JURNAL TEKNIK MESIN
(JOURNAL OF MECHANICAL ENGINEERING)

**VOL. 14, NO. 2, June 2025**
https://publikasi.mercubuana.ac.id/index.php/jtm/index
DOI: 10.22441/jtmv14i2.33587

# Finite Element Analysis and Optimization of Medium Bus Frame Structure

**Ninda Kurniadi[1], Kaleb Priyanto[1], and Farda Pega Libra Gojandra[1]**
[1]Mechanical Engineering Department, Faculty of Engineering, Universitas Negeri Semarang, 50229, Indonesia.

E-mail: nindakurniadi@mail.unnes.ac.id

***Abstract--****Buses are very important transportation for people all over the world, but many still use non-renewable fuel, causing harmful exhaust emissions and noise. Electric buses are becoming more important as they don't produce exhaust and make less noise. The main goal of this research is to analyze the torsion strength and optimization of a medium electric bus frame structure using the Finite Element Method (FEM). This is done because a conventional chassis is being converted to electric by replacing engine components with electric parts like motors and batteries. The study performed a torsion strength analysis under static conditions, using the stress and displacement results as constraints for optimization. Size optimization was applied, changing the thickness of the frame rectangular hollow to find the lightest design. The results show that the frame structure's weight was successfully reduced by 12%, decreasing from 886 kg to 772 kg. This significant weight reduction was achieved without compromising the frame's strength and safety requirements, with maximum stress and displacement values actually decreasing after optimization.*

***Article History:***
*Received: May 16, 2025*
*Revised: May 31, 2025*
*Accepted: June 02,2025*
*Published: June 09, 2025*

***Keywords:*** *frame structure, torsion, optimization*



## 1. INTRODUCTION

Buses serve as an essential mode of public transportation globally. However, traditional buses that primarily utilize non-renewable fuels contribute significantly to environmental issues through exhaust emissions and noise pollution. The growing concern for environmental sustainability has led to a strong interest in electric buses as a more environmentally friendly alternative, offering minimal or zero tailpipe emissions and reduced noise levels [1]. The amount of pollutant emission reduction varies according to the source of electricity, for example from solar power the reduction is about 80-90% [2].

This research addresses the critical need to evaluate the structural strength of a conventional bus chassis, specifically the NQR71, when undergoing conversion into an electric bus. Replacing engine components with electric motor systems, batteries, and other support systems requires careful structural validation to ensure the frame's strength and safety are maintained post-modification. In essence, the NQR71 served as the baseline conventional structure upon which the electric conversion was simulated and analyzed, fitting the research's purpose of studying this conversion process

Therefore, the primary objective of this study is to conduct a torsion strength analysis of the converted medium electric bus frame structure and perform a subsequent size optimization using the Finite Element Method (FEM) under static load conditions. The goal of the optimization is to identify the minimum attainable weight for the frame structure while ensuring it still meets the required torsion strength criteria. Results from the torsion analysis, specifically stress and displacement values, serve as essential constraints for this optimization process. Reducing the weight of a vehicle frame is a significant factor, potentially leading to decreased energy consumption and production costs while maintaining performance and safety standards [3], [4], [5], [6], [7]. FEM is widely utilized in vehicle analysis due to its effectiveness and cost-efficiency compared to physical prototyping [8–10].

## 2. METHODOLOGY

This research employed the Finite Element Method (FEM) to analyze and optimize the structure of a medium electric bus frame. FEM was chosen because it enables effective results using only digital models or drawings, avoiding the high costs associated with creating physical prototypes.

The methodology proceeded through several key stages:

- Initially, the 3D CAD geometry of the bus frame was imported into CAE software. This frame originated from a conventional NQR71 chassis modified for an electric powertrain. The frame structure is composed of six main parts: the left and right sides, front and rear sections, and the upper (roof) and lower (floor) frames [11]. The imported model for analysis contained 364058 elements and 359422 nodes.
- The material for the entire frame was defined as steel, Table 1. shows the materials properties.
- The total mass of the electric bus included structural components (like the frame, floor plate, batteries, motor) and non-structural components (such as passengers, seats, air conditioning, glass, etc.). The bus has capacity of 33 passengers each weighing 60 kg, were applied as mass distributions on the frame nodes. Table 2. shows the bus component.
- A torsion load simulation was performed to generate the stress conditions experienced by the frame when the bus is running and passing through uneven roads or potholes. This load incorporated the weight of structural components based on material density and the distributed weight of non-structural components. Supports were applied at three points, two on the rear axle and one on the front axle. The simulation produced results for deformation (displacement) and stress distribution.
- Following the torsion analysis, size optimization was carried out on the entire frame structure. The objective was to achieve the minimum mass without violating the strength. The design variables for this optimization were the thicknesses of the frame pipes, which are of rectangular box type. Four thickness values were considered as variables: 2 mm, 2.5 mm, 3 mm, and 4 mm.
- The constraints used in the optimization process were the stress and displacement values obtained from the initial torsion analysis. This ensured that the optimized, lighter frame would still meet the necessary strength and safety criteria. The optimization solver completed the process in three iterations to find the optimal result.

This methodology allowed the researchers to evaluate the structural performance under load and systematically reduce the frame's weight through thickness adjustments while maintaining crucial strength and safety requirements.

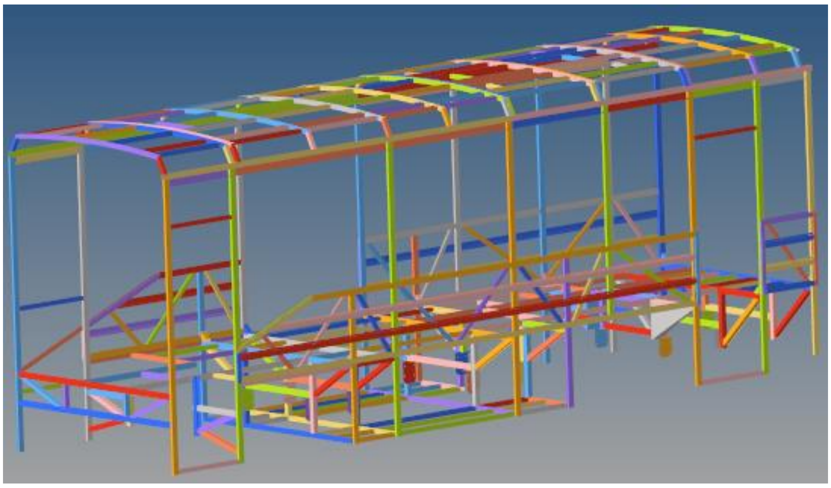

Figure 1. Bus Frame 3D Model

Table 1. Material Properties

| Properties | Value |
|---|---|
| Density (kg/m$^3$) | 7,850 |
| Poisson Ratio | 0.3 |
| Modulus Elasticity (GPa) | 210 |

Table 2. Bus Component

| Component | Mass (kg) |
|---|---|
| Frame structures | 886 |
| Seat and passangers | 2,350 |
| Battery | 475 |

| | |
|---|---|
| Motor | 475 |

## 3. RESULTS AND DISCUSSION

### 3.1 Torsion Load

Figure 2. shows the torque load condition refers to the torque function on the front axle when one wheel is free and one wheel on the other side is up, this condition is also called extreme torque. Torsional loading will produce a twisting effect on the frame structure, such as when a bus is passing over an uneven road or a pothole.

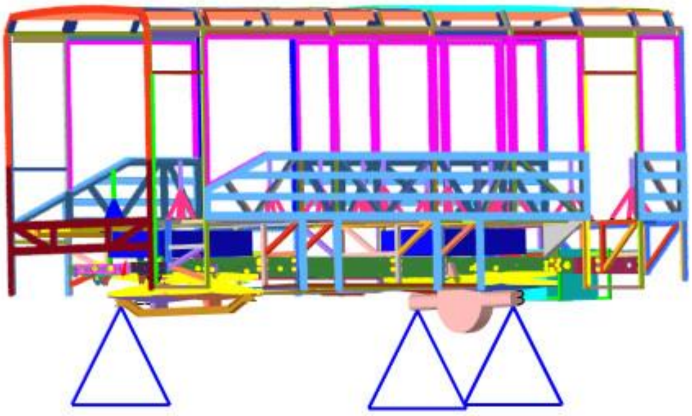

Figure 2. Torsion Load Condition

Torque simulation was carried out, the maximum displacement result on the roof frame was 19.755 mm. The largest displacement occurred at node 6229, namely the left front roof frame area, precisely in the area where the front door is attached. Figures 3. show the results torque displacement before optimization process. Figures 4. show the results of the torque simulation displacement after optimization process and the maximum displacement at nodes 6229, which is located in the upper door area.

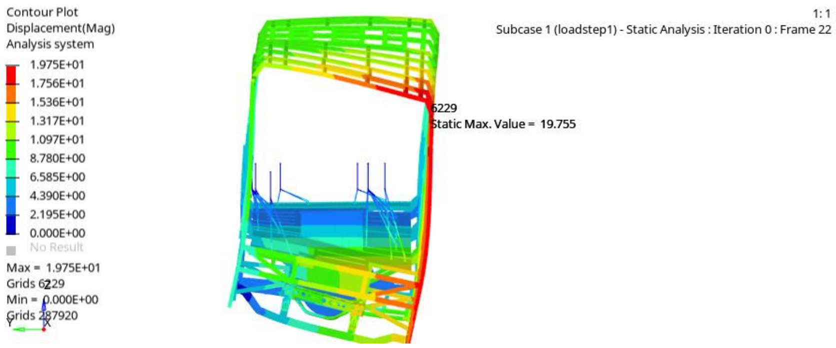


Figure 3. Displacement Results

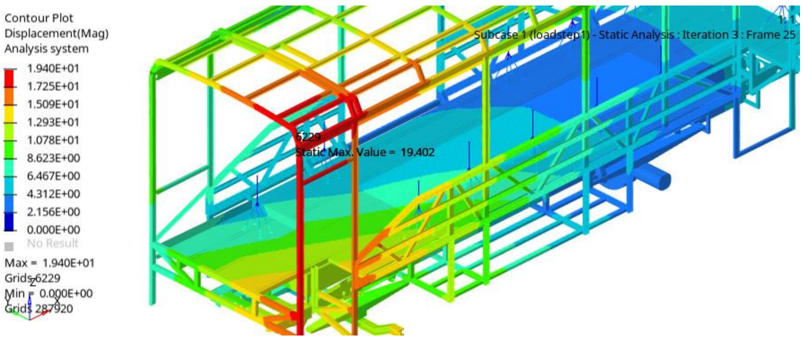


Figure 4. Maximum displacement at nodes 6229

Figure 5. shows the results of stress distribution. The maximum stress shows 210 MPa located on the floor frame, precisely in the front left battery area with node 856889. Figures 6. show the maximum stress that occur in the bus frame structure after optimization process done.

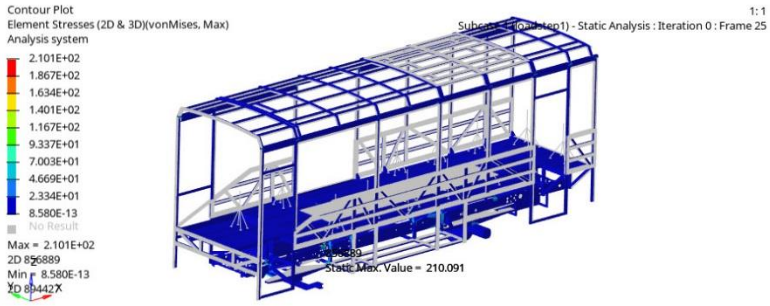


Figure 5. Stress Distribution

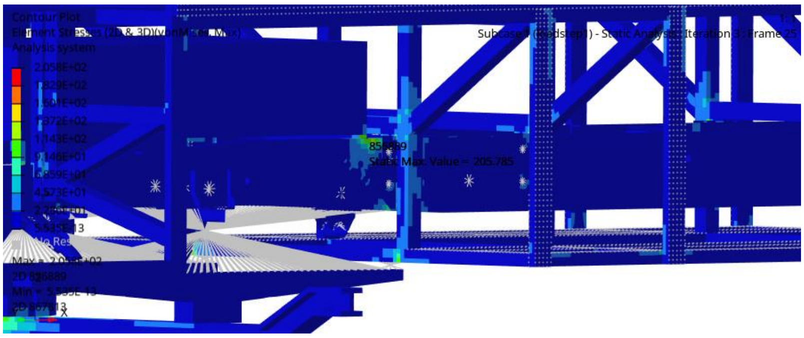

Figure 6. Maximum stress at nodes 856889

Figure 7. expresses factors of safety during optimization process. The minimum factors of safety located near rear battery at nodes 887703 with value 1.81.

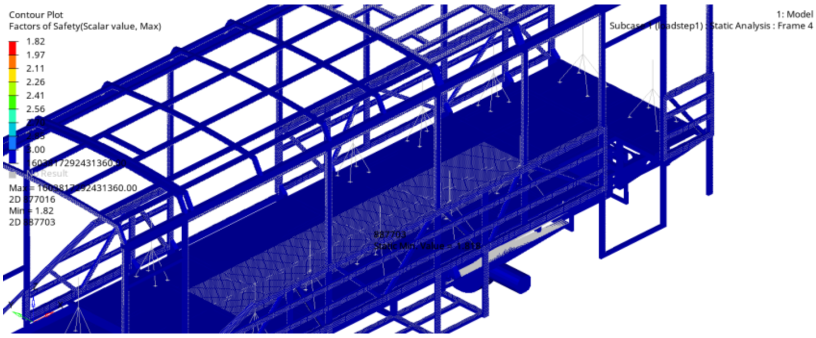


Figure 7. Factors of Safety

3.2 Size Optimization

Reducing the weight of the bus frame structure is proposed as a way to lower production expenses and boost energy efficiency without compromising the structural integrity. Vehicles that are lighter consume less energy to move. Furthermore, a lighter vehicle structure positively impacts its dynamic performance [12], [13]. In this specific study, steel remains the material used for the frame, and the optimization process is employed to decrease its weight.

Other researchers have frequently conducted vehicle optimization using constraints like stress, displacement, and torsional stiffness [1,7,14]. The particular optimization technique utilized in this research is size optimization. This optimization method was applied across the entire bus frame, encompassing the roof, floor, and both the right and left sides.

The primary goal of this optimization process is to achieve the lowest possible weight for the frame structure. This is pursued by making the pipes that form the frame thinner without diminishing the structure's strength. Four main design variables were involved in the optimization, table 3. shows the optimization setup. These variables correspond to the thicknesses of the pipes making up the frame structure components, which are characterized by different dimensions of hollow, rectangular box-type pipes.

The limitations or "optimization responses" serving as constraints during this optimization were displacement and stress. The target function to be minimized was the Total Mass (or Minimum Mass).

Table 3. Optimization Setup

| | |
|---|---|
| **Design Variables** | 2 mm |
| | 2.5 mm |
| | 3 mm |
| | 4 mm |
| | 2 mm |
| **Optimization Responses** | Displacement |
| | Stress |
| | Total Mass |
| **Objective Function** | Minimum Mass |

Based on the results of torsion load test, the outcome of size optimization process demonstrated a considerable reduction in the weight of the frame structure. The initial weight of the structure was 886 kg, and after successful optimization, it was reduced to 772 kg. This represents a weight reduction of 114 kg, equivalent to approximately 12% of the original weight.

This weight decrease was achieved by adjusting the design variables, specifically the pipe thicknesses. Table 4 in the source illustrates how the initial thicknesses were changed to smaller optimum thicknesses after the optimization.

Crucially, even after the optimization process, the frame structure continued to satisfy the strength and safety requirements. The optimization results show that the maximum displacement slightly decreased. Similarly, the maximum stress also decreased. These changes in displacement and stress values, remaining within safe limits, indicate that the strength and safety of the bus frame structure were maintained despite the reduction in weight.

Table 4. Design Variables Change

| **Beam** | **Inital (mm)** | **Optimum (mm)** |
|---|---|---|
| Beam 1 | 2 | 1.6 |
| Beam 2 | 2.5 | 2.1 |
| Beam 3 | 3 | 2.7 |
| Beam 4 | 4 | 2 |
| Total Mass | 886 kg | 772 kg |

The weight of the frame structure from 886 kg has been reduced to 772 kg and still meets the structural strength and safety requirements. Table 5. shows the changes in displacement and stress distribution after the optimization process. At the current optimization process, the solver performs three iterations and getting the optimal result.

Table 5. Optimization Results

| | **Initial** | **Optimum** |
|---|---|---|
| **Maximum displacement of torsion (mm)** | 19.75 | 19.40 |
| **Maximum stress of torsion (MPa)** | 210 | 205 |

As shown in Table 5, the displacement conditions and stress distribution in the bus frame structure decreased from 19.755 mm to 19.405 mm and stresses from 210 MPa to 205 MPa after the optimization process was carried out. This shows that the bus frame structure still meets the strength and safety requirements.

## 4. CONCLUSION

This study successfully demonstrated that optimization of the electric bus frame structure can be done without sacrificing strength and safety, and has the potential to increase the efficiency of electric vehicles. The results showed that the frame weight was successfully reduced by 12%, from 886 kg to 772 kg, which means a weight reduction of 114 kg. Most importantly, this weight reduction was achieved without compromising frame strength, minimum factors fo safety achieve with value 1.81. In fact, after the optimization process, the maximum displacement value decreased slightly to

19.40 mm and the maximum stress decreased to 205 MPa. Thus, this study successfully achieved its objectives, namely analyzing the strength of the electric bus frame (initial analysis results) and finding a lighter frame design (optimization results) while still meeting the specified strength and safety requirements.

Further studies could explore variations with bus conditions when turning, braking or other dynamic loads conditions. Optimization can be done using topology, gauge or free shape optimization.

## REFERENCES


1. Pravilonis T, Sokolovskij E, Kilikevičius A, Matijošius J, Kilikevičiene K. The usage of alternative materials to optimize bus frame structure. Symmetry (Basel). 2020 Jun 1;12(6).
2. Abbasi M. Study of Electric Buses and Their Impact on the Environment in Urban Networks Investigating the impact of battery electric buses on air pollution View project [Internet]. 2018. Available from: https://www.researchgate.net/publication/323336803
3. Wang Y, Hao C, Ge Y, Hao L, Tan J, Wang X, et al. Fuel consumption and emission performance from light-duty conventional/hybrid-electric vehicles over different cycles and real driving tests. Fuel. 2020 Oct 15;278.
4. Huang Y, Surawski NC, Organ B, Zhou JL, Tang OHH, Chan EFC. Fuel consumption and emissions performance under real driving: Comparison between hybrid and conventional vehicles. Science of the Total Environment. 2019 Apr 1;659:275–82.
5. Wu T, Han X, Zheng MM, Ou X, Sun H, Zhang X. Impact factors of the real-world fuel consumption rate of light duty vehicles in China. Energy. 2020 Jan 1;190.
6. Chong HS, Kwon S, Lim Y, Lee J. Real-world fuel consumption, gaseous pollutants, and CO2 emission of light-duty diesel vehicles. Sustain Cities Soc. 2020 Feb 1;53.
7. Sawulski J, Ławryńczuk M. Optimization of control strategy for a low fuel consumption vehicle engine. Inf Sci (N Y). 2019 Aug 1;493:192–216.
8. Arcieri E V., Baragetti S, Fustinoni M, Lanzini S, Papalia R. Study and modelling of the passenger safety devices of an electric vehicle by finite elements. In: Procedia Structural Integrity. Elsevier B.V.; 2018. p. 212–9.
9. Li S, Feng X. Study of structural optimization design on a certain vehicle body-in-white based on static performance and modal analysis. Mech Syst Signal Process. 2020 Jan 1;135.
10. Miao BR, Luo YX, Peng QM, Qiu YZ, Chen H, Yang ZK. Multidisciplinary design optimization of lightweight carbody for fatigue assessment. Mater Des. 2020 Sep 1;194.
11. Haryanto I, Raharjo A, Kurdi O, Haryadi GD, Santosa SP, Gunawan L. Optimization of Bus Body Frame Structure for Weight Minimizing with Constraint of Natural Frequency using Adaptive Single-Objective Method. Vol. 1, International Journal of Sustainable Transportation Technology. 2018.
12. Corviniensis AT, Vii ET, Belingardi G. Lighweight Design of Vehicle Body A Contribution Toward. 2014;
13. Reducing the Weight of a Bus Body Frame by 17% Without Compromise to Performance or Safety Success Story [Internet]. Available from: www.altairproductdesign.com
14. Yang Z, Deng B, Deng M, Sun G. A Study on Finite Element Analysis of Electric Bus Frame for Lightweight Design. Available from: https://doi.org/10.1051/matecconf/2018175