Journal of Analysis and Computation (JAC)
(An International Peer Reviewed Journal), www.ijaconline.com, ISSN 0973-2861
Volume XVI, Issue IV, Jan-June 2022

EXPERIMENTAL AND ANALYTICAL STUDY OF HEAD IMPACT SIMULATION FOR AUTOMOTIVE DISPLAY

Sampat N Pathare, Hanumant P Jagtap, and Pramod B. Magade,

Zeal College of Engineering and Research. Pune Survey No, 39, Dhayari Narhe Rd, Narhe, Pune, Maharashtra 411041

ABSTRACT

During the development of vehicle instrument panels (IP) and electronic devices, occupant head impact simulations on IP and electronic devices are frequently done as part of an integrated design process. The head impact zone for electronic devices and IP will be determined by the US FMVSS No. 201U and the UN ECE R21 Guideline. From this, the crucial location for the system's head impact zone is determined, which is then utilized to select the numerous head impact locations to be tested/analyzed. There are two types of head impact test rigs available: one with a pendulum and one with a linear head impactor. Either of these two test rigs can be utilized for the testing. A high-speed camera and an accelerometer are used to collect test data. Data collection system collects and processes test data in the form of time and acceleration of components. Geometrical requirements, component deacceleration, and no sharp edges are all design requirements for passing the head impact simulation. The completed design is examined in order to investigate the various parameters that influence the product's performance in simulation. This research will aid in the CAE simulation of head impacts for IP and electronic gadgets (Automotive Display). This research also assures that head impact CAE assessments on electrical devices mounted on IP are performed in a consistent manner.

Keywords: Head Impact Simulation, Electronic Devices, Automotive Display, U.S. FMVSS No. 201U, UN ECE R21, deacceleration, Crumping Zone, Radioss

1. INTRODUCTION

Passenger cars are the most popular means of transportation all over the world, and when it comes to cars, the most important design consideration is passenger car occupant safety. Today, we'll discuss head impact in the automobile interior, which occurs frequently during car accidents. The human body, propelled by inertia forces, flies against the cockpit in a random direction. Also, as seen in Fig. 1, the occupant's head may collide with electronic gadgets situated in front of the instrument panel.

Fig. 1. Real accident test image

The time it takes for the head impact to appear is measured in milliseconds. The human eye blinks every 300 to 400 milliseconds, whereas head impact takes only 10 to 100 milliseconds. The maximum energy that a human head can withstand before fracture is 0.865 J, and the event occurs in a very short time with a high kinetic energy. The total energy during a head impact test, on the other hand, is 151 J. The HIC value should not exceed 1000 in order to ensure safe design. If the HIC value is more than 1000 at that point, a head injury will occur. [1-7], or product deacceleration is observed.
In short duration high energy Head impact may cause different lesions of the head that could lead to specific injuries [2]

1. Skull deformation - skull fracture.
2. Relative motion between the brain and skull - increase the CSF (cerebrospinal fluid) pressure, which may cause collection of blood between brain and skull.
3. Intracerebral stress/strain (Pressure gradient) - Diffuse axonal injuries.
4. Most common facial skull fracture is the broken nose.

In the design and development of an automotive instrument panel, occupant head impact simulations are a useful tool. This study describes head impact regulation pertaining to Instrument Panels, identifying head impact zones, physical testing, This Study will help prepare a guideline for CAE simulation for Head Impact simulation for IP and Electronic devices, this study also ensures standardization in performing head impact CAE analyses on IPs and electronic devices mounted on IP. [3]

This paper also includes the outcomes of a case study, as well as the variability in results due to operator influence. It is important to emphasize that the process described in this article is the best interpretation based on experience, existing regulations, SAE, and OEM suggested practices, and does not claim to be the only or even the best way to comply with existing regulations.

2. PROBLEM STATEMENT

Head Impact Simulation help to design the IP and electronic components in such way that, parts will allow sufficient deformation space to reduce the loads on the head and reduce the Injuries to human and increase the survival chances form the crash accidents.
While testing the electronic device for head impact, during the testing the following failures are

observed. The main goal of this research is to provide a quick prediction of failure location and solution to reduce the Head injuries of occupant by reducing the deacceleration using explicit simulation

The objective of this project is to re simulate the test failure of product in software and optimize the mounting bracket in such a way that it will absorb the impact energy and reduce the overall g level on the product. Optimization of bracket can be done using the CAE tools.
Mounting bracket is one of the strongest Structural members in Product assembly, to reduce the overall g level of the product in head impact testing, Reduced stiffness of mounting bracket will help, using crumple zone it is possible to reduce the overall g level on product in impact testing along with this, for impact at top center plastic parts shows failure, to reduce this failure, Support is required at this section
Impact simulation using CAE tools will help to optimize the mounting basket to absorb the impact energy and reduce the deacceleration for Head impact simulation, in this Project Software which used are CATIA and Hypermesh and Radioss. [7] [12]
CATIA software is used for creating models of Mounting Bracket Concepts. Hypermesh software is used meshing and Radioss is used for analysis.

3. PASSING CRITERIA

For the safe design point of view the deacceleration after the impact on dummy head should not more than 80g. If the deacceleration value is more than 80 g at that particular point, then there will be head injury will take place. for head impact test. which includes requirements concerning the maximum head acceleration.[1][3][7].

1. Geometrical requirements -Rounding radii R >2.5 mm, Area A >2 cm^2 ref fig 3.4.1 [11]
2. Deceleration - should be less than 80 g for continuous 3ms time. Using 20% safety margin for sub system level analysis, the acceptance criteria for deacceleration 64 g for continuous 3 ms.[11]
3. No sharp edges or any dangerous design are allowed. After the impact no flying particles detached from investigated structure towards the occupants are allowed too. (Sharp edge is when R<2.5 mm.).[11]

4. DESIGN AND SIMULATION SETUP

CAD modeling for Automotive display is carried out in CATIA V5, and then same model is transferred to Hypermesh for FEA modeling, all plastic parts are meshed with 2D shell mesh and Bracket is mesh with tera mesh with 2 layers in critical area. A dummy rigid head impactor having diameter of 164 mm and mass of 6.8 Kg is modeled. Impact analysis is performed with current mounting bracket and to study the effect of crumple zone 3 proposal are made.

4.1. Simulation Setup:

Explicit dynamic Analysis is performed at sub system level, in this analysis 80 J energy is considered for calculation, dummy impactor is impacted at desired location as shown in Fig. 2. with initial velocity of 17.46 km/hr. Following output are recorded in simulation, Energy distribution, Deaccelerations of dummy impactor for 3 ms, energy absorbed by glass and bracket. Due to product confidentiality, representation of automotive device was used. All simulations in this report were performed on a detailed FEM model meshed with midsurface and tetra solid elements. Meshed model consisted more than 100 parts, 1st order 213715 elements, nonlinear contacts and materials with failure criteria.

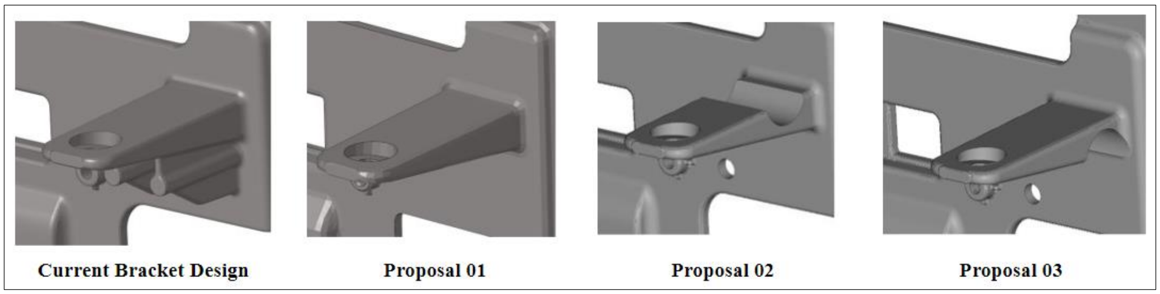

Fig. 2. Assembly of Display with head impactor

Analysis is performed with current mounting bracket design and 3 mounting bracket proposal with reduced localize stiffness as shown in Fig. 3., following are the design for current design and proposal for mounting bracket. In proposal 1 ribs provided for mounting lug are removed and for Proposal 02 and Proposal 03 notch is provided to break the bracket to reduce the output g acceleration.

4.2. Material Properties for Display and Mounting Bracket

Parts of display are made up of plastic, Glass, Steel, and magnesium alloys, following are the properties are used in simulation, shown in Table 1. Plastic Strain 0.027 at break of mounting bracket is calculated form yield strength 150 MPa, 3% elongation at break and ultimate tensile strength 230 MPa of material. Material properties are referred for MatWeb [14]

Part	Density mg/mm³	Young's Modulus (E) MPa	Poisson's ratio
Glass	2.39E-09	69000	0.22
Mounting Bracket	1.75E-09	47600	0.33
Plastic	1.13E-09	2250	0.36
Steel	7.88E-09	210000	0.29

Table 1. Material Properties used in simulation

4.3. Simulation results

- In current mounting bracket design, output acceleration observed is 77 g as shown in Fig. 5, this output acceleration should be reduced to less than 64 g, to reduce the head injuries. In assembly Mounting bracket is main structural component which contribute for stiffness of assembly. Hence, Mounting Bracket is chosen to increase deformation off assembly, and this will help to decrease the deacceleration in assembly. Due to Strong Mounting bracket, Impact Energy is get absorbed by other decorative parts and shows failure, to reduce this failure Mounting bracket need to modify such way that it will absorb the significant Impact energy and reduce the output acceleration.
- Comparative study is performed for impact at the center of display glass for mounting bracket current design and 3 mounting Bracket Proposals.
- Energy balance is converging well, and for proposal 02 and proposal 03 bracket shows failure at 6.6 ms. These two brackets are broken before maximum internal energy is achieved. Bracket is broken before the maximum internal energy it will help to reduce the stress on glass. shown below Fig.4

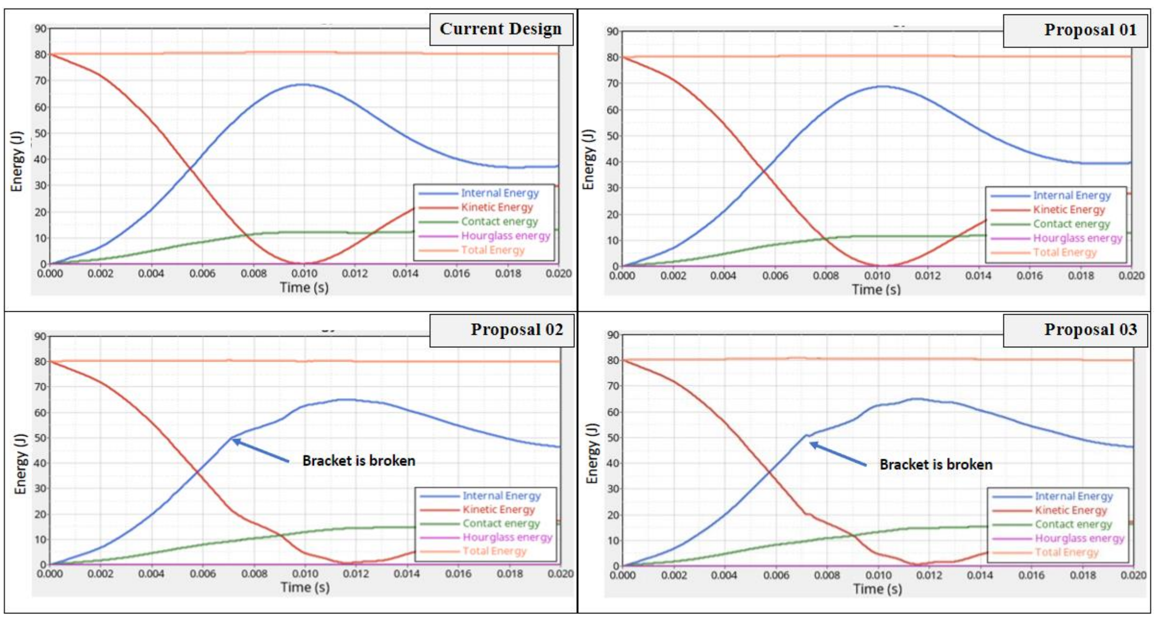

Fig. 4. Energy Convergence

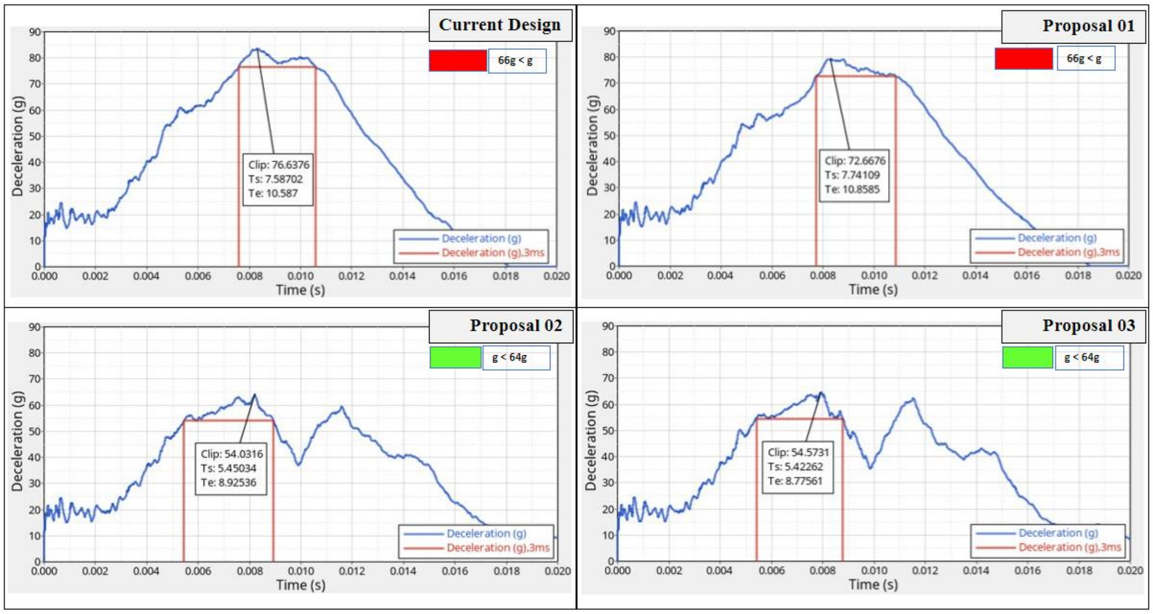

Fig. 5. Output Acceleration (g)

- Internal energy of parts will give the idea about how the bracket will absorb the energy in impact simulation and due to rupture of mounting bracket internal energy of the glass get reduce, this will help to reduce the stress on glass. Due to crumple zone on mounting tab, Internal energy of glass get reduced from 6.5J to 4.5J as shown in Fig. 6.

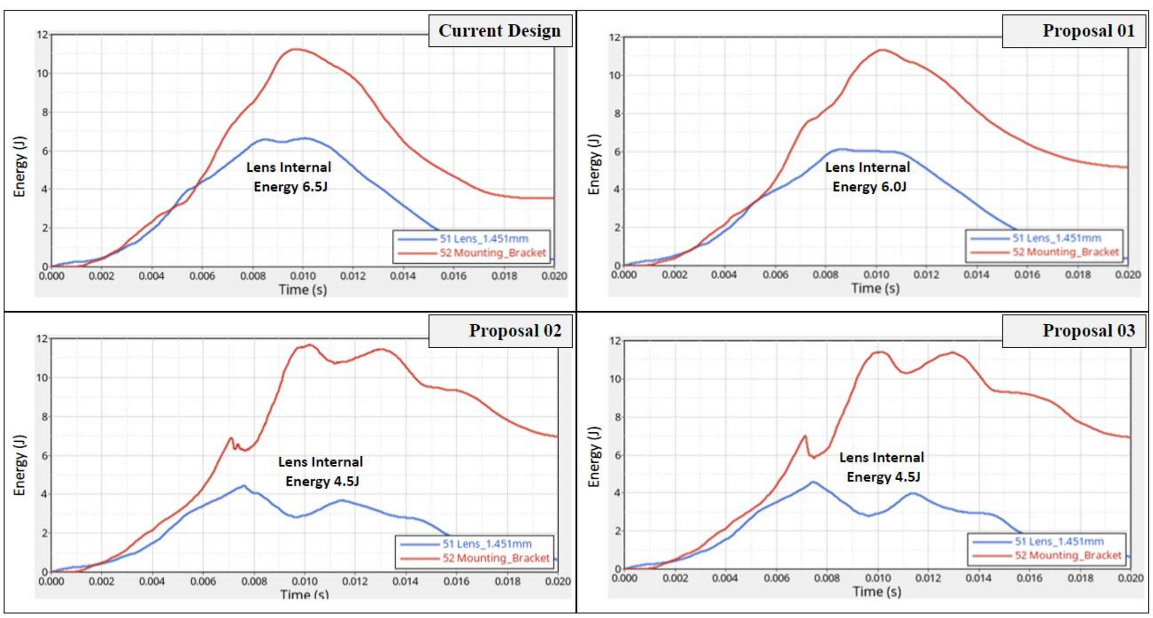

Fig 6. Internal Energy of Mounting Bracket and Display Lens (g)

- Plastic strain on current bracket design is 0.014. Plastic strain bracket keeps on increase as we reduce the stiffness of bracket, in proposal 02 and Proposal 03 bracket exceeds the braking strain as shown in Fig. 7. After 6.6 ms bracket will break and absorb the impact energy.

+

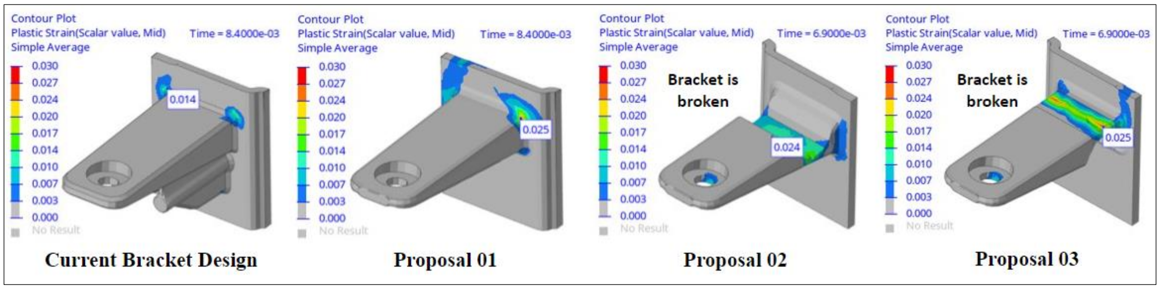

Fig. 7. Plastic Strain on Mounting Bracket

4.4. Experimental validation

From the CAE simulation Proposal 03 is selected as best candidate in terms of head impact requirement and manufacturing. Head impact testing is carried out it in testing laboratory for Proposal 03, test sample mounting lug design before and after testing. Output acceleration of dummy head is found to be 52 g. and one mounting bracket is broken in head impact testing.

Test Observation:

1. Output acceleration for 3 ms observed on dummy head impactor is 52 g.
2. One mounting bracket lug is broken as shown in Fig. 8. Another mounting bracket is holding the product on its place.
3. No crack is found on lens.

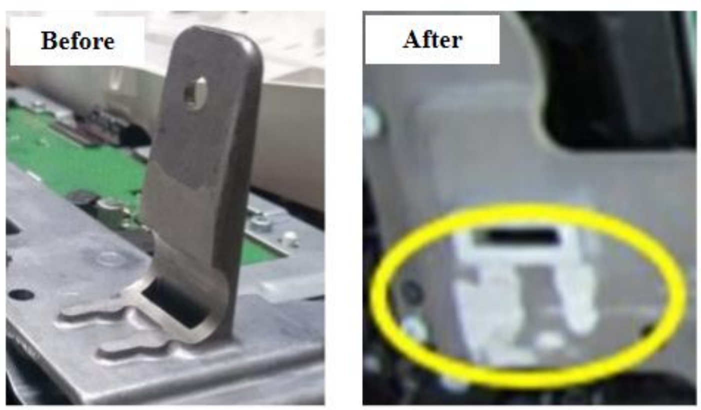

Fig. 8. Mounting Bracket Before and After impact testing

5. RESULT DISCUSIONS

1. Proposal 01, Proposal 02 and Proposal03 show reduced acceleration level after impact compared to current design, hence this concept will produce relatively lower stress on lens after impact.
2. In Proposal 01 acceleration level after impact are more that 64g and stress on Lens are also not meeting the acceptance criteria for stress.
3. Proposal 02 and proposal 03 are showing equivalent results, out of this two proposals Proposal 03 is considered as final Design for mounting Bracket design.

4. CAE results are validated in testing.

Load Case	Results	Current Design	Proposal 01	Proposal 02	Proposal 03
HIT Point 1 Display Centre (V=17.46 km/hr)	Deceleration (g)	77	73	54	55
	Bracket Plastic Strain	0.014	0.025	Bracket Broken	Bracket Broken
	Conclusion	NOK	NOK	OK	OK

Table 2. Result Summary

6. CONCLUSION

Head Impact Simulation help to improve the design of IP or IP mounted electronic system design at very initial stage. This will reduce the actual testing cost by testing combination of parameter. After finalizing the parameter design team can go for soft part and the Design phase actual testing.

1. If the part is weak to sustain the head impact load, then this part has made stiff such way that they will not deform significantly and get survive in head impact load. on other hand, if part is significantly stiff then we have to make crumple zone to deform the geometry to absorb the impact energy.
2. Proposal2 and Proposal3 will help to reduce the peak acceleration in head impact testing
3. Crumple zone is used to minimize the acceleration by changing acceleration direction vector by rupture and this will help to reduce deceleration and stress on front glass.
4. Simulation results are validated by performing actual impact test, and Product pass the validation test for given specification.

REFERENCES

[1] Gerald Joy Sequeira, Thomas Brandmeiera, "Evaluation and characterization of crash-pulses for head on collisions with varying overlap crash scenarios," Science Direct.

[2] Erik G. Takhounts, Matthew J. Craig, Kevin Moorhouse, Joe McFadden, "Development of Brain Injury Criteria (BrIC)" Stapp Car Crash Journal, Vol. 57 (November 2013), pp. 243-266

[3] Mike Keranen, Srikanth Krishnaraj, Kumar Kulkarni, Li Lu and, Ravi Thyagarajan, "Automating Instrument Panel Head Impact Simulation," SAE Int'l 2005-01-1221.

[4] Chris Jurewicz, Amir Sobhani, Jeremy Woolley, Jeff Dutschke, Bruce Corben, "Exploration of vehicle impact speed-injury severity relationships for application in safer road design," SAE Int'l 2005-01-1221.

[5] Michael Walter, Hartmut Chaladek, Dr.Armin Hub, "Optimization of a Cockpit Structure according to ECR-R21 Regulation," Tooling and Production, Nov. 1990, 54-61.

[6] S. lskander Farooq, Peter J. Schuster, Ford Motor Company, "Body Concept Design for Pedestrian Head Impact," SAE Int 2003.01·1300

[7] Shreyansh Vijay Patil, Prof. S. A. Pachapuri, "OPTIMIZATION OF INSTRUMENTAL PANEL TO REDUCE TRAUMATIC BRAIN INJURY," IRJET,Volume: 04 Issue: 09 | Sep -2017.

[8] S.Srinivasa Ragavan, Dr.A.Siva Kumar, N.Prabhu Kishore and Dr.K.Vijaya Kumar Reddy," INVESTIGATIONS INTO HEAD IMPACT FOR HEADLINER DESIGN WITHOUT COUNTER MEASURES", Vol. 2, Issue 9, September 2013.

[9] Z. Ahmad and M.S. Othman," ENERGY ABSORPTION PERFORMANCE OF A RAIN FOREST VEHICLE UNDER FRONTAL IMPACT", ISSN (Print): 2289-4659; e-ISSN: 2231-8380; Volume 6, pp. 807-817, June 2014

[10] STATUS OF UNITED NATIONS REGULATION, "UN ECE R21 Regulations".

[11] US Department of Transport National Highway Traffic Safety Administration, "U.S. FMVSS No. 201U Laboratory Test Procedure for FMVSS 201U, Report TP201U-01, 1998".

[12] https://www.altair.com/learning-center/?product=RADIOSS

[13] https://www.sciencedirect.com/

[14] https://www.matweb.com/search/datasheet.aspx?matguid=07baafbb9c364fb18fd413bceced867f