STRUCTURE ANALYSIS OF LOW FLOOR ELECTRIC BUS USING THE FINITE ELEMENT METHOD

Worakiat Prasatsup, Kontorn Chamniprasart*

School of Mechanical Engineering, Suranaree University of Technology

*** e-mail : kontorn@sut.ac.th**

ABSTRACT

Currently, the industrial sectors are realized the importance of energy. Because the fuel is used widely, the current volume of available fuel is limited and phased down indefinitely. The automotive industry is another large industry that still depends on fuel energy. Because the fuel cannot stand longer in the future and the combustion of fuel causes pollution to environment. The automotive industry has researched and developed alternative energy vehicles to reduce the use of fuel. The electric vehicle is one choice for alternative energy vehicles. It is the most popular and has been developed for higher performance. Currently, the use of private automobile have increased, especially in Thailand, the average orders of automobile per year has increased. Therefore, the traffic jam in the city can causes air pollution. A campaign of using the public bus transportation instead of private automobiles is seriously applied in community. A project “Electric Bus”, that uses electric energy to drive and is environmentally-friendly, has been made the electric bus is a low floor bus and is suitable to use in the city. All kinds of passenger can use the low floor electric bus, the bus is comfortable, safe and easy to use for disable and elderly. In the design and production processes of the low floor electric bus, the safety must be the first priority. The purpose of this work is to analyze the structural strength of the low floor electric bus using Finite Element Method. The battery module location on the low floor electric bus is a main subject to be analyzed its effect of the bus structure. The analysis can forecast the structural strength in terms of stress, strain, and displacement under several load and constrain conditions. The methodology of the analysis is applied through case studies, which reflect the actual duty cycle of the bus. The results are expected to effectively enhance the improvement and development of the low floor electric bus structure.

1. INTRODUCTION

The statistics of road accidents take place every year causing many fatalities and severe injuries to the passenger. The safety must be the first priority for design and production of the new bus. If making the bus are safer, this problem can be reduced. Currently, the structural design and analysis of the bus are modern and more comfortable for engineers. Computer simulations become more important for the bus simulation. They can analyze the structural strength using Finite Element Method. FEM is a powerful numerical engineering analysis, and widely used in static and dynamic stress analyses of vehicles. The results of the numerical analysis revealed that the location of maximum deflection and maximum stress agrees well with theoretical maximum location of simple beam loaded by uniform force (Veloso, et al. 2009). Croccolo, et al. (2011) presented an analyzed structural of an articulated urban bus chassis, with a total length of 18 m, two chassis, using Finite Element Method. The structural response was expressed in terms of stress, strain, and displacement, under several loading, at reflecting the actual duty cycle of the bus. Hemant B.Patil, et al. (2013) proposed the stress analysis of automotive chassis with various thickness and change the position of cross member in order to reduce the magnitude of stress at critical point of the chassis.

The purpose of this work is to simulate and forecast the structural response of the low floor electric bus, in terms of stress, strain and displacement, under bending loading and torsion loading that are exerted on the chassis structure, and represent the actual duty cycle of the bus, using Finite Element Method.

2. EXPERIMENT

2.1 Experimental Apparatus

2.1.1 Computer Aided Design. The model of low floor electric bus is modeled from the generation of CBL.EV (Cherdchai Industrial Factory Co., Ltd.). It is created using computer aided design (CAD) SolidWorks 2013. The model composes of 3D beam elements. The electric bus model is chassis structure combined with the body structure as shown in Figure 1. The chassis structure and body structure have assembly parts about 540 parts.

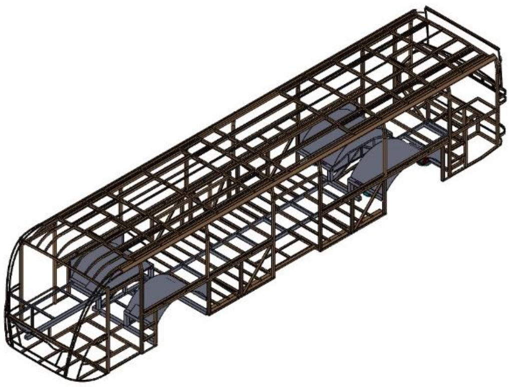

Fig. 1 CAD model of low floor electric bus

2.1.2 Computer Aided Engineering. The structural analysis of the low floor electric bus using computer aided engineering (CAE) ANSYS for creating a mesh and simulation of structural strength using Finite Element Method. The accuracy of results depends upon the accuracy of CAD geometry and quality of meshing.

2.2 Technique

2.2.1 Material Data. The material properties of the model are considered under the linear elastic and isotropic behavior. The chassis structural is stainless steel grades RST4003 with Yield strength = 539 N/mm^2, a Tensile strength = 541 N/mm^2, a Young's modulus = 193 GPa and a Poisson's ratio = 0.26. The external dimensions of stainless steel rectangular section tubular is 80x40 mm and 80x80 mm with the wall thickness of 4 mm. The body structural is steel grades ss400 with Yield strength = 450 N/mm^2, a Tensile strength = 505 N/mm^2, a Young's modulus = 173 GPa and a Poisson's ratio = 0.30. The external dimensions of steel rectangular section tubular is 50x50 mm, 50x25 mm. The wall thickness is the range of 2 to 4 mm. The external dimension of angle steel is 40x40 mm with the wall thickness of 4 mm and 6 mm.

2.2.2 Bending Loads. Each load applied to the chassis and body structure is introduced as a lumped mass. For remote boundaries conditions, the lumped masses are defined. Ansys allows the control of the specific geometry behavior, which can be defined as either rigid or deformable. Lumped masses, that are applied to the chassis as shown in Figure 2, consist of 315 kg of distributed masses belonging to the air condition and ventilators on the roof, 545.20 kg of distributed masses on the front chassis (e.g. driver, steering pump, battery), 2,739 kg of distributed masses belonging to passengers' mass on the middle chassis, 907 kg of distributed masses on the rear chassis (e.g. passengers, air pump, cooling pack, control units) and 2,530 kg of 11 battery modules. Fixed supports have been applied to the rear axle. Moreover, the standard gravitational acceleration g (9.81 m/s^2) has been applied to the whole mass system.

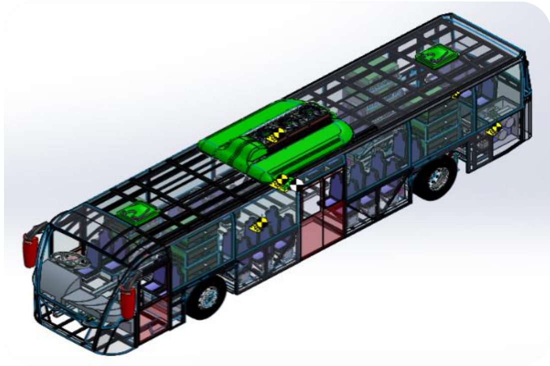

Fig. 2 Lumped masses are applied to the chassis

2.2.3 Torsion Loads. When the electric bus run on an uneven road, the chassis could be subjected to torsion loads. The supports to be fixed have been chosen as follows:

(I) Left front axle constraint and right rear axle constraint.

(II) Right front axle constraint and left rear axle constraint.

3. ANALYSIS

3.1 Governing Equations

3.1.1 Stress Theory

The von Mises stress is an equivalent or effective stress at which yielding is predicted to occur in ductile materials (for example, steel or aluminum alloy), can be evaluated from:

$$\sigma_{vm} = \frac{1}{\sqrt{2}}[(\sigma_1 - \sigma_2)^2 + (\sigma_2 - \sigma_3)^2 + (\sigma_3 - \sigma_1)^2]^{1/2} \qquad (1)$$

The derivation of this form of the von Mises stress is based on the principal axes

The maximum von Mises stress failure criterion is based on the von Mises-Hencky theory, also known as the scalar-energy theory or the maximum distortion energy theory. The theory states that a ductile material starts to yield at a location when the von Mises stress becomes equal to the stress limit. In most cases, the yield strength is used as the stress limit. According to the von Mises failure criterion, the factor of safety (FOS) is expressed as:

$$\text{FOS} = \frac{\sigma_{limit}}{\sigma_{vm}} \qquad (2)$$

3.2 Finite Element Analysis

3.2.1 Bending Load Case.

In the case of bending analysis, when the chassis structure receives the load of lumped mass, the maximum deformation was 0.202 mm, which occurred on the structure of the roof, air condition area. The maximum strain was 212.750 με, which occurred on the floor between the structure of body and chassis, near the rear axial, and the maximum stress was 35.488 MPa, which occurred in the maximum strain point. Figure 3 to Figure 5 show the stress, strain and deformation in the chassis of low floor electric bus, respectively.

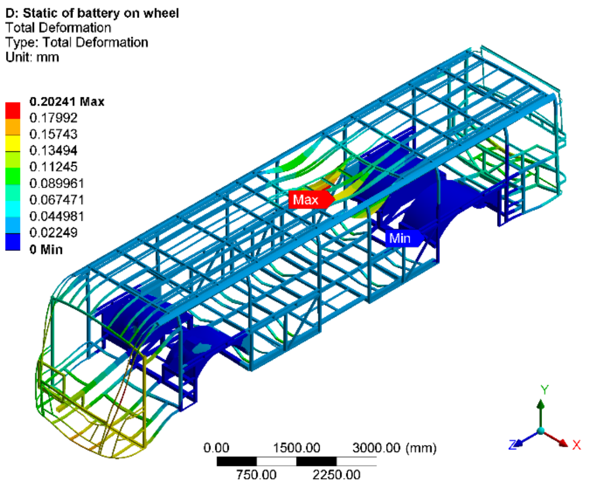

Fig. 3 Total displacements – Bending load case (Scale 3.1e+003)

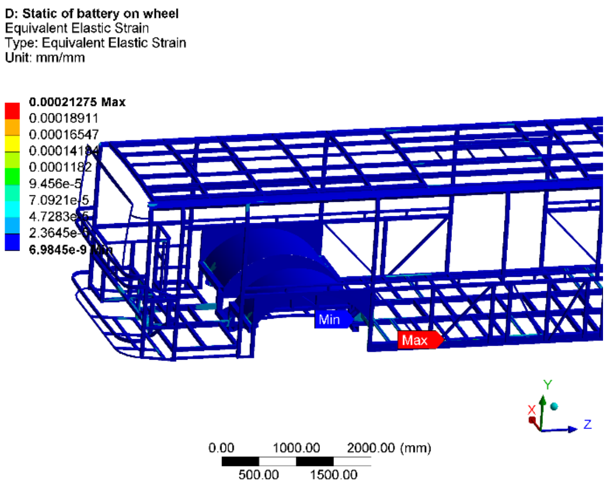

Fig. 4 Von-Mises equivalent strain

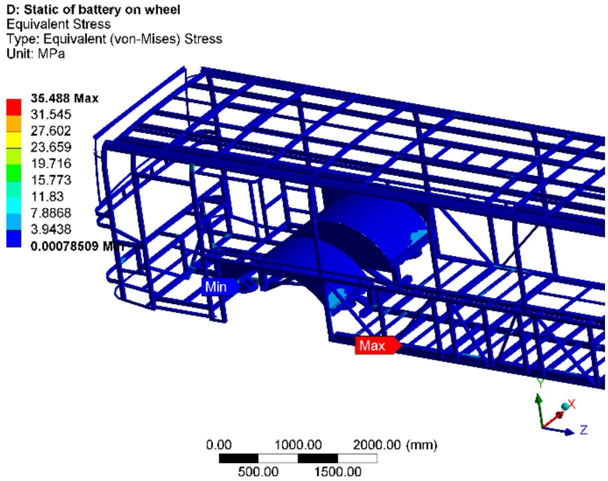

Fig. 5 Von-Mises equivalent stress

3.2.1 Torsion Load Case.

In the case of torsion analysis, when the electric bus run on an uneven road, the chassis could be subjected to torsion load. The maximum deformation was 0.245 mm as shown in Figure 6.

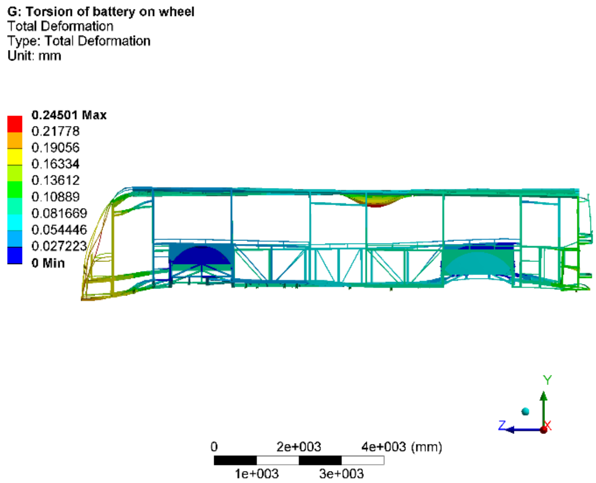

Fig. 6 Total displacements – Torsion load case (Scale 2.6e+003)

CONCLUSION

The static structural analysis of low floor electric bus was performed via the Finite Elements Method using computer aided engineering ANSYS Workbench 14.5. The results of this analysis showed the maximum stress in the chassis structure, when the chassis is subjected to the actual duty cycle. The results of this analysis are expected to use for designing the low floor electric bus structural and adjust the corresponding weaknesses of the structure. It can reduce time consumption in the design engineering process and also reduce the cost of actual test that can enhance high effective productions of the low floor electric bus.

REFERENCES

Veloso, V., Magalhães, H.S., Bicalho, G.I., and Palma, E.S., Failure investigation and stress analysis of a longitudinal stringer of an automobile chassis, *Journal of Engineering Failure Analysis*, vol. 16, issue 5, pp. 1696-1702, 2009

Croccolo, D., De Agostinis, M., and Vincenzi, N., Structural Analysis of an Articulated Urban Bus Chassis via FEM: a Methodology Applied to a Case Study, *International Journal of Mechanical Engineering*, vol. 57, no. 11, pp. 799-809, 2011

Patil, B., Kachave, D., and Deore, R., Stress Analysis of Automotive Chassis with Various Thicknesses, *IOSR Journal of Mechanical and Civil Engineering*, vol. 6, issue 1, pp.44-49, 2013

NOMENCLATURE

σ_{vm} : Von Mises stress
σ_{limit} : Yield strength

Subscripts
CAD : Computer Aided Design
CAE : Computer Aided Engineering
FEM : Finite Element Method
FOS : Factor of safety

Worakiat Prasatsup
received the B.E. (2012) degrees in automotive engineering from Suranaree University of Technology.

Kontorn Chamniprasart
Received the B.S. (1980) degree in Mechanical Engineering from (First Class Horner) from Royal Thai Air Force Academy, Thailand, M.Sc. (1987) degree in Mechanical Engineering from University of Pittsburgh, USA, and Ph.D. (1992) degree in Mechanical Engineering from University of Pittsburgh, USA