.5 kg	0.68064 kg	0.60054 kg	4.5175 kg	4.3448 kg

Table 5.1 Engineering data

5.3 Geometry

Geometry, the second step after engineering data, requires the model of bumper. This model is designed on Solidworks and saved as 'IGS' and then imported to ansys by using this option. The imported geometry is shown in fig 5.3

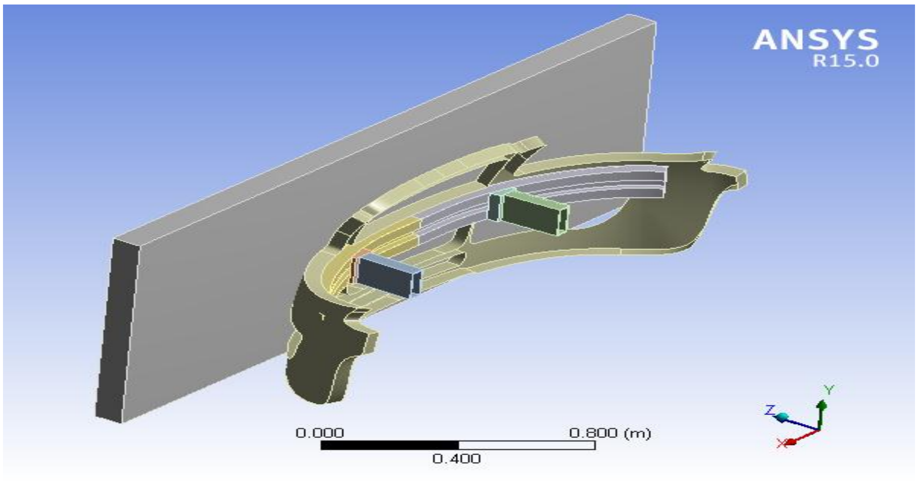


Fig 5.3 Model geometry

5.4 Meshing

Meshing is discrete representation of the geometry that is involved in the problem. Essentially, it partitions space into elements (or cells or Zones) over which the equation can be approximated. Zone boundaries can be Free to create computationally best shaped zones, or they can be fixed to represent internal or external boundaries with in a model.

5.4.1 Types of Element

- 2D
  - Triangular
  - Quadrilateral
  - Quad-triangular

- 3D
  - Tetrahedron
  - Pyramid
  - Triangular prism
  - Hexahedron

5.4.2 Meshing Steps:

Meshing can be done by two ways either manually assign the meshing type or do automatically according to geometry. In this project meshing is done automatically using quad-triangular meshing.

Object Name	Mesh
Sizing	
Use Advanced Size Function	On: Curvature
Smoothing	High
Transition	Slow
Span Angle Centre	Coarse
Curvature Normal Angle	Default (30.0 °)
Min Size	Default (7.061e-003 m)

Max Face Size	Default (3.5305e-002 m)
Max Size	Default (3.5305e-002 m)
Growth Rate	Default
Minimum Edge Length	1.6146e-002 m
Inflation	
Inflation Option	Smooth Transition
Statistics	
Nodes	13369
Elements	11013

Table 5.2 Meshing steps

Automatic meshing is done according to our geometry and, the meshed model of geometry is shown in the fig 5.4

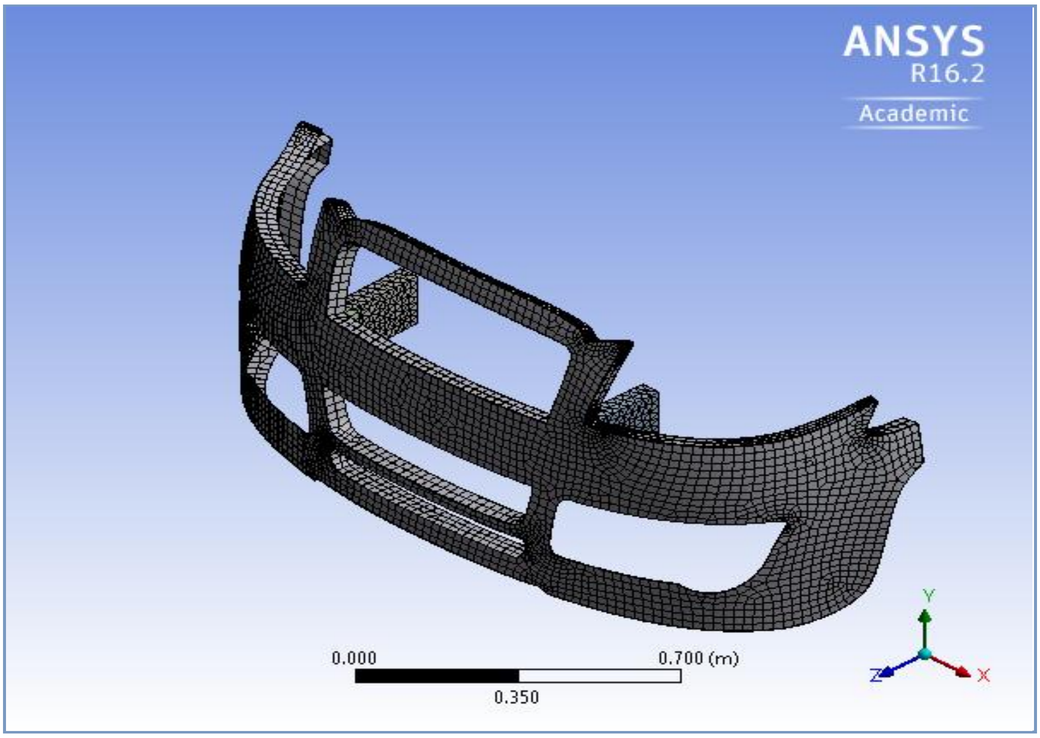


Fig 5.4 Meshed model

5.5 Modal Analysis

Modal analysis is the study of the dynamic properties of structures under vibrational excitation. Modal analysis is the field of measuring and analyzing the dynamic response of structures and or fluids during excitation. Examples would include measuring the vibration of a car's body when it is attached to an electromagnetic shaker, analysis of unforced vibration response of vehicle suspension, or the noise pattern in a room when excited by a loudspeaker.

You use modal analysis to determine the vibration characteristics (natural frequencies and mode shapes) of a structure or a machine component while it is being designed. It also can be a starting point for another, more detailed, dynamic analysis, such as a transient dynamic analysis, a harmonic response analysis, or a spectrum analysis. You use modal analysis to determine the natural frequencies and mode shapes of a structure. The natural frequencies and mode shapes are important parameters in the design of a structure for dynamic loading conditions. They are also required if you want to do a spectrum analysis or a mode superposition harmonic or transient analysis. You can do modal analysis on a prestressed structure, such as a spinning turbine blade. Another useful feature is modal cyclic symmetry, which allows you to review the mode shapes of a cyclically symmetric structure by modeling just a sector of it.

Modal analysis in the ANSYS family of products is a linear analysis. Any nonlinearity, such as plasticity and contact (gap) elements, are ignored even if they are defined. You can choose from several mode extraction methods: subspace, Block Lanczos, Power Dynamics, reduced, unsymmetrical, and damped. The damped method allows you to include damping in the structure.

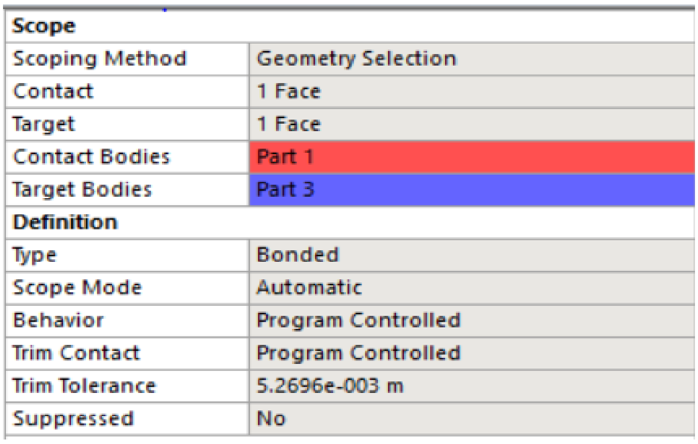
5.5.1 Schematic Setup for Modal Analysis (Free Vibration)

- Modal analysis can be employed on any type of geometry:
- Solid bodies, surface bodies and line bodies.
- The Point Mass feature can be used. A point mass adds mass without additional flexibility to the structure thus reducing the natural frequency.

- Material properties: Young's Modulus, Poisson's Ratio, and Density are required. Structural and thermal loads are not available in free vibration:
- If no supports (or partial) are present, rigid-body modes will occur at or near 0 Hz.
- The choice of boundary conditions will affect the mode shapes and frequencies of the part. Careful consideration was given for how the model is constrained. Symmetry, which allows reviewing the mode, shapes of a cyclically symmetric structure by modeling just a sector of it.

5.5.2 Contact

In general the contact types used are Bonded, Frictionless, Rough, Frictional and No separation. Out of which the default value for contact type is bonded type. In this project the default value is used.



Scope	
Scoping Method	Geometry Selection
Contact	1 Face
Target	1 Face
Contact Bodies	Part 1
Target Bodies	Part 3
Definition	
Type	Bonded
Scope Mode	Automatic
Behavior	Program Controlled
Trim Contact	Program Controlled
Trim Tolerance	5.2696e-003 m
Suppressed	No

Fig 5.5 Contact type

5.5.3 Solution Setup

Within Mechanical Analysis Settings:

- We have Specified the number of modes to find 10 (default is 6).
- Optionally we have specify a frequency search range (defaults from 0Hz to 1e+08Hz)..
- After all setup additional result output were requested by solving the model

When the solution completes, the solution branch displayed a bar chart and table listing frequencies and mode numbers

The bar chart shown is selected randomly for a particular mode of aluminum material with second cross-section.

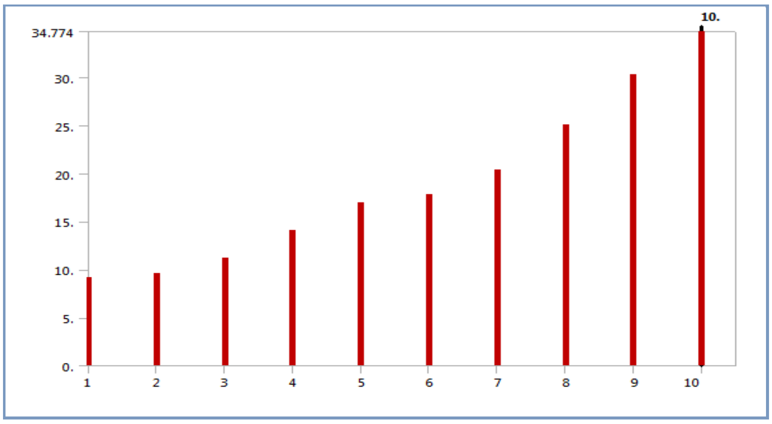


Fig 5.6 Bar chart at Particular Mode Shape

5.5.4 Modal Results

Modal analysis result gives the information about total deformation for the selected mode at respective frequencies, which is tabulated and shown in table 5.3. This information is for all 9 (3 materials with 3 cross-sections) combinations.

PARAMETERS	ALUMINIUM			MAGNESIUM			MILD STEEL		
	C/S 1	C/S 2	C/S 3	C/S 1	C/S 2	C/S 3	C/S 1	C/S 2	C/S 3
TOTAL DEFORMATION (m)	1.4278	2.8312	2.5589	1.4270	2.8289	2.5437	1.4287	2.8346	2.5621
FREQUENCY (Hz)	5.6351	17.838	13.880	5.6306	17.791	13.858	5.6403	17.892	13.896

Table 5.3 Modal analysis result

*NOTE: TOTAL DEFORMATIONS AND FREQUENCIES ARE NOTED DOWN FOR MODE NUMBER 6.

The total deformation for mode 1 of aluminum with second c/s after analysis is shown in fig 5.7

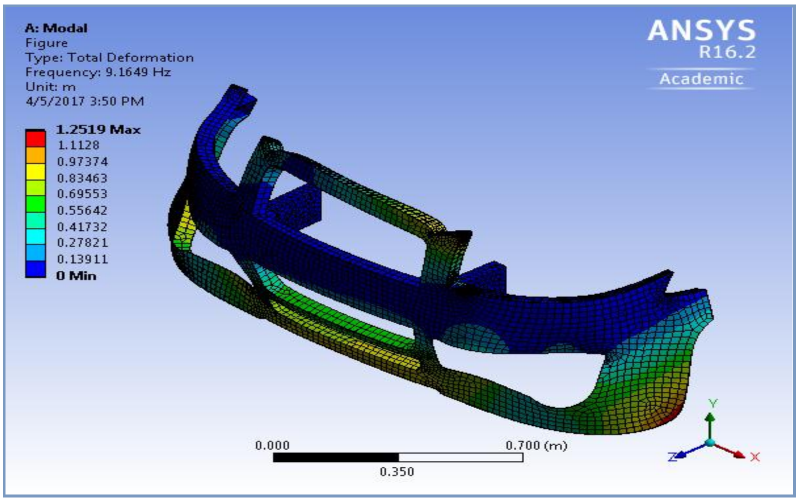


Fig 5.7 Total deformation

5.6 Explicit Dynamics Analysis (LS-DYNA)

An Explicit FEM analysis does the incremental procedure and at the end of each increment updates the stiffness matrix based on geometry changes (if applicable) and material changes (if applicable). Then a new stiffness matrix is constructed and the next increment of load (or displacement) is applied to the system. In this type of analysis the hope is that if the increments are small enough the results will be accurate. One problem with this method is that you do need many small increments for good accuracy and it is time consuming. If the numbers of increments are not sufficient the solution tends to drift from the correct solution. Furthermore this type of analysis cannot solve some problems. Unless it is quite sophisticated it will not successfully do cyclic loading and will not handle problems of snap through or snap back. Perhaps most importantly, this method does not enforce equilibrium of the internal structure forces with the externally applied loads.

5.6.1 Schematic Setup For Explicit Dynamics

First three steps i.e. engineering data, geometry and model, are same as that of modal analysis.

1) Initial condition:

In this there are 2 choices as velocity and angular velocity out of which velocity is selected as input type and defined by components (in X-direction v=16.67m/s and in Y-direction v=0m/s to constrain the deformation in Y-direction).

Input Type	Velocity
Coordinate System	Global Coordinate System
X Component	0. m/s
Y Component	0. m/s
Z Component	16.67/s

Table 5.4 Initial Condition

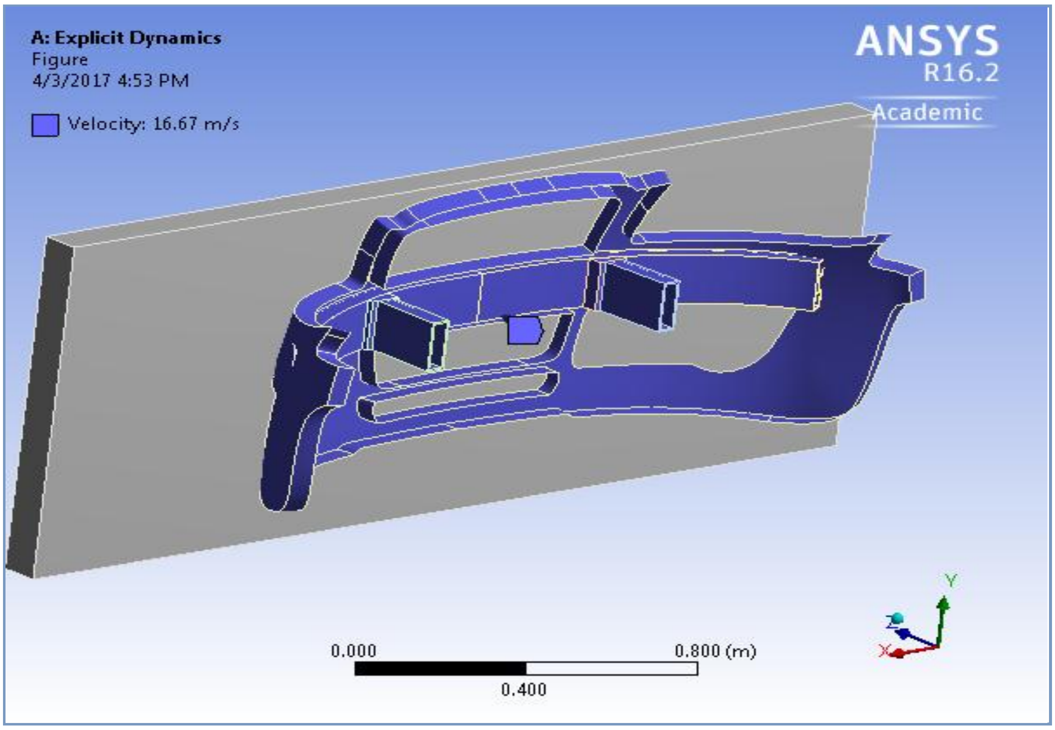


Fig 5.8 Velocity direction and magnitude

2) Analysis setting:

In this, fixed supports are selected as 2 faces of wall and 6 faces of bumper system. The co-ordinate system used is of global system. Also no. of points, for which graph is to be plotted, is taken as 100 for end time of 0.024 sec with maximum energy error as 0.5.

Object Name	Fixed Support	Displacement
Geometry	2 wall Faces	6 Faces of Bumper System
Define By		Components
Coordinate System		Global Coordinate System
X Component		Free
Y Component		0. m (step applied)
Z Component		Free

Table 5.5 Fixed Support

Object Name	Analysis Settings
Step Controls	
Maximum Number of Cycles	10000000
End Time	0.024 s
Maximum Energy Error	0.1
Output Controls	
Save Results on	Equally Spaced Points
Number of points	100

Table 5.6 Analysis setting

When the solution completes, the solution branch displayed a Energy vs Cycle graph and table listing total deformation and equivalent stress (von messes stress)

The Energy vs Cycle graph shown is selected randomly for aluminum material with second cross-section.

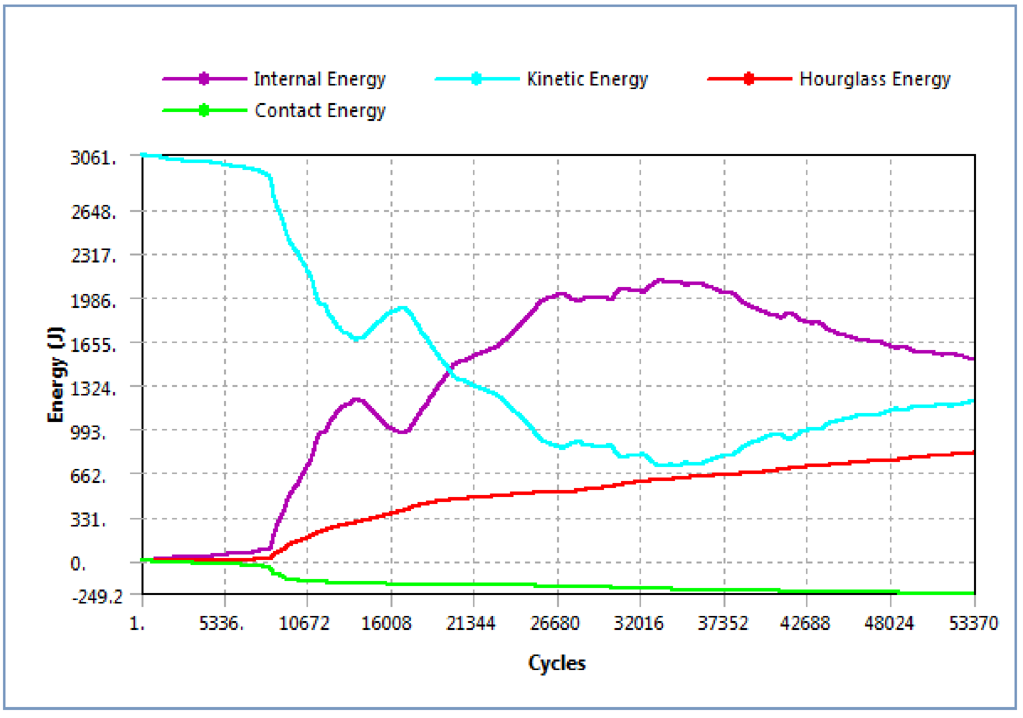


Fig 5.9 Energy graph

5.6.2 Explicit Dynamic Result

Explicit Dynamic analysis result gives the information about total deformation, Equivalent stress and also energy absorption which is tabulated and shown in table 5.7 and 5.8. This information is for all 9 (3 materials with 3 cross-sections) combinations.

PARAMETERS	ALUMINIUM			MAGNESIUM			MILD STEEL		
	C/S 1	C/S 2	C/S 3	C/S 1	C/S 2	C/S 3	C/S 1	C/S 2	C/S 3
TOTAL DEFORMATON (m)	0.20887	0.24277	0.20068	0.20607	0.24550	0.19993	0.21283	0.25392	0.20488
EQUIVALENT STRESS (MPa)	369.18	104.68	190.26	246.62	105.84	114.26	1076.2	364.39	593.94
YIELD STRESS (Mpa)	280			193			250		
WEIGHT (kg)	13.268	13.193	15.107	10.143	10.095	11.338	29.6342	29.422	34.845

Table 5.7 Explicit dynamic result

Parameter	ALUMINUM			MAGNESIUM			MILD STEEL		
	C/S 1	C/S 2	C/S 3	C/S 1	C/S 2	C/S 3	C/S 1	C/S 2	C/S 3
Cycles	29870.8	19476.4	19563.5	30179.35	20822.1	20839.2	29562.75	1920	16562
Kinetic Energy (KJ)	3081.8	3061	2495.3	2213.9	2200.4	1875.9	7626.8	7568.1	5738.9
Max. Energy (KJ)	1415.4	1423.23	1150	984	1008	873	3801.6	3792	2705.85
Energy Absorbed (%)	45.92	46.69	46.08	44.44	45.80	44.61	49.84	50.01	47.17

Table 5.8 Energy absorption

The total deformation for aluminum with second c/s after analysis is shown in fig 5.10

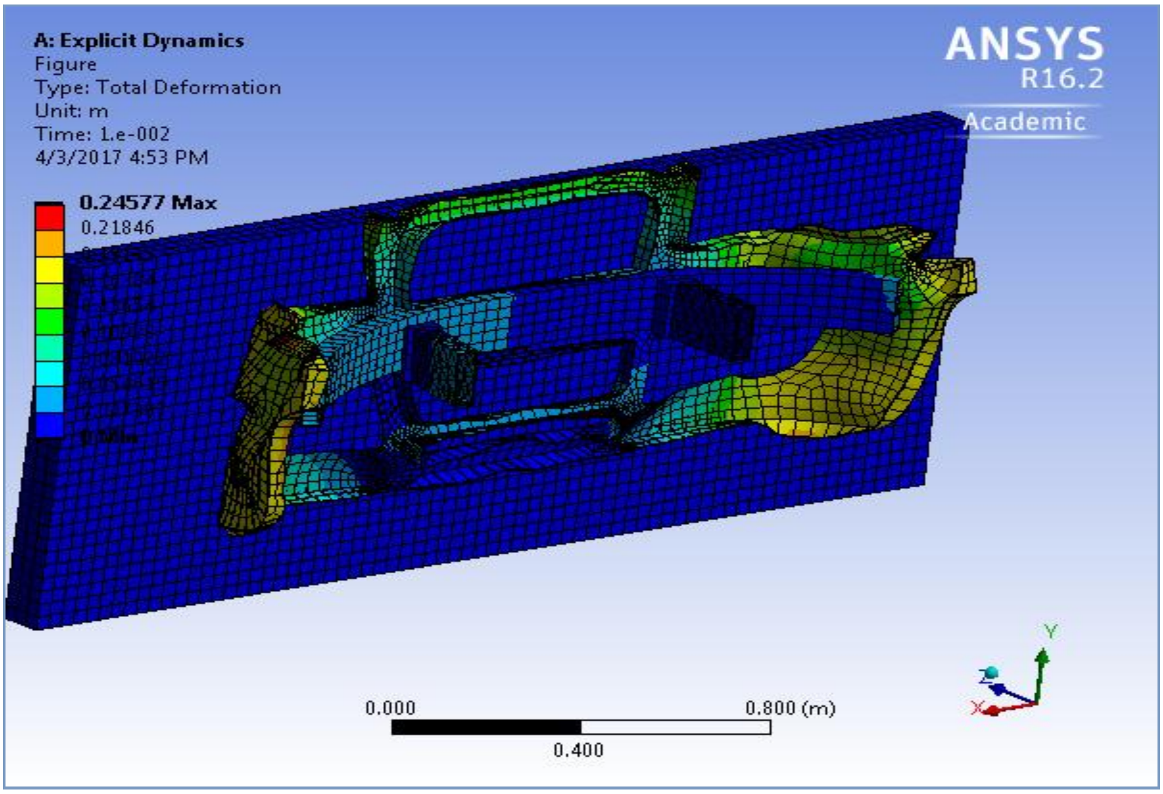


Fig 5.10 Deformation of bumper in dynamics

The Equivalent stress for aluminum with second c/s after analysis is shown in fig 5.11

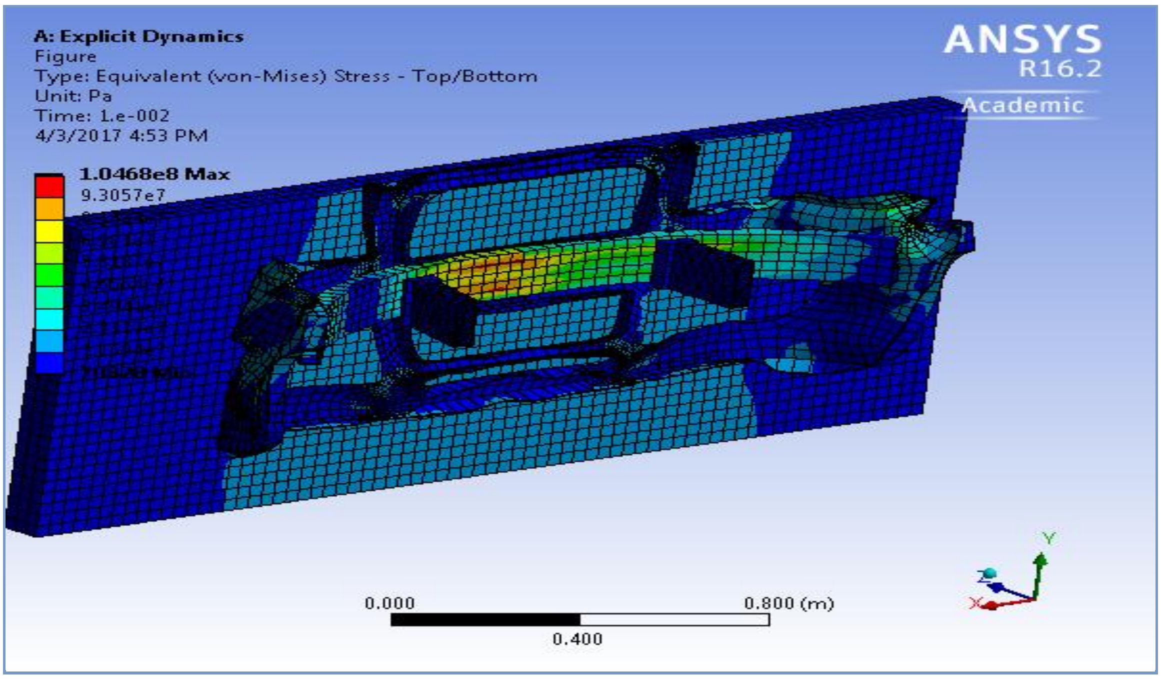


Fig 5.11 Equivalent stress in dynamics

5.7 Fatigue Analysis

As we know when a component is subjected to varying loads, fatigue criteria play a vital role in the failure of that component. Similarly, bumper of a car is also subjected to fatigue loading which affects life of the bumper.

Fatigue is a phenomenon associated with variable loading or more precisely to cyclic stressing or straining of a material. Just as we human beings get fatigue when a specific task is repeatedly performed, in a similar manner metallic components subjected to variable loading get fatigue, which leads to their premature failure under specific conditions.

According to ASTM (American society for testing and materials) fatigue life define as the number of stress cycles of a specified character that a specimen sustains before failure of a specified nature occurs. As previously highlighted, the fatigue life of a structural component significantly depends on the loading.

Standard fatigue tests under constant-amplitude loadings are valuable for determination of fatigue characteristics of materials. They also allow to initial selection of appropriate materials for a structure and are used in the algorithms to evaluate fatigue lifetime or endurance as material constants. It is also known that the obtained test results and proposed models under a constant amplitude.

5.7.1 Schematic Setup For Fatigue analysis

In this analysis also, first three steps i.e. engineering data, geometry and meshing, are same as that of modal analysis.

1) Initial condition:

In this step, we have selected two corner faces of the fascia to be fixed by using edge selection option, out which one is shown below by red colour in fig 5.12

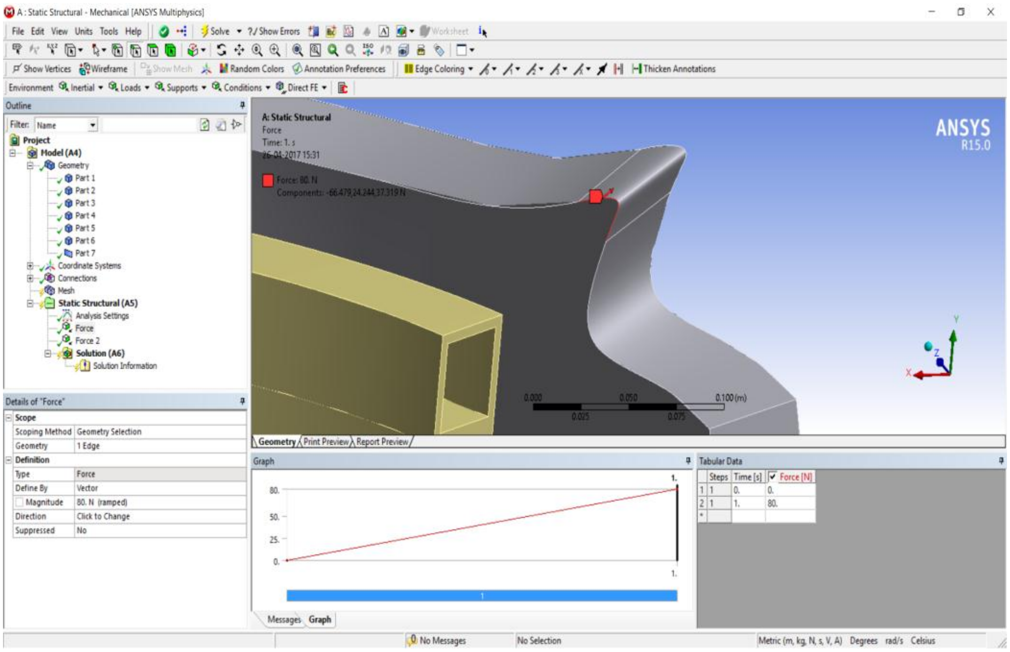


Fig 5.12 Direction of forces

At this fixed edges we have applied force of 80N as input type defined by vector, so direction is given manually. The force used is of fixed support type.

Scoping Method	Geometry Selection	
Geometry	2 Faces	1 Edge
Definition		
Type	Fixed Support	Force
Suppressed	No	
Define By		Vector
Magnitude		80. N (ramped)
Direction		Defined

Table 5.9 Fixed supports for fatigue analysis

2) Analysis setting:

In this step, fatigue tool is used which gives information about biaxiality indication. The used fatigue tool is shown in table no. in this, loading is of fully reversed type and theory used is Goodman with equivalent stress as component.

Object Name	Fatigue Tool
State	Solved
Materials	
Fatigue Strength Factor (Kf)	1.
Loading	
Type	Fully Reversed
Scale Factor	1.
Definition	
Display Time	End Time
Options	
Analysis Type	Stress Life
Mean Stress Theory	Goodman
Stress Component	Equivalent (Von Mises)
Life Units	
Units Name	cycles
1 cycle is equal to	1. cycles

Table 5.10 Fatigue tool

Fatigue material properties are based on uniaxial stresses but real world stress states are usually multi axial. This result gives the user some idea of the stress state over the model and how to interpret the results.

Biaxiality indication is defined as the smaller in magnitude principal stress divided by the larger principal stress with the principal stress nearest zero ignored – Stress Biaxiality Indication Values:

• Biaxiality of zero corresponds to uniaxial stress.

• Biaxiality of –1 corresponds to pure shear.

• Biaxiality of 1 corresponds to a pure biaxial state.

Comparing biaxiality with safety factor, the most damaged point occurs at a point of nearly uniaxial stress. As you can see in the Biaxiality Figure no., this model is under both nearly pure shear and nearly pure biaxiality.

5.7.2 Fatigue result

When the solution completes, the solution branch displayed biaxiality indication with minimum and maximum values. These values are tabulated in table no. for selected cross-section with three different materials.

Biaxiality indication

Material	Biaxiality indication value	
	Minimum	Maximum
Aluminium	-0.9999	0.9913
Magnesium	-0.9999	0.9926
Mild steel	-0.9999	0.9898

Table 5.11 Fatigue analysis result

Minimum and maximum biaxial indication values for second cross section with aluminum is shown in fig 5.13

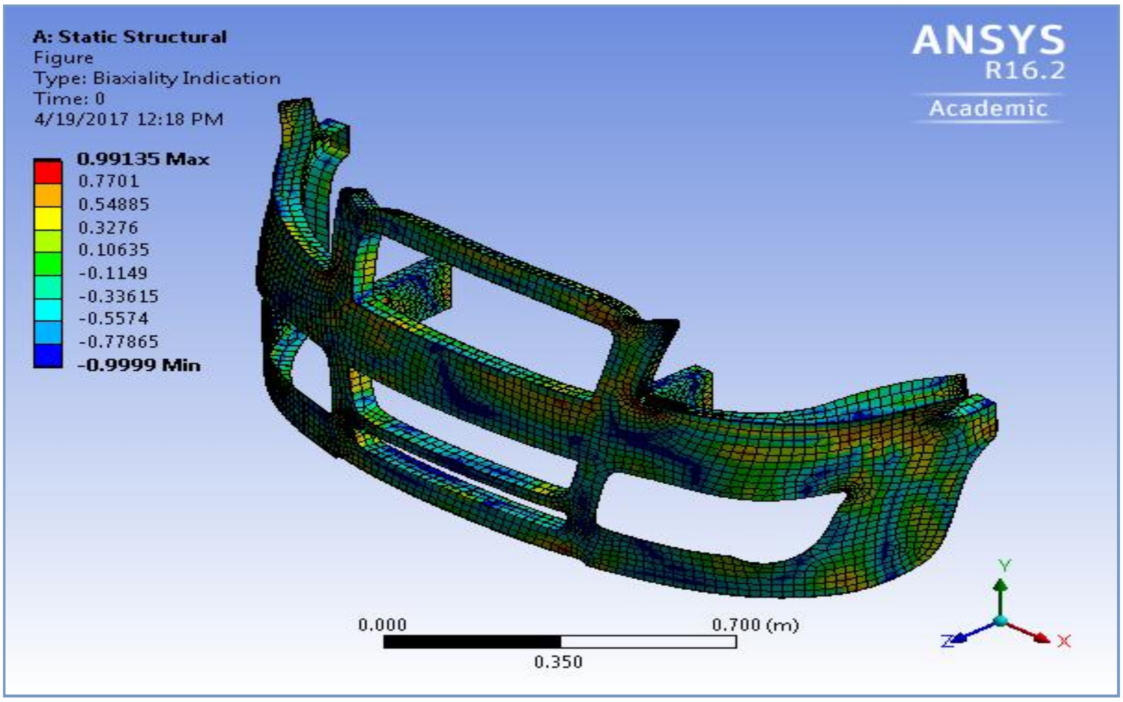


Fig 5.13 Biaxiality Indication

Chapter 6

RESULTS

1) Modal analysis :

PARAMETERS	ALUMINIUM			MAGNESIUM			MILD STEEL		
	C/S 1	C/S 2	C/S 3	C/S 1	C/S 2	C/S 3	C/S 1	C/S 2	C/S 3
TOTAL DEFORMATION (m)	1.4278	2.8312	2.5589	1.4270	2.8289	2.5437	1.4287	2.8346	2.5621
FREQUENCY (Hz)	5.6351	17.838	13.880	5.6306	17.791	13.858	5.6403	17.892	13.896

2) Explicit Analysis:

A) Total Deformation And Equivalent Stress

PARAMETERS	ALUMINIUM			MAGNESIUM			MILD STEEL		
	C/S 1	C/S 2	C/S 3	C/S 1	C/S 2	C/S 3	C/S 1	C/S 2	C/S 3
TOTAL DEFORMATON (m)	0.20887	0.24577	0.20068	0.20607	0.24250	0.19993	0.21283	0.25392	0.20488
EQUIVALENT STRESS (MPa)	369.18	104.68	190.26	246.62	105.84	114.26	1076.2	364.39	593.94
YIELD STRESS (MPa)	280			193			250		
WEIGHT (kg)	13.268	13.193	15.107	10.143	10.095	11.338	29.6342	29.422	34.845

B) Energy Absorption

Parameter	ALUMINUM			MAGNESIUM			MILD STEEL		
	C/S 1	C/S 2	C/S 3	C/S 1	C/S 2	C/S 3	C/S 1	C/S 2	C/S 3
Cycles	29870.8	19476.4	19563.5	30179.35	20822.1	20839.2	29562.75	1920	16562
Kinetic Energy (KJ)	3081.8	3061	2495.3	2213.9	2200.4	1875.9	7626.8	7568.1	5738.9
Max. Energy (KJ)	1415.4	1423.23	1150	984	1008	873	3801.6	3792	2705.85
Energy Absorbed (%)	45.92	46.69	46.08	44.44	45.80	44.61	49.84	50.01	47.17

3) Fatigue analysis:

Material	Biaxiality indication value	
	Minimum	Maximum
Aluminium	-0.9999	0.9913
Magnesium	-0.9999	0.9926
Mild steel	-0.9999	0.9898

The optimum results obtained in each analysis has been discussed in analysis chapter.

Chapter 7

CONCLUSION & DISCUSSION

On the basis of modal analysis results the value for total deformation is minimum for second cross-section at respective high frequencies. So the combination of materials with second cross-section comes under the main consideration.

On the basis of modal analysis results the two combinations that is magnesium with second cross-second and aluminium with second cross-section are in desired limit and all remaining combinations are out of the limits from equivalent stress and/or weight point of you. But when comes to energy absorption the maximum percentage of energy is absorbed by mild steel which is failed in equivalent stress and after mild steel it comes aluminium which absorbs about 46.69% energy that is higher than magnesium. So till now aluminium with second cross-section appears to be best combination.

As from two analysis results second cross-section is most suitable so the fatigue analysis is performed only for this cross-section along with varying materials. After the biaxiality indication value are interpreted I fatigue analysis, aluminium materials gives the best range hence aluminium with cross-section 2 (invert B-type) is selected.

After considering all required criteria’s and comparing all the results, we concluded that the best suitable combination is aluminium with second cross-section. This is the final proposed solution of this project.

REFERENCES

[1] T. Ambati, Srinath, P. Veeraraju “simulation of vehicular frontal crash test” international journal of applied research in mechanical vol-2 (2012)

[2] Byeong S.K, K. Park, Youn-Min Song “finite element frontal crash analysis of nev-vehicles platform with upper and sub frame body journal of dept. of automotive engineering hoseo university (2013)

[3] Erika Fechova, Jozef Kmec, Alena Vagaska Drazan Kozak “material properties and safety of car at crash test”, journal of international conference on manufacturing engineering and materials, science direct (2016)

[4] Saeed Barbat, Xiaowei Li, Priya Prasad “vehicle to vehicle front to side crash analysis using a CAE based methodology “journal of passive safety research and advanced engineering (2004)

[5] Touraj Gholami, Jürgen Lescheticky, Ralf Pabmann “crashworthiness simulation of automobile with ABAQUS/Explicit”, journal of Abaqus user conference BMW Group (2003)

[6] Wlodzimierz Bedkowski “assessment of the fatigue life of machine components under service load – A review of selected problems” journal of theoretical and applied mechanics, (2014)

[7] Jadav Chetan, Panchal Khushbu, Moulvi Nauman “the fatigue analysis of a vehicle suspension system – A review article” international journal of advanced computer research (2012)

[8] Hariharan Sankarasubramanian Sudipto Mukherjee,” optimization of vehicle front for safety of pedestrians” Indian institute of technology, Delhi, India, 2014.

[9] C. Sadhasivam & S. Jayalakshmi,” simulation of car body by development of static and dynamic analysis” department of mechanical engineering, pet engineering college, Tirunelveli, Tamilnadu, India 2011.

[10] Mandeep Kumar Vandanapu1, Ratnakar Pandu,” design optimization of passenger car front bumper” international journal of science and research (IJSR) 2015.

[11] Mr. Siddique Ahmed, Mr. Gajendra Patil," bumper shape optimization for pedestrian safety" international journal of engineering research and applications (IJERA), Vol. 3, Issue 1, January -February 2013.

[12] Intaek Lee Katsutoshi Ainaka Junji Saiki Minoru Yubuchi," parameter optimization for vehicle to vehicle crash compatibility using finite elementmethods" altair engineering Ltd. Japan, 2011.

[13] Shokri El Housini,"front bumper crashworthiness optimization",daan engineering s.n.c,2010.

[14] Steven Hoffenson1, Panos Papalambros, Michael Kokkolaras1,, Matthew Reed," an optimization approach to occupant safety and fuel economy in vehicle design" department of mechanical engineering, university of Michigan, Ann arbor, Michigan, USA,2013.

[15] Sankarasubramanian Hariharan, Sudipto Mukherjee, Anoop Chawla," a passenger car front optimization using Genetic Algorithm for safety of pedestrian during a vehicle pedestrian Impact",2015.

[16] K. Preston White, Jr., W. Thomas HollowelI H. Clay Gabler, Walter D. Pilkey," simulation optimization of the crashworthiness of a passenger vehicle in frontal collisions using response surface methodology", university of Virginia Charlottesville, VA, 2011.

[17] M. Pohlaka, J. Majak and M. Eerme," optimization of car frontal protection system", dept. of machinery, Tallinn university of technology ehitajate tee 5, 19086 Tallinn, Estonia, 2014.

[18] Hao Chen, Yali Yang and Liangjie Wang," vehicle front structure energy absorbing optimization in frontal impact" college of automotive engineering, shanghai university of engineering science, shanghai, 201620, china,2013.

[19] www.wikipidia.org

[20] www.rts.i-car.com