

# Optimization of Bumper Column of a Low Speed Vehicle

Sagar H N<sup>1</sup>, Pavana Kumara B A<sup>2</sup>, Manjunath Siddappa Bailwad<sup>3</sup>, Punith H<sup>4</sup> Bharath HB<sup>5</sup>

Assistant Professor, Department of Mechanical Engineering, Sri Krishna Institute of Technology, Bangalore, India<sup>1</sup>

U.G. Student, Department of Mechanical Engineering, PNS Institute of Technology, Bangalore, India<sup>2,3,4,5</sup>

**ABSTRACT:** Bumper beam column is a structural component is placed between the bumper and the engine mounts. During the crash events the bumper beam get deformed itself and not transfer any amount of energy to the next element because of this the engine and other element next to the bumper Beam or totally safe. The impact analysis of segmented bumper beam was conducted and getting a stress in the first segments of 522 Mpa and second segmented have 43.5 Mpa, But what we seen was in the first segment is totally failed because it exceeds the ultimate strength of mild steel so in order to reduce the stress in the first segment, we go for Optimization using tosc software for both the segment of bumper beam. In the Optimization we are varying the parameter of connection angle by an zero to 90degree, segmented height bearing ratio  $h1/h2 = 40/60$  and load of 55 km per hour has been applied on the column. After Optimization we are getting a stress value in first segments was 343.5 Mpa and in the second segment of 28.7 Mpa that shows the two segments are totally within the yield stress of mild steel so the element next to the bumper columns are totally safe.

**KEYWORDS:** solid works, Abaqus Explicit, Tosca.

## I. INTRODUCTION

Bumper column are the latent wellbeing segments, because of impact it experiences twisting and it watches the effect vitality by the bumper beam itself. . Bumper beams are a thin walled structures set in halfway of car body and guard which is appeared beneath figure, which are utilized as vitality entrancing amid crash occasion. Bumper beam made substantial part from metal added substances. Beneath frontal crash mishaps, crash holder is relied upon to break down and itself spellbinding the crash quality. With the goal that harms to other casing in which components, lodge is limited. The formulation to calculate axial weigh down conduct of aluminum beam column in which proposed via Wierzbicki and Abramowicz. However the analytical guesses had been confirmed experimentally by means of Abramowicz and Jones, and Langseth and Hopperstad. Take a look at of segments beam column continues to be restrained; where this layout can possible to growth the perfect important load and unique substances in every section may be used. The crash field designs the usage of hybrid material of aluminum and metallic have been developed which will reduce crash box weight as much as 17.5%.

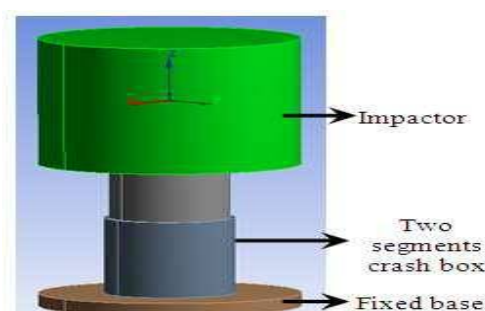


Figure.1 The bumper beam model component were impactor, bumper beam, and fixed rigid base

## II. PROBLEM DEFINITION

The aim of this project is to study the stress induced in two segmented body on energy absorbing bumper beam and optimize the both segmented bumper beam in order to reduce the stress. Above configuration is optimized using TOSCA by using the parameters given below

- Connection angle between segments ( $\theta$ ) will be of the range 0-45 degree of the interval of 5 degree each.
- Connection length between segments ( $p$ ) will be varied from 5-10mm of the interval 2.5 mm.
- Outside diameter ( $d_o$ ) will be varied from 100-200 of the interval 10mm each.

Segment Height and thickness ratio will be constant 40/60mm and 2/3mm respectively.

## III. METHODOLOGY

The basic step is to create a geometric model of bumper beam is done using solid works modeling tool. Rigid mass is modeled and applied with a velocity equivalent to maximum speed specification. This is impacted on the bumper beam. Once the geometry is created as per specifications it is imported into hyper works for messing. The finite element model is prepared by meshing it with appropriate elements like linear hexahedral, contact elements and constraining the model by applying material properties and boundary conditions. This finite element model is imported to ABAQUS EXPLICIT to evaluating the explicit dynamic analysis and optimized the model by TOSCA software Results are analyzed and the impact damage is studied. The reaction forces are observed.

## IV. GEOMETRIC MODELING

Figure shows the geometric model of a section of the bumper beam. The modeling has been done using SOLID works modeling software. The dimensions of bumper beam are considers from the literature survey. It consists of two segment bumper beam which has the Connection angle between segments ( $\theta$ ) of 45 degree. Design parameters in two segments bumper beam is as shown in figure.

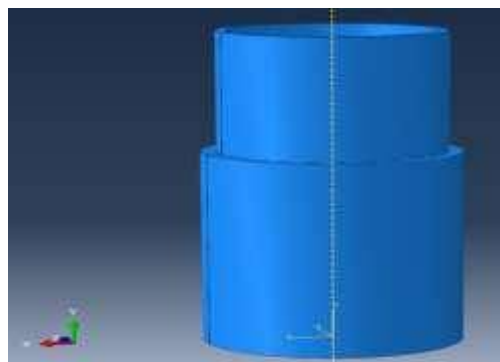


Figure- 2: Geometric models of two segments of crash box.

## V. MESHING

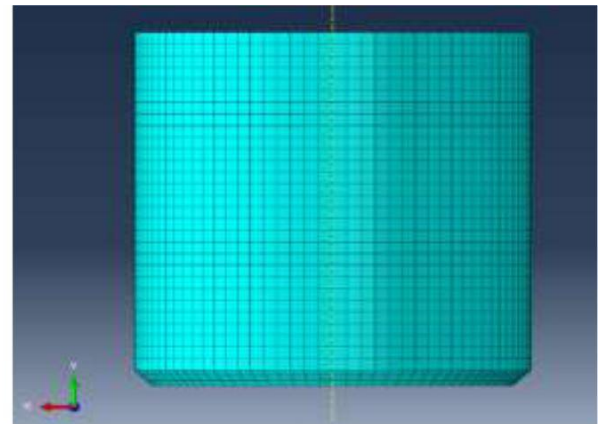
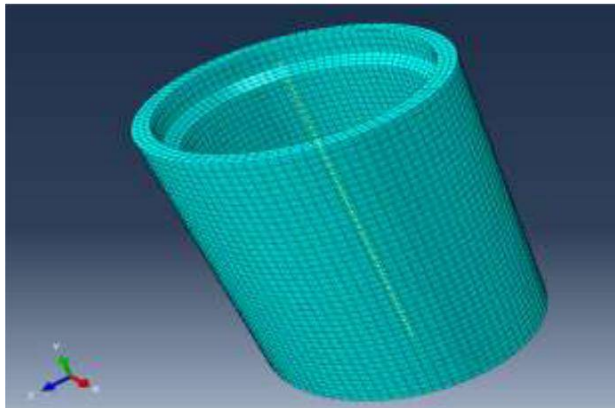


Figure: 3 Mesh models of two segments of crash box

Table No. 1 Elements and nodes count

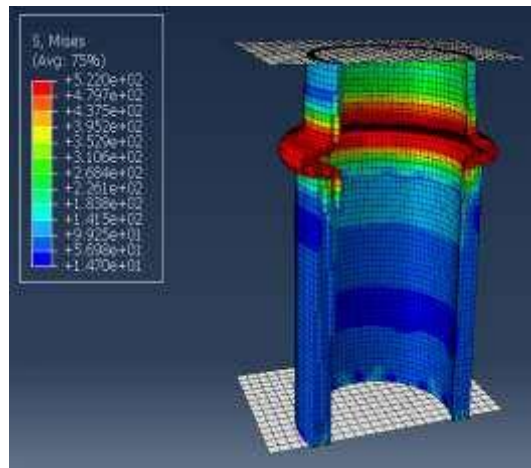
|                          |                                    |
|--------------------------|------------------------------------|
| TYPE OF ELEMENT          | linear hexahedral elements (C3D8R) |
| TOTAL NUMBER OF NODES    | 30684                              |
| TOTAL NUMBER OF ELEMENTS | 23594                              |

## VI. LOADS AND BOUNDARY CONDITIONS

As shown in the fig, above the segmented bumper beam is placed between the two rigid bodies at the top and bottom. The bottom rigid body is fixed completely which doesn't have any motion in any direction. The Boundary condition for the top rigid body will be free to move in Y axis and constrained to X and Z axis. The velocity boundary condition is given to top rigid body in negative Y direction with the magnitude of 25m/s. The top rigid body acts as the impact load which will be of an object moving at 25m/s on the column. The analysis has been done in ABAQUS /CAE.

## VII. RESULTS AND DISCUSSION

The FE model of bumper beam with loads and boundary conditions was run for analysis in ABAQUS. The following figures give the results of both stress contour and displacement contours.



The above figures show the amount of stress in the segment box column. As the force applied from the top rigid body the first segment tends to deform. Maximum stress of 522Mpa is in the first segment and the minimum stress concentration is of 43.5Mpa in the second segment. The analysis shows the stress in the first segment doesn't intrude stress in the second segment.

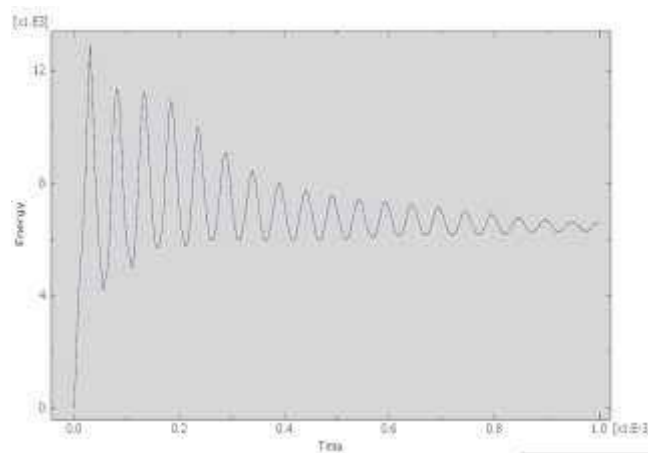


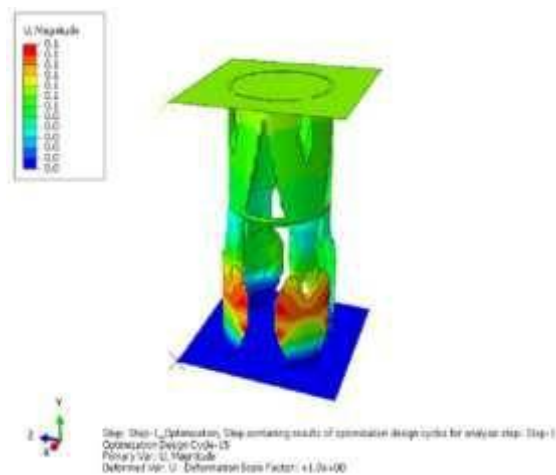
Figure 4: The plots shows the strain energy of the system

The above graph shows the history of the strain energy with respect to change in time when the load is applied on the system.

## VIII. OPTIMISATION USING TOSCA

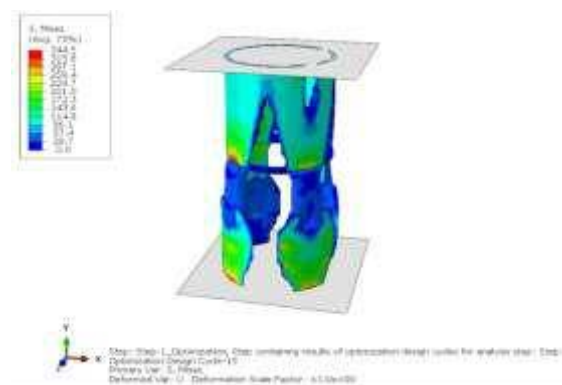
The Tosca optimisation suite built into ABAQUS/CAE platform takes full advantage of any improvement potential while leveraging advanced simulation capabilities. It creates optimised design concepts to achieve the highest performance and quality in shortest development time.

## 1. Displacement counter:



The above figures shows the total displacement in the segments when the load is applied. The total displacement is the mean of the initial position to final position of the segment. The maximum displacement in the negative Y direction is of 0.1mm which is red in color. The yellow color shows the displacement in the range of 0.1mm to 0.01mm. All the displacement is in the first segment. In the second segment there is very small displacement. Final Maximum displacement is 0.1 mm.

## 2. Stress counter:



The above figures display the quantity of strain within the section container column. As the force implemented from the top inflexible frame the primary section tends to deform. Maximum pressure of 344.4Mpa is within the first phase and the minimal pressure attention is of 28.7Mpa within the 2d phase. The analysis indicates the pressure within the first section doesn't intervene pressure within the 2d section. The pressure cost is decreased after optimization.

**International Journal of Innovative Research in Science, Engineering and Technology**

*An ISO 3297: 2007 Certified Organization*

*Volume 7, Special Issue 7, June 2018*

**One Day National Conference on “Recent Trends in Mechanical Engineering & Applied Sciences 2018”  
(RTME & AS-2018)”**

**17<sup>th</sup> May 2018**

**Organized by**

**Department of Mechanical Engineering and Applied Science, Sri Krishna Institute of Technology, Bangalore, India**

## **IX. CONCLUSION**

- The maximum stress in the segment is found to be 522 Mpa which exceeded the ultimate strength of the material to failure. After the optimisation, the stress reduced to ~344 Mpa.
- The optimised stress is within ultimate limit which will just undergo plastic deformation and avoid permanent failure.
- Displacements are found to follow the expected buckling pattern in explicit dynamic analysis.
- Total Energy is absorbed by the first segment crash box by affecting the rollup of the segment and doesn't intrude inside the second segment.
- Total 70 % of both segment weight is reduced by this topology optimisation process.

## **REFERENCES**

- [1] “Crash Analysis of Car Cross Member Bumper Beam” by G.Yedukondalu Asst. Professor K L University Guntur t.-522502, India. Dr. A. Srinat UGC Research Awardee Professor Mechanical Engg., K L University Guntur Dt.-522502. Dr. J. Suresh Kumar Professor JNTUH University Hyderabad-500085, India.
- [2] “Lumped Parameter Model for Design of Crash Energy Absorption Tubes” By Piyush Dube, M. L. J. Suman, Faculty of Engineering and Technology, M. S. Ramaiah University of Applied Sciences, Bangalore 560 054.
- [3] “Design and Analysis of an Automotive Front Bumper Beam for Low-Speed Impact” By Mahesh Kumar V. Dange, Dr. Rajesh. B. Buktar, Dr. Nilesh. R. Raykar, Department of Mechanical Engineering, Sardar Patel College of Engineering/ Mumbai University, India.