GSJ: Volume 10, Issue 6, June 2022, Online: ISSN 2320-9186

www.globalscientificjournal.com

Numerical Investigation of Dynamic Analysis of Bus Structure

Tewodros Gamea [a]*, Albetel Melsew [b]

[a]*Automotive engineering department, AASTU, Adiss Ababa 16417, Ethiopia*

[b]*Automotive Engineering Department, Gondar University 196, Ethiopia*

[a]*Email:* tewodros.game@aastu.edu.et.

[b] *Email:elbetel.Melsew@uog.edu.et*

Abstract

This research presents the numerical investigation of the bus body frame structure during braking and cornering loads. The study is very important to fill the gap in the area of vehicle structure response due to maneuverability. Here it is aimed to optimize the weight of bus body structure by evaluating its stress and deformation. The study worked at Bishoftu bus model ZK6118HGA, modelled with solid work 18 and analysis has been done with ANSYS 19.0. Braking, left cornering, and right cornering test has been performed. Optimization has done by replacing overdesigned RHS (rectangular hollow section) with lower sized RHS. The improved structure also investigated with the cornering and braking tests. The weight of the improved structure is significantly lower than the original structure. The study revealed that the weight of the bus body structure reduced by 93.58 kg. Which is about 3.07% of the whole structure weight.

Keywords: Braking, deformation, left cornering, optimization right cornering, steel structure, stress.

1. Introduction

Over the last decades, the transport sectors are accountable for the consumption of a large percentage of oil reserves worldwide[1]. Moreover, it accounts for a large contribution to total greenhouse gas emissions. For energy reduction and environmental protection application of lightweight methods in the automotive industry is helpful. A small reduction in overall weight will result in huge operational cost savings. For conventional combustion vehicles, it is stated that 6-8% of fuel consumption and 4% of emission could be saved for every 10% mass reduction[2]. The current trend in bus design is to reduce weight to minimize fuel consumption. Optimization techniques applied for structural design are a major research area to reduce the overall weight of vehicles without compromising the safety of the passengers[3]. Design experts usually try to maintain all required structural parts and elements, which may increase the production cost and the product price. Moreover, the increase in weight leads to weight increase and thereby an increase in fuel consumption of the bus[4]. The structural analysis involves the determination of the force and displacements of the structures or components of a structure and assessing its stability and safety. Further braking and cornering boundary condition applied for simulating response of frame structure. The dynamic analysis of the braking system is defined from the dynamic boundary condition and it is almost similar to the static distribution of the static weight and acceleration. The dynamic behaviour of structure during braking is the analysis of the longitudinal dynamics of the bus. The main part of the cornering load lies on the side frame while the car turns left most of the mass shifts to the right side in a lateral direction since a portion of the load is applied on the right side of the frame lateral force and self-aligning torque transient response has been studied. The general objective of this research is to analyze the dynamic behaviour of bus structure during braking and cornering.

Different scholars used a different methodology to achieve a weight-reduced structure. Generally, two classes of methods are used such as substitute the sub structure with completely different material and with the same material but reduced size. The composite material was the main constituent for the replacement of the frame structure and chassis of the bus. To reduce the weight of the chassis structure, lightweight materials (carbon fiber composite) were used to replace the full steel chassis to be composite-steel chassis[5]–[8]. An existing vehicle chassis frame of a Mahindra Bolero vehicle is taken for chassis frame with three different composite materials namely, Carbon/Epoxy, E-glass/Epoxy, and S-glass /Epoxy subjected to the same pressure as that of a steel chassis frame. Uniquely AISI 4130 Chromyl steel for the frame design[8], [9].

Various literature obtained different results on deformation and stress. Lan et al. found that stress levels lower than 10 Mpa on the sides, the roof, the front, and rear pans. Besides, it is obtained that torsional rigidity does not change significantly after the removal of some of the supporting members between longitudinal beams. Chinta and Vengopal [38] concluded that better results are obtained by reducing the thickness and also using composite materials than the original model and conventional steel. Foucault et al. [10] verified experimental test and bodyweight reduction was found to be 5.7%. Zhang et al. [11] find out the optimized frame bending, the full-load braking condition, and the full-load torsional operating condition stresses decreased by 44.499%, 23.364%, and 31.303%, respectively. The bending stiffness of the improved structure is higher compared to the original structure. Sunderji et al. [12] concluded that hybrid bolted composite-steel chassis structure, the total weight of the chassis reduces by about 22.7 % relative to the complete steel chassis structure, thus it could be presumed to prolong the mileage of electric vehicles by an additional 20 %. Mohd Khalid Ahmed et al. [13] stated that, for the similar load carrying capacity Composite material with heavier vehicle chassis, the weight reduced with a range between 73%-40%. Natural frequencies of polymeric composite heavy vehicle chassis are 32% - 54% higher than steel chassis and 66 -78% stiffer than steel chassis. In the structural analysis the Stress. Filho et al.[5] concluded that the optimized structure presented an increase of 75% in torsional stiffness, an increase of the height of the center of gravity of only 30mm, an increase of 6% in the total mass.

Some kinds of literature presented a procedure in numerical simulation for improving the structural features of a bus frame with a synchronized decrease in the mass by multi material optimization whereas added with sensitivity and robustness analysis. Topology optimization was done by taking torsional, bending stiffness, and frequency as design constraints to design identified space of the bus. In addition, design optimization is conducted for a newly developed aluminum-steel electric bus body structure[14], [15].

Most optimization techniques resulted in a positive economic impact without losing rigidity and stiffness of the system structure. The body structure is partitioned into seven components based on stress distribution, load path, and deformation characteristics of the benchmark model under prescribed load cases. A parametric study of the thickness variations of the cores and faces of the sandwich structures reveals the levels of significance of changing each design parameter on the specific stiffness's, specific frequencies, and specific side deformation of the bus. Reduced size has been achieved with a different optimization technique.

The literature reviewed shows that the study on the frame structure of the bus body studied under both static and dynamic analysis through structural and modal analysis as discussed before. The static structural analysis of the bus frame mainly applied the static load of the bus including the self-weight of the bus body. In most studies, the authors concentrate only based on bump cases in addition to static structural to obtain the dynamic response of the structure so that torsional load is studied. The main dynamic loads such as cornering and braking have not been addressed properly. The dynamic analysis was considering the boundary condition from maneuvers such as braking and cornering. Static distribution of static weights and acceleration, braking, cornering, displacement direction magnitude was observed during the dynamic analysis of bus structure.

2. Material and methods

The modeling of the bus structure is made with data collected from Bishoftu Automotive Engineering Industry bus bodybuilding section. Before measuring the structure, a drawing of the bus structure is made. This helped for easy placement of the measured length on the corresponding members on the drawing. During the data collection of the structure, all members of the structure are measured with a length measuring tape. The modelling of the bus structure is made with ANSYS 2019(ANSYS 19.0). The bus structure is made with steel beams of rectangular hollow sections (RHS), of different sizes. The sizes of RHS used are for example RHS 80×40×3, RHS 50×30×2, and 40×40×2. The bus body is created by 3-D wireframe modeling as shown the dimension is 2.55-meter height, 11.89-meter length, and 3.27-meter width the total weight is 3046 kg. There are 328 elements mostly are T-joints. The assembled model can be divided into six frames; driver side, passenger's side, floor side, roof side, rear side, and front side. The material type used is the DIN standard of St 37.2 with a variety of shapes.

GSJ: Volume 10, Issue 6, June 2022
ISSN 2320-9186

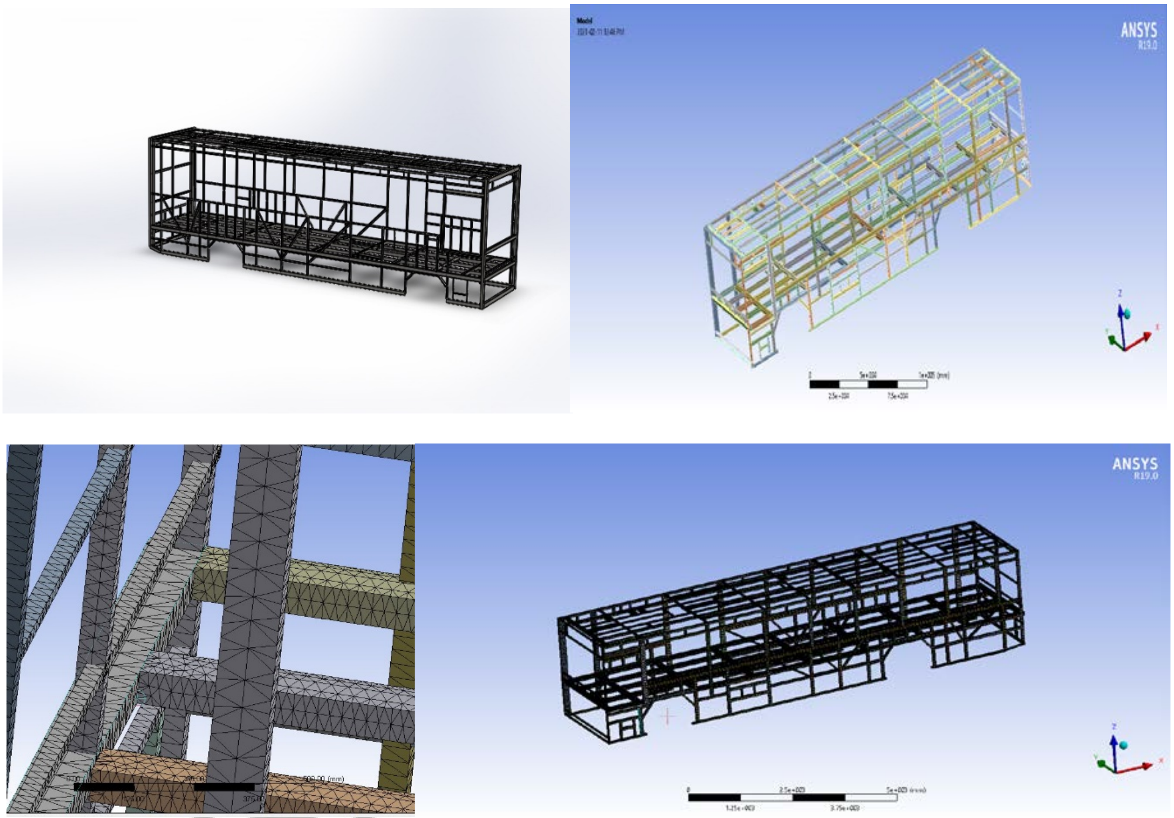


Figure 1: - solid work model, model imported to ANSYS and Meshing.

Finite element modelling made with ANSYS 19 Workbench. The structure modelling is by SOLIDWORK software exported as SAT. Format and is imported to ANSYS 19.0 Workbench. Finite element meshing is made with ANSYS 19 workbench. Meshing is an integral part of the computer-aided engineering (CAE) simulation process. The mesh influences the accuracy, convergence, and speed of the solution. Furthermore, the time it takes to create a mesh model is often a significant portion of the time it takes to get results from a CAE solution. Tetrahedral mesh elements are used in the meshing of the bus structure.

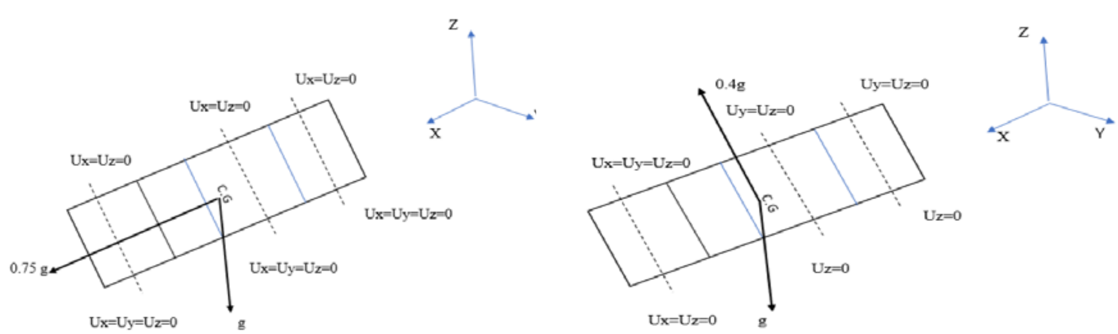


Figure: Boundary condition for braking and left cornering

3 Analysis Result

load result

The boundary condition for the case of braking load - both front and rear wheels (left and right wheel) must be constrained in three main directions at the nodes, and the offsets in the x, y, and z directions are specified as explained in figure 4.12. By applying these boundary conditions to the finite element model, the stresses and displacements on critical structural elements were calculated from the ANSYS 19.0 desktop. In the case of a braking load, the structure experiences both bending loads and torsional rigidity. Figure 4.1 shows the normal stress distribution in the frame structure of a bus during braking, with critical stress occurring at the front of the body due to the shift in vehicle weight. The maximum normal stress is 49.545 MPa in section 160, and the minimum normal stress is 61.95 MPa in section 68. The elements were subjected to both compressive and tensile stresses from -61.95 MPa to 49.545 MPa.

Figure 4.2 shows the equivalent stress on the front part of the body frame has more value than the remaining part, the maximum equivalent stress is 115.99Mpa at part 160 and the minimum equivalent stress occurs at part 90. The maximum equivalent stress that occurred at part 160 (115.99Mpa) is less than the maximum allowable stress of steel structure which is 230Mpa. The rare side of the bus body experience the least von misses stress. The braking effect also showed on the deformation counter plot obtained from the analysis workbench tool, the bus structure frontal area got a high displacement value. Part 170 has the total deformation value of 2.3961mm and the minimum total deformation creates at part 90 as shown in figure 4.3.

Figure 4.4 shows the distribution of directional deformation along the x-axis, the maximum directional deformation is 0.45183mm and 0.27303mm along the positive x-axis and negative x-axis respectively. The negative sign in displacement shows the direction of deformation.

Figure 4.5 shows the shear stress distribution of the XY plane, the minimum shear stress is -11.614Mpa at part 137 and the maximum shear stress is 26.376Mpa at part 160.

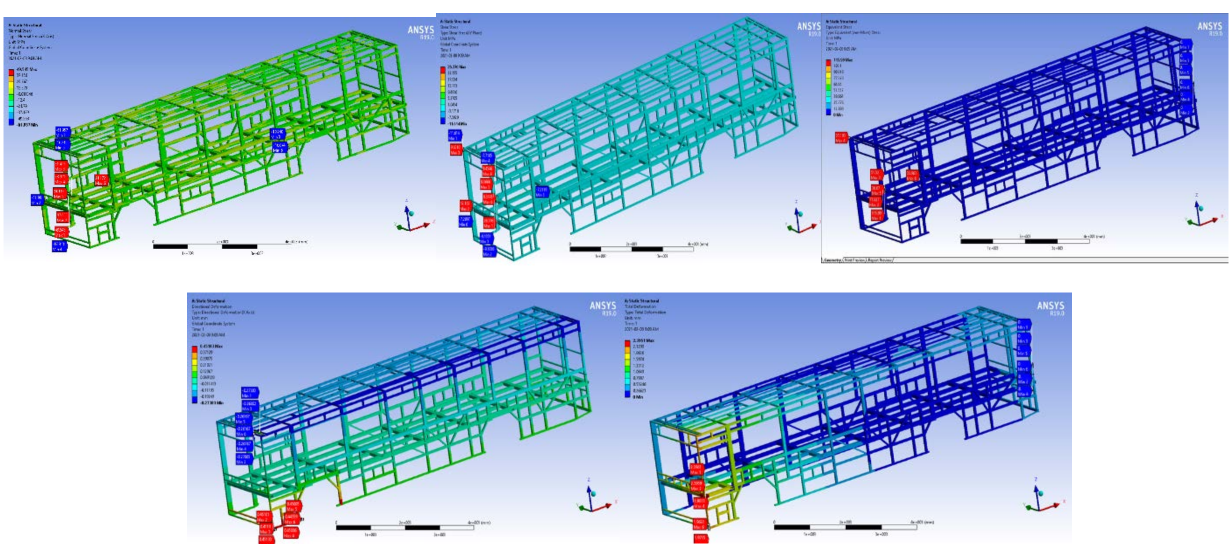


Figure: Braking load result

The boundary condition for the right cornering load case is both front and the rear wheels (left and right side wheel) are to be constrained in the three principal directions at the nodes and the displacements in x, y, and z directions are set as explained at the figure. Normal stress distribution during right cornering clearly shows the critical stress values that occurred at the junction point between the side frame and the bed structure. It is due to weight transformation from the outer wheel to the inner wheel. Figure 4.6 shows normal stress along the x-axis, the sidewall structure experience the maximum tensile stress of 91.586Mpa at part 26 and the minimum compressive stress of – 46.006Mpa at part 306.

Figure 4.7 shows von misses stress of bus structure during the right cornering. The total effect of stress on the bus structure is mainly expressed by the equivalent stress of the structure. On the contour plot, the maximum equivalent stress is 139.46Mpa, which is less than the allowable stress of a steel structure. The minimum stress 0Mpa at part 306 and the maximum equivalent stress occurred at part 313.

Figure 4.8 shows the total deformation of the structure, the maximum total deformation occurs at the part by which minimum equivalent stress is obtained and the minimum total deformation occurs on the part that the maximum equivalent stress recorded. The maximum total deformation is 38.123mm and the minimum total deformation is 0mm.

The negative sign in displacement shows the direction of deformation. Figure 4.9 shows the distribution of directional deformation along x-axis, the maximum directional deformation is 0.5932 mm (part146) along positive x-axis and - 0 .6371mm (part326) along Negative x-axis.

Figure 4.10 shows the shear stress of the structure, the maximum shear stress is 26.387 Mpa occurs at part 306 and the minimum shear stress is -43.988Mpa recorded at part 6. The body structure can stand the maximum shear load along the XY plane. The shear stress is directly related to the torsional rigidity of the structure.

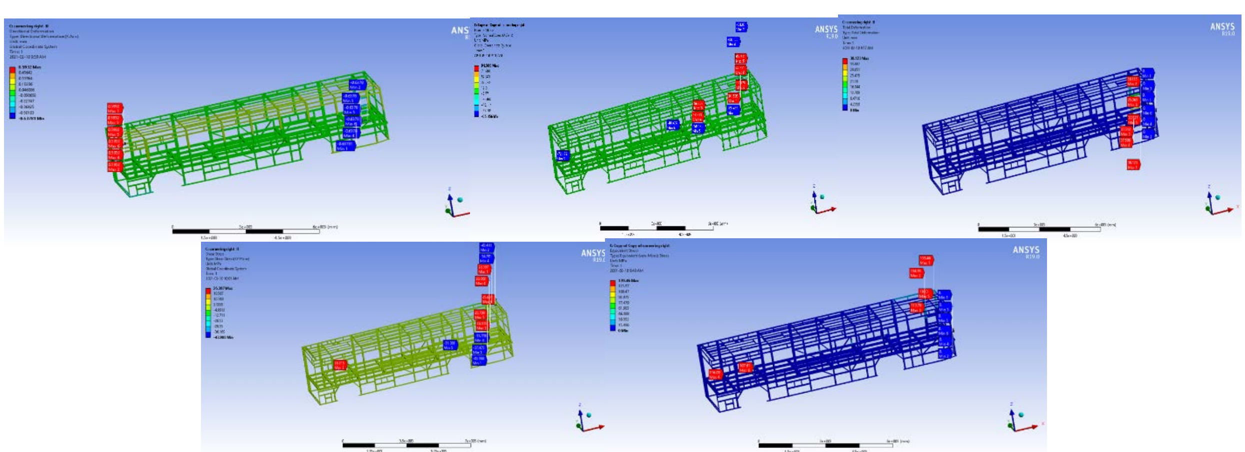


Figure: Right cornering load Result.

During left cornering; the structure experience both tensile stress (106Mpa) and compressive stress (-40.84Mpa) occurs at part 145 and part 99 respectively. The junction point between the side and bed structure developed the critical stress as shown in figure 4.11.

Figure 4.12 shows the equivalent stress of the bus structure; maximum stress is 153.33 Mpa occurs at part 15 and the minimum equivalent stress is zero. The rare side structure more likely to develop minimum von-misses stress values. The members on the sidewall and front side structure relatively develop higher equivalent stress values. The maximum equivalent that occurs is less than the maximum yield stress of steel structure so the structure is in the safe range.

As shown in Figure 4.13 the maximum total deformation values of 6.3194mm. Here also the minimum total deformation develops at the backside of the frame structure and the maximum total deformation comes along the frontal area of the structure.

Figure 4.14 shows the distribution of directional deformation along the x-axis, the maximum directional deformation is 4.6852mm along the positive x-axis and – 0.34662mm along the Negative x-axis.

Figure 4.15 shows the shear stress of the structure, the maximum shear stress is 15.149Mpa occurs at part 41 and the minimum shear stress is -32.866Mpa recorded at part 22. The body structure can stand the maximum shear load along the XY plane.

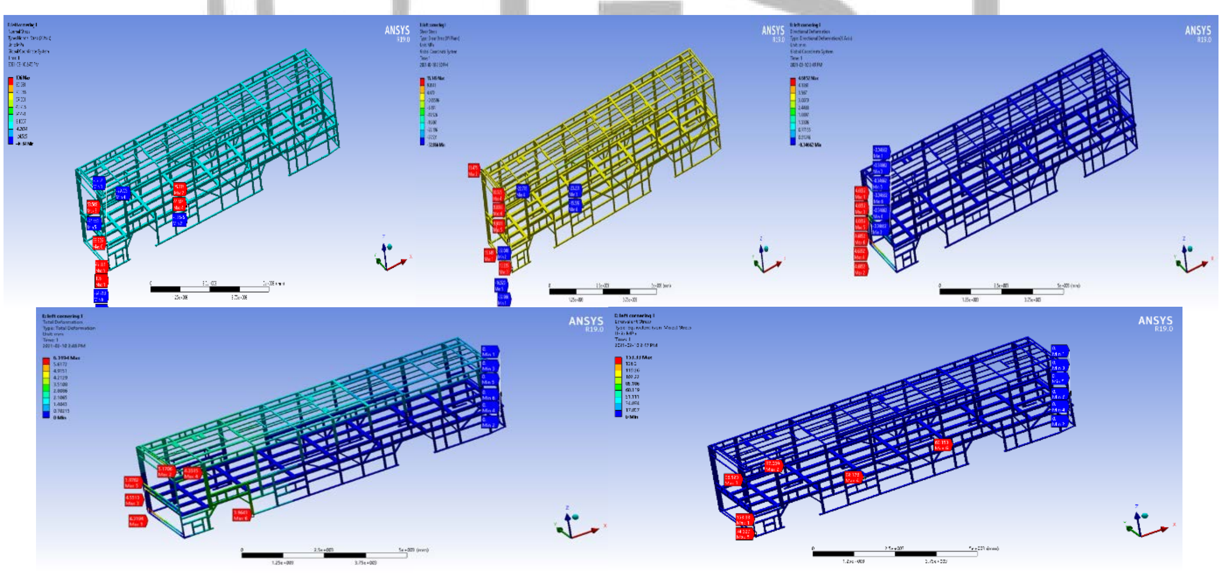


Figure left cornering results

Improved bus

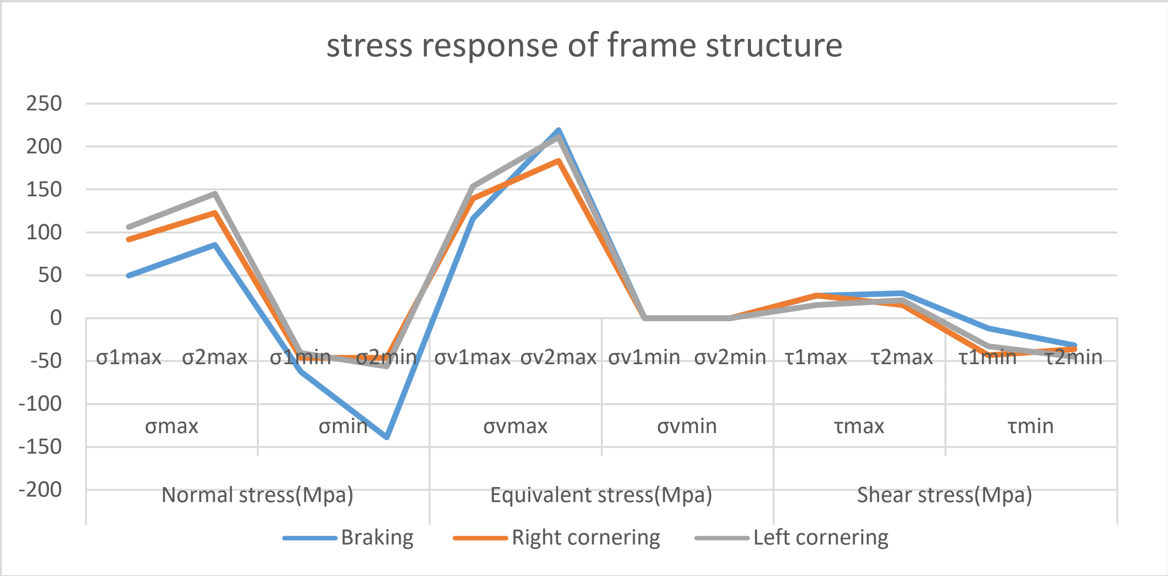
It was found that the modified bus body structure weighs 93.5778 kg less than the original body structure weight. The amounts to a 3.07% improvement in the weight without compromising the

GSJ: Volume 10, Issue 6, June 2022
ISSN 2320-9186

safety and performance of the vehicle. Stress and strain parameters of improved bus structure with the original one were compared in the following tables.

Table: comparison of response of frame structure original and improved bus structure.

parameter			Braking	Right cornering	Left cornering
Normal stress(Mpa)	σ_{max}	σ_{1max}	49.545	91.586	106
		σ_{2max}	85.5	122.73	144.75
	σ_{min}	σ_{1min}	-61.95	-46.1	-40.84
		σ_{2min}	-138.82	-46.006	-56.32
Equivalent stress(Mpa)	σ_{vmax}	σ_{v1max}	115.99	139.46	153.33
		σ_{v2max}	219.14	183.28	210.88
	σ_{vmin}	σ_{v1min}	0	0	0
		σ_{v2min}	0	0	0
Total deformation(mm)	ε_{max}	ε_{t1max}	2.3961	38.123	6.3194
		ε_{t2max}	8.492	7.8018	8.689
	ε_{min}	ε_{t1min}	0	0	0
		ε_{t2min}	0	0	0
Directional deformation(mm)	ε_{xmax}	ε_{x1max}	0.45183	0.5922	4.6852
		ε_{x2max}	1.1251	5.1048	0.71974
	ε_{xmin}	ε_{x1min}	-0.273	-0.6371	-0.34662
		ε_{x2min}	-1.0679	-0.37266	-5.97
Shear stress(Mpa)	τ_{max}	τ_{1max}	26.376	26.387	15.149
		τ_{2max}	28.996	15.356	20.83
	τ_{min}	τ_{1min}	-11.62	-43.09	-32.86
		τ_{2min}	-31.72	-36.06	-45.2



Conclusion

- The research work started with the measurement of the complete dimensions of the bus body, and from that, a model was created in Solid work for further analysis using ANSYS software.
- Through the analysis identified strong and weak points in the structure. The analysis is done through the longitudinal and lateral performance of braking and cornering loads and boundary conditions of the bus structure.
- The results obtained from the analysis give us room to further improve the bus structure by redesigning. In the improvement process, the members of the original structure are replaced with reduced thickness and cross-section profile.
- Accordingly, a new model was recreated using the reduced size elements and again analysis was carried out under the same loading conditions. The result of the analysis confirmed that the modified structure is safe in all aspects.
- It was found that the modified bus body structure weighs 93.577 kg less than the original body structure weight. The amounts to a 3.07% improvement in the weight without compromising the safety and performance of the vehicle.
- Based on the previous works of literature analogy, it is expected to reduce 2.4% fuel consumption and a 1.2 % reduction in emission.

Reference

[1] G. Heppeler, M. Sonntag, and O. Sawodny, *Fuel Efficiency Analysis for Simultaneous Optimization of the Velocity Trajectory and the Energy Management in Hybrid Electric Vehicles*, vol. 47, no. 3. IFAC, 2014. doi: 10.3182/20140824-6-ZA-1003.00286.

[2] X. Tag and C. D. X. X, "Fuel consumption and CO 2 emissions from passenger cars in Europe À Laboratory versus real-world emissions I," *Prog. Energy Combust. Sci.*, vol. 60, pp. 97–131, 2017, doi: 10.1016/j.pecs.2016.12.004.

[3] F. Lan, J. Chen, and J. Lin, "Comparative analysis for bus side structures and lightweight optimization," *Proc. Inst. Mech. Eng. Part D J. Automob. Eng.*, vol. 218, no. 10, pp. 1067–1075, 2004, doi: 10.1177/095440700421801001.

[4] L. Pelkmans, D. De Keukeleere, and G. Lenaers, "Emissions and fuel consumption of natural gas powered city buses versus diesel buses in real-city traffic," *Adv. Transp.*, vol. 8, pp. 651–660, 2001.

[5] E. Sundberg, "Weight Optimization of a Composite Chassis for a Multimodal Lightweight Vehicle," 2014.

[6] F. Lan, J. Chen, and J. Lin, "Comparative analysis for bus side structures and lightweight optimization," *Proc. Inst. Mech. Eng. Part D J. Automob. Eng.*, vol. 218, no. 10, pp. 1067–1075, 2004, doi: 10.1177/095440700421801001.

[7] Y. Jung, S. Lim, J. Kim, and S. Min, "Lightweight design of electric bus roof structure using multi-material topology optimisation," *Struct. Multidiscip. Optim.*, vol. 61, no. 3, pp. 1273–1285, 2020, doi: 10.1007/s00158-019-02410-8.

[8] T. Y. K. and S. H. LEE, "Combustion and Emission Characteristics of Wood Pyrolysis Oil-Butanol Blended Fuels in a Di Diesel Engine," *Int. J. …*, vol. 13, no. 2, pp. 293–300, 2012, doi: 10.1007/s12239.

[9] S. Lu, H. Ma, L. Xin, and W. Zuo, "Lightweight design of bus frames from multi-material topology optimization to cross-sectional size optimization," *Eng. Optim.*, vol. 51, no. 6, pp. 961–977, 2019, doi: 10.1080/0305215X.2018.1506770.

[10] G. Foucault, J. C. Cuillière, V. François, J. C. Léon, and R. Maranzana, "Adaptation of CAD model topology for finite element analysis," *CAD Comput. Aided Des.*, vol. 40, no. 2, pp. 176–196, 2008, doi: 10.1016/j.cad.2007.10.009.

[11] H. Zhang, G. Huang, and D. Yu, "Numerical modeling for the frame structure of light van-type electric truck," *Sci. Prog.*, vol. 103, no. 2, pp. 1–27, 2020, doi: 10.1177/0036850420927204.

[12] M. R. Chandra, S. Sreenivasulu, and S. A. Hussain, "Modeling and Structural analysis of heavy vehicle chassis made of polymeric composite material by three different cross sections," *Int. J. Mod. Eng. Res. www.ijmer.com*, vol. 2, no. 4, pp. 2594–2600, 2012, [Online]. Available: https://s3.amazonaws.com/academia.edu.documents/56474455/PROPUES_TPM.pdf?response-content-disposition=inline%3B filename%3DDiseno_de_un_plan_de_Mantenimiento_Prod.pdf&X-Amz-Algorithm=AWS4-HMAC-SHA256&X-Amz-Credential=AKIAIWOWYYGZ2Y53UL3A%2F20200314%2Fus-e

[13] T. C. Nguyen, "Research on Application of 3D Parametric Software for Design and Shaping of Bus Frame Structure," *Appl. Mech. Mater.*, vol. 894, no. September 2019, pp.

96–103, 2019, doi: 10.4028/www.scientific.net/amm.894.96.

[14] R. R. Prabhu and V. R. Deulgaonkar, “Finite element analysis and experimental validation of residual stress analysis in T-welded zone of aluminum alloy 6061-T6,” *Int. J. Mech. Prod. Eng. Res. Dev.*, vol. 8, no. 4, pp. 1169–1176, 2018, doi: 10.24247/ijmperdaug2018120.

[15] J. Ahokas and S. Kosonen, “Dynamic behaviour of a tractor-trailer combination during braking,” *Biosyst. Eng.*, vol. 85, no. 1, pp. 29–39, 2003, doi: 10.1016/S1537-5110(03)00035-7.