Finite Element Analysis of an Electric Bus Body Structure in Real Driving Conditions

Dittapoom Shinabuth[1,*], Chi-na Benyajati[2], Sittikorn Lapapong[2], Monsak Pimsarn[3] and 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 order to evaluate structural strength of a bus body, it could involve a lot of materials cost and time by using an experimental approach. This research proposed procedures for reducing such drawbacks by a transient response study using finite element analysis. In this work, a computational model was created to replicate a chassis and body framework of an electric bus prototype. In analysis, four driving conditions were studied to obtain the dynamic responses. It was found that the maximum stress of 57.21 MPa, 93.41 MPa, 154.64 MPa, and 141.94 MPa were generated in bumping, braking, cornering, and torsional case respectively. The bumping condition displayed the lowest stress level while the maximum stress was in the cornering case. Moreover, torsional stiffness of 25,234 N-m/deg was calculated during the torsional load application. It was concluded that the strength of the electric bus prototype was satisfactory to design specification and safety aspect under general driving applications. Furthermore, possible improvement of an electric bus body design was discussed regarding a structural weight optimization to increase a driving performance as well as to reduce energy consumption.

***Keywords**:* Transient response, Electric bus prototype, Driving conditions, Torsional stiffness

1. Introduction

Due to an increasing demand for bus transport in Thailand, many buses are being constructed especially for tourism industries and public transportations. Most small or medium-sized bus structures were fabricated by the local mechanic skills with a few being supervised by qualified engineers. These buses may be subjected to various problems such as tire wear, inadequate structural durability, and excessive structural weight. Computer-Aided Engineering (CAE) was introduced to solve various engineering problems via numerical technique called Finite Element Analysis (FEA). CAE has become a necessary tool in bus body analysis for preventing any potential problems. Because of environmental issues, automotive industries and research centers have started concentrating their research on alternative energy applications such as Electric Vehicle (EV) and Fuel Cell Vehicle (FCV). An electric vehicle has become one of the solutions to reduce the usage of fossil fuel on land vehicles. In Thailand, while most buses are generally operated with an internal combustion engine (ICE), there are only few driven by an electric motor.

Fig. 1 Electric bus prototype

The bus shown in Figure 1 is an 8-metre bus prototype which could be classified as a medium-sized bus vehicle. In addition, it was powered by electricity from battery sources. A design and analysis of an electric bus structure might be significantly different from an ICE bus structure. In this study, the purpose was to investigate strength of an electric bus prototype under real driving conditions. Moreover, structural members of the body construction which tend to be less important would be identified with a view to obtaining a lightweight electric bus structure.

2. Bus structure

To construct a bus structure, there are several factors that need to be considered such as structural strength, fabrication difficulty and structural configuration. In fabrication process, a bus structure is generally divided into two sections. One is a chassis construction which is a main component built by steel sheets designing to resists an external loads either from the upper or lower structure such as reaction forces from the ground. Another part that placed on the top of the chassis is called a body structure creating to accommodate passengers and bus utilities. The body structure is constructed similar to a skeleton frame that consists of many beam elements with various cross sectional shape. For this reason, the main focus on reducing a structural weight or increasing structural stiffness could be on the body structure. In a studied electric bus structure, a significant difference from the conventional bus structure was a location and weight of the power sources. Geometry of the electric bus structure employed in this study is shown in Figure 2.

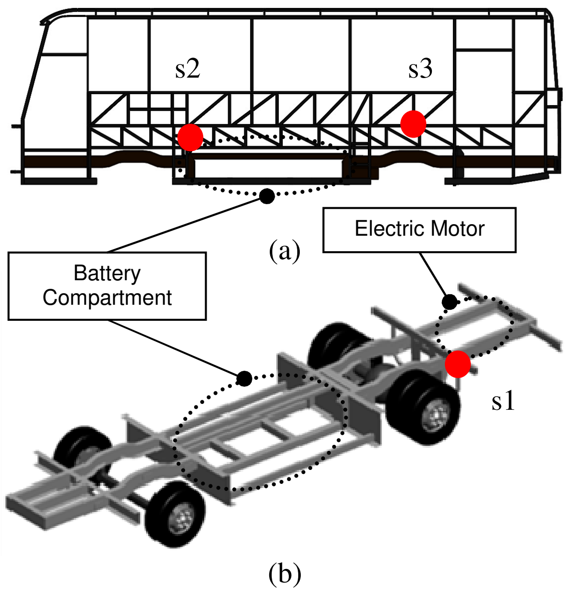


Fig. 2 Geometry of the studied electric bus:
(a) Side view of the frame (b) Chassis

From Figure 2, around the center of the chassis, an empty space between front and rear wheels was reserved. This was designed for carrying battery packs which supplied current to an electric motor. Battery packs combined with high electricity voltage were necessary to be placed at the center of the chassis in order to keep a good stability. On the other hand, many structural members were observed around a waist location of the body frame above battery compartment. The s1-s3 points on the chassis and body were identified for a stress monitoring locations in this study. They were placed behind left rear wheel, at connecting joint between the chassis and body, and on the top of the right rear wheel member for s1, s2, and s3 respectively as designated in Figure 2.

3. Finite element analysis

Finite Element Analysis (FEA) is a numerical technique using partial differential equations to approximate the engineering solutions such as structural, thermal and vibration analysis. In order to take into account the effect of large moving masses, this study employed a transient mode of structural analysis to investigate strength of the bus structure under different four operating conditions.

3.1 Computational modeling

In this process, Finite Element (FE) model was generated to convert a physical geometry of the actual bus structure into an analytical model. The FE model is relatively important to obtain a precise analysis solution. Consequently, the computational modeling was separated to two types because of an appropriate computational time and accuracy. The employed model was consisted of 45,237 beam elements and 14,934 shell elements as shown in Figure 3.

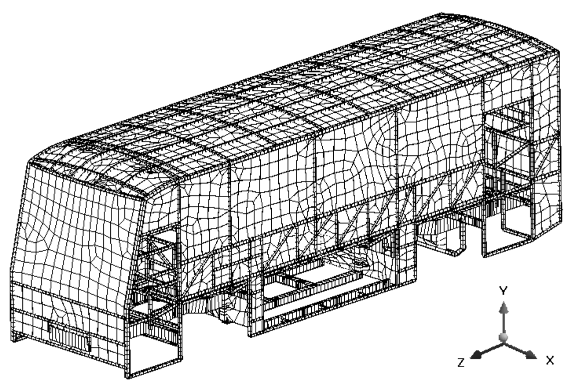


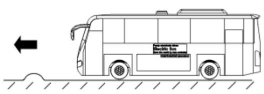
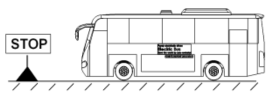
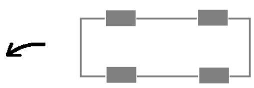
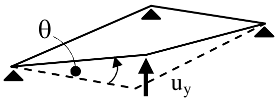
Fig. 3 FE model of the electric bus

The chassis was assigned by c-channel beams while square and rectangular tubes were assigned to the body structure. Material properties were of common steel which had a mass density, elastic modulus, yield strength, and Poisson's ratio of 7,850 kg/m^3, 210 GPa, 324 MPa, and 0.3 respectively [1].

3.2 Boundary conditions

Driving situations affecting either small or large a structural deformation could appear in two main modes [2]. Firstly, it was a bending case happened when the bus was subjected to the vertical load or displacement. Another was a torsional case, an external load attempted to deform the bus structure in rotating fashion along a longitudinal axis. In this case, a twist angle was utilized to calculate a torsional stiffness. In previous study, it was found that a large torsional deformation directly influenced the fatigue fracture of window pillar members [3]. In order to determine structural strength of the bus body operating in the different situations, several boundary conditions were considered as summarized in Table 1.

Table. 1 Driving conditions and applied loads

Driving Condition		Applied Loads
Bump		u_y = 200mm
Brake		a_z = - 0.75g
Turn		a_x = ±0.75g
Twist		u_y = 200mm (at wheel hub)

From Table 1, in a bumping situation, both front and rear axles were activated by sequential vertical forced displacement equal to double amplitude of typical speed bump height of 200 mm. A braking case was a situation where the bus was driven at constant velocity and a brake pedal was suddenly pressed. The braking deceleration of 0.75g was applied in longitudinal direction. Furthermore, a lateral loading value of 0.75g was employed to obtain both sides of a turning response. Finally, a vertical displacement of 200 mm was applied to lift one wheel upward while other three wheels were constrained to simulate a twisting behavior of the bus. In addition, distributed loads were included in the analysis comprising a battery loading of 2,000 kg, a passenger mass of 2,300 kg, and an air conditioner system of 200 kg [4].

4. Results

As mentioned in previous section, a transient dynamic analysis was employed to obtain the dynamic response of the bus structure under specified time-dependent loads and constraints. To display a solution, combined stress which was a combination between axial and bending stress was determined to distribute the calculated stress throughout the electric bus structure. Moreover, the critical location on each case was magnified for better visibility.

4.1 Bumping case

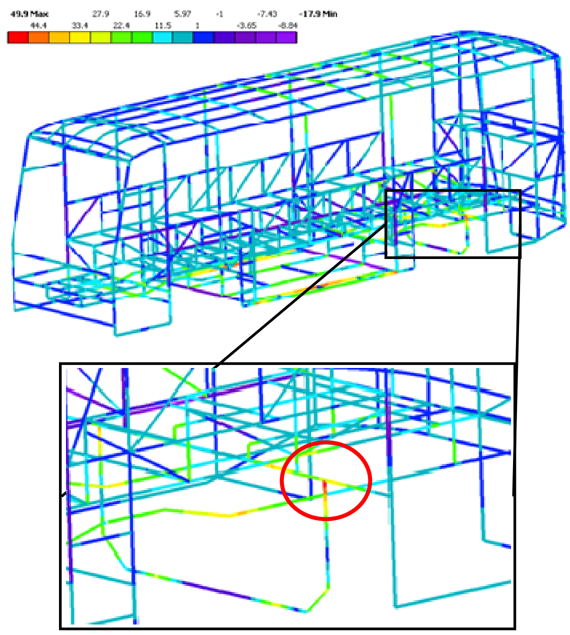


Fig. 4 Calculated stress distribution in a bumping case

Figure 4 shows a combined stress plot that was calculated by vertical displacement applying to the front and rear axles in sequence. It was found that the critical point was illustrated at location s1 that was close to an access to the rear door. The maximum stress of 57.21 MPa was exhibited for the duration of rear bump together with another significant stress observed at the lower frames carrying battery packs on both sides of the bus. Additionally, the central tubular construction was not on duty to resist the bumping condition.

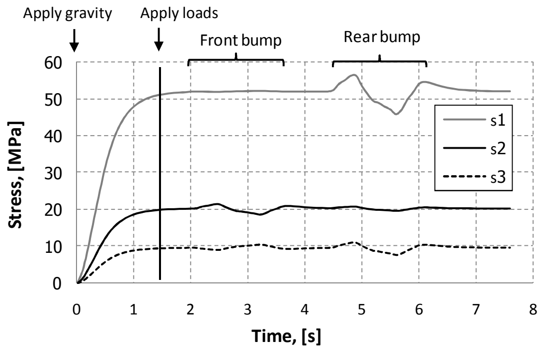


Fig. 5 Bumping responses at selected monitoring locations

Dynamic stress responses were displayed through three different monitoring locations as shown in Figure 5. Before performing a driving condition, the bus body was subjected to a gravitational acceleration of 9.81 m/s^2 as part of dynamic calculation by the software. As a result, an initial state of three monitoring points had different stress magnitudes. As can be seen in Figure 5, s1 showed 52.27 MPa followed by 19.65 MPa and 9.14 MPa at location s2 and s3 respectively. During bumping sequence, a significant stress variation was generated to be 47.5 MPa at location s1 during the rear axle climbed to the peak of the speed bump while others two locations presented minute stress level on both front and rear bumping.

4.2 Braking case

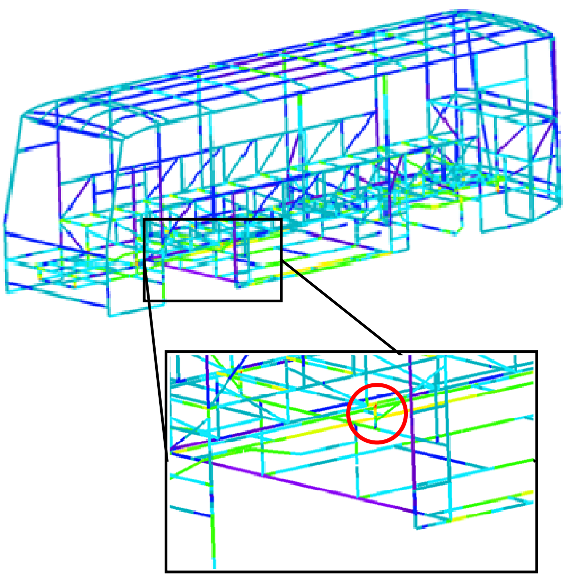


Fig. 6 Calculated stress distribution in a braking case

During a braking period, the structure was subjected to a longitudinal loading and the maximum stress of 93.41 MPa in tension was observed at location s2. This was a tubular member at the front of battery compartment located between the chassis and the body structure. In addition, tubular members around the central body construction and roof frame received little effect from the deceleration load.

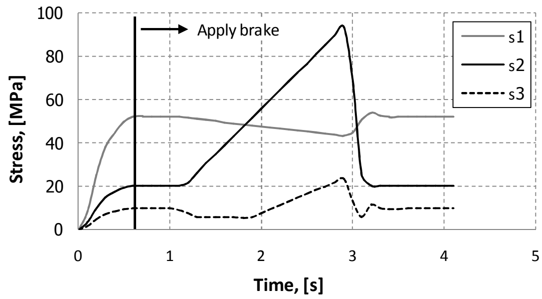


Fig. 7 Braking responses at different locations

The effect of deceleration at measurement points from braking maneuver was displayed in Figure 7. Similar to the bumping case, the stress response at s1 was the highest compared to s2 and s3 after applying only gravity to the model. The noticeable stress response was seen at location s2. The resulting stress of s2 steadily increased when a brake pedal was pressed and reached its peak at 93.41 MPa within 1.9 seconds in relation to the profile of applied deceleration. Furthermore, a dynamic stress at s3 showed two other distinct variations. It could be seen that there was a first region of small stress drop which was accompanied by a rise to the highest point of 22.35 MPa. After that, the stress dropped rapidly with small vibration response coinciding with the bus stopping completely.

4.3 Cornering case

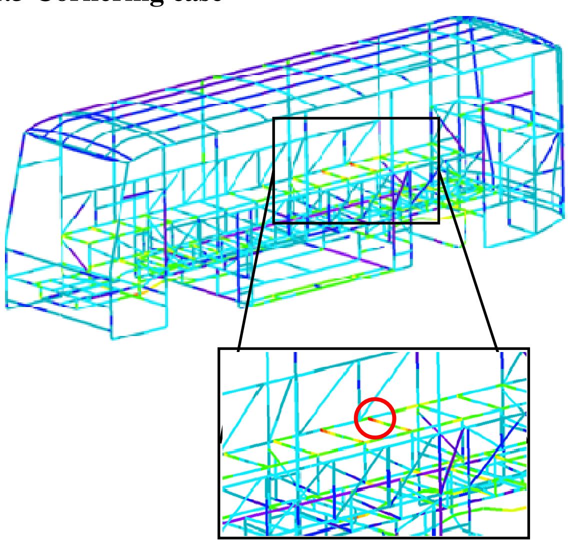


Fig. 8 Calculated stress distribution in a cornering case (Right turn)

In cornering case, a beam member underneath passenger seat above the right rear wheel was estimated to be the critical point under the lateral loading. It was calculated that maximum stress of 154.64 MPa occurred during a right turning. Although both side frames were subjected to the lateral load, the roof frame was found to be under relatively small amount of stress.

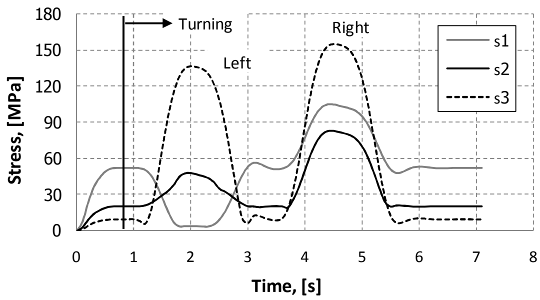


Fig. 9 Cornering responses at different locations

The cornering responses were exhibited in Figure 9. In general, there were two noticeable stress variations in the response from each monitoring location coinciding with left and right turning sequentially. However, the main focus was directed to location s3. The s3 line first climbed up smoothly to 137.23 MPa during the left turning condition followed by the second peak of the maximum stress at the right turn.

4.4 Torsional case

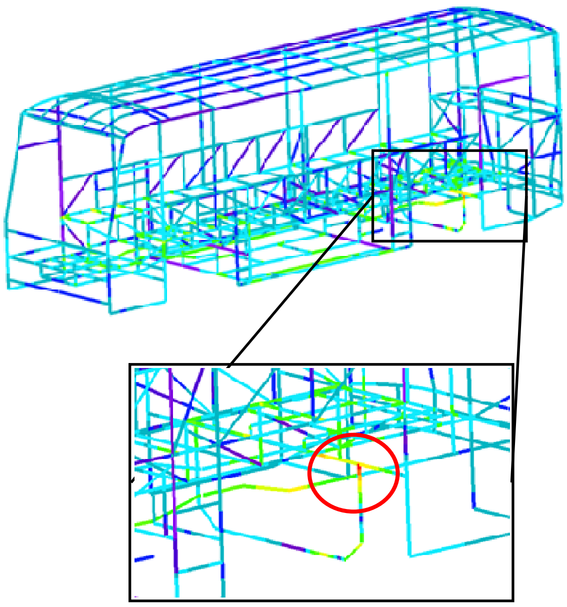


Fig. 10 Calculated stress distribution in a torsional case

In this case, the vertical displacement was used to induce a torsional load to the structure. It is important that the vehicle structure subjected to the torsional load has significant effect to the side frame of a bus structure [5]. The computed solutions showed that stress of 141.94 MPa was generated at the critical point when the left rear wheel was lifted off the ground by 200 mm. The location of maximum stress was the same as the case of rear axle bump. Consequently, a torsional stiffness of the studied bus structure was needed to assess the structural rigidity in comparison with the standard value ranging from 18,000 - 40,000 N-m/deg for a bus structure [6]. To calculate a torsional stiffness (K), the applied torque (T) and twist angle (θ) were the major parameters. On the other hand, a reaction force (F_z), a distance between the wheels in the same axle (L), and a vertical displacement (Δy) may be applicable as follows:

$$K = \frac{T}{\theta} = \frac{F_z \cdot L}{tan^{-1}\left(\frac{\Delta y}{L}\right)} \tag{1}$$

The calculation following Eq. (1) showed that a torsional stiffness could be obtained from a slope between an applied torque and twist angle as displayed in Figure 11. In this case, the corresponding torsional stiffness of the electric bus prototype was calculated to be 25,234 N-m/deg.

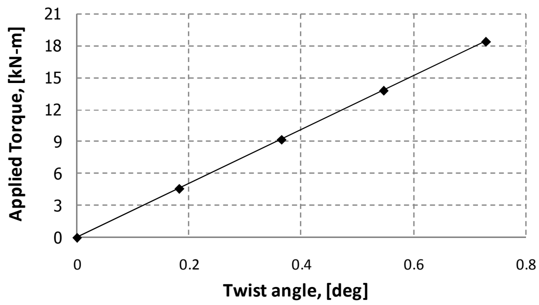


Fig. 11 Torsional stiffness

5. Discussion

In a bumping case, the structure was subjected to the vertical load. It was found that stress mainly appeared at the left rear wheel during the rear axle bump. It could be said that the members constructed around a rear door was a main reason due to lack of supporting members because of a space needed for the door frame. Besides, several areas on the body, namely the waist, the roof frame, and the construction around electric motor seemed to be diminutively affected by the bending load as shown from monitoring location s2 and s3 in Figure 5.

The calculated tress distribution in a braking case was simulated by longitudinal load as shown in Figure 6. According to the load transfer of large moving mass of battery packs, the maximum stress was indicated at the joint in front of the battery compartment. It might be due to low strength by too few supporting members as well as small thickness of material used in this area. Moreover, a small stress level occurred on the roof frame and along the central body forming passenger section above the battery compartment. Additionally, significant stresses were still generated along the beams supporting battery packs in the battery compartment.

A cornering case was representing the effect of lateral load. In Figure 8, significant stress concentrations on the structure during the left and right turning were not only noticed above rear wheel positions but also on the front section of the chassis. Moreover, it was observed at the location opposite of s3, i.e. on the left side of the body structure, a similar stress response pattern to those of s3 shown in Figure 9 but in reverse order. This showed that both side of waist members were affected by a lateral loading in any direction.

Torsional loading case was studied in order to obtain a torsional rigidity of the structure during twisting action as shown in Figure 11. The vertical displacement was applied to the left rear wheel for twisting the bus structure. As a result, the torsional stiffness of the whole structure was estimated to be approximately by 71.3 % higher than the minimal requirement. Therefore, the bus structure was capable of resisting certain degree of external load which came in torsional fashion. Moreover, the safety factor in each driving condition was calculated with respect to the yield stress of steel to be 5.66, 3.46, 2.09, and 2.28 in case of a bumping, braking, cornering and torsional maneuver respectively. All analysis results illustrated that strength of the electric bus body was reasonably high enough to operate in majority of normal driving conditions.

6. Conclusions

Transient dynamic analysis was employed to determine strength of the electric bus structure in four driving situations that frequently happened during the operation. The calculated maximum stress of 57.21 MPa, 93.41 MPa, 154.64 MPa, and 141.94 MPa was presented during bumping, braking, cornering and torsional case respectively. It was clear that a speed bump with the height of 200 mm had insignificant effect to the structure while the cornering case gave the highest resulting stress in all four situations. From the simulated results, it could be concluded that overall structural strength of the electric bus prototype was satisfactory under four common driving situations and further structural optimization could be carried out. The waist beams of the body and roof frame have been observed to exhibit a common area of relatively low resulting stress from all the cases considered. These areas were generally fabricated from many square tubes in 38x38 mm with 3.2 mm thickness. Furthermore, optimum structural weight of the electric bus could be achieved by changing cross-sectional area, removing zero force members, and rearranging the alignment of the beam fabrication. These parameters will be investigated further in a future work.

7. Acknowledgements

The author would like to specially thank Automotive Laboratory, National Metal and Materials Technology Center (MTEC), Thailand for their support.

8. References

[1] Santosh B. Belure, Satish S. Kadam and S. B. Wadkar (2008). Criticality Analysis of Passenger Bus Body Structural Design Using Finite Element Method, paper presented in *National Conference on Computational Methods in Mechanical Engineering,* India.

[2] Masahiro, S. and Yukio, K. The strength of Automobile Body, 1998 (AddTech, Korea)

[3] M H Kim, MW Suh and D H Bae. (2001). Development of an Optimum Design Technique for the Bus Window Pillar Member, *IMechE*, vol. 214 Part D, pp. 11-20.

[4] D. Croccolo, M. D. Agostinis and N. Vincenzi. Structural Analysis of an Articulated Urban Bus Chassis via FEM: a Methodology Applied to a Case Study, *Journal of Mechanical Engineering*, vol.57(11), October 2011, pp. 799 – 809.

[5] Lonny L. Thompson, Srikanth R. and E. Harry Law. Design of a Winston Cup Chassis for Torsional Stiffness, SAE Technical Paper Series, paper presented in *Motorsports Engineering Conference and Exposition*, November 16-19, 1998, Michigan, U.S.A.

[6] F. Lan, J. Chen and J. Lin. (2004). Comparative Analysis for Bus side Structures and Lightweight Optimization, *IMechE*, vol.218 Part D, May 2004, pp. 1067 – 1075.