Crash analysis of sandwich composite electric microbus

P. Jongpradist1,2, S. Limpanawisut1, A. Siritana1, P. Saarding1, N. Saingam1, P. Tangthamsathit1, P. Chanpaibool1

1Department of Mechanical Engineering, Faculty of Engineering, King Mongkut’s University of Technology Thonburi, 126 Pracha Uthit Rd., Bang Mod, Thung Khru, Bangkok, Thailand 10140

* Corresponding Author: pattaramon.jon@mail.kmutt.ac.th

Abstract
Sandwich composite is ideal for lightweight monocoque structure owing to not only its superiority in strength-to-weight ratio and specific flexural properties but also its manufacturability to any desired complex shape. In this work, an optimum lightweight design of monocoque composite sandwich-structure microbus subjected to bending and torsion stiffness criteria and natural frequency constraints is further investigated under crash conditions through finite element analysis. A composite sandwich panel made of 5.4-mm woven glass fabric-epoxy face sheets and 50-mm rigid polyurethane foam core is tested under low-velocity impact with a hemisphere headed impactor following ASTM D7136 to characterize the failure characteristics. An enhanced composite damage material model so-called MAT54 in LS-DYNA with damage progression in each lamina utilizing strength-based Chang-Chang criteria is implemented with various formulations to validate and compare for their efficiency and accuracy under impact test. The reduced-integration Balytschko-Tsay shell element with 5 through-thickness integration points is applied to the monocoque composite microbus model. The structural crashworthiness under the frontal crash of the bus at 20, 40 and 50 km/h to a rigid wall barrier is examined. The front structure absorbs the most frontal impact energy up to 60% and shows severe damage when the crash velocity is 40 km/h or higher. For the bus impact velocity of 50 km/h, the maximum deformation of the bus front is 809 mm resulting in the intrusion of the structure into the specified driver survival space. Design improvement of the sandwich structure should be further explored to enhance its energy absorption and safety of the bus under crash events.

Keywords: Crash simulation, Sandwich structure, Composite material, Microbus

1. Introduction
An electric vehicle is a green alternative to traditional internal combustion engine vehicle that benefits in reducing emissions of greenhouse gas and air pollution contributing to climate change and smog. One of the key challenges in designing electric vehicles lies in lightweight design and energy efficiency. However, the safety of the vehicle structures must not be compromised. Sandwich composite is ideal for lightweight monocoque structure owing to its superiority in strength-to-weight ratio and specific flexural properties as well as its manufacturability to any desired complex shape. Many researchers have been focused on developing composite automotive parts to enhance vehicle strength and
crashworthiness. Kang et al. [1] appended carbon composite roll bar components to the bus structure to improve safety under rollover in which the maximum deformation of the bus structure could be reduced by 40% at the end of the collision. Ko et al. [2] designed a composite bus structure made of glass fabric/epoxy with stainless steel inserts as the bus frame. Safety of the bus under frontal crash to a rigid wall at a speed of 60 km/h and rollover according to ECE-R66 standard were verified via finite element analysis using LS-DYNA. Several studies [3-5] designed lightweight superstructure for the bus roof strength based on ECE-R66 rollover test. It was found that pillar strength highly affects the bus crashworthiness under rollover accident [6-7].

Recently, Kunakorn-ong et al. [8] designed a lightweight monocoque composite sandwich-structure microbus based on bending and torsion stiffness criteria and natural frequency constraints. The monocoque design proves to considerably reduce the body weight while the bus stiffness under working conditions are reassured. The current study further investigates the performance of the bus structure under full frontal crash conditions by using finite element analysis using LS-DYNA. Recommendations on safety improvement are also provided.

2. Modelling and Validation

2.1 Validation of material models through an impact drop test

The material models defined in LS-DYNA are verified following ASTM D7136 Measuring the Damage Resistance of a Fiber-Reinforced Polymer Matrix Composite to a Drop-Weight Impact Event [10]. This test method determines the damage resistance by using a drop weight machine with a hemispherical striker tip as displayed in Figure 1a. A sandwich structure composes of 4 plies of 0/90 G600 glass fiber fabric and 5 plies of ±45 G600 glass fiber fabric for each skin and 50-mm H100 foam core. Four composite specimens made by vacuum infusion process are tested under impact energy of 100 J, 150 J, and 200 J. The impactor mass is 10 kg and the impact velocity ranges are between 3.1 and 5.4 m/s. Damages of the specimens for different impact energy are shown in Figure 1b. It can be seen that the damage area increases as the impact energy increases. Fiber fracture and delamination are observed in all specimens and permanent foam deformations occur without perforation of the composite panels.

The impact test scenarios are simulated in LS-DYNA by using a mesh size of 2 mm. For explicit dynamic simulation using plate and shell elements in LS-DYNA, the critical time step is automatically determined by the wave propagation velocity for membrane deformations governed by the size of the smallest element. The MAT024 Piecewise linear plasticity is assigned to H100 foam core while the composite material model is assigned as MAT054 Enhanced composite damage with Chang-Chang strength-based damage criteria. The 2-way fiber behavior option is triggered to account for the properties of woven fabric. Thus, the failure criteria $e$ in material coordinates $a$- and $b$-directions (corresponding to the fiber direction of 0 and 90 degrees) are given as follows:

![Figure 1. (a) Impact testing machine and test set-up (b) Specimen damages after impact test](image)
for the tensile fiber mode and
\[ \sigma_{aa} > 0, \varepsilon_f^2 = \left( \frac{\sigma_{aa}}{S} \right)^2 + \beta \left( \frac{\sigma_{ab}}{S_c} \right) - 1 \left\{ \begin{array}{ll}
\geq 0 & \text{failed} \\
< 0 & \text{elastic}
\end{array} \right. \]  
and
\[ \sigma_{bb} > 0, \varepsilon_f^2 = \left( \frac{\sigma_{bb}}{S} \right)^2 + \beta \left( \frac{\sigma_{ab}}{S_c} \right) - 1 \left\{ \begin{array}{ll}
\geq 0 & \text{failed} \\
< 0 & \text{elastic},
\end{array} \right. \]

for the compressive fiber mode,
\[ \sigma_{aa} < 0, \varepsilon_f^2 = \left( \frac{\sigma_{aa}}{S} \right)^2 - 1 \left\{ \begin{array}{ll}
\geq 0 & \text{failed} \\
< 0 & \text{elastic} \quad \text{and} \quad \sigma_{bb} < 0, \varepsilon_f^2 = \left( \frac{\sigma_{bb}}{S} \right)^2 - 1 \left\{ \begin{array}{ll}
\geq 0 & \text{failed} \\
< 0 & \text{elastic},
\end{array} \right. \]

for matrix failure criterion,
\[ \varepsilon_m^2 = \left( \frac{\sigma_{ab}}{S_c} \right)^2 - 1 \left\{ \begin{array}{ll}
\geq 0 & \text{failed} \\
< 0 & \text{elastic}. \right. \]  

After the material failure criterion is satisfied, the material properties are perfectly plastic until the defined DFAIL. Information on the damage status at each integration point and each element can be plotted using additional integration point history variables written to the LS-DYNA database by the *DATABASE_EXTENT_BINARY definition as variable NEIPS. The history variables 1, 2, 3 are failure flags for tensile fiber mode \( e_t \), compressive fiber mode \( e_c \), and matrix mode \( e_m \), respectively.

Sandwich structure is modelled through PART_COMPOSITE. Laminated Shell Theory is activated using parameter LAMSHT = 1 to correct the default assumption of uniform transverse shear strain through thickness when the composite shell has layers of dissimilar materials where large stiffness changes across thickness such as sandwich structures. Results from different shell element formulations ELFORM=2 and 16 are investigated for model accuracy and robustness. Both formulations are based on Reissner-Mindlin kinematic assumption. The shell deformation is described by the mid-surface displacements and rotations with 5 DOF in a local coordinate system which globally yields 6 DOF. The shell cross-section remains straight and unstretched while shear deformations are possible. The ELFORM=2 is Balytschko-Tsay shell which uses one-point integration and bi-linear nodal interpolation. Thus, the element is highly efficient, but warpage may be an issue in coarse models. The ELFORM=16 corresponds to 2x2 full-integration shell formulation and uses Bathe-Dvorkin transverse shear treatment to eliminate w-mode hourglassing. This formulation generally provides more accurate results than using ELFORM=2.

The number of through-thickness integration points can be defined in *PART_COMPOSITE. The BETA angles contribute to the initial orientation of the material coordinate system for each integration point. In this study, the number of through-thickness integration points of 5 points (one point for each fiber orientation for skin and one point for foam core), 19 points (one point for each ply and one point for the core), 29 points (one point for each fiber direction and 25 points for the core), and 38 points (one point for each ply and 25 points for the core) are studied. By default, the output results in the composite shell are reported at 3 integration points, i.e., top, middle and bottom points. More results at the through-thickness integration points can be requested by defining MAXINT in *DATABASE_EXTENT_BINARY.

Figure 2a demonstrates comparison of the force-displacement curves of impact test obtained from different models. It is found that simulation results when ELFORM=2 generally return a slightly higher maximum force and displacement before the impactor rebound as well as a larger permanent deformation. Impact modelling of sandwich structure using ELFORM=16 is accurate despite the number of through-thickness integration points. However, models using ELFORM=16 requires the computational cost of approximately 3-4 times larger than the similar model with ELFORM=2 (Figure 3b). In views of accuracy, robustness and efficiency of a simple impact problem the use of 5 integration points and ELFORM=16 is recommended. For the current application to the microbus model consisting of a large number of nodes and elements, the ELFORM2 element with one integration point for each fiber direction in the sandwich faces and one integration point in the core which delivers moderately stiffer results is therefore chosen for implementation to achieve a cost-effective model.
Figure 2. Simulated results under impact drop and CPU times using various model formulations

Table 1 illustrates the force-displacement curves obtained from numerical simulation are compared with the maximum forces recorded from the experiments. For all impact energy of interest, the sandwich structures are damaged with permanent deformations after the impactor rebound. The higher the impact energy, the higher the impact force and permanent deformation. It is also noticed that the maximum forces from impact test are somewhat less than that obtained from FEA whereas the predicted displacements at the maximum force are comparable. Results discrepancies are more apparent when the impact velocity is higher. The maximum force obtained from the experiment of 200-J impact seems to be rather lower than the expected value. This could be owing to inhomogeneous material properties of the composite laminate which cannot be captured in the simulation results. The results can be further investigated via reliability analysis to account for this contribution.

Table 1. Experimental and simulation result for forces and displacement under an impact drop test.

| Test no. | Impact energy (J) | Maximum force | Maximum displacement |
|---------|------------------|---------------|----------------------|
|         |                  | Exp (kN)      | Sim (kN)             | Error | Exp (mm) | Sim (mm) | Error |
| 1       | 100              | 31.1          | 35.6                 | 14.4% | 6.8      | 7.5      | 9.8%  |
| 2       | 150              | 43.6          | 47.0                 | 7.8%  | 8.4      | 8.5      | 1.6%  |
| 3       | 200              | 44.3          | 55.2                 | 24.4% | 8.9      | 9.4      | 6.4%  |

2.2 Baseline design
The sandwich composite microbus used in the current study is adopted from Kunakorn-ong [8] as illustrated in Figure 3. The bus consists of nine components distinguished by their functions including the bottom, floor, roof, beam, ceiling, pillar, battery tray, subfloor, and vent. The bus is 2.4 meters wide, 7.8 meters long and 3.2 meters height with 24 passenger seats. To correctly compute the dynamic behavior of the bus under crash events, the mass of other important components is uniformly distributed in the FE model at their corresponding locations. The mass elements for axle systems, windows and doors of 1892 kg are assigned to the bottom component. The driver and 24-passenger load and seats of 2282 kg are given to the floor. The 1536-kg battery is applied to the battery tray. The total body weight of the baseline bus 7,610 kg.

In this paper, the FE simulation is performed by using LS-DYNA. The shell elements of size 30 mm$^2$ were implemented to all components of the bus model. The total numbers of elements in the baseline model are 104,829 elements. The sandwich structure is composed of H100 foam core and E-glass/epoxy faces with their thicknesses and ply orientations as listed in Table 2 with the color code explanation in Table 3. The glass fiber fabric with a surface density of 400 g/m$^2$ (G400) and 600 g/m$^2$ (G600) was mainly selected for the design of structure due to its lightweight and high specific stiffness, strength, and toughness except that the plain-weave carbon fiber fabric with a surface density of 200 g/m$^2$ (C200) was implemented at the battery tray where much higher bending stiffness of the component was required.
Figure 3. Components of composite microbus.

Table 2. Material layup for composite microbus components.

| Component       | Laminate layup | Thickness (mm) | weight (kg) |
|-----------------|----------------|----------------|-------------|
| Bottom          |                | 61.2           | 524         |
| Vent            |                | 38.8           | 130         |
| Battery Tray    |                | 69.2           | 270         |
| Pillar          |                | 55.2           | 204         |
| Floor           |                | 47.2           | 232         |
| Roof            |                | 4              | 125         |
| Ceiling         |                | 70             | 264         |
| Beam            |                | 51.2           | 44          |
| Subfloor        |                | 6.6            | 5           |

Table 3. Material, ply thickness and fiber orientation for color codes used in Table 1.

| Material  | Ply thickness (mm) | Fiber direction (degree) | Color code | Fiber direction (degree) | Color code |
|-----------|--------------------|--------------------------|------------|--------------------------|------------|
| G400      | 0.4                | 0/90                     | ±45        | ±45                      |            |
| G600      | 0.4                | 0/90                     | ±45        | ±45                      |            |
| C200      | 0.4                | 0/90                     | ±45        | ±45                      |            |
| H100 (core)| 5                  | -                        | ±45        | ±45                      |            |

to support the weight of the battery packs. Each layer of fabric has a nominal thickness of 0.4 mm. The zero-degree fiber directions for all components were assigned in the orientation along the horizontal axis parallel to the floor in the plane of each component panel. Mechanical material properties for the composite faces and foam core are obtained from [8].

3. Frontal crash simulation

The bus structure under full frontal crash to a rigid wall barrier is simulated. The bus is constrained in z-axis at the wheel locations and the initial velocities \( v \) at 20, 40 and 50 km/h are prescribed in +x-direction (see Figure 4a). Automatic single surface contact interaction is assigned to all parts. The driver survival space is measured according to the survival space in ECE-R29 [11] corresponding to the reference point of the driver’s center of gravity. Therefore, the maximum deformation of the bus frontal structure should not be greater than 600 mm after full-frontal crash. An example of bus deformation
behavior for the frontal crash with an initial velocity of 50 km/h is represented in Figure 4b. The bus impacts the wall at the time = 29 ms and structural damage start occurring at 60 ms until the crash finished at 160 ms.

The energy absorption of each component under the initial impact velocity of 20, 40 and 50 km/h are displayed and compared in Figure 5. The values indicated in the plot is the internal absorbed energy by each component after the crash while the y-axis shows the percentage of energy absorption for each part. For all crash velocities, most of the kinetic energy is transferred to the bottom and floor of the microbus by which approximately 90% of crash energy are absorbed. The bottom component especially the front panel absorbs more percentage of impact energy when the crash velocity increases.

![Figure 4](image)

**Figure 4.** (a) Boundary and initial conditions (b) Deformation behavior for a frontal crash at 50 km/h.

![Figure 5](image)

**Figure 5.** Percentage energy absorbed by each component under the frontal crash.

![Figure 6](image)

**Figure 6.** Deformations of (a) front panel and (b) right front pillar.
Figure 6 provides the deformations of the front panel at 400-mm height from the lower floor of the bus and the deformations along with the right pillar’s height for different impact velocity. The deformation of the frontal part at a slow impact velocity of 20 km/h is found to be less than 50 mm. Nonetheless, damages in the compressive fiber mode are already detected at the outer ply of the sandwich face in the front panel of the bus. At an impact velocity of 40 km/h, the front panel has damaged in all failure modes with the largest damage in compressive fiber mode. However, the deformations of the front panel and pillar are still less than the criteria for survival space. When the crash speed reaches 50 km/h, the maximum deformation is found to be 809 mm in which the driver’s survival space is intruded by 59 mm. This implies that the driver is not safe during the crash accident and the front panel should be re-design to improve the bus crashworthiness under the frontal crash.

4. Conclusions
This study set out to present the method to simulate an impact drop test to a sandwich structure with a different number of integration points and shell element formulation. The use of highly computing-efficient Belytschko-Tsay element with the minimum through-thickness integration points has proved to provide comparable and reliable results with much more detailed elements. The simple model is thus applied to crash analysis of a complex sandwich composite microbus. Most of the impact energy from the frontal crash are found to dissipate via the bottom components through front panel’s deformation. Consequently, the baseline bus model is not safe under the frontal crash. Overall, this study strengthens the idea that the design of these two parts should be modified to increase the energy absorption of the part to improve the crashworthiness of the structure under full-front collision.

References
[1] Kang K Chun H Na W Park J Lee J Hwang I and Hong H 2011 Proc. the 18th International conference on composite materials (Jeju, Korea)
[2] Ko H Y Shin K B Jeon K W and Cho S H 2009 J. Mech. Sci. and Technol. 23(10) 2686-2693
[3] Gursel K T and Gursesli S 2010 Gazi Univ. J. Sci. 23(1) 71-79
[4] Liang C C and Le G N 2009 Int. J. Crashworthiness 14(6) 623-639
[5] Karlinski J Ptak M Dzialak P and Rusinski E 2014 Arch Civ. Mech. Eng. 14(1) 342-343
[6] Kongwat S Jongpradist P and Hasegawa H 2020 Int. J. Automot. Technol. 21(4) 981-991
[7] Kunakorn-ong P and Jongpradist P 2020 Int. J. Automot. Technol. 21(1) 215-25
[8] Kunakorn-ong P Ruangjirakit K Jongpradist P Aimmanee S and Laoonual Y 2020 Proc. Inst. Mech. Eng., Part C 3-1
[9] Sirisobhana P Ruangjirakit K Jongpradist P and Aimmanee S 2019 Proc. the 22nd International Conference on Composites Materials (Melbourne, Australia)
[10] ASTM D7136 Standard Test Method for Measuring the Damage Resistance of a Fiber-Reinforced Polymer Matrix Composite to a Drop-Weight Impact Event
[11] United Nations 2011 Economic Commission for Europe agreement regulation No.29: Uniform Provisions Concerning the Approval of Vehicles with regard to the Protection of the Occupants of the Cab of a Commercial Vehicle