TRANSIENT ANALYSIS OF IMPACT LOADS ON BUMPER BEAM AT DIFFERENT OFFSETS

Hassan A. Khayyat
Assistant Professor, Department of Mechanical Engineering College of Engineering, Shaqra University, Kingdom of Saudi Arabia

ABSTRACT

The objectives of this study were to increase the physical understanding of the different phenomena taking place during the offset impact of an automotive bumper beam-longitudinal system as well as to validate a modeling procedure for the system’s crash performance. The experimental database was used for the development and validation of modeling procedures for the crash performance of the bumper beam-longitudinal system with the use of the FE-code ANSYS-DYNA. The numerical model should be able to predict the collapse mode with a high level of certainty in order to ensure robust design.

Key words: Crash Analysis, Bumper Beam, Impact Loads.

Cite this Article: Hassan A. Khayyat, Transient Analysis Of Impact Loads On Bumper Beam At Different Offsets Journal of Mechanical Engineering and Technology, 7(2), 2016, pp. 91–100.

http://www.iaeme.com/currentissue.asp?JType=IJMET&VType=7&IType=2

INTRODUCTION

In a frontal or rear crash, the bumper beam is the primary component which undergoes damage and transfers the forces to the rest of the structure. Thus the modern bumper beam systems should play a key part in the safety concept of an automobile, ensuring that minimal accelerations are transferred to the passenger. Further the automotive producers are demanding for robust bumper beam systems showing good and reproducible impact behaviour. Manufacturing of bumper beam from aluminium extrusions often involve series of forming operations performed in the soft W- temper condition, and then age hardening of the components to the materials peak hardness condition. Thus it is clear that for proper crash performance of the systems the FE model must rely upon the geometry obtained from a simulation of the process route i.e., including simulations all major forming operations.

The bumper beam system in this study consists of a bumper beam directly connected to a longitudinal at both ends, hereafter named a bumper beam-longitudinal
system. That is, the system does not include any cashboxes. This is also the preferred system for some automotive producers, since a longitudinal will offer higher resistance to deformation, and thus give higher energy absorption than if cashboxes are used. Frontal offset crash testing has gained acceptance worldwide as an assessment of the frontal crashworthiness of vehicles. However, assessing the impact performance of bumper beam-longitudinal system through full-scale crash tests of a car is not easy as the view of the system is hidden. Thus, a separate study on the bumper beam-longitudinal system is required to understand the involved physics. This is the main motivation for the present study on bumper beam-longitudinal system at offset impact.

LITERATURE SURVEY

Toshiyuki Sawa, Yoshihito Suzuki, and Shoichi Kido used finite element method to analyze the stress wave propagations in adhesive joints of similar hollow cylinders under static and impact tensile loadings in elastic deformation range. They used DYNA3D to start the analysis and applied the impact loading to the joint by dropping a weight. The effects of the Young’s modulus of the adhesive on the stress wave propagation at the interfaces were examined and finally they found that the characteristics of the joints subjected to impact loadings were opposite to those subjected to static loadings. Thomas J. Trella, Randa Radwan, Samaha (1995) described the development and validation of a computer based model of the moving deformable barrier developed for side impact safety performance simulations using LS-DYNA3D. They investigated the effects of important factors central to FEA modeling such as material node merging, mesh density, and element type and then found that the material damping coefficient and compacted Young’s modulus both had a strong influence on the simulated impact responses.

David H. Johnson, Richard B. Englund, Brian C. McAnlis, Kevin C. Sari, and David Colombet presented a modeling technique used to create a “mostly-brick” meshed 3D model of a nut and bolt joint using ANSYS and the created 3D modeling can simulate the conditions of joint tightening and sliding along the helical thread flanks when the nut is turned.

Ford engineers developed a target-vehicle model used for computer simulation of vehicle crash compatibility. For the target-vehicle model they chose five frontal impact modes to test it, which included full frontal impact and corner frontal impact. After running the analysis the model would provide the vehicle responses and component characteristics such as compression, tension, bending stiffness and rate effects which were used to compare with the results of vehicle-to-vehicle test. The target-vehicle model was then be calibrated and optimized based on the results of comparison until an ideal target-vehicle was reached in the end. The methods and ideas utilized in the modeling process were kind of enlightening.

S. W. Kirkpatrick, J. W. Simons, and T. H. Antoun (2000) developed and validated a high fidelity finite element model of a full size car for crashworthiness analysis, which was part of an overall program to develop a set of detailed finite element models for various vehicles. In the program, they selected the Ford Crown Victoria as the representative full size car and briefly described the modeling procedure including the vehicle teardown and digitization and model generation. The techniques used in vehicle digitization; the mesh and the element type used in the FE model were introduced, and the developed FE model was presented too. The authors performed the component crash tests and vehicle crash tests separately and obtained a
set of data from the tests for validating the crash model. In their paper, they introduced the test Conditions and analyzed and compared the test results thereby concluded the overall collision response of the vehicle and verified the validity of the developed model.

From above demonstration, tremendous advancements have been made on the computer simulation of impact analysis and the FEA methods and the CAE tools had been intensively applied for solving such problems. And in this work, the path put forward in those previous literatures is followed to create and validate a computer model and it is proved that the methodology used in the study could be spread into other impact problems.

**SCHEMATIC MODEL FOR ANALYSIS:**

![Schematic Model for Analysis](image)

**RESULTS AND DISCUSSIONS**

![Results and Discussions](image)

**Figure 1** Meshed Model of Bumper Beam

**Figure 2** Boundary conditions of Bumper Beam subjected to 100% offset impact
From the Figure 3, it is observed that that the maximum deflection is 0.272755 mm for the bumper in X direction which is indicated as MX and minimum deflection is 0.016108 which is indicated as MN. From the Figure 4, it is observed that that the maximum deflection is 0.021255 mm for the bumper in Y direction which is indicated as MX and minimum deflection is -0.97787 (compression) which is indicated as MN.

**Figure 3** Deformation of Bumper beam 40% off set impact; the deflection is 1.001 mm

**Figure 4** X displacement Bumper Beam Subjected to 40% Offset Impact

**Figure 5** Y displacement Bumper Beam Subjected to 40% Offset Impact
Figure 6 Vonmises stress of Bumper Beam Subjected to 40% Offset Impact

Fig 6. shows the variation of Von-mises stress induced in the bumper beam subjected to 40% offset impact. It is observed that the maximum stress is 39.262 MPa and is less than the ultimate strength of bumper material (425 MPa). Where as in the beam stresses induced are much less and ultimate strength of steel is 650 MPa. Hence the design is safe based on strength.

Figure 7 Deformation of Bumper beam 60% offset impact

From the Figure 7, it is observed that that the deflection is 1.961 mm. From the Figure 8, it is observed that that the maximum deflection is 0.37968 mm for the bumper in X direction which is indicated as MX and minimum deflection is -0.162534 (compression) which is indicated as MN.
Figure 9 Y displacement Bumper Beam Subjected to 60% Offset Impact.

From the figure 9, it is observed that that the maximum deflection is 0.28311 mm for the bumper in Y direction which is indicated as MX and minimum deflection is -1.954 (compression) which is indicated as MN. Fig 10 shows the variation of Von-mises stress induced in the bumper beam subjected to 60% offset impact. It is observed that the maximum stress is 55.061 MPa and is less than the ultimate strength of bumper material (425 MPa). Where as in the beam stresses induced are much less and ultimate strength of steel is 650 MPa. Hence the design is safe based on strength.

Figure 10 Von-mises stress of umper Beam Subjected to 60% Offset Impact.

Figure 11 Deformation of Bumper beam 80% offset impact.
Figure 12 X displacement Bumper Beam 80% offset

From the Figure 11, it is observed that the deflection is 2.076mm. From the Figure 12, it is observed that the maximum deflection is 0.342425mm for the bumper in X direction which is indicated as $M_X$ and minimum deflection is -0.255394 (compression) which is indicated as $M_N$.

Figure 13 Y displacement Bumper Beam Subjected to 80% Offset Impact

Figure 14 Von-mises stress of Bumper Beam Subjected to 80% Offset Impact

From the Figure 13, it is observed that the maximum deflection is 0.02423mm for the bumper in Y direction which is indicated as $M_X$ and minimum deflection is -2.075 (compression) which is indicated as $M_N$. Fig 14 shows the variation of Vonmises stress induced in the bumper beam subjected to 80% offset impact. It is observed that the maximum stress is 52.146MPa and is less than the ultimate strength of bumper material (425MPa). Where as in the beam stresses induced are much less and ultimate strength of steel is 650MPa. Hence the design is safe based on strength.
Figure 15 Deformation of Bumper beam 100% offset impact

From the Figure 15 it is observed that that the maximum deflection is 1.663mm.

Figure 16 X displacement Bumper Beam Subjected to 100% Offset Impact

From the Figure 16 it is observed that that the maximum deflection is 0.24353mm for the bumper in X direction which is indicated as MX and minimum deflection is -0.24353(compression) which is indicated as MN.

Figure 17 Y displacement Bumper Beam Subjected to 100% Offset Impact
Transient Analysis Of Impact Loads On Bumper Beam At Different Offsets

**Figure 18** Von-mises stress of Bumper Beam Subjected to 100% Offset Impact

From the Figure 17 it is observed that the maximum deflection is 0.016563mm for the bumper in Y direction which is indicated as MX and minimum deflection is -1.663(compression) which is indicated as MN. Fig 18 shows the variation of Vonmises stress induced in the bumper beam subjected to 100% offset impact. It is observed that the maximum stress is 41.228MPa and is less than the ultimate strength of bumper material(425MPa). Where as in the beam stresses induced are much less and ultimate strength of steel is 650MPa. Hence the design is safe based on strength.

**Figure 19** Displacement Velocity and Acceleration at 40%, (ii) 60%, 80% and 100% offsets
Table 1 Von-mises stress of Bumper Beam Subjected to various Offset Impacts

<table>
<thead>
<tr>
<th>S.No</th>
<th>Bumper Beam</th>
<th>Von-mises Stress (N/mm^2)</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>40% offset Impact</td>
<td>39.262</td>
</tr>
<tr>
<td>2</td>
<td>60% offset Impact</td>
<td>55.061</td>
</tr>
<tr>
<td>3</td>
<td>80% offset Impact</td>
<td>52.146</td>
</tr>
<tr>
<td>4</td>
<td>100% offset Impact</td>
<td>41.228</td>
</tr>
</tbody>
</table>

CONCLUSIONS

[1] Transient Analysis of Bumper Beam-Longitudinal System has been carried out using ANSYS software.
[2] The bumper beam is modeled with BEAM188 element and the induced stresses (55MPa) are within the allowable stresses (225MPa). Hence the design is safe based on strength criteria. The factor of safety (f.o.s) is 2.
[3] The deflections induced in bumper beam is well within limits (max is 2.076mm). Hence the design is safe based on rigidity criteria.
[4] The Transient Analysis of bumper beam reveals that the maximum displacement, velocity and acceleration respectively are 2.15mm,0.215mm/s and 0.115mm/s^2.
[5] The bumper beam model is stable and static and dynamic loading.

REFERENCES