A Numerical and Experimental Measurement in a Dynamic Strain Response of an Electric Bus Body Structure

Dittapoom Shinabuth[1,*], Chi-na Benyajati[2], Anek Phuchamnong[2], Monsak Pimsarn[3], Masaaki Okuma[4]

[1]TAIST-Tokyo Tech, Automotive Engineering, Bangkok, Thailand 12120

[2]Automotive Laboratory, National Metal and Materials Technology Center, National Science and Technology Development Agency, Bangkok, Thailand 12120

[3]Department of Mechanical Engineering, Faculty of Engineering, King Mongkut's Institute of Technology Ladkrabang, Bangkok, Thailand 10520

[4]Department of Mechanical and Aerospace Engineering, Graduate School of Science and Engineering, Tokyo Institute of Technology, Tokyo, Japan 152-8550

*Corresponding Author: E-mail dit_4@windowslive.com

Abstract

In this paper, a structural analysis of a medium-sized electric (EV) bus is proposed. Due to an increasing demand of bus transport in Thailand, a numerical technique called Computer-Aided Engineering (CAE) has now become a necessary tool in a bus analysis for verifying the strength of a structure and predicting any potential problems. Generally, computational modeling of a bus is often investigated via the use of finite element analysis in comparison with strain responses that are experimentally measured by strain gauges at various locations during a test run. In this study, a numerical analysis was carried out on both chassis and body structure of an 8m-bus prototype which was purely powered by electricity. A computational model of the bus structure was consisted of shell and beam elements for chassis and body structure respectively. Linear springs were applied on the chassis model to represent actual suspension components such as leaf spring and air bellows in order to give the computational model a dynamic response in various driving conditions. A bumping test where the bus was driving pass a single speed bump with a front axle followed by a rear axle was considered in this study. A maximum stress of 10.97 MPa and a maximum deviation of 17.35% compared to experimental results were calculated to occur at a front section of the chassis where front chassis side-rail members and battery compartment were connected. Furthermore, the results from this study could be used as a guideline to an EV bus design in Thailand by giving approximated solutions to examine strength of the bus body structure.

***Keywords**:* Electric bus, Computer-Aided Engineering, Linear spring, Dynamic response, Speed bump

1. Introduction

Nowadays, bus is still a popular road transportation mode in Thailand in both short and long-range applications such as in cities, or tourism and inter-state routes. Most buses are constructed by local mechanics while only few are imported from other countries. In a fabrication of local buses, a main concern is usually given to an overall stiffness of a bus with little focus on resulting structural weight. This could have detrimental effect on various issues such as structural durability, unbalance of a vehicle weight, fuel consumption, and driving performance. In recent times, Computer-Aided Engineering (CAE) has been increasingly accepted among major local bus manufacturers as an important tool to achieve an effective design within existing governing standards as well as production cost and time reduction.

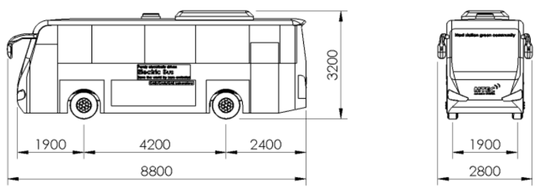
There have been many studies in the literature focusing on a bus design and analysis using CAE. F Lan and J Chen [1] demonstrated a structural analysis of a bus frame side with/without supporting member before a structural optimization was performed. Cross sectional shape and component thickness were the design variables of interest. Manokruang S. and Butdee S. [2] proposed a bus body design using a wireframe modeling on Computer-Aided Design (CAD). It was found that wireframe of a bus structure modeling was suitable for a beam or frame analysis in CAD/CAE systems. K. Chinnaraj and M. Sathya Prasad [3] studied a dynamic strain response of a truck chassis under braking and cornering maneuver conditions. They reported that deviation of the results was possibly due to the unknown residual stress, which was contained during fabrication process. Moreover, the study concluded that the residual effect would not have affected strength of the frame rail but it would accelerate a fatigue failure.

From literature reviews, majority of work has been done on a vehicle operated with a combustion engine. However, in electric bus applications, a presence of battery packs results in an additional concentrated weight on the chassis such that it could directly affect the structural stiffness specifically during a drive. Therefore, the purpose of this study is to investigate strength of an electric bus body structure using a dynamic capability of a finite element analysis and experimental strain measurement of a strain gauge attachment.

2. Methodology

In order to investigate a bus body structure, a finite element analysis and an experimental strain measurement from a field test were carried out. An electric bus investigated in this study was an 8m 6-wheels bus prototype as displayed in Figure 1.

(a)



(b)

Fig. 1 (a) Electric bus prototype
(b) Bus general dimensions (in mm.)

Strain responses of a bus structure usually come from contacts between wheels and the road before being transmitted to a chassis via suspension system. These responses vary continuously with time indicating a dynamic application. However, most studies from the literature have carried out a computational analysis in a static fashion i.e. without taking the effect of suspension system into account. Therefore, the aim of the current study was to perform a dynamic analysis on the electric bus prototype.

2.1 Computational modeling and analysis

In general, an overall bus body structure is integration between a primary chassis structure which is a frame configuration consisted of formed sheet metals, and a body structure which is a network of tubular members forming a bus body. An analysis of an electric bus might not be similar to that of a conventional bus because the main loading comes from a battery weight.

In order to simplify an analytical model of the bus structure, a simple line and surface bodies were used to represent the bus geometry. Apart from a physical geometry of the bus, a suspension system was also considered. Suspension of a bus typically contains leaf springs and dampers. Furthermore, an additional air bellows could be installed to improve a passenger comfort further. In a finite element modeling, a suspension system for each wheel containing leaf springs, dampers, and air bellows could be represented by linear springs and a damper model as presented in Figure 2 [4]. In this study, a stiffness of front and rear portion of leaf spring, a tire, and air bellows was 170 kN/m (K_{s1}) and 85 kN/m (K_{s2}), 900 kN/m (K_t), and 800 kN/m (K_{air}) respectively.

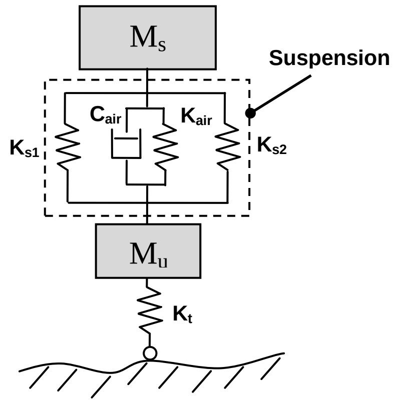


Fig. 2 Schematic of a suspension modeling

In addition, the air bellows was also assigned with a damping of 150 kN-s/m [5,6]. Furthermore, an unsprung mass (M_u) consisted of mass underneath a suspension system such as axle and wheel. On the other hand, a sprung mass (M_s) was comprised of the components supported by suspension system. Details of these external loading are listed in Table 1 [7].

Table. 1 External loadings for bus structural analysis

Battery	2000	kg
*Unsprung mass, M_u	400	kg
Electric motor	190	kg
Air conditioning	200	kg
Passengers and accessories	2300	kg
Electrical control system	300	kg
Total	5390	kg

* A quarter car modeling

In a finite element model of the bus structure, chassis and bus body was represented by shell and beam elements respectively. The completed model shown in Figure 3 was comprised of 56,746 shell elements and 35,887 beam elements with additional element mid-side nodes. Multi-Point

Constraints (MPC) [8,9] was employed to connect a shell and beam elements. Material properties of conventional steel were applied with Young's modulus, yield strength, and Poisson's ratio of 195 GPa, 245 MPa, and 0.29 respectively.

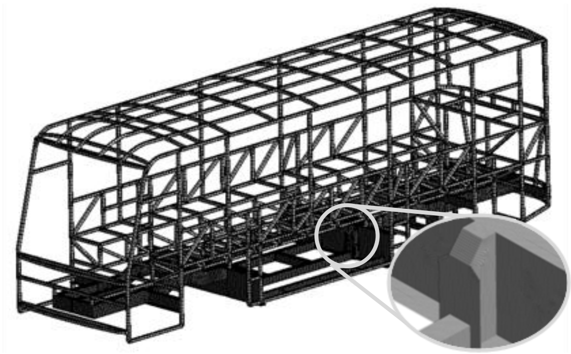


Fig. 3 Finite element model of electric bus

For an applied load, bumping situation displayed in Figure 4 was inspected in this study. A bus structure received a vertical loading effect when a vehicle travelled over a small bump. Hence, a boundary condition of a vertical displacement loading equal a height of speed bump in an actual test was sequentially applied to raise the front axle and rear axle.

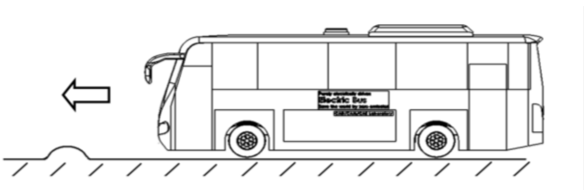


Fig. 4 Bumping situation test

2.2. Experimental strain measurement

In experiment, uniaxial strain gauges were employed to detect a minute of surface deformation of the bus structure during a test run as demonstrated in Figure 5. Eight strain gauges were attached at various locations on both chassis and body structure. Strain gauge attachment locations on the bus structure are shown in Figure 6. During the test, the bus was driven to a speed bump and then slowly climbed over it at a constant speed.

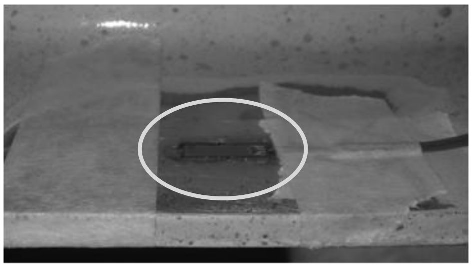
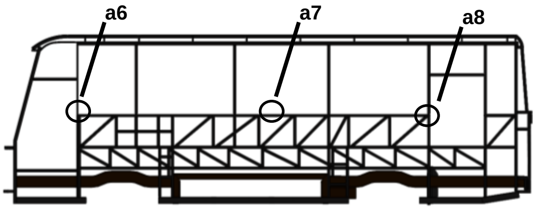
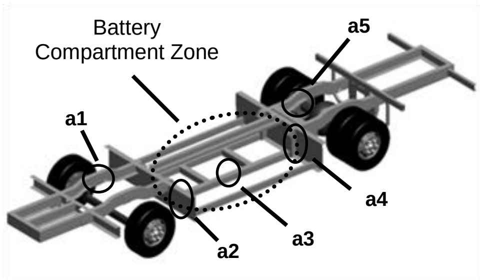


Fig. 5 Example of a uniaxial strain gauge attachment on the bus chassis

The tested speed bump had 70 mm height and 1100 mm length. The generated strains on the bus structure were recorded since front wheels came into contact until rear wheels subsequently preceded with a bump.



(a)



(b)

Fig. 6 Strain gauge attachment locations (a) Right side of body frame (b) Chassis

3. Results and Discussion

3.1 Numerical simulation

Results of a bumping simulation were presented by contour of a strain distribution as shown in Figure 7. Calculated strain seemed to locally concentrate on sensitive areas especially at a front section of the bus chassis where there was no supporting feature apart from the welding lines between front chassis side rails and battery compartment plate.

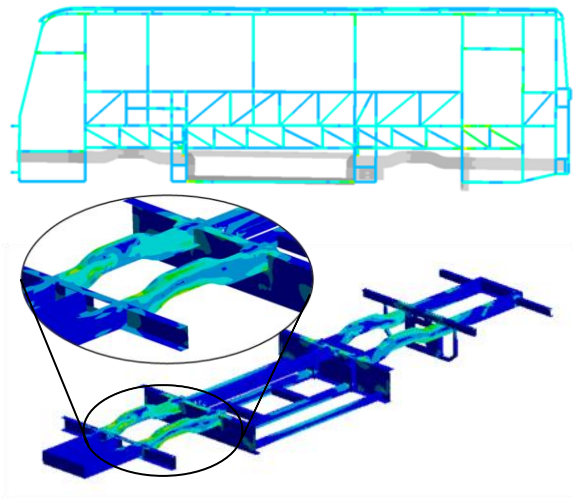


Fig. 7 Strain distribution on a bumping simulation

Furthermore, the calculated strain was monitored locally from 8 defined locations at approximately the same place as those displayed in Figure 6. A maximum strain of 6.23e-5 in compression was found at location “a1”. Thus, it could be concluded from the bumping simulation that the area on the chassis subjected to risk of failure were around the connecting positions between chassis side rails and battery compartment by the welding process. Another area was on the chassis side rails where they were connected to a suspension system. Moreover, on the body structure, a relatively high strain was observed on the members close to the rear door.

3.2 Experimental measurement

A dynamic strain history of the bus was measured during an actual driving. An example of strain history at location ”a1” is shown in Figure 8. It can be seen that the measured data had to be filtered first to allow an effective investigation of a specified area.

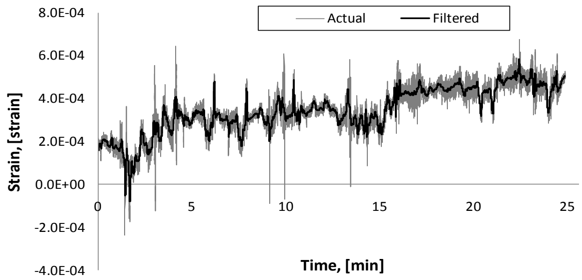


Fig. 8 Dynamic strain history on a normal test

The filtered strain history was then considered only during a period of bumping movement with the aim of a comparison both experimental and numerical data.

3.3 Simulation vs. Measurement

Figure 9 presents an investigation of dynamic strain at location ”a1” under a bumping test. It can be seen that a numerical simulation displayed a similar trend to that of the experiment. At both sets of result, the highest strain magnitude at this location occurred when the front wheels were at the highest position of the speed bump i.e. “front-bumped”. The maximum values were 5.63e-5 and 6.23e-5 for measured and analyzed strain respectively.

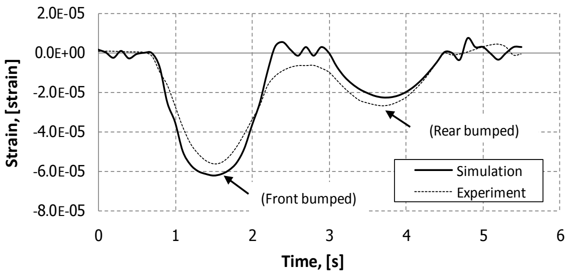


Fig. 9 Dynamic strain comparison at location “a1” during a bumping maneuver

Furthermore, a relatively much smaller strain peak response was observed at location ”a1” in both measured and analyzed strain variation when the rear wheels went over the speed bump afterwards i.e. “rear-bumped”. A deviation of the strain results during a bumping was calculated to be 10.65% and 17.35% for "front-bumped" peak and "rear-bumped" peak respectively.

In addition, investigation of the whole bus structure at all measurement locations was carried out in order to validate the computational model of the bus and the employed boundary conditions. Additionally, all strain data were converted to a corresponding stress in order to assess the resulting safety factor of the bus structure. A stress comparison between numerical simulations and experiments at all measurement locations is shown in Figure 10.

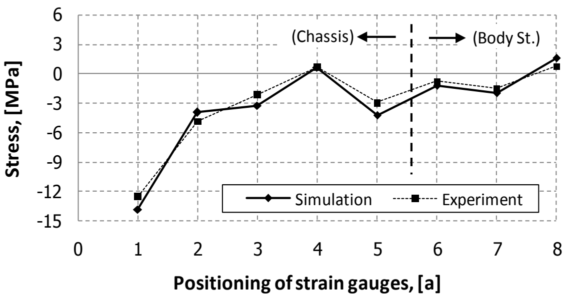


Fig. 10 Stress comparison of an entire structure on a bumping test

A stress comparison of both simulation and experiment at 8 strain gauge locations showed a similar trend with various degree of discrepancy at each point. Deviations on chassis structure locations were calculated to be 10.66% and 9.6% for maximum and minimum respectively while 23.12% and 11.03% for maximum and minimum were determined on the body structure locations. These deviations might come from a complexity of converting the actual bus structure into a CAD system, and an assumption of rigid connection for all the welding joints. Moreover, the corresponding stress that was found on the bus structure under the bumping test in this study would give a significantly higher safety factor than that required by the transportation regulation set.

4. Conclusion

In this study, the dynamic investigation of an electric bus was carried out on a bumping test using a dynamic analysis of a numerical calculation and strain measurement of a strain gauge setup. Computational model of chassis and body structure were combined and analyzed together. In order to acquire a strain history, strain gauges were attached at various locations over an entire structure. Results of this research can be summarized as followed:

- A maximum stress of 10.97 MPa was found at location "a1" on the front section of the chassis while a maximum deviation of 23.12% was found at location "a8" on the rear part of the body structure.
- A strain comparison at location "a1" showed a very similar trend despite an offset of the magnitudes ranging between 10-17%.
- The deviations between both numerical and experimental data could be due to a complexity of the body structure, and the welding joint assumption.
- Under the bumping test investigated, the area on the chassis subjected to risk of failure were around the connecting positions between the chassis and the battery compartment while another location was the body members close to the rear door.

The higher strain area especially members on the body structure would require an improvement by adding or changing some design features or material that would affect overall strength of the bus structure. However, this study was only conducted on a bumping test. A braking and cornering maneuver will be investigated in order to further validate the employed physical modeling and to obtain a lightweight optimization of the bus. Furthermore, the results from this study could be used as a guideline to an electric bus design by a finite element analysis.

5. Acknowledgements

The authors are very grateful to Automotive Laboratory, National Metal and Materials Technology Center (MTEC), Thailand for their supports to this work.

6. References

[1] F Lan, J Chan and J Lin, Comparative Analysis for Bus Side Structure and Lightweight Optimization, *Proc. Instn. Mech. Engrs*., Part D: J. Automobile Engineering, 218, pp. 1067-1075.

[2] Manokruang S. and Butdee S., Methodology of Bus-Body Structural Redesign for Lightweight Productivity Improvement, *AIJSTPME,* (2009) 2(2), pp. 79-87.

[3] K. Chinnaraj, M. Sathya Prasad and C. Lakshmana Rao, Experimental Analysis and Quasi-Static Numerical Idealization of Dynamic Stresses on a Heavy Truck Chassis Frame Assembly, *Applied Mechanics and Materials*, Vols. 13-14 (2008), pp. 271-280.

[4] M.M.M. Salem and Ayman A. Aly, Fuzzy Control of a Quarter-Car Suspension System, *World Academy of Science, Engineering and Technology*, 53 (2009), pp. 258-263.

[5] T.-T. Fu and D. Cebon, Analysis of a Truck suspension Database, International *Journal of Vehicle Design*, *Heavy Vehicle Systems*, March (2002).

[6] NCHRP (report 505), Review of Truck Characteristics as Factors in Roadway Design, ISBN:0-309-08779-1, Washington D.C. (2003).

[7] S. Butdee and F. Vignat, TRIZ Method for Lightweight Bus Body Structure Design, *Journal of Achievements in Materials and Manufacturing Engineering*, Vols. 31, Issue 2, December (2008), pp. 456-462.

[8] T.H.T. Chan, Y.Yu, K.Y.Wong and Z.X.Li, Condition-Assessment-Based Finite Element Modeling of Long-span Bridge by Coupling Method, *National Natural Science Foundation of China*, (No. 50278017).

[9] V. Adams and A. Askenazi. *Building Better Products with Finite Element Analysis*, 1st edition, ISBN: 978-1566901604, On Word Press (1999).