Crash analysis of torque box beam column of an automobile vehicle

Sagar H N[1] , Dr.B.Ravindra[2]

[1] *Department Of Mechanical Engineering Dr. Ambedkar Institute Of Technology Bangalore, India*
[2] *Professor and HOD Department of Mechanical Engineering Dr. Ambedkar Institute of Technology Bangalore, India*

Abstract- Crash boxes other thin walled structure which is mounted between automotive chassis and the bumper. Geometric model of torque box beam column is done using solid edge ST8 modelling tool. Once the geometry is created as per specification and it is imported to ABAQUS for meshing. This finite element model is imported to ABAQUS explicitly to evaluate the explicit dynamic analysis .segmented torque box beam columns are placed between two rigid bodies at the top and bottom. But bottom rigid bodies fixed completely it doesn't have any motion in any direction. The boundary condition for the top rigid body is free to move in Y direction only and apply the velocity of 55 km per hour on column and evaluate the result.

***Keywords—* torque box beam column; crash analysis;solid edge ST8; ABAQUS explicitly**.

1. INTRODUCTION

Crash box is one of the passive safety components, because of collision it undergoes deformation and it observes the effect energy by the crash field itself. Crash containers are a thin walled structures mounted among car chassis and bumper (fig-1) which can be used as power absorption all through crash event. Those factors play an essential role has crash worthy machine and used to reduces the numerous accidents due to frontal crash and additionally play a function to soak up the impact strength of the frontal crash by way of present process plastic deformation to shield the occupants.

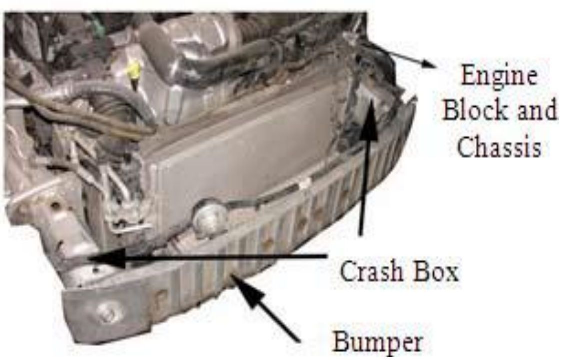

Figure- 1: Crash boxes positions mounted between automotive chassis and bumper.

Because of friction, fraction, metal slicing, torsion, crushing, cyclic plastic deformation, shear, Extension and fluid float which thanks to friction, to power soak up were designed to deplete electricity. a few of the entire above element the number one energy absorbing mechanism are cyclic plastic deformation, extension and friction.

Have a look at on two segments crash field remains restricted, wherein this layout can possible to growth the allowable crucial load and exceptional materials in every phase may be used. The crash box designs the use of hybrid cloth of aluminum and metallic had been developed as a way to reduce crash container weight up to 17.5%.

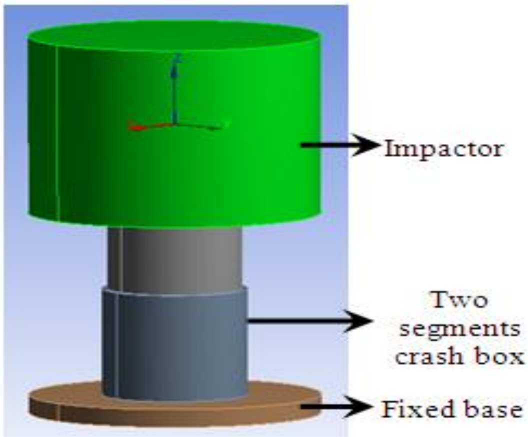

Figure -2: Two segments crash box model.

1.1 Material

Mild metallic is used for the layout of torque container beam column. It's far an Iron-carbon alloy containing less than 0.25 percentage carbon which makes it more ductile and much less hard as a result rendering it flawed for structural work.

1.2 Problem Definition

The aim of this project is to study the impact of rigid body on energy observing bumper column and optimize.
Therefore a crash simulation is performed and validated using analytical method.

And optimize the geometry of torque box bumper column.Above configuration is optimized using TOSCA by using the parameters given below

1. Connection angle between segments (θ) will be of the range 0-90 degree of the interval of10 degree each.
2. Connection length between segments (p) will be varied from 10-20mm of the interval 2mm.
3. Outside diameter (do) will be varied from 50-100mm of the interval 10mm each.
4. Segment Height and thickness ratio will be constant 40/60mm and 2/3mm respectively.

2 Methodology

The primary step is to create a geometrical version of torque container beam Column is performed the use of strong aspect ST8 modeling tool. Rigid mass is modeled and implemented with a pace equivalent to most speed specification. That is impacted at the torque container beam column. Once the geometry is created as according to specs it is imported into ABAQUS for messing. The finite detail model is ready by meshing it with suitable elements like linear hexahedral, contact factors and constraining the version via making use of cloth homes and boundary conditions. This finite element model is imported to ABAQUS explicit to evaluating the express dynamic evaluation and subsequently results are analyzed and the effect damage is studied. The response forces are located.

Following are the steps involved in Methodology:

Step1: The 3D model of the torque box beam Column is done using Solid Edge ST8, created meshing and analysis has been done using ABAQUS/CAE.

Step 2: Rigid mass is modeled and applied with a velocity equivalent to maximum speed specification. This is impacted on the torque box beam column.

Step 3: Explicit dynamic analysis is performed using ABAQUS/EXPLICIT.

Step 4: Results are analyzed and the impact damage is studied. The reaction forces are observed.

2.1 Geometric Modeling

Figure 3 shows the geometric model of a section of the torque box beam Column. The modeling has been done using SOLID EDGE ST8 modeling software. The dimensions of torque box beam Column are considers from the literature survey. It consists of two segment crash box which has the Connection angle between segments (θ) of 45 degree. Design parameters in two segments crash box is as shown in figure 3.

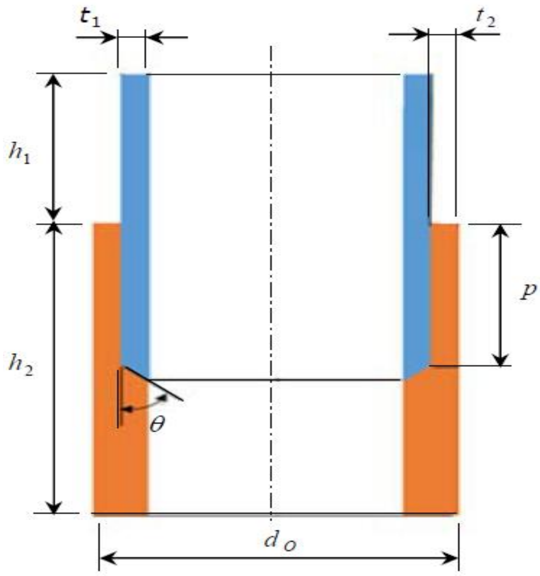

Figure -3: Design parameters in two segments crash box.

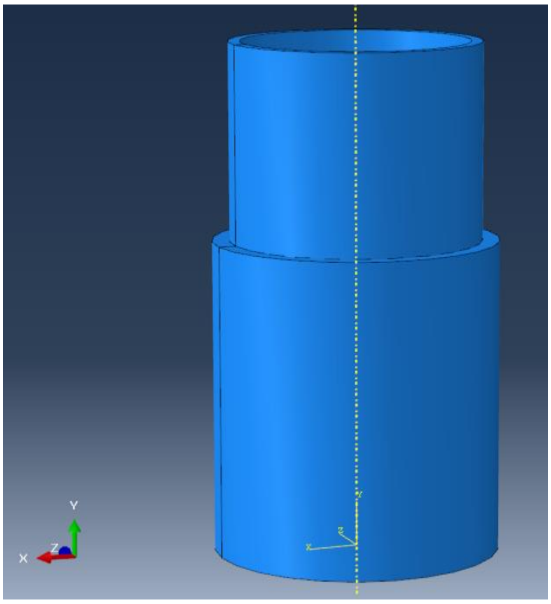

Figure- 4: Geometric models of two segments of crash box

2.2 Meshing

fig 5 suggests the Finite detail version generated for the geometric model of the 2 segments of crash field using linear hexahedral factors of kind C3D8R. those factors are selected due to the fact the quadratic end result interpolation at the combination point.

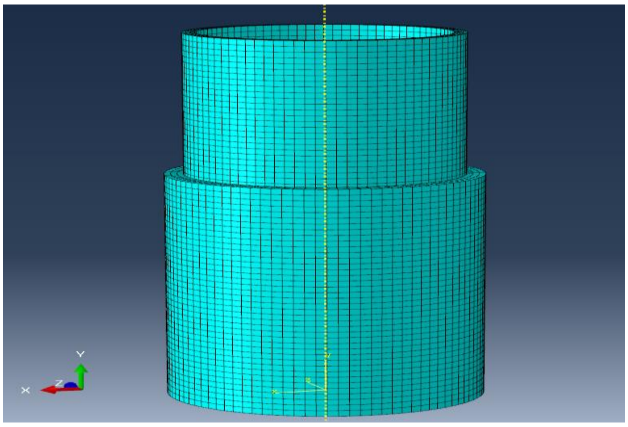

Figure- 5: Mesh models of two segments of crash box

2.3 Loads and Boundary Conditions

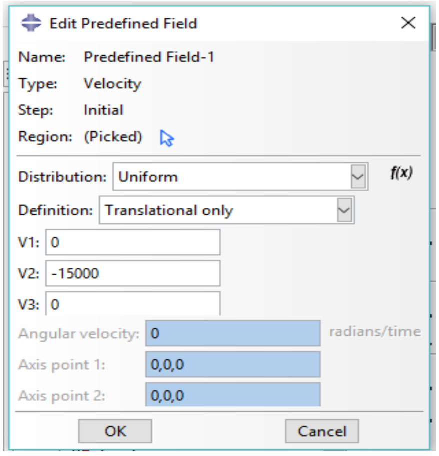

Figure- 6: Application of loads and boundary condition

The segmented crash column is places between the 2 inflexible bodies at the top and backside. The lowest inflexible frame is constant completely which would not have any motion in any route. The Boundary condition for the top inflexible body could be free to transport in Y axis and constrained to X and Z axis.

The speed boundary condition is given to pinnacle rigid body in poor Y course with the importance of 15m/s. The top inflexible body acts because the impact loads so that it will be of an item moving at 55km/h on the column. The analysis has been achieved in ABAQUS /CAE.

3 RESULTS & DISCUSSION

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

3.1 Stress Contour

The figure 7 shows the stress in the segment box column after the load is applied

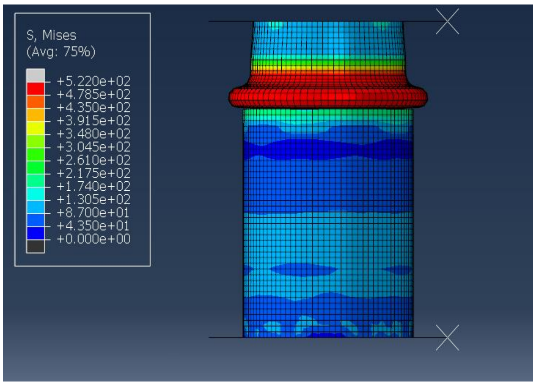

Figure -7: stress in the segment box column

The above figures show the amount of strain inside the section box column. As the pressure carried out from the pinnacle rigid body the first phase tends to deform. Maximum pressure of 522Mpa is inside the first phase and the minimal stress attention is of for 43.5.5Mpa within the second section. The evaluation shows the stress inside the first section would not intrude pressure within the 2d phase.

3.2 Displacement Contour

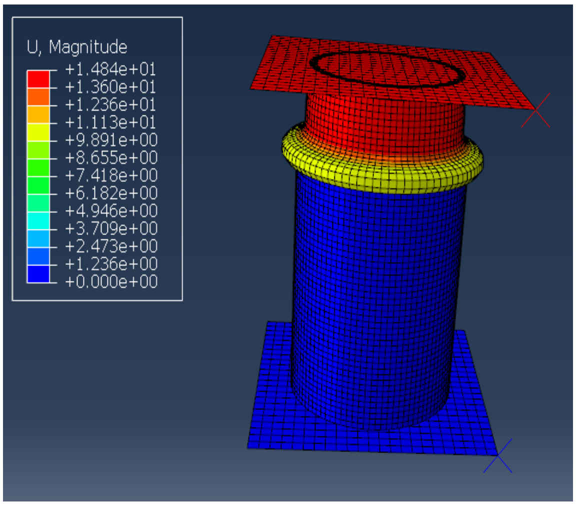

Figure -8: total displacement in the segment box column

The above figures indicate the total displacement within the phase whilst the burden is applied. The whole displacement is the suggest of the preliminary role to final function of the section. The maximum displacement inside the bad Y path is of 14.84mm which is red in shade. The yellow color suggests the displacement in the variety of nine.891mm to 11.13mm. all of the displacement is in the first section. Inside the second section there's very small displacement which is of the range0-1.23mm displacement.

4 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.

4.1 Optimization Results

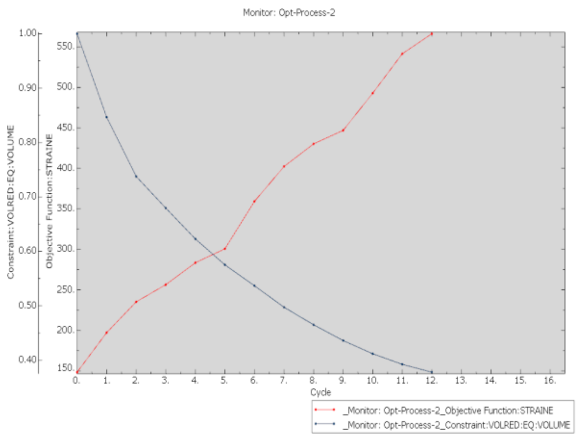

Fig -9: Starin Graph

Graph suggests the tracking of goal feature and constraint thru the optimisation steps. It shows convergence to the satisfactory possible answer. With up to 60% of the weight reduction within the first segment. pressure power is inside perfect limits.

4.2 Stress Counter

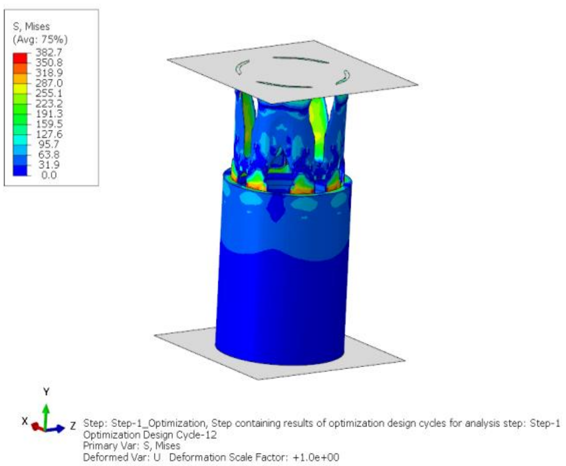

Fig-10: Stress In Segment

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 382.7Mpa is within the first phase and the minimal pressure attention is of 31.1Mpa 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.

4.3 Displacement Counter

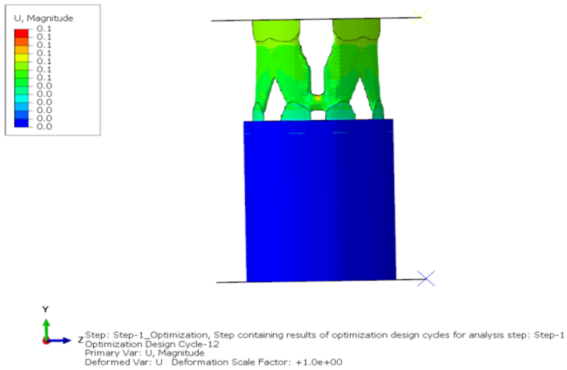

Fig-11: Totale Displacement

The above figures shows the total displacement in the segment when the load is applied.
Final Maximum displacement is 0.1 mm.

4.4 Topology of the Final Configuration of the First Segment

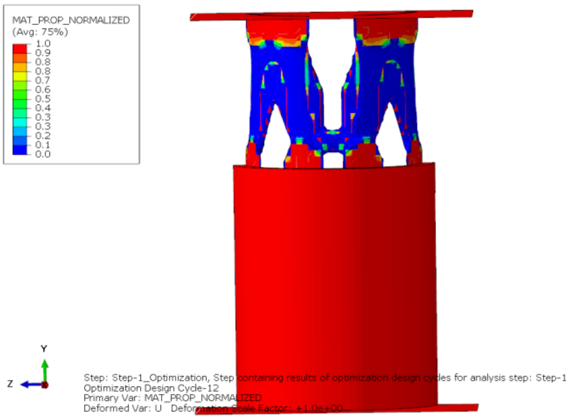

Fig-12: Topology of the Final Configuration

5 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 ~345 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 60 % of first segment weight is reduced by this topology optimisation process.

SCOPE OF FUTURE WORK

Connection angle between the segment change to 30^0, variation of diametrical ratio, thickness variation, change the material like aluminum because too much malleable, use metal -metal composite. And go for other analysis like design response from exist, static analysis, strain energy design response etc.

REFERENCE

[1] A survey paper on factors controlling the energy absorption of crash box" by Ashutosh S. Kalshetti, Sanjaysingh Vijaysingh Patil, Associate professor, Department of Mechanical Engineering RMD Sinhgad School of Engineering.
[2] "Application of analytical hierarchy process in the design concept selection of automotive composite bumper beam during the conceptual design stage", by A. Hambali1, S. M. Sapuan1, N. Ismail1 and Y. Nukman2, Malaysia.
[3] "Simulation of Automobile Side Member Collapse For Crash Energy Management"
[4] "Experimental Investigations with Crush Box Simulations for Different Segment Cars using LS-DYNA" by A.Siva Kumar Ȧ*, G.Himabindu Ȧ, M.Sankar Raman Ȧ and K.Vijaya Kumar Reddy Ḃ . India Accepted 10 January 2014, Available online 01 February 2014, Special Issue-2, (February 2014).
[5] "Simulation Of Automobile Side Member Collapse For Crash Energy Management" By Ichiro Hagiwara,Masaaki Tsuda,Vashihirosato And Yuichi Kitegawa, Nissan Moter Co Pvt Ltd.
[6] "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.
[7] "Design and Analysis of an Automotive Front Bumper Beam for Low-Speed Impact" By Maheshkumar V. Dange, Dr. Rajesh. B. Buktar, Dr. Nilesh. R. Raykar, Department of Mechanical Engineering, Sardar Patel College of Engineering/ Mumbai University, India.