Stress Analysis and Validation of Superstructure of 15-meter Long Bus under Normal Operation

Lapapong S., Pitaksapsin N., Sucharitpwatkul S.*, Tantanawat T., Naewngerndee R. and Phuchamnong A.

National Metal and Materials Technology Center

E-mail address: sedthaws@mtec.or.th

Abstract

The strength of the superstructure of a bus is very critical to the safety of passengers, both in normal operation and in the event of accident. During the normal operation, the structure of the bus is subjected to several loads, which may be induced by its inertia during vehicle maneuvering (i.e. braking and cornering) or by external loads from the road (i.e. crossing over a speed bump). Moreover, there is a substantial possibility that these loads may lead to a structural failure. Hence, it is necessary to determine stresses occurred in the bus's superstructure to ensure its integrity under these driving scenarios. This paper presents techniques implemented to analyze stresses on the superstructure of a newly designed 15-meter long bus subjected to loads previously mentioned using Finite Element Method (FEM). The stress analysis technique used in each scenario is selected based upon the frequency intensity of load excitations and the dynamic responses of the structure. Besides, the results obtained from FEM, particularly strains, are validated with the experimental ones to investigate the fidelity of the selected techniques.

Keywords*: Finite Element Method, Stress Analysis, Bus, Structure*

1 Introduction

A bus production is one of a few sectors in automotive industry in Thailand that involves structural designs. A domestic bus maker purchases a bus chassis and other components from its suppliers. Based on the acquired chassis and components, the bus maker designs and builds a bus frame, called a superstructure, to form the skeleton of a bus. Designing the superstructure is one of the critical steps in the production process. If the superstructure is overdesigned, the bus will be heavy and result in high fuel consumption and short lifetime. If it is under-designed, the bus will be easily damaged and require frequent repairs. The goal of the design is to have a superstructure with sufficient strength to withstand loads while keeping the weight as minimal as possible. Once the design of the superstructure is obtained, the bus maker builds it on the acquired chassis. Seats, floors, panels, glasses, engine, and other components are put together to complete the bus production process.

Most of the buses built in Thailand typically have the length not exceeding 12 meters. To increase the capacity of the bus, while maintaining the same amount of space for each occupant, the length of the bus has to be increased. The chassis for a 15 meter-long bus is commercially available. The task of the bus maker is to redesign the superstructure of the bus for this new specification. Designing the superstructure based on trial and errors may be carried out as a conventional design method. However, the cost of such method is too expensive for the bus maker to survive in the competitive business. One approach to help reach a highquality design while keeping the development cost low is to employ a finite element method. Nonetheless, the results of the finite element method will be valuable only when the data, such as loading characteristics, used in the finite element method is closely related to reality. The objective of this paper is to present techniques used to analyze stress on a superstructure for a 15-meter-long bus that is subjected to loads under a normal operation. Three scenarios: braking, double-lane change, and speed-bump crossing, are arranged in a field testing to represent frequent situations that the bus typically experiences in the normal operation. In the field test, acceleration and strains of the superstructure are acquired. The acceleration is later inputted as a load acting of the superstructure in the stress analysis of each scenario. The technique used to perform the stress analysis in each scenario is determined by evaluating how intense the associated load excites the structural dynamics. To investigate the effectiveness of the applied techniques, the strains obtained from the field testing are compared to the ones from the FEM simulations.

2 Finite element modelling

To construct a finite element model of the superstructure of the bus shown in Figure 1, a three-dimensional solid model of the superstructure is first formed in a Computer-Aided Design (CAD) software as illustrated in Figure 2. The superstructure basically consists of several beam sections and thin-plate components. Based on this CAD model, the finite element model is further developed in a commercial Computer-Aided Engineering (CAE) software as shown in Figure 2. In this finite element model, the beam structures are meshed with one-dimensional nonlinear beam elements that have six degrees of freedom per node (CBEAM) [1]. Two dimensional shell elements (Kirchhoff element) [2] are applied to the thin-plate components such as floor, interior wall, etc. The density of the superstructure is adjusted, taking masses of body panels and windows into consideration. Regarding this assumption, the density is non-uniform and considerably varied throughout the superstructure. Masses of passengers, seats, engines, air conditioning systems, etc. are represented by lumped masses located at their original centers of gravity. This model is entirely composed of 6,718 elements and 6,225 nodes. The mechanical properties of the superstructure are acquired from the ASTM E8 tension testing [3] and summarized in Table 1.

Figure 1: 15-meter long bus.

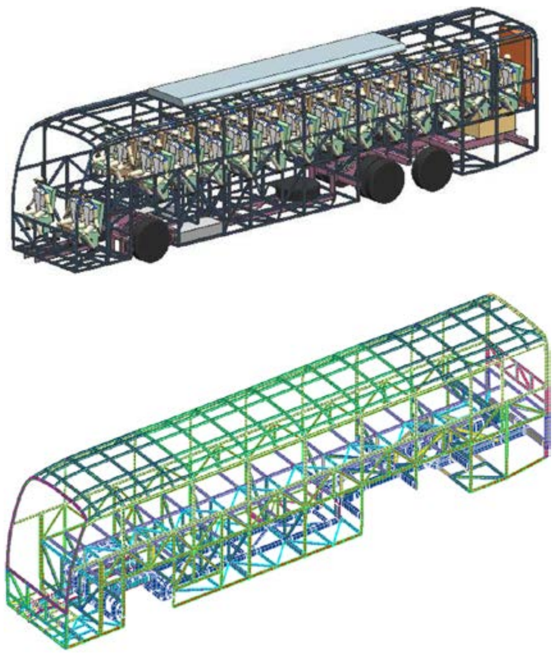

Figure 2: Complete CAD model and finite element model of the superstructure.

Table 1: Mechanical properties of the superstructure.

Modulus of elasticity (E)	193 GPa
Poisson ratio (ν)	0.30
Density (ρ)	7,850 kg/m^3
Yield strength (σ_{yt})	333.87 MPa

3 Field testing

This section describes a field setup implemented to measure acceleration (and/or deceleration) and structural strains induced during the scenarios of braking, double-lane change, and speed-bump crossing. In this work, the acceleration is necessary to define a load exerting on the superstructure. The strain is essential to evaluate the fidelity of simulation results. To measure these, four tri-axial accelerometers are installed on the chassis of the bus, and seven foil strain gauges are attached to the superstructure. Their output signals are recorded through a DAQ system. Figure 3 shows the rough locations of these devices. In the figure, an accelerometer and strain gauge are denoted by “A” and “S”, respectively. The convention of the coordinates used throughout this work is defined by the Society of Automotive Engineers [4].

After the field testing is conducted, the acceleration data are post-processed with a low-pass filter to eliminate noises. Figure 4 illustrates the accelerations of suddenly braking bus with the initial longitudinal velocity of 16.7 m/s (60 km/hr). The maximum longitudinal acceleration in this case is estimated to be 7.8 m/s2 (0.8g's). The lateral accelerations of the bus manoeuvring the double-lane change are shown in Figure 5, and the peak acceleration is approximately 3.4 m/s2 (0.35g's). Figure 6 presents the accelerations while the bus is crossing over the speed bump at the longitudinal velocity of 8.3 m/s (30 km/hr). Evidently, from the figure, this driving condition may be considered as an impact. In these figures, the longitudinal, lateral, and vertical accelerations are respectively represented by the blue, green, and red lines as well as the subscripts "x", "y", and "z".

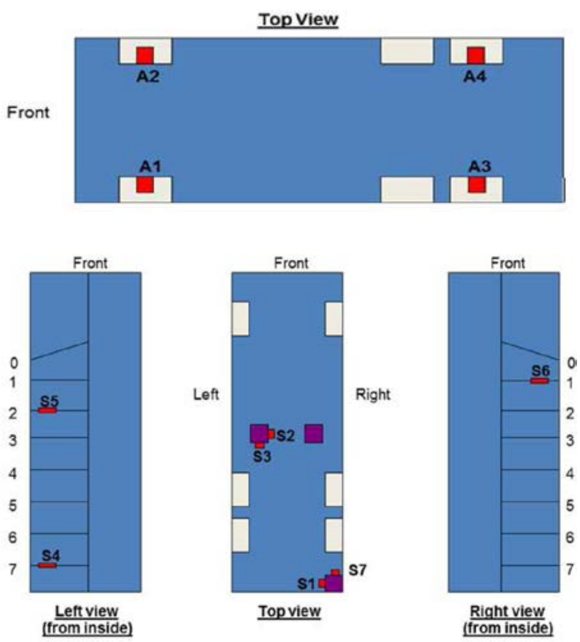

Figure 3: Locations of accelerometers (A) and strain gauges (S).

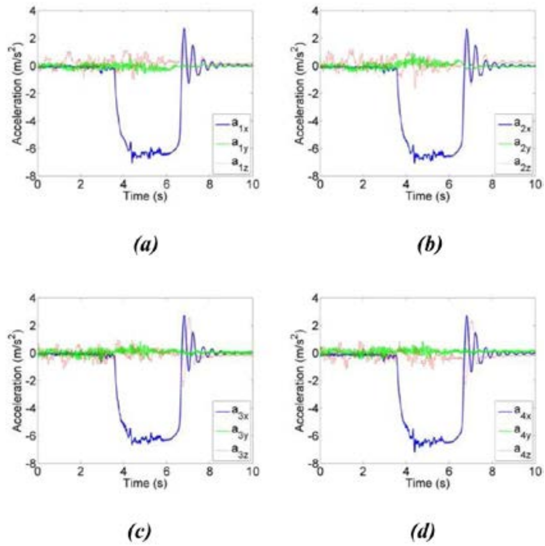

Figure 4: Accelerations recorded during braking with the initial longitudinal velocity of 16.7 m/s (60 km/hr). (a)Accelerometer A1. (b) Accelerometer A2. (c)Accelerometer A3. (d) Accelerometer A4.

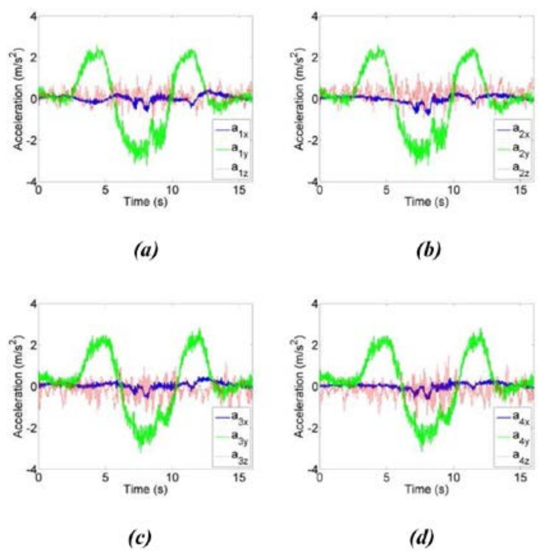

Figure 5: Accelerations recorded during double-lane change at the longitudinal velocity of 16.7 m/s (60 km/hr). (a) Accelerometer A1. (b) Accelerometer A2. (c) Accelerometer A3. (d) Accelerometer A4.

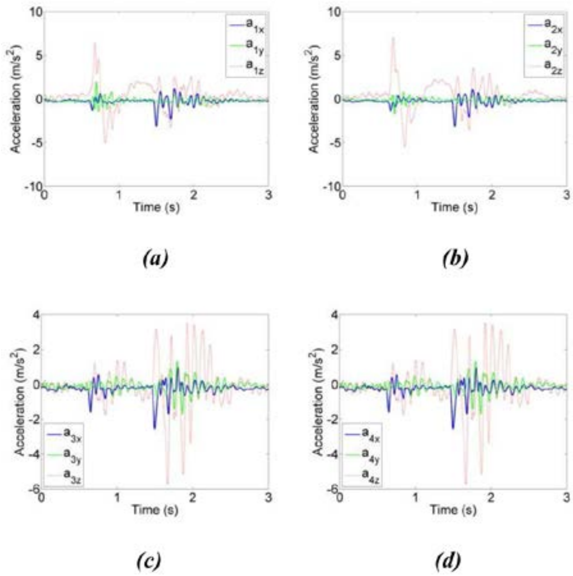

Figure 6: Accelerations recorded during speed-bump crossing at the longitudinal velocity of 8.3 m/s (30 km/hr). (a) Accelerometer A1. (b) Accelerometer A2. (c) Accelerometer A3. (d) Accelerometer A4.

4 Stress analysis

This section presents the results of FEM-based stress analysis of the superstructure in the interested scenarios, which are braking, double-lane change, and speed-bump crossing. Before the stress analysis is performed in each driving condition, the relevant analysis technique must be selected. Thus, the frequency intensity of load excitations and the dynamic responses of the superstructure are first considered. Figure 7 shows the frequency-domain representation of the longitudinal acceleration data acquired from four accelerometers during the braking. From the figure, it is evident that this acceleration data inherently contain relatively low frequency contents (< 1 Hz). Further, the FEMbased modal analysis of the superstructure reveals the natural frequency of its first mode to approximately be 3.8 Hz. Based upon these, it may be concluded that the load, which is the acceleration, is not able to excite the dynamics of the superstructure in this case. The similar conclusion may also be drawn in the case of double-lane change. Hence, the technique of nonlinear quasi-static stress analysis, which does not concern the structural dynamics, is adopted in these cases.

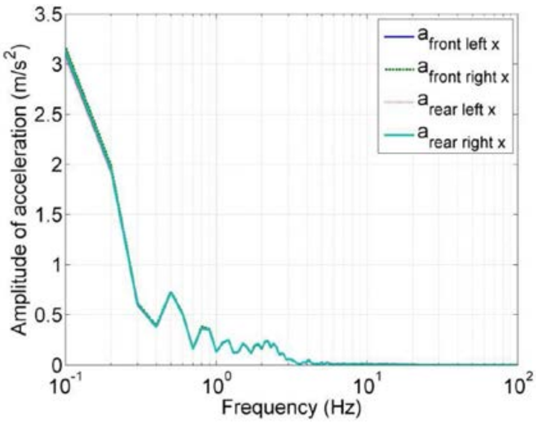

Figure 7: Frequency-domain representation of the longitudinal acceleration during braking.

In the stress analysis of the braking and double-lane change scenarios, the superstructure is constrained with a simple support as shown in Figures 8 and 9. The front and rear supports are the pinned and fixed translation ones, respectively. The fixed translation supports allow the superstructure to move in the longitudinal direction. In the braking scenario, the longitudinal deceleration with the magnitude of 7.85 m/s2 (0.8g's) is applied on the superstructure as a body force. Figure 8 illustrates the distribution of maximum combined stress [1] in the superstructure in this case. The maximum stress is 189 MPa at the point near the front axle. In the case of double-lane change, the superstructure is acted by the lateral acceleration with the magnitude of 3.43 m/s2 (0.35g's) in the same fashion. The structural maximum combined stress of this case is shown in Figure 9. Located near the rear axles, the largest stress in this case is computed to be 119 MPa.

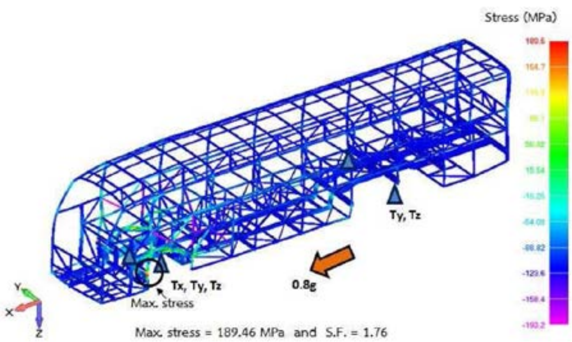

Figure 8: Distribution of maximum combined stress in the superstructure in the case of braking.

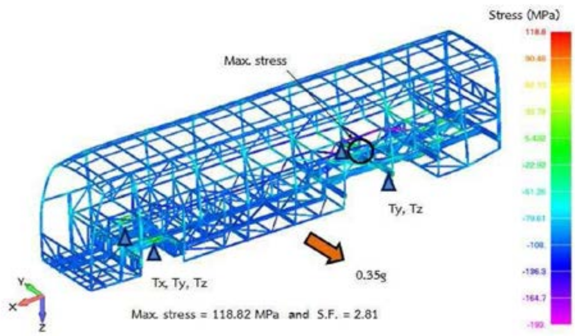

Figure 9: Distribution of maximum combined stress in the superstructure in the case of double-lane change.

To analyze the structural stress in the speed-bump crossing case, the response spectrum method [5] is employed, since the external loads contains the frequency content that is in the range of dynamic excitation of the superstructure as shown in Figure 10. Figure 11 presents the stress distribution in the superstructure for this case. The input loads are the frequency-domain representation of the vertical accelerations given at the points of supports (indicated by the arrows in the figure). The stress in each mode is combined by the method of the Square Root of the Sum of the Squares (SRSS) [5]. From the figure, the maximum stress is approximately 188 MPa located in the right side-member above the rear axles.

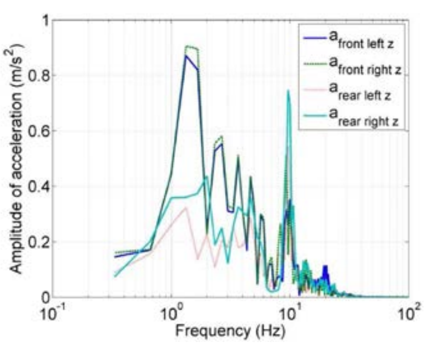

Figure 10: Frequency-domain representation of the vertical acceleration during braking.

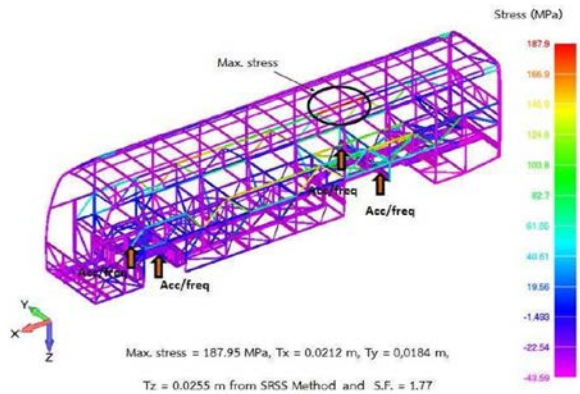

Figure 11: Distribution of combined stress in the superstructure in the case of speed-bump crossing.

5 Validation of stress analysis

The fidelity of the stress analysis technique employed only in the cases of braking and double-lane change is investigated in this section. However, the validation in the speed-bump crossing case is not presented, because of the experimental limitation based on the fact that the response spectrum method and SRSS combination method are used to estimate the probable maximum structural stress in this case. As previously described, the structural strains are recorded during the field testing and subsequently used as a validation index. Figures 12 and 13 illustrate the experimental strains compared to the ones computed from the simulations in the cases of braking and double-lane change, respectively. In the figures, the horizontal axes indicate the strain gauges' identification numbers as shown in Figure 3, and the vertical axes represent the maximum stresses at their defined locations. The plots show that the finite element simulations give an acceptable strain prediction in both scenarios; however, there are a minority of validation points in the braking case that introduce errors. Thus, it may be concluded that the technique used to perform the stress analysis in this work is rather valid, and a further investigation is required to determine the source of errors.

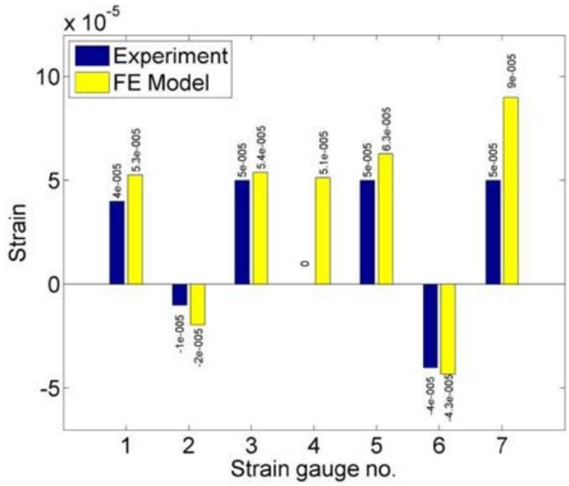

Figure 12: Comparison of strains obtained from the field experiment and finite element model in the case of braking.

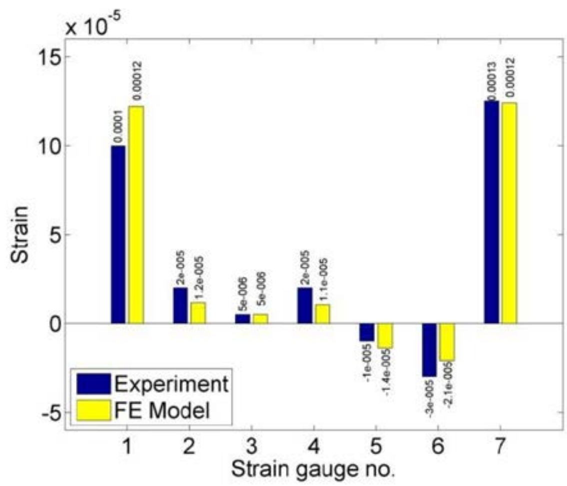

Figure 13: Comparison of strains obtained from the field experiment and finite element model in the case of double-lane change.

6 Conclusions

This work outlines a procedure to compute the stress distribution of a bus superstructure that undergoes a daily operation. Three driving conditions including braking, double-lane change, and speed-bump crossing are candidates to represent the scenarios that a bus frequently experiences. These conditions are arranged in a proving ground to acquire accelerations exerting on the structure and structural strains at various locations. The acceleration data are further analysed to determine their frequency contents. Based on frequency contents and the modal analysis of the structure, a nonlinear quasi-static FEM is selected to perform the stress analysis in the cases of braking and double-lane change, and the response spectrum method is applied in the speed-bump crossing case. The FEM results reveal that the maximum induced stresses in all three driving conditions do not exceed the yield strength of the material used to construct the superstructure. In addition, the strain values obtained from the finite element simulations are found to be close to those measured by strain gauges, which confirm the validity of the applied stress-analysis techniques.

Acknowledgements

The authors gratefully acknowledge the financial support of this work from Thepsombattour Co., Ltd. through the National Metal and Materials Technology Center under the project no. P-11-00440.

References

[1] MSC Software Corporation, 2012., *MSC Nastran 2012 Quick Reference Guide*, Santa Ana, CA.

[2] Zienkiewicz, O.C. and Taylor, R.L., 2000., *The Finite Element Method: Volume 2*, Fifth Edition, Butterworth-Heinemann.

[3] *ASTM E8/E8M-11*: Standard Test Method for Tension Testing of Metallic Materials.

[4] *SAE International Surface Vehicle Recommended Practice*, “Vehicle Dynamics Terminology,” SAE Standard J670e, Rev. July 1976.

[5] Gupta, A.K., 1990. *Response Spectrum Method in Seismic Analysis and Design of Structures*, Blackwell Scientific Publications.