Engineering Innovations
ISSN: 2813-1002, Vol. 12, pp 41-48
doi:10.4028/p-j2BDms

Submitted: 2023-10-24
Revised: 2024-05-13
Accepted: 2024-05-13
Online: 2025-01-03

Weight Optimization of the Electric Bus Carbody Structure

Wijianto[1,a], Giri Fahrezy Istighfarrullah[1,b*], Agus Dwi Anggono[1,c] and Taurista Perdana Syawitri[1,d]

[1]Department of Mechanical Engineering, Faculty of Engineering, Muhammadiyah University of Surakarta, Indonesia

[a]wijianto@ums.ac.id, [b]girifahrezy28@gmail.com, [c]ada126@ums.ac.id, [d]tps123@ums.ac.id

Keywords: Car body; CAD; Design; Electric Vehicle.

Abstract. Car body is a lightweight construction unit in the form of welded steel construction, has high strength and rigidity against dynamic loading without a fixed change in shape. In this study, the design of the car body bus frame consisted of roof, mascara, side wall, end wall, front part, and underframe. All of these components are designed using Autodesk Inventor Professional 2020 software. The frame material used is SS201. For data retrieval purposes, weight calculations are used in Autodesk Inventor. In the study, we succeeded in making the car body design according to regulations, measuring the weight of the car body, and optimizing the car body. The results showed that with a little optimization on the side wall frame, the weight of the car body bus can be reduced by 11.7% from the initial weight of 461.5 kg to 407.4 kg without changing its characteristics.

Introduction

This mode of transportation is largely responsible for the consumption of a large portion of the world's oil reserves over the last few centuries. This sector also makes a large contribution to total greenhouse gas emissions. Most of it is caused by road transportation based on gas-fueled motorbikes. Today, the concept of alternative drivers is becoming increasingly important as a result of climate change and increasing global resource crises. Due to advances in technology, greater attention is now being paid to the concept of battery-powered electric propulsion. In addition, electric vehicles are also believed to be an effective solution for reducing greenhouse gas emissions [1]. The Merah Putih Electric Bus produced by PT Industri Kereta Api (Persero) is one of the superior products of domestic work in terms of electricity-based public transportation. One of the important components of the electric bus is the Car body. Car body is a passenger-filled space structure built with a plate-like frame construction and as an important component in transportation facilities. In accordance with its function, to maintain the safety and comfort of passengers during the journey to stop at their destination. The car body is a monocoque construction unit that integrates with the base frame. The main body of the car body consists of the underframe, flooring, roof, side wall, mascara and end wall [2]. As is the case in the industrial world, vehicles themselves have been regulated in Presidential Regulation no. 55 of 2012 which aims to regulate the technical specifications of the dimensions, engine and carrying capacity of motorized vehicles [3]. However, the main drawback of electric buses lies in their battery life. The battery must provide sufficient power to run the vehicle. The heavier the vehicle, the more power required. Thus, weight reduction must be made to increase battery life, especially in reducing the weight in the car body frame sector [4]. This research is concerned with optimizing and analyzing the bus structure using the Finite Element Method (FEM) under static conditions using ANSYS software [5]. In this study, steel pipe elements are widely used to make bus body frame structures because they cannot only maintain accurate rigidity but also reduce production costs a lot. Bus body geometry is obtained directly from three-dimensional Computer-Aided Design (CAD) files using Autodesk Inventor. After that, weight optimization was carried out on the frame structure of the car body to determine the minimum frame bus weight [6]. The design variable is the position of the steel pipe in the frame structure on the side wall, front frame and end wall.

Literature Review

In the previous study, Haryanto et al, conducted a study entitled "Optimization of Bus Body Frame Structure for Weight Minimizing with Constraint of Natural Frequency using Adaptive Single-Objective Method". This study discusses the analysis and optimization of the bus frame structure using the Adaptive Single Objective method for the optimization method. Optimization of the bus body frame is carried out using size optimization with variable thickness and with the aim of minimizing the mass of the structure. The software used is ANSYS R18. The material used as the frame of this research bus is Steel. The results showed that by using Adaptive Single Objective method the structural weight of the bus frame can be reduced by up to 8.4% without a significant effect on the change in the natural frequency of the first 6 orders. This reduction in structural weight is essential for possible improvements in bus design to achieve lower production costs and higher energy efficiency [4].

In research on optimizing the bus frame structure Nandhakumar et al, conducted a study entitled "Weight Optimization and Structural Analysis of An Electric Bus Chassis Frame". In this study Nandhakumar et al. Optimize bus chassis weight using the finite element method. The software used is ANSYS Mechanical 20.0. The materials used as the frame of this research bus are aluminium 6061-T6 and aluminium 7075-T6. In this study, the results were obtained in the form of successfully optimizing the weight of the electric bus frame by replacing the material from conventional steel to aluminium alloy. The two types of aluminium alloys selected are aluminium 6061-T6 and aluminium 7075-T6. The results of the analysis show that the optimized electric bus frame using aluminium alloy 7075-T6 has a lighter weight compared to the electric bus frame using conventional steel. In addition, the analysis results also show that the optimized electric bus frame has lower voltage and deformation compared to the electric bus frame that uses conventional steel [7].

The next research analyzed the monocoque type chassis conducted by Wikarta and Yudha with the title “Strength Analysis of Solar Electric Bus Chassis Using Finite Element Method”. The purpose of this research is to determine the strength of the frame due to vertical loads, acceleration, braking, and turning conditions. The parameters are Von-Mises stress and safety factor. The chassis is considered safe if its safety factor is more than 1.5. The results show that the chassis is able to withstand vertical loads and acceleration, but is unsafe when subjected to braking and turning. A redesign of the chassis is required to strengthen its structure [8].

Model Car Body Bus

Car body is a lightweight construction unit in the form of welded steel construction, has high strength and rigidity against dynamic loading without permanent deformation. Car body construction can be either fused construction (monocoque or semi-monocoque) or not fused with the base frame (platform)· Car body consists of Roof, Mascara, Side Wall, End Wall, Front Part, and Underframe [9].

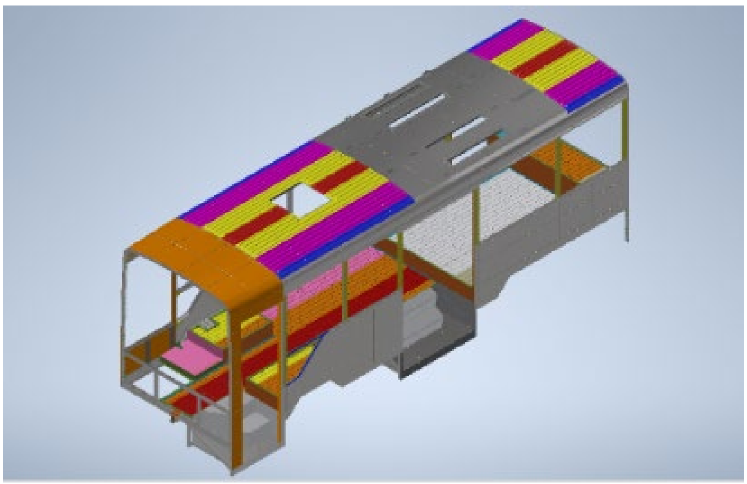

Fig. 1. Car body bus Merah Putih image

Table 1. Car body bus specifications

Dimensions		
Length	mm	7.900
Wide	mm	2.100
High	mm	3.200
Capacity	seats	19 + 1
Frame		
Material	SS201	
Structure	*Semi-monocoque*	

Figure 1 shows the car body bus used in this study, with specifications that can be seen in Table 1. The bus has a passenger capacity of 19 seats accompanied by 1 special seat for the disabled that can be folded.

Material Car Body Bus

In this study, the material used for *the* car body bus was SS201. The material properties of SS201 can be seen in Table 2 [10].

Table 2. Material property SS201

Material Properties SUS 201		
Properties	**Value**	**Dimension**
Density	7.86	g/cm^3
Tensile Yield Strength	292	MPa
Tensile Ultimate Strength	685	MPa
Young's Modulus	193-200	GPa
Poisson's Ratio	0.27	

Loading Analysis Simulation

Simulation The loading analysis in this study was carried out to determine the amount of voltage distribution, maximum voltage value, total deformation, and safety factor of the bus car body under static conditions. This research uses finite element analysis (FEA) software ANSYS Workbench 2021 R2 [11,12].

Convergence Test

Convergence tests are carried out to ensure that the tests carried out are in a convergent condition. The size variations ranging from 15 mm to 11 mm for convergence tests. The output of the convergence test is the value of Equivalent-stress, total deformation, safety factor, number of elements, and skewness.

Loading on the Car Body

In testing the strength and safety of a vehicle can be done through two methods. The first method is experimental, but this method is considered less effective and efficient because it requires a long time and costs money, while the second method is simulation. In the simulation method, the car body is subjected to static loading which represents the actual loading. Loading on the car body using vertical loading. Vertical loading is shown in Figure 2.

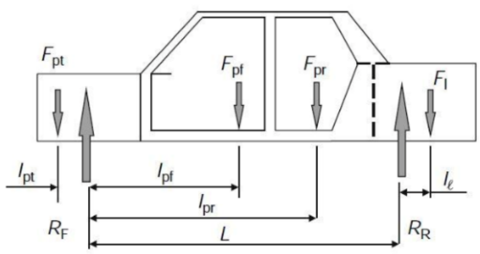

Fig. 2. Vertical load distribution

In Figure 2, it shows the vertical load distribution on the car with the *wheelbase* (L) and the distance of each load to the front wheels. Therefore, the derivative of the equation is obtained as follows:

$$R_F = \frac{F_{pt}(L-l_{pt})+F_{pf}(L-l_{pf})+F_{pr}(L-l_{pr})-F_l l_l}{L} \quad (1)$$

$$R_R = \frac{F_{pf}l_{pf}+F_{pr}l_{pr}+F_l(L-l_l)-F_{pt}l_{pt}}{L} \quad (2)$$

Where:
Fpt = Force caused by the power train;
Fpf = Force caused by the front passenger;
Fpr = Force caused by rear passenger;
Fl = Force caused by items in the baggage;
ll = Distance of load *Fl* to the rear wheel;
lpf = Distance of load *Fpf* to front wheel;
lpr = Distance of load *Fpr* to front wheel;
lpt = Distance of load *Fpt* to front wheel;
L = Wheelbase;
RF = Reaction force by the front wheel;
RR = Reaction force by rear wheel.

Safety Factor

Safety factors are used because of the uncertainty of a load in real conditions so correction factors are needed to ensure that mechanical components are safe in real conditions. The use of safety factors aims to guarantee safety from a mechanical problem. The calculation of the safety factor is as follows:

$$SF = \frac{\sigma_y}{\sigma} \quad (3)$$

Where:
SF = safety factor;
σ_y = yield strength (N/mm^2);
σ = voltage (N/mm^2).

Weight Optimization

In this study, optimization of the car body bus structure uses optimization of shape and size, where the position of the steel pipe in the frame structure on the side wall of the car body bus becomes a parameter in optimization. With maximum voltage limitation so that it gets the lightest structure.

Optimization Process for Carbody Structure

In this study, the optimization problem was simplified by considering design variables with iron pipe structures that make up the side wall frame. Iron pipes are used to form the frame structure of side walls, namely U iron pipes. Figure 3 and Figure 4 show the results of optimization of the side wall frame in this research. Optimization is used to reduce the mass of side wall structures in each group.

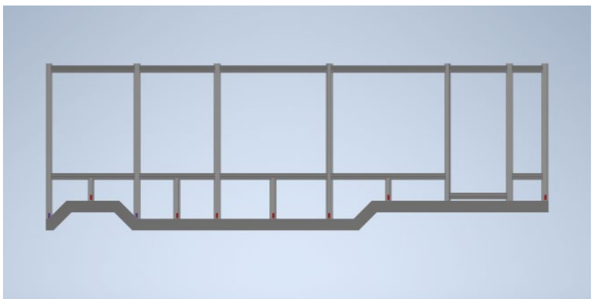

Fig. 3. Right wall structure after modified

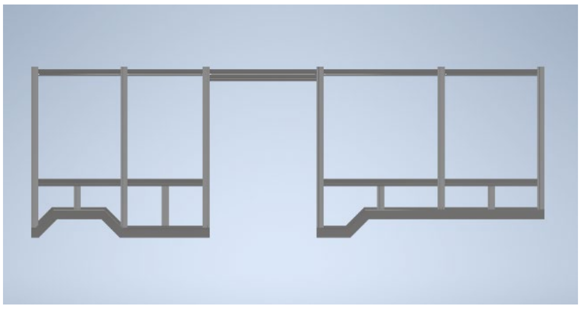

Fig. 4. Left wall structure after modified

Convergence Test Output

The convergence test is used to confirm and prove that the car body bus analysis data taken from changing element dimensions is already in a convergent state, with element variations ranging in size from 15 mm to 11 mm.

The output taken is in the form of Equivalent-stress value, total deformation, safety factor, number of elements, and skewness. The output of the convergence test can be seen in Table 3.

Based on the results of the convergence test, the resulting skewness value is 0.06 and the value is classified as very good. The value of the equivalent stress and resulting deformation forms a graph that tends to be linear with an error of no more than 5%. The convergence analysis graph is shown in Figure 5 and Figure 6.

Table 3. Convergence test output

Element	***Equivalent Stress*** **(MPa)**	**Total Deformation (mm)**	**Element**	**Skewness Average**
15	126.4	2.472	433093	0.062
14	139.44	2.455	495287	0.063
13	138.76	2.478	654325	0.061
12	123.56	2.497	687656	0.05
11	141.04	2.492	781952	0.043

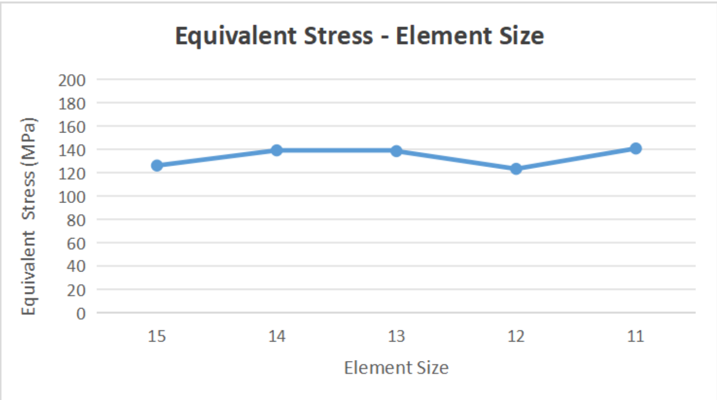

Fig. 5. Equivalent-stress graph with Element size,

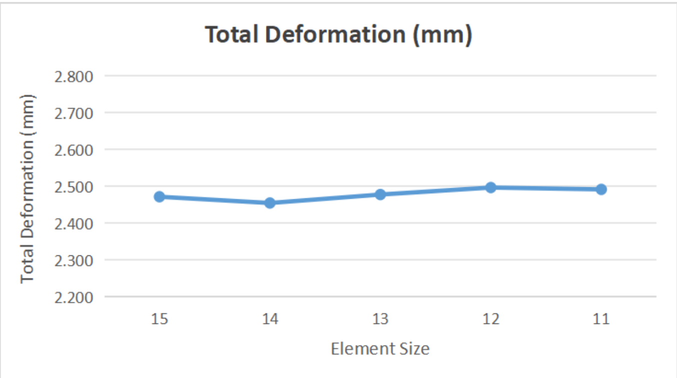

Fig. 6. Total deformation graph with Element size

Based on the convergence test results, the resulting skewness value is 0.06 and the value is classified as very good. The value of the equivalent stress and deformation produced forms a graph that tends to be linear with an error of no more than 5%. This shows that the meshing flow has been well formed so that it can flow the loading evenly. The meshing results produced are classified as convergent, so it is feasible to do the next simulation.

Finite Element Analysis Output

Equivalent-stress and deformation are taken as design constraints. FEA results obtained for the carbody frame made of U iron pipe with SS201 material and have been optimized on the iron pipe on the side wall frame, front frame and end wall. The results of FEA can be seen in Figure 7 until Figure 10 and have been summarized in Table 4.

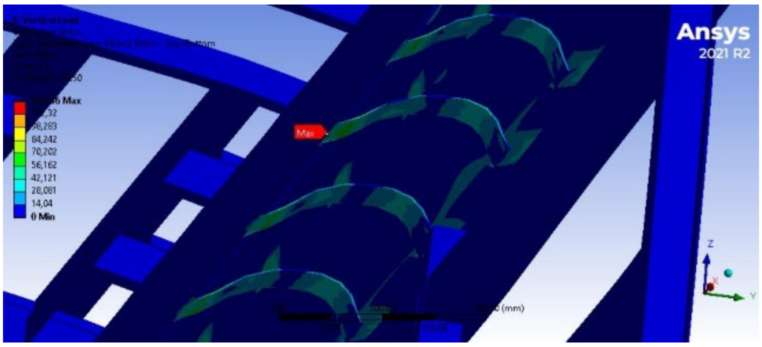

Fig. 7. Equivalent stress

Based on the results of the loading simulation that can be seen in Figure 6, there is a maximum force distribution on the roof of 126.36 MPa. This value is still relatively safe because it is far below the yield strength of SS 201 material, which has a strength of 292 MPa. This shows that it is certain that it did not happen at that point. Meanwhile, the underframe has the least force distribution.

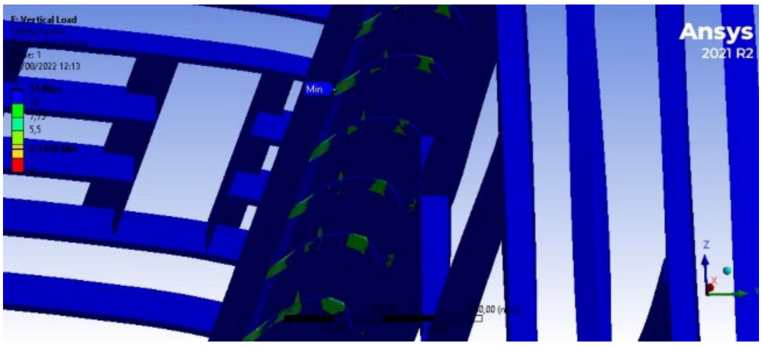

Fig. 8. Safety factor

The resulting safety factor value is 2.31, which is obtained from the comparison of force distribution with material yield strength. This shows that the component can still accept forces up to 2.31 times before failing or breaking. The resulting safety factor value is still relatively safe because the standard used for bus vehicles, the safety factor must be above 1.5.

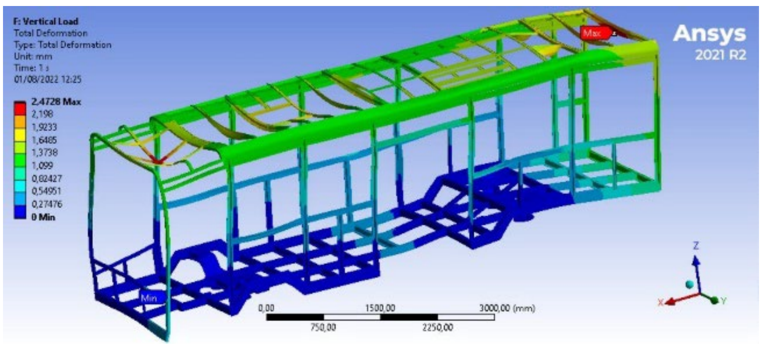

Fig. 9. Total Deformation

Based on the results of vertical loading, the area that experienced the greatest total deformation was located at the back of the roof, which was rated at 2.4728 mm. While the front underframe experiences the smallest deformation compared to other parts.

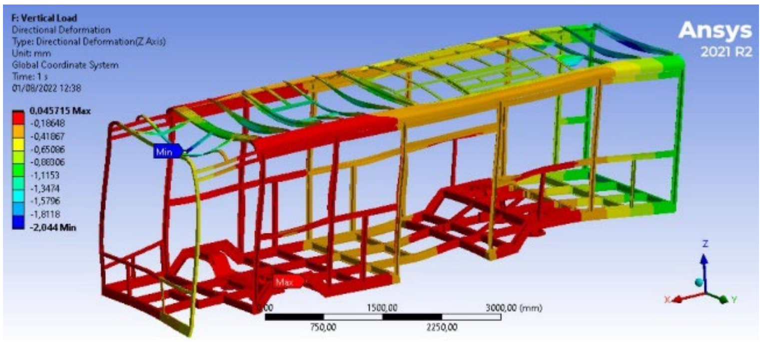

Fig. 10. Directional deformation

Directional deformation in the direction of the Z axis due to vertical loading in the direction of the Z axis (-). The results of Z-axis unidirectional deformation showed the highest deformation towards Z (-) at the top of the bus mascara, which was 2.044 mm, besides that there was also Z-direction deformation (+) in the chassis underframe located in the air suspension mounting support area, amounting to 0.04 mm.

Table 4. FEA output

Vertical Loading	
Material	SS201
Distribution Force	126.36 MPa
Total Deformation	2.472 mm
Directional Deformation	2.044 mm
Safety Factor	2.31

Results

The results of carbody structure optimization use shape and size optimization to determine the minimum structural weight by changing the location and shape of the pipe in each group mentioned in Figures 3 and 4. Structural optimization is proven to reduce the weight of the carbody bus frame. For comparison of initial and optimized conditions on the carbody bus structure can be seen in Table-5.

The results showed that by using structural optimization (shape, size) the weight of the carbody bus frame structure can be reduced by 3% from the initial weight of 548.1 kg to 487.27 kg without significant influence. This reduction in structural weight is critical to enable bus designs to achieve lower production costs and also higher energy efficiency.

Table 5. Initial and optimum conditions

Component	**Mass (kg)**		**Reduction (kg)**
	Initial	**Optimum**	
Left wall frame	177.1	152.79	-24.31
Right wall frame	177.1	169.61	-7.49
Left wall panel	53.65	39	-14.65
Right wall panel	53.65	46	-7.65
Total Mass	461.5	407.4	-54.1

Conclusion

The conclusion of the static analysis of the semi-monocoque chassis structure of the electric bus with vertical loading, the loading analysis shows that the semi-monocoque chassis experiences a force distribution of 126.36 MPa, a total deformation of 2.472 mm, the largest Z direction deformation at 2.044 mm, and a safety factor of 2.31. Optimization of the bus body frame is carried out by optimizing the structure (shape and size) with variable steel pipe positions in the frame structure at the side wall, front frame and end wall with the aim of minimizing the mass of the bus body structure. By optimizing these variables, it is proven that the mass of the bus body frame can be reduced by 11.7% without having a significant effect on the character of the frame.

Acknowledgements

Special thanks to The Directorate General of Higher Education, Research and Technology, Ministry of Education, Culture, Research and Technology, Republic of Indonesia for the scholarships contributed in this study.

References

[1] MR Cagney, Electric Bus Technology, Transport Research Report, Australia, 2017.

[2] Nguyen, T. C., Research on Application of 3D Parametric Software for Design and Shaping of Bus Frame Structure, Applied Mechanics and Materials, 894 (2019) 96–103.

[3] President of the Republic of Indonesia, Peraturan Pemerintah Republik Indonesia Nomor 55 Tahun 2012 Tentang Kendaraan, State Institution of the Republic of Indonesia, 2012.

[4] Haryanto, I., Raharjo, F. A., Kurdi, O., Haryadi, G. D., and Santosa, S. P., Optimization of Bus Body Frame Structure for Weight Minimizing with Constraint of Natural Frequency using Adaptive Single-Objective Method, International Journal of Sustainable Transportation Technology, 1 (2018) 9–14.

[5] Chen, X., and Liu, Y., Finite Element Modeling and Simulation with ANSYS Workbench, CRC Press, Taylor & Francis Group, 2015.

[6] Wang, D., Xie, C., Liu, Y., Xu, W., and Chen, Q., Multi-objective Collaborative Optimization for the Lightweight Design of an Electric Bus Body Frame, Automotive Innovation, vol. 3, no. 3, (2020) 250–259.

[7] Nandhakumar, S., Seenivasan, S., Saalih, A. M., and Saifudheen, M., Weight Optimization and Structural Analysis of an Electric Bus Chassis Frame, Mater Today Proc, vol. 37, (2021) 1824–1827.

[8] Wirakarta, A., and Yudha, Y. A., Stress Analysis og Solar Electric Bus Chassis Using Finite Element Method, Mechanical Engineering, Institute of Technology Sepuluh November, 2020.

[9] Yao, C., and Yang, M., Research on Lightweight Design and Finite Element Analysis of a 9 Meter Bus Body Frame, J Phys Conf Ser, vol. 1748, (2021)

[10] Callister, Jr. W. D., and Rethwisch, D. G., Materials Science and Engineering, Tenth ed, Wiley, Hoboken, New Jersey, 2018.

[11] Haldankar, M., and Shirahatti, A. M., Finite Element Analysis and Optimization of Commercial Bus Body Structure, International Journal of Engineering and Technical Research, vol. 2 (2014) 175–178.

[12] Chintada, V., Seela, C. R, and Matta, V. R., Modal and Static Analysis of Automotive Chassis Frame by Using FEA, International Journal of Applied Engineering Research, vol. 10 (2015) 19775–19777.