NONLINEAR STATIC SIMULATION OF AUTOMOTIVE BUMPER OF A PASSENGER CAR IN LOW SPEED IMPACT CRASH SIMULATION

Rajshekhar A Kharde[1], S. F. Patil[2]

[1]*M.Tech Student, KLE Dr. M.S Sheshgiri College of Engg & Tech, Belgaum, Karnataka, India*
[2]*Mechanical Engineering, KLE. Dr. M.S Sheshgiri College of Engg & Tech, Belgaum, Karnataka, India*

Abstract

Bumpers are important structural components of vehicles in terms of safety, static strength, and styling. This work describes various aspects of the nonlinear static FEA of vehicle front bumpers using the commercial software Ls-Dyna, and the correlation with physical tests. The basic concepts, procedures and techniques of nonlinear FEA are presented. The effects of various aspects of nonlinear simulations like numerical integration, contact parameters, non-conformance of the mesh and material law are studied. A nodally integrated element formulation for plates is proposed for analysis, and the formulation is implemented in Ls-Dyna using the UEL user subroutine. Numerical tests are carried out on the new element. Suggestions are proposed in simulation to improve the correlation of CAE results with physical test.

Keywords: *CATIA V5, HYPERMESH v11, EPS, Ls-Dyna, ASTM D880.*

---***---

1. INTRODUCTION

Bumpers are automotive parts situated at the front and rear of vehicle to ensure the protection of a vehicle body in case of a low speed crash. This is enabled via the formability of bumpers and energy absorption during an impact. The crash capability of the bumper should also meet worldwide regulations regarding the safety of passenger this dissertation describes the nonlinear static simulation of stiffness performed on an automotive front bumper. A brief introduction to automotive front bumpers, and to nonlinear static simulation is presented [1].

1.1 Bumpers

Bumpers are fixed on the front and on the back side of a car to provide protection. They reduce the effects of collision with other cars and objects due to their large deformation zones. The bumpers are designed and shaped in order to deform itself and absorb the force (kinetic energy) during a collision [2].

Various national regulations have been enacted with regard to the performance of bumpers under low speed impact such as 49 CFR (Code of Federal Regulations) Part 581 of the National Highway Traffic Safety Administration (NHTSA) in the United States, the United Nations Economic Commission for Europe (ECE) regulation No. 42, and the CMVSS (Canada Motor Vehicle Safety Standard) 215. There are no AIS (Automotive Industry Standards) regulations for bumpers in India as of 2011.

With the increasing emphasis on energy efficiency in automobiles due to the rising cost of fuel, there is a push towards reducing the weight of all components, including bumpers. Thus, the challenge in bumper design is to maintain a delicate balance between vehicle weight reduction, having sufficient stiffness, and meet safety standards at the same time [1].

The whole frontal bumper system consists fascia, absorber and grille.

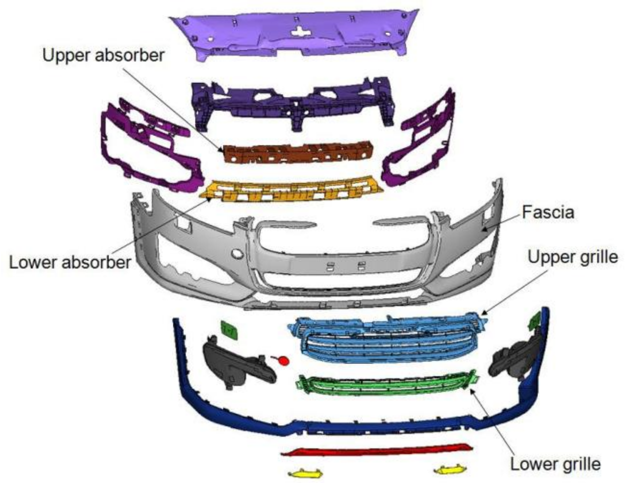

Fig -1: Frontal bumper system components

In this work simulation results are validated by experimental results and displacements at critical points such as 5, 7, 8 and 9 are computed. In this paper, the focus is on improving the accuracy of the nonlinear static simulations performed on vehicle front bumpers to evaluate their stiffness and strength.

2. TYPES OF NONLINEARITY

There are two common sources of nonlinearity: (1) geometric and (2) material. The geometric nonlinearity arises purely from geometric consideration (e.g. nonlinear strain-displacement relations), and the material nonlinearity is due to nonlinear constitutive behavior of the material of the system. A third type of nonlinearity may arise due to changing initial or boundary conditions [3].

3. EXPERIMENTAL TESTING OF BUMPER

The physical tests on the bumper were carried out at the laboratory of Faurecia Automotive Exteriors. The test conditions for subjective stiffness are known. They define the acceptable maximum displacement under a defined effort, the type of impactor and the test locations on the bumper fascia. This data will also vary as per the bumper model for the same client (sedan, utility, etc.).

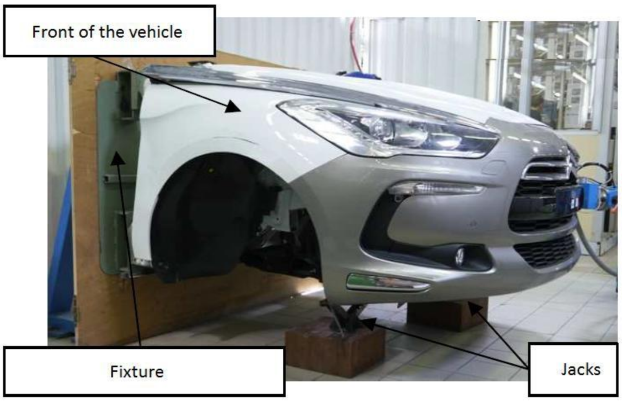

Fig -2: Test setup for full bumper system

The front or rear of a vehicle is cut and mounted on a plaque that can adapt to the arrangement of the Faurecia laboratory. It is also supported by two jacks on the bottom to avoid excessive vertical movement during the performance of stiffness tests. This setup is provided by the manufacturer. The bumper is attached to this unit via the standard attachment points defined for this purpose.

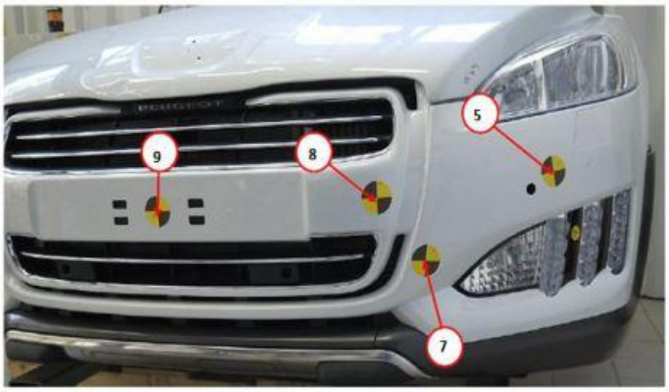

Fig -3: Test positions

4. MODEL PREPARATION

The first step in the model preparation is to mesh all the individual parts based on the CAD file which is created in Catia V5. The CAD file is meshed using a pre-processing software like ANSA and Hyper mesh. While meshing, it is necessary to maintain minimum quality criteria for every element. A complete meshed model is shown in Fig. 4

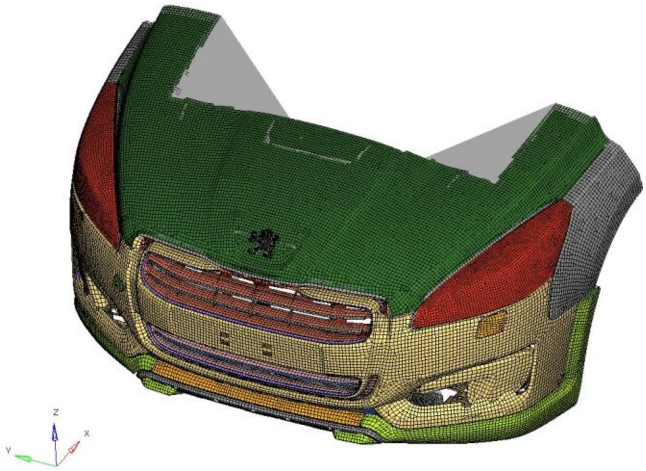

Fig -4: Complete meshed model of bumper

5. SIMULATION OF BUMPER

The vehicle manufacturer's specification gives the maximum displacement of the areas to be tested under a given static load. The compression force is applied normal to the skin (fascia). A mark is applied to the fascia to indicate the position to be tested.

5.1 Simulation Results

5.1.1 Point 5

Fig.5 shows the displacement contour for subjective rigidity test on test position 5.Where the maximum displacement is about 14.41mm for applied load of 250N.

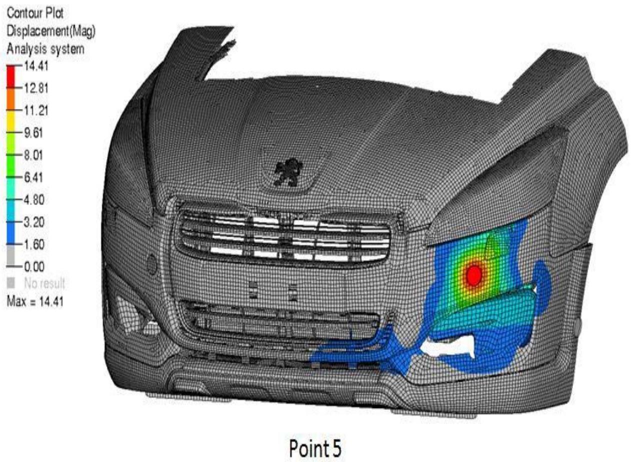

Fig -5: Displacement contour plots for various positions of bumper

5.1.2 Point 7

Fig.6 shows the displacement contour for subjective rigidity test on test position 7.Where the maximum displacement is about 5.01mm for applied load of 250N.

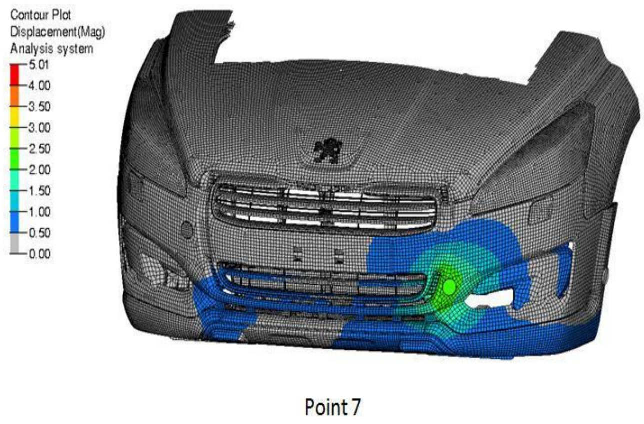

Fig -6: Displacement contour plots for various positions of bumper

5.1.3 Point 8

Fig.7 shows the displacement contour for subjective rigidity test on test position 8.Where the maximum displacement is about 31.50mm for applied load of 250N.

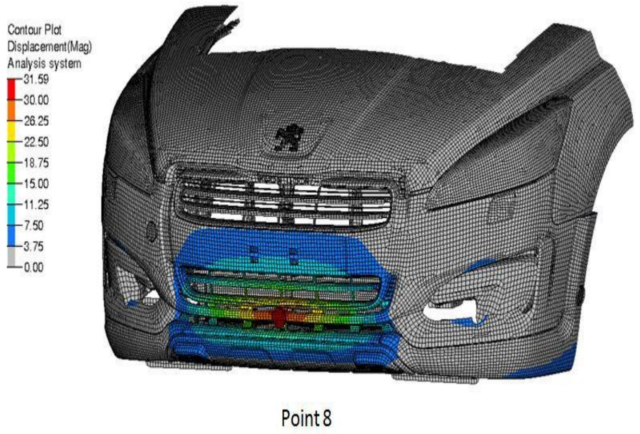

Fig -7: Displacement contour plots for various positions of bumper

5.1.4 Point 9

Fig.8 shows the displacement contour for subjective rigidity test on test position 9.Where the maximum displacement is about 10.10mm for applied load of 250N.

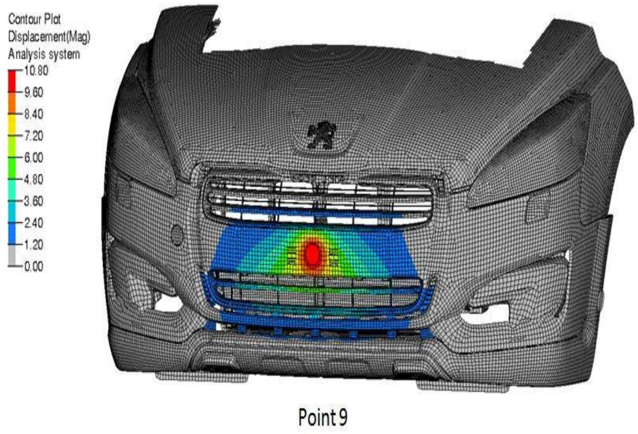

Fig -8: Displacement contour plots for various positions of bumper

6. RESULTS AND DISCUSSION

6.1 Point 5

Fig. 9 shows correlation between physical test result and CAE test result at Point 5. In physical test the displacement is about 10.1mm for applied load of 250N and in CAE test the displacement is 11.5mm for applied load of 250N. The percentage of error based on test is 28.36%.

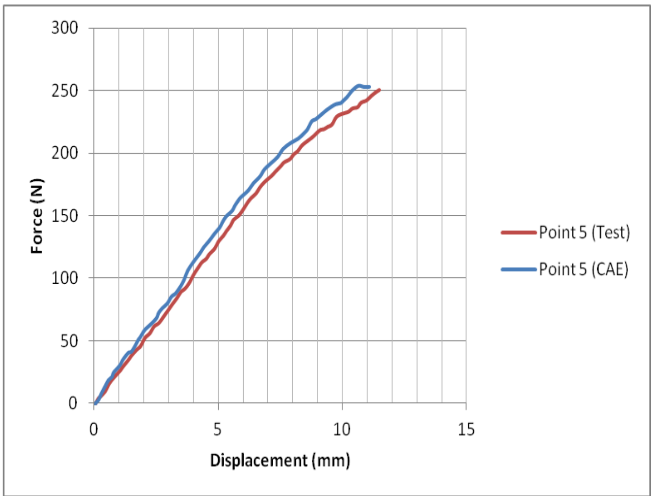

Fig -9: Correlation Load-Deflection curves at Point 5.

6.2 Point 7

Fig.10 shows Correlation between physical test result and CAE test result at Point 7. In physical test the displacement is about 5.86mm for applied load of 250N and in CAE test the displacement is 5.1mm for applied load of 250N. The percentage of error based on test is 3.4%.

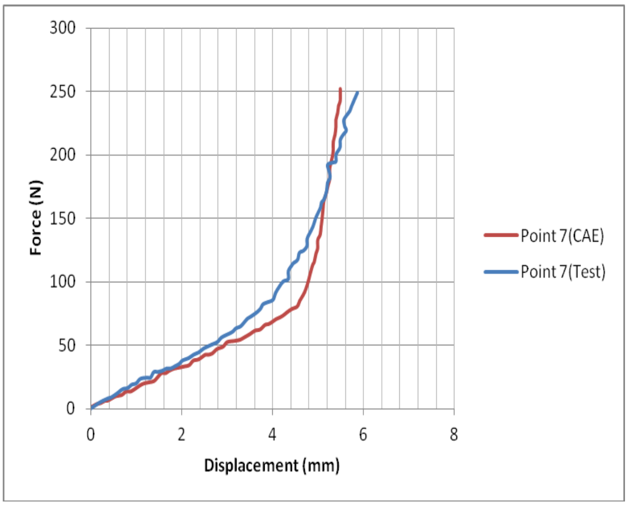

Fig -10: Correlation Load-Deflection curves at Point 7.

6.3 Point 8

Fig.11 shows Correlation between physical test result and CAE test result at Point 8. In physical test the displacement is about 30mm for applied load of 250N and in CAE test the displacement is 31.50mm for applied load of 250N. The percentage of error based on test is 12.02%.

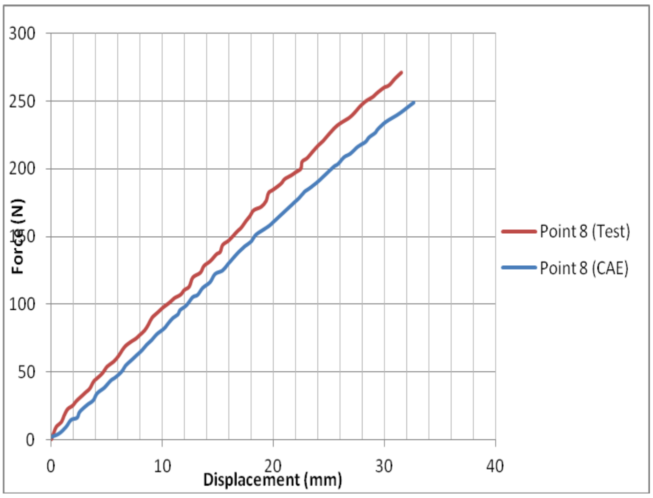

Fig -11: Correlation Load-Deflection curves at Point 8.

6.4 Point 9

Fig.12 shows Correlation between physical test result and CAE test result at Point 7. In physical test the displacement is about 7.8mm for applied load of 250N and in CAE test the displacement is 7.8mm for applied load of 250N. The percentage of error based on test is 27.77%.

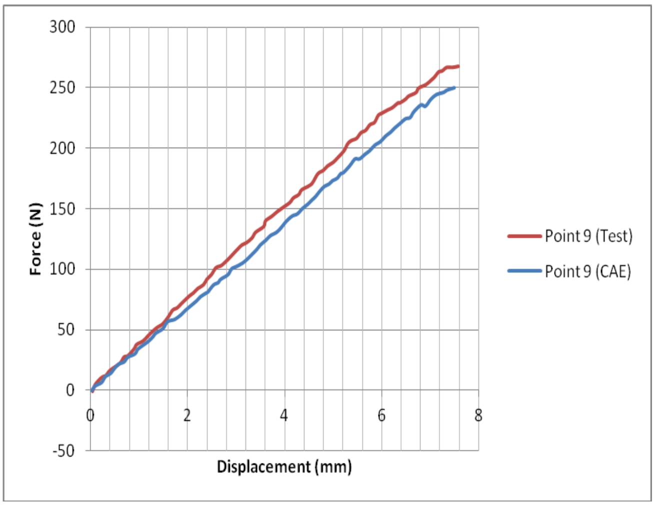

Fig -12: Correlation Load-Deflection curves at Point 9.

The improved percentage of error based on test is shown below in table-1,

Table -1: Improved percentage of error based on test.

Points	**Test**	**Mesh conformed**	**Mesh conformed, Contact and penetration corrected**	**Mesh conformed, Contact and penetration corrected, Material Law modified**	**Error %**
5	10.1mm	14.1mm	13.5mm	11.5mm	28.36
7	5.86mm	5.6mm	5.3mm	5.2mm	3.4
8	30mm	34.1mm	34.0mm	31.50mm	12.02
9	7.8mm	10.8mm	10.5mm	7.8mm	27.77

7. CONCLUSION

The contact and penetration errors, mesh conformance and material law do have important effects on the results. The main causes for non-correlation were inaccuracy of contact and penetration modeling, mesh and conformance errors, and material law difference. However, the contribution of each of these factors varies as per the load case. As can be seen from Figs. 9, 10, 11 and 12, after these aspects are taken care, the results are improved and there a good correlation of 88% between the CAE and physical test results.

REFERENCES

[1]. Bhavesh A. Bohra,Pawar. D.B, Comparative analysis of frontal car bumper during impact, ISSN 2319 – 4847, 2014.
[2]. Kee Joo Kim, Jun-Hyub Park, Crash simulation and development of automotive bumper impact beam design for weight reduction, Tongmyong University, 2012.
[3]. Reddy, J. N., An Introduction to Nonlinear Finite Element Analysis, Oxford University Press, 2004.
[4]. Ls-Dyna Analysis User's Manual, ver 6.10.
[5]. Radioss Theory Manual, ver 10.0, Part 2, Section 9.3, Elasto-plasticity of isotropic materials, 2009.