Finite Element Method for Predicting the Behavior of Sandwich Structure Luggage Floor of Passenger Cars

Alok Raj$^{1, a}$, Meenakshi Akkiraju$^{2, b}$, Balaji Pillai$^{3, c}$, Naresh Kulkarni$^{4, d}$, Rathnakara Somashekara$^{5, e}$

$^{1, 2, 3, 4, 5}$ Mercedes Benz Research and Development India, 560066, Bangalore, India

E-mail: $^a$alok.raj@daimler.com, $^b$meenakshi.akkiraju@daimler.com, $^c$balaji.pillai@daimler.com, $^d$naresh.kulkarni@daimler.com, $^e$rathnakara.somashekara@daimler.com

Abstract. Sandwich structures are gaining more importance in automobile industry due to its low weight, high flexural stiffness, and corrosion free properties. The composite structure with paper core sandwiched between two Polypropylene Glass-Filled (PP-GF) faces are being utilized for load floor of luggage compartments in passenger cars. To design, develop, and optimize the luggage floors, reliable Finite element (FE) methods are essential. This study deals with developing a finite-element method to predict the behavior of the luggage floor of a typical Sedan car during ball impact test and static stiffness test. FE simulations are performed using LS-Dyna explicit code. Honeycomb core is modelled using solid elements and Modified honeycomb material model in LS-Dyna is assigned to the honeycomb structure. The two faces, bottom and top, are modelled using shell elements and modified-piecewise linear plasticity material model is assigned to them, respectively. Experimental investigations are done for quasi-static four point bend test. FE simulations are performed for quasi-static four point bend test and validated with experimental results. Based on validated FE method, the ball impact test and static push test are simulated. Experiments are performed for the ball impact test and static push test. The results are compared and correlation between experiment and FE simulation is studied by comparing the deformation pattern, forces, and displacements. This developed FE method is for predicting the behavior of the luggage floor.

1. Introduction

Honeycomb sandwich composites must have high bending stiffness to be used as luggage floors in automobiles (Figure 1). Honeycomb composite structures, which comprises of three layers viz., two face sheet and a core material, have good bending stiffness. Over the past few decades, various researchers all around the world have shown keen interest to get mechanical properties experimentally and tried to develop modelling techniques to predict its behavior. Heimbs et al. [1, 2] predicted the structural response of honeycomb sandwich material and focused on modelling and numerical analysis. Seemann and Krause [3] analyzed various approaches of modelling Nomex honeycomb and suggested orthotropic elasto-perfectly plastic material model to be perfect. Shipsha and Zenkert [4] did detailed experimental investigation showing core crushing damage and compression after effect for composite sandwich panels. The Honeycomb structure studied in this paper consists of hexagonal paper honeycomb sandwiched between anisotropic polypropylene glass fibers (PP-GF).
In the literature, many of the simulations are done using detailed modelling of honeycomb cores. A Finite Element method with simpler approach of modelling honeycomb core needs to be developed to enable the use of FE method for faster design and development of sandwich structures in automobiles. The current work aims to use the shell-solid-shell concept to develop a simple and reliable FE modelling method to predict the behavior of the sandwich structures. Experimental studies are conducted to validate the proposed FE method.

![Image](a) Rear view of car with luggage compartment (b) Detailed view of luggage floor

2. Experimental Study

The experimental results are required to validate the behavior of the sandwich composites in proposed FE method. Experiments are performed under room temperature at both static and dynamic loading conditions. The following experiments are conducted.

2.1. Static Four Point Bend Test

In accordance with DIN standards 53293 [5], the flexure experiments are performed on various specimens. The specimen dimensions are tabulated in table 1 and test specifications are shown in table 2. The specimens are subjected to displacement-controlled loading. The test set-up is shown in figure 2(a) and a typical force Vs displacement plot is shown in figure 2(b).

| Test specimen | Specimen | Measurement | Length [mm] | Width [mm] | Thickness [mm] |
|---------------|----------|-------------|-------------|------------|----------------|
| Table 1       |          |             | 380         | 60         | 10             |

| Test specification | Description |
|--------------------|--------------|
| 4-Point bend test   | DIN 53293    |
| Test Standard/Work Instruction | Stroke |
| Test control mode | Stroke |
| Rate of Loading [mm/min] | 3 |
| Load span Length [mm] | 150 |
| Support Span Length [mm] | 300 |
| Diameter of support and loading roller [mm] | 10 |
2.1.1. Results. The normalized force vs displacement curves are plotted as in Figure 2(b).

2.2. Static Push Test.
A static test is performed on the load floor using a rectangular loading device. The load is held constant for 5 sec and then unloaded. Maximum deflection and permanent deformation are checked.

2.2.1. Results. The maximum vertical deflection measured. After unloading, permanent deflection experienced by floor is also measured.

2.3. Dynamic Ball Drop Test

2.3.1. Test Set up. To get dynamic response of the part, a rigid ball with a specific mass is raised to a specific height as shown in figure 4(a). The Impactor freely falls due to gravity onto the luggage floor. The deflection of floor is measured using a transducer positioned in touch condition beneath the floor in the spare-wheel tub as shown in figure 4(b). An accelerometer mounted on top of ball records the acceleration. The set-up of ball drop test is shown in Figure 4(c).
2.3.2. Results. The maximum vertical deflection measured. The Permanent deflection experienced by floor is also measured.

3. Numerical Study

3.1. Material Modelling

The luggage floor consists of honeycomb structured paper and PP-GF face sheet. The mechanical properties of honeycomb paper depends upon direction of loading. Six normal and shear stress strain curves [6] are used to define the orthotropic properties of solid honeycomb core. The FE method under investigation proposes to use simple solid model instead of actual honeycomb geometry to avoid complexity. But representing honeycomb structures with simple solid layer would result in subsequent increase in strength and stiffness. Hence the material stress-strain curves need to be scaled accordingly.

To obtain the scaling factor for curves, an empirical scaling approach is utilized.

3.1.1. Empirical Scaling of Material Properties for Finite Element Simulations. The purpose of this method is to scale the stress-strain curves of honeycomb core material when replaced by solid element. A FE model is prepared taking actual geometrical dimensions of honeycomb. Figure 5 demonstrates the actual FE model of honeycomb and solid structure.
The FE mesh of hexagonal geometry is prepared by taking measurements from the sample. A compression load is applied to actual geometry and simple solid model. Figure 6 illustrates the force direction and FE model at compressed state.

![Honeycomb and Solid Models](image)

**Figure 6.** (a) Compressed honeycomb FE model (b) Compressed simplified solid FE model

The maximum reaction force offered by actual honeycomb structure (Factual) normalized is 100% whereas by simple solid structure (Fsolid) is 416%. Based on force comparison, an empirical scaling factor is obtained. Empirical scaling factor (Factual / Fsolid) = 0.24. The stress-strain curve defining the off-axis properties of honeycomb paper is scaled by this factor. The scaled stress-strain curves are used to simulate the compression test again with solid elements. The results are compared with the actual honeycomb geometry and forces are found to be similar. Similar approach is taken by Kayran and Aydineak [7] to replace aluminium honeycomb core with equivalent continuum core. A good level of correlation between actual and scaled model is observed. Table 3 exhibits the resultant force and corresponding displacement for actual honeycomb and simple solid model with and without tuning the stress-strain curves.

| Material                                | Normalized Force (in %) |
|-----------------------------------------|-------------------------|
| Solid with Honeycomb structure          | 100%                    |
| Simple Solid structure with original material | 416%                  |
| Simple Solid structure with scaled material | 102%                  |

3.2. **FE Modelling.**

Replication of the actual geometry of honeycomb structure is complex and computationally expensive. This limits the usage of proposed FEM in digital development of sandwich parts. Hence, a simple solid element based FE modelling technique is proposed, where honeycomb structures are represented as hexagonal solid elements with element size of 3mm. Commercially available LS-Dyna Explicit code is used for simulation. Modified honeycomb material with scaled stress-strain properties are used. The PP-GF material on top and bottom of the paper is modelled as shell elements and Modified-piecewise linear plasticity material [8] is assigned to it. Solid honeycomb and shell face of PP-GF are in contact through nodal merge. A contact definition of Automatic Single Surface is used. Delamination modelling is not considered, as the real life use cases under consideration of present study does not load the honeycomb part to the extent of considerable delamination.

3.3. **Simulations**

3.3.1. **Four Point Bending Simulation.** Specimen rests on two supports as shown in figure 2(a) and dimensions are similar to as mentioned in Table 1. For the two loading devices, all rotational degrees
of freedom are constrained and allowed to move vertically only. A displacement-controlled quasi-static movement is given to the rollers placed with touch condition on top of the specimen. In the simulation, the reaction force is measured from both the two loading devices. Figure 7 (a) & (b) shows the simulation view and corresponding variation in total force with imposed displacement.

![Simulation view](image1)

![Force-displacement curve](image2)

**Figure 7.** (a) Simulation view (b) Force-displacement curve

3.3.2. *Push Simulation*. Simulation is done similar to the experiment. Force is applied quasi-statically and held for sufficient time for load stabilization, followed by unloading. Maximum and permanent deflections are measured, respectively. Figure 8 (a) & (b) shows the initial and deformed state of floor in push test simulation.

![Initial state of luggage floor](image3)

![Deformed state of luggage floor](image4)

**Figure 8.** (a) Initial state of luggage floor (b) Deformed state of luggage floor

3.3.3. *Dynamic Ball Drop Simulation*. Simulation is done similar to the experiment. The reaction force is measured on the impactor. A local indentation mark is clearly visible in simulation same as seen in the test. The maximum force and deflection are noted. The permanent deflection is also noted. Figure 9 (a) & (b) shows the initial and deformed state of floor in ball drop simulation.

![Initial stage of luggage floor](image5)

![Deformed state of luggage floor](image6)

**Figure 9.** (a) Initial stage of luggage floor (b) Deformed state of luggage floor
4. Validation of FE with Experimental Results
The proposed FE method of composite material with honeycomb core sandwiched between two face sheets is validated by using the four-point bend experiment results. Simulation results are normalized for force and deflection from the experimental results and are compared. Comparison is described in Table 4. The deformation pattern is observed to be similar.

Based on the aforementioned validation, the FE model is used for static push and ball impact. Simulation results for normalized force and normalized deflection are compared in Table 4 for static push test and the ball impact test. Good correlation is observed which suggests the applicability of the proposed FE method for predictive engineering. Minor variation in force could be attributed to the failure and the delamination of face sheet from the core, which is not expected in the simulation. Experiment and simulation results of the all the studies have been tabulated below (Table 4).

![Figure 10](image_url)

**Figure 10.** (a) F-D curve of simulation overlaid with curve obtained from experiment (b) Simulation and Test specimen at deformed state

|                  | Flexural Test vs Simulation | Push Test vs Simulation | Impact Test vs simulation |
|------------------|-----------------------------|-------------------------|--------------------------|
|                  | Force (N) | Deflection (mm) | Force (N) | Deflection (mm) | Force (N) | Deflection (mm) |
| Experiment       | 100 %     | 43.3 %          | 100 %     | 102.5 %         | 100 %     | 93.5 %          |
| Simulation       | 105.6 %   | 41.6 %          | 100 %     | 97.5 %          | 118.7 %   | 95 %            |

5. Conclusion and Future Scope
Finite Element Method using simple shell-solid-shell with scaled material properties are proposed to model sandwich structures. A good level of correlation for four point bend test, push test, and ball drop test, with the proposed FE method is achieved, respectively. This proposed FEM is suggested to be used for digital design and development of the luggage floor or similar components made of honeycomb structures during the early phase of design. This could reduce the number of hardware tests and get the perfect designs during the initial phase itself. This in turn could lead to reduction in costs and turnaround time. Delamination prediction, element size sensitivity, and more test cases, need to be experimentally validated to improve the method. This is suggested as future work.

6. Acknowledgements
The authors gratefully acknowledge the support provided by the Mercedes-Benz Research and Development India (MBRDI) and Daimler AG (DAG), for the research work leading to this paper. Authors would like to express their gratitude to Mr. Parthiv Shah, Head of the Department, Safety, MBRDI who rendered his valuable support during the research period. Authors expresses their gratitude to colleagues of RD/KIA (Interiors) department at Daimler AG for their valuable support during the research.
7. References
[1] Heimbs, S., Mehrens, T., Middendorf, P., Maier, M. and Schumacher, A., 2006. Numerical determination of the nonlinear effective mechanical properties of folded core structures for aircraft sandwich panels. In 6th European LS DYNA User’s Conference.
[2] Heimbs, S., Vogt, D., Hartnack, R., Schlattmann, J. and Maier, M., 2008. Numerical simulation of aircraft interior components under crash loads. International journal of crashworthiness, 13(5), pp.511-521.
[3] Seemann, R. and Krause, D., 2017. Numerical modelling of Nomex honeycomb sandwich cores at meso-scale level. Composite Structures, 159, pp.702-718.
[4] Shipsha, A. and Zenkert, D., 2005. Compression-after-impact strength of sandwich panels with core crushing damage. Applied Composite Materials, 12(3), pp.149-164.
[5] Feldhusen, J., Torsakul, S., Brezing, A. and Krishnamoorthy, S., 2008. Numerical Modeling and Experimental Investigation of the Failure Modes of the Cellular Foam Sandwich Structures. Journal of Metals, Materials and Minerals, 18(2), pp.111-115.
[6] Heimbs, S., Middendorf, P. and Maier, M., 2006, June. Honeycomb sandwich material modeling for dynamic simulations of aircraft interior components. In Proceedings of the 9th international LS-DYNA users conference. Dearborn (MI).
[7] Kayran, A. and Aydincak, İ., Assessment of Effective Elastic Properties of Honeycomb Cores by Finite Element Analysis of Sandwich Panels.
[8] L.S DYNA Keyword user manual. Version 8, 2008.