INTRODUCTION
The development process of automobiles with regard to crashworthiness behaviour depends strongly on virtual testing and simulation. Thus, development work based on cost intensive prototype construction and testing has been extensively reduced for the body in white as well as for the exterior and interior trim. The dramatic shortening of the total development time during the last years needs a much more front loaded development process which has been realized by numerical simulation. The numerical simulation accompanying the design of a car may be divided into three main phases – the concept, the series development and the optimization phase. During the concept phase the passive safety concept and its needed packaging space have to be determined. The series development is finished by prototype testing which should confirm the virtual development in an ideal case. Optimization work should close the development before the car’s launch. During these phases the focus of the numerical simulation changes from a more global view to a very detailed local analysis.

The aim of this work is to study front bumper of one of the existing passenger car in Indian market. Design modifications can be suggested or tried out on following basis:
• Performance related parameters of bumper
• Deformation/ Energy absorption capability
• Shape/ Size/ Thickness (Geometry)

The study will focus on modifying few of above stated parameters to suggest improvements in existing bumper of passenger car present in Indian markets. First, study will focus on studying existing design and based on observations, design improvements will be suggested. Modified front bumper design will be tested using FEM software for deflection, impact force and stress distribution. Results of modified bumper will be compared against existing design.

LITERATURE REVIEW
Few of the literatures at the initial study are cited below.
• Javad Marzbanrad, et al., Design and analysis of an automotive bumper beam in low-speed frontal crashes [1], studied the most important parameters including material, thickness, shape and impact condition for design and analysis of an automotive front bumper beam to improve the crashworthiness design in low-velocity impact. The simulation of original bumper under condition impact is according to the low-speed standard of automotives stated in E.C.E. United Nations Agreement, Regulation no. 42, 1994. The bumper beam analysis is accomplished for composite and aluminum material to compare the weight and impact behavior. The strength in elastic mode is investigated with energy absorption and impact force in maximum deflection situation. A good design of this part of automotives must prepare for the safety of passengers; meanwhile, should have low weight. Beside the role of safety, fuel efficiency and emission gas regulations are being more important in recent years that encourage manufacturer to reduce the weight of passenger cars. In this research, a front bumper beam made of three materials: aluminum, glass mat thermoplastic (GMT) and high-strength sheet molding compound (SMC) is studied by impact modeling to determine the deflection, impact force, stress distribution and energy-absorption behavior. The mentioned characteristics are compared to each other to find best choice of material, shape and thickness. The results show that a modified SMC bumper beam can minimize the bumper beam deflection, impact force and stress distribution and also maximize the elastic strain energy. In addition, the effect of passengers in the impact behavior is examined. Furthermore, beside the above-mentioned benefits, some more advantages like easy manufacturing due to simple shape without-ribs, economical aspects by utilizing low-cost composite material and reducing weight with respect to others can be achieved by SMC material.
• Han J. et al., Maximization of the Crushing Energy Absorption of Tubes [2], studied crushing energy absorption of square tubes. Four node shell and solid finite elements in DYNA 3D program were used to model the tubes. The tubes had a rigid mass attached to one end. They were crashed into a rigid wall at some initial velocity. Several cases were solved with different thickness and width of the tubes. Initial imperfections were introduced in some of the cases. Depending on the dimensions, Axis-symmetric, non-Axis symmetric or column buckling modes got generated during crushing. Some of the results were compared with the available experimental data. It was found that Maximum energy was absorbed for the Axis-symmetric crush mode. Also, the design of experiments approach was used to generate the approximate response surface as a function of the width and thickness of the tube. The tube was optimised to maximize the energy absorption during impact with side constraints on the dimensions. It was found that the energy absorption increased with decrease of width and increase of thickness. However, if the width was reduced too much, the tube could go into a column buckling mode, which had a dramatic decrease in...
the energy absorption capacity. It was also found that the mean impact force increased linearly with the width multiplied by the thickness squared.

- Willem Witteman et al., Adaptive frontal structure design to achieve optimal deceleration pulses [4], discuss possibilities to design an adaptive vehicle structure that can change the stiffness real time for optimal energy absorption in different crash situations. Besides that all the energy which is absorbed is also important to manage the intensity during the crash time, because the resulting crash pulse has a large influence on the injury level due to predetermined crash velocity. This implies that in a given vehicle concept the structure must have a specific stiffness. Normally, the two main frontal rails have to absorb most of the crash energy with a progressive folding deformation of a steel column.

- Bahig B.Fileta. et al., Design of vehicle structures for crash Energy Management [6], provides an immense resource that has quenched the author for literature on crashworthiness engineering in the automotive domain. Manuscript provides the basic introduction for crashworthiness and the various techniques used for styling and packaging considerations drove the design of vehicle body structures. Author has discussed several analytical design capabilities based on pure axial collapse mode and bending collapse mode of frontal rail structures, providing engineers with a variety of tools to design modern vehicle structures that can meet the growing customer demands for better crashworthiness performance, quieter ride and reliability. Here the author has also discussed about thin-walled box columns, composed of plate elements and subjected to axial compression, which will buckle locally when critical stress is reached. Local buckling initiates the processes that lead to the eventual collapse of the section and a subsequent folding of the column. The collapse strength of the section is related to its thickness/width (t/b) ratio and material properties. For very small t/b ratios representing the so called ‘non-compact’ sections, the mode of collapse of a section will be influenced predominantly by the geometry, since its local buckling strength is considerably below the material yield strength. The mode of collapse of ‘non-compact’ sections is characterized by large irregular folds reminiscent of crumpling, which give rise to a bending type instability that is induced by fold irregularities. For larger t/b ratios, the ‘compact’ sections in which the elastic buckling strength exceeds material yield strength, the material strength properties are expected to govern the mode of collapse and consequently the post-buckling stability.

**PROBLEM DEFINITION**

The problem consists of design and analysis of front bumper of a passenger car for its performance enhancement and effecting compliance to the standard practices in the Industry using Explicit Finite Element Code.

**PRESENT THEORIES & PRACTICES**

Within the passive safety simulation a lot of different load cases are tested numerically. During the last years the number of load cases has been increased significantly. The variety of applications might be divided into three main topics.

**Structural Crashworthiness of Whole Car:** When frontal, rear and side crash computations are widely used in the development process, compatibility is a growing application. Usually, these models represent most of the body in white, chassis, driveline and exterior trim components. Interior trim components are modeled when the influence on the overall load paths has to be considered, like seats for side crash analysis. These components might be discretized or their masses are smeared over neighbouring parts.

**Occupant Protection:** The finite element method is mainly used for occupant protection analysis during side impact. The optimization of structural response, interior trim deformation, and airbag behaviour is done in a parallel and iterative process. Because of the high number of needed simulations, a major number of analyses are done in so called substructure runs which use boundary conditions of the whole car analysis. The same car model as for the side crash computation of the whole car is used. Interior trim and a dummy model are added.

**Crashworthiness of Components:** A wide range of applications have to be covered by simulation, some of which are mentioned here. These analyses can be performed independently of the whole car crash simulations, but same meshes are used when possible. The discretization of the components is in general much finer than in whole car analysis but the number of represented components is smaller.

FEM is backbone of today’s automotive industry. In recent times FE analysis is widely used to validate the complex designs like bumper. Use of FEA not only reduces product development time but also saves lot of cost. Hence, this work proposes FE analysis of bumper to validate the design modifications in a bumper of car.

**OBJECTIVES OF THE WORK**

The following are the objectives of the study:

1. To study existing passenger car front bumper in Indian market for possible design modifications.
2. To carry out impact analysis using CAE to benchmark the performance of the existing bumper.
3. To carry out design improvements in the current model of the bumper.
4. To perform impact analysis for the modified bumper.
5. To validate virtually the FE results of the modified bumper against the existing bumper FE results in terms of deflection, impact force and stress distribution.

**PROPOSED WORK AND METHODOLOGY**

This study will use commercial FEA tools for carrying out FE analysis limited to the Bumper as a single component though; the boundary conditions would take into consideration its fitment with the mating parts in the assembly.

1. Study the literature related to bumper design and its performance improvement by referring to books, journal papers and related manuals.
2. Pre-processing of the finite element model of existing bumper (Baseline design) using HyperMesh.
3. Impact analysis of a FE model of existing bumper (Baseline design) using LS-DYNA as a solver.
4. Post-processing of the results of the Baseline design using LS-POST / Hyperview as post processor.
5. CAD modeling of modified bumper using CATIA.
7. Applying loads and boundary conditions on the modified bumper using HyperMesh.
10. Comparing the results of the two simulations for its improvement in crashworthiness and validate against the Test results of the Baseline design.

SOFTWARE FOR DESIGN AND ANALYSES
CATIA V5, Hypermesh and LS-DYNA can be used for Modeling, Pre-processing and Analysis.
CATIA V5 can be used for the modeling of the bumper. Hypermesh can be used for Pre-processing of the finite element model.
The general purpose FEA software, LS-DYNA Explicit Finite Element Code can be used for impact simulation of a FE model of bumper. LS-POST / Hyperview can be used for Post-processing.

REFERENCES