Journal of Mines, Metals and Fuels, *73(3): 575-587; 2025. DOI: 10.18311/jmmf/2025/47543*

Print ISSN : 0022-2755

Journal of Mines, Metals and Fuels

Contents available at: *www.informaticsjournals.co.in/index.php/jmmf*

A Review of Enhancing Passenger Safety and Comfort in Indian Buses by Finite Element Analysis (FEA) Approach

Anurag Ashok Kumar Nema* and Shantanu M. Jagtap

Department. of Mechanical Engineering, School of Engineering and Sciences, MIT Arts, Design and Technology (MIT-ADT) University Loni, Pune - 412201, Maharashtra, India; anurag.nema@mituniversity.edu.in

Abstract

In India, buses are considered a leading mode of transport for travelling across cities. Since a large population of India travels using buses, it is a fundamental need to understand the structural ability of passenger buses to withstand crash energy and provide maximum comfort against vibrations by onboard components. In India, there are rules in place to ensure the safety of buses. These rules cover important areas such as the strength of the bus structure, fire safety measures, the availability of emergency exits and the performance of the brakes. These regulations are detailed in the Automotive Industry Standards (AIS) such as Code of Practice for Bus Body Design Approval as per AIS-052, Rollover and frontal pendulum impact test as per AIS-031/AIS-119, Urban Bus Specifications-II (UBS-II), Additional Requirement for Bus Construction as per AIS-153, which are established by the Automotive Research Association of India (ARAI) and the Government of India. The design of the standard bus body depends mainly on the performance constraint under various types of loads and operating circumstances besides those of the road conditions. It is important to prioritise the safety and comfort of passengers, which includes understanding how buses can withstand crashes and reduce vibrations. This study delves into the use of Finite Element Analysis (FEA) to assess the structural strength of buses by means of correlating the physical behaviour with the analytical calculations and physical tests. Finite Element Analysis software such as ANSYS, LS-Dyna and Optistruct provide effective tools for simulating how buses react to different stress levels and situations.

Major Findings: *Focusing on the advancement made in safer designs and more comfortable ergonomics in the automotive sector, the FEA methods have provided a cost-effective and time-efficient solution, if a rollover kind of analysis is considered. If the natural frequency domain is tested as per global standards, the safety of passengers could be provided with the bare minimum requirements of comfort. Comfortable journey marks the success of designers and to justify the performance, numerical methods have proven its eligibility over decades.*

Keywords: *Accident, CAE, Crashworthiness, Protection, Simulation, Vibration*

1.0 Introduction

The Maharashtra State Road Transport Corporation (MSRTC) runs bus services for passengers with 18,449 buses daily. From 21 May 2020, the MSRTC started goods transportation, private bus body building, private vehicle tyre remoulding, etc. To build a standard bus, the bus body structure is required to be balanced, so that the bus structure can handle normal loads and accidental loads. To regulate the design and manufacturing of bus bodies, the Government of India has commissioned the Central Motor Vehicle Rules (CMVR), which ensures that the design of buses follows the guidelines mentioned in the AIS. Automotive testing and certification agencies like

**Author for correspondence*

ARAI, Pune, India, Central Institute of Road Transport (CIRT, Pune, India), International Centre for Automotive Technology (ICAT, Haryana, India), etc. possess the ability to test such bus structures as per the AIS and IS standards.

It is important to analyse the natural vibration frequencies and the mode shapes of the bus body for learning the vibration and noise. In complex structures like buses, due to a lack of theoretical information regarding the reduction of vibration of a passenger bus, most bus-body builders opt for empirical methods in the initial stages of bus body building. To reduce vibrations, it is important to carry out the modal analysis and design optimisation for the bus structure. Many research scholars have conducted studies to determine the natural frequencies and vibration modes of mechanical components. Natural frequency and resonance show the overall structural stiffness of the bus structure. Mode shapes like torsional and bending show the behaviour of the bus body under natural frequency analysis.

Across the globe, various FEA software are used to perform different types of analysis like static structural, vibration, computational fluid flow, electromagnetics, fatigue, etc. When solving complex problems, mainly two types of analysis are conducted; implicit and explicit. Implicit type of analysis is more suited for linear, static or time-dependent (quasi-static), slow dynamic problems. This type of analysis is conducted with low strain rates. For implicit analysis, software like Altair Optistruct, ANSYS, Nastran, etc. shows higher convergence with actual results.

The explicit type of analysis is conducted where highly non-linear dynamic problems are to be solved. Problems that include very high-speed rates, high degrees of freedom among subsystems or components, different behaviour of material in different axes, etc. come under explicit analysis. To solve such problems, FEA software like LS-Dyna, Altair Radioss, Abaqus, etc. are widely used. In case, any computational fluid dynamics problem is to be solved, software like Open FOAM, ANSYS Workbench (Fluent module), STAR CCM++, COMSOL, Autodesk CFD, Simscale, etc. are widely used. To conduct fatigue-related analysis, Altair FEMFAT, Siemens Simcenter, etc. are the most popular among the basic fatigue analysis solvers or software.

In the case of electrical components, electromagnetic FEA analysis is widely performed. Software like Dassault Systèmes SIMULIA, ANSYS HFSS, etc. provide relatively accurate simulation results, similar to the physical aspect of behaviour.

2.0 Types of Analysis Performed in Finite Element Regarding Bus Structure

2.1 Fire Safety Analysis of Bus Materials

A study was conducted to analyse the bus materials under the fire accidents and regulations in 2023. This study was proposed to analyse the safety of passengers from fire hazards while travelling. Often, when a bus is subjected to an accident, the entire bus catches fire and passengers face difficulties during evacuation due to smoke. In such cases, computational fluid dynamics tools can be used just as FEA. Since buses operate on highways, buses and coaches should be evaluated against fire scenarios. Bus interior material contains types of rubbers, polycarbonates, ABS, etc.[1] The type of test is briefly mentioned in UN ECE Reg. No. 118, but it is observed that the mentioned test methods need to be replaced by test methods that can evaluate the reaction to fire when exposed to the source of ignition. Safety against smoke and toxic gases should be considered in the fire requirement standards of UN ECE 118. During a crash and fire scenario, there is only a few minutes window for passengers to exit from the bus and smoke gases strongly affect the evacuation of passengers[1].

To generate an idea about how the computational methods can be effectively used in research on the spread of fire smoke in a bus. Fire Dynamics Simulator (FDS) was analysed in a study sponsored by the National Natural Science Foundation of China and Scientific Research Foundation of Shandong University of Science and Technology. The computational methods were used to describe the factors affecting bus fires, by employing temperature and hot flue gases in a simple single-deck bus. By the computational method, it was observed that the diffusion rate increases when flue gases are produced more. The concentration of carbon monoxide in the bus was found to increase as the flue gas rate increased[1].

2.2. Evaluation of Bus Structure Under Strength Requirements

To verify the strength of the bus structure in case of a frontal crash, a study was proposed in 2022, consisting of FEA as per ECE R 29. The FEM, which is widely used in research and development is preferred in the automotive industry for structural analysis. A comprehensive evaluation of the bus structure was conducted and the structure part of the bus is tested as per the test procedure mentioned under ECE R29 regulation of European Standards[2]. The deformation of the frontal structure can be improved by two different zones, 1st is the crumple zone and 2nd is the driver compartment zone. The Crumple zone absorbs maximum deformation energy and the driver compartment zone resists the deformation after impact. Figure 1 shows the depiction of deformations.

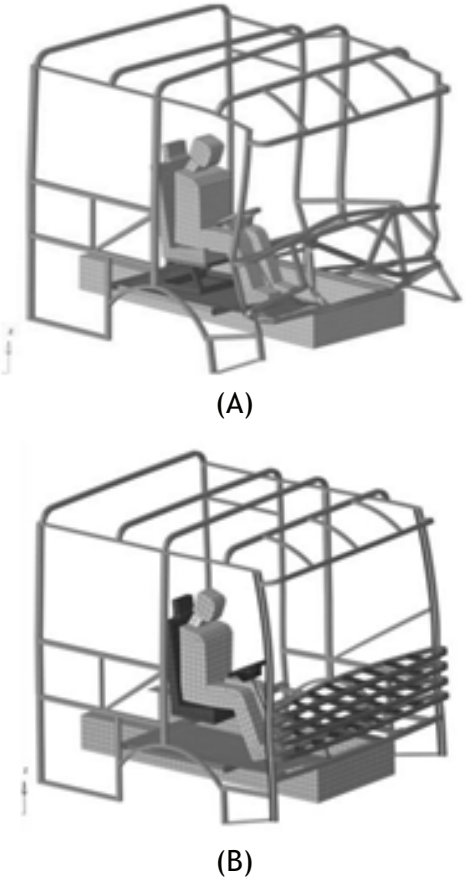


Figure 1. **(A)** Structural deformation before optimisation[2] **(B)** Structural deformation after optimisation[2].

In the Indian context, for an optimised bus body design, a newly optimised and developed model must be chosen for manufacturing the body. These design specifications are available in the local automotive industry[3]. A study conducted in 2019 suggested that general design parameters which included the length, width, height, etc. should conform with the AIS-052 (revision 1) and the testing methodology must conform with the AIS-031[4]. If we see the current trend, there is a demand for buses with improved vehicle features and overall work performance. If we observe a typical bus structure, it mainly comprises tubular sections or hat sections. Research conducted in 2014 showed the effect of quasi-static loading in the bus carline model.

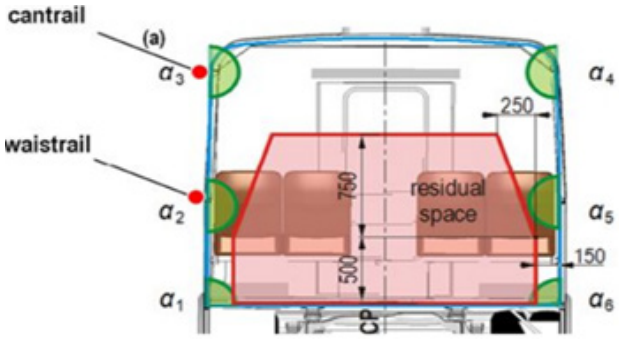


Figure 2. Carline bus model with residual space and dimensions[5].

In this carline study, the application of load i.e. 10475 N is done through a quasi-static approach. As per AIS-031, residual space should be placed on the bus structure as shown in Figure 2. It is evaluated that; the sectional strength of hat-section is lesser than the rectangular tubular section[5].

2.3 Material Response Analysis by Finite Element Method for Bus Structure

When a transient loading condition is considered for any structural member, parameters like Von Mises stress, total deformation, etc. are considered crucial[6]. A study proposed by Kishan A H in 2021, showed the transient loading effect on electric bus chassis frame. To verify material response in the chassis frame, transient load analysis can be conducted under static and dynamic conditions. Generally, the chassis frame is comprised of a rectangular box section, channel section and channel

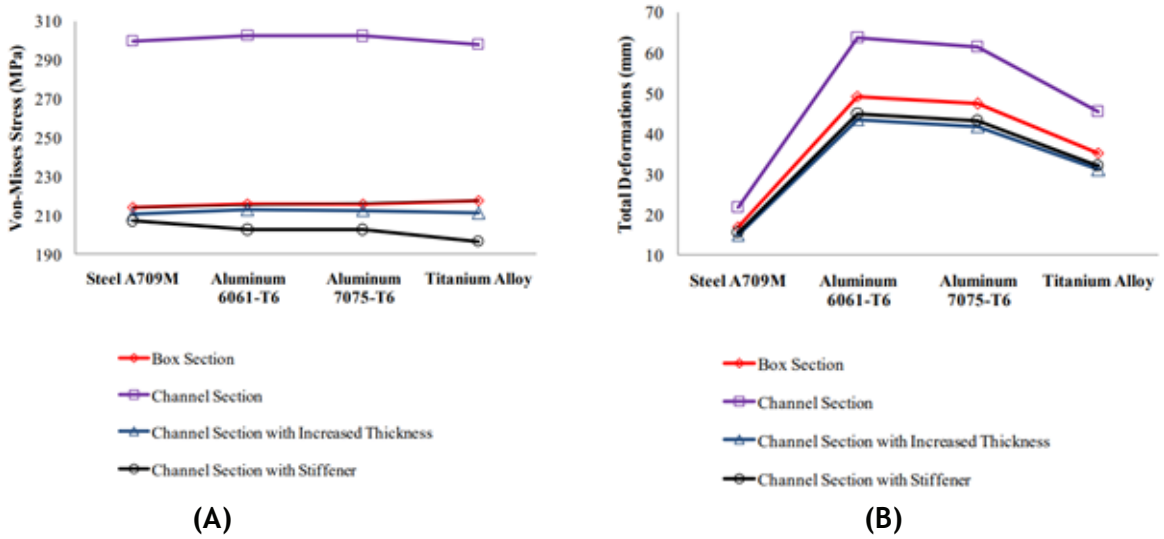


Figure 3. **(A)** Von-mises stress comparison between materials **(B)** Total deformation comparison between materials.

section incorporated with stiffeners inside[6]. The author proposed a comparison of four different materials, such as steel, aluminium 6061-T6, aluminium 7075-T6 and titanium alloy in Figure 3.

If the acceleration and braking concept is adopted for bus structure analysis, a study showed that certain cases can be performed in ANSYS 14.1. Under a static load of 180 N, titanium material produces more stress and deformation than aluminium material. The stress range for titanium can vary from 310 to 330 MPa when a load of 180N is applied. However, the use of aluminium alloy can potentially reduce the weight of the vehicle body and the impact on efficiency can also be noted[7].

The usage of fibreglass in bus body construction would significantly reduce structural weight, but the yield strength of fibreglass is lesser than steel by 11%[8]. To determine the strength of alternate materials, both numerical and experimental dynamic tests are required, which would evaluate material properties too. Considering Poisson's ratio of composite material, in the longitudinal direction it can be observed up to 0.035. If a simple tension test is conducted on pultruded fibreglass composite material, steel and aluminium, the comparison would show that fibreglass withstands 1.5 times more force than steel and 2.3 times more force than aluminium as shown in Figure 4[8].

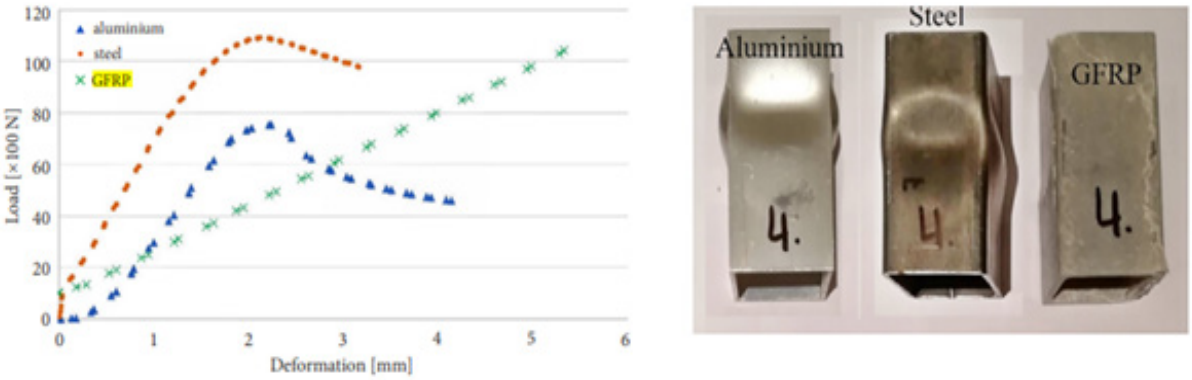


Figure 4. Load vs deformation plot for analysed materials and corresponding deformation in specimen.

In general cases, if a bus body designer conducts FEA analysis, the quasi-static loading conditions are performed with the yield ranging between 240 MPa and 380 MPa material. A study performed by Padmavati, *et.al.* concluded that the Kevlar material could withstand higher loading and it gave minimum deformation when subjected to prescribed acceleration and impact load[9]. If the cost of material is taken into account, Kevlar costs are higher as compared to structural steel and aluminium alloy 6061.

2.4 Natural Frequency and Vibration Response by Finite Element Method for Bus Structure

FEA is an extensively established and globally used instrument for numerical structure analysis, The typical steps involved in FEA analysis of the bus body are shown in Figure 5. For any FEA, the generation of a 3D CAD model is the very first stage[10]. Considering modal analysis (natural frequency and eigenmodes), free-free analysis case and boundary-constrained analysis could be performed[11]. In 2020, Anand Raj, *et al.,* performed a fatigue analysis of the bus structure with aluminium alloy material, for 10,00,000 cycles of starting-up and shutting-down, the fatigue life outcome which was obtained in the test was more than 1,00,000 cycles, which confirmed the safety of optimized bus structure[11].

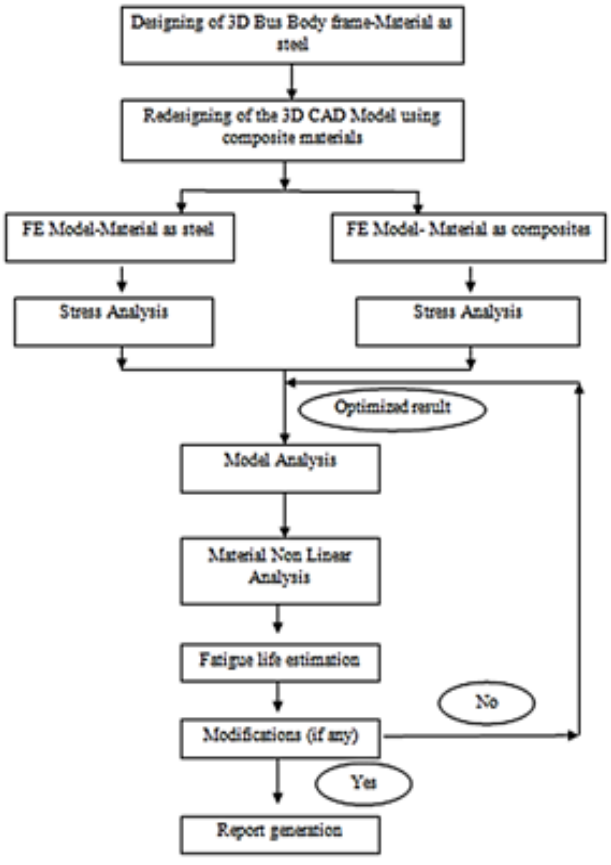


Figure 5. Typical phases in bus body analysis in FEA.

For higher stiffness and construction of the bus structure, different types of section members are used. Every bus body configuration consists of a tubular cross-section like 80X40mm, 60X40mm, 40X20mm, a square cross-section of 40X40, a U-beam of c/s 60X30mm, etc,[12]. In 2018, free modal analysis was performed by Haryanto, *et.al.* for verification of natural frequencies of the bus frame and further optimisation. The problem of optimisation was formulated as,

$$f = f(\{x\})$$

Subjected to,

$$g_k\ (\{x\}) \leq 0,\ \forall k = 1,2,\ldots,K$$

$$h_1\ (\{x\}) \leq 0, \forall\ l = 1,2,\ldots,L$$

$$\{x_L\} \leq \{x\} \leq \{x_U\}$$

Where, x_L and x_U are upper and lower bound of design variables[12].

In 2018, Deng S., *et.al.* conducted a modal analysis of a bus structure and optimisation of the bus body frame in such a manner that the structure could avoid the resonance between the side structure and the rear of the cavity[13]. The theoretical natural frequency parameter (ω) could be found out by,

$$\omega = \sqrt{\frac{K}{M}}$$

where K is the stiffness matrix

M is the mass of the component

The generalised equation of motion in matrix form could be written as,

$$[M]\{\ddot{x}(t)\} + [C]\{\dot{x}(t)\} + [K]\{x(t)\} = \{f(t)\}$$

Where, [*M*], [*C*], [*K*] are the mass, damping and stiffness matrices respectively along with the corresponding vectors of acceleration, velocity and displacement[13].

In 2016, Hu K, *et al.,* performed a modal analysis of the chassis frame structure of an electric bus, so that the theory of frequency analysis and natural modes could be resolved. The optimisation of the bus chassis frame should be carried out in such a way that the natural frequency should be lower. If a modal frequency value obtained in the finite element analysis is lower than the externally excited vibrations such as transmission, onboard components, road roughness, etc. the resonance phenomenon could be avoided[14].

To improve dynamic parameters like noise and vibrations, the bus structure can be validated under modal analysis to understand structural resonance frequencies. By performing such analysis, the determination of spatial characterisation of vibration patterns would be possible. A study was conducted in 2020 by Tautvydas P, *et.al.*, to justify the usage of alternative materials in bus body construction. This study showed the possibility of the usage of steel and fibreglass composite materials in safety arcs, roof members, side structures, etc,[15]. The dynamic properties of bus structure can be evaluated by determining the response of the structure to forced excitation. Table 1 shows the results of modal analysis when material modifications are made in the bus body.

Topology optimisation of a bus structure could reduce the weight of the bus structure, making it lightweight. A study was conducted by Xin-Chun Wu, *et.al.*, for topology optimisation by using finite element software named Optistruct. The weight of the bus structure was reduced by 3.8% and the results of optimisation showed a positive scope for further optimisation in design[16].

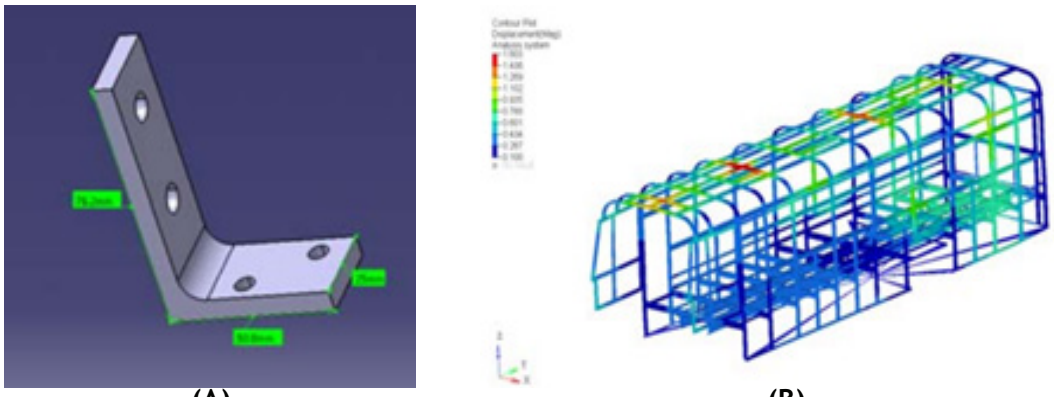
If the bus structure is considered, the entire bus frame is attached to its structural members using various joints[17]. Often it is observed that when the bus body is subjected to several loads such as standing, cornering and bump loads, the joints of the bus floor structure form a weak linkage. In such cases, an additional stiffener like a small clit, stump pillars, support plates, etc. provides an improvement in the overall stiffness of the bus structure. A study conducted in 2016 showed that by using such type of arrangement structure construction, bus structures show a reduction in stress by almost 45%. Due to additional stiffness by clit design, the bus structure could show a natural frequency increment of 4Hz. Figure 6 shows the clit design and respective displacement contour.

Case Study: The effect of vibration characteristics of an automotive seating system on ride comfort – A finite element study.

In general, an automobile comprises various sub-systems like engine assembly, transmission, suspension, linkages, electronic components, structural members, seating,

Table 1. Results of numerical modal analysis of material modifications to the bus body[15].

Mode No.	Resonant Frequencies, Hz and the Change						
	Option1	Option2		Option3		Option4	
	Steel	Steel and Fiberglass (Safety Arcs)		Steel and Fiberglass (Roof and Sides)		Fiberglass	
	Frequency, Hz	Frequency, Hz	Change $\Delta = f_{FE\ Steel}/f_{FE}$	Frequency, Hz	Change $\Delta = f_{FE\ Steel}/f_{FE}$	Frequency, Hz	Change $\Delta = f_{FE\ Steel}/f_{FE}$
1	6.711	4.789	1.401	6.009	1.117	2.654	2.529
2	8.115	7.421	1.094	6.811	1.191	3.033	2.676
3	9.912	9.421	1.052	8.785	1.128	3.692	2.685
4	14.230	12.063	1.180	10.068	1.413	4.846	2.936
5	14.863	12.173	1.221	11.442	1.299	4.962	2.995



(A) (B)

Figure 6. **(A)** Structure support clit design[17] **(B)** displacement contour[17]

etc. Considering the automotive seating system, the need for vibration analysis is very important to increase the comfort levels of drivers or passengers. To generate an idea about how seating assembly can be modified in such a way that the whole sub-system provides maximum comfort to human anatomy parts like the spinal column, abdomen, etc., a case study was conducted in 2019[18]. The general natural frequency range for the spinal column varies from about 10 – 12Hz and abdomen about 4 – 8Hz. Detailed frequency ranges for other anatomical parts of the human body are mentioned in Table 2.

Table 2. Natural frequency of human body parts

Sr. No.	Body Part	The natural frequency of human body parts (Hz)
1	Shoulder	4 – 5
2	Abdomen	4 – 8
3	Hand	30 – 50
4	Spinal column	10 – 12
5	Head	20 – 30
6	Chest Wall	50 – 60

Considering the natural frequency of the seating system, there are various critical zones on the system where vibration transmissivity is more towards the human body, as shown in Figure 7. The entire complex problem can be solved in finite element analysis and it is feasible to correlate the FEA results with previously experimented results that are readily available in the literature[18].

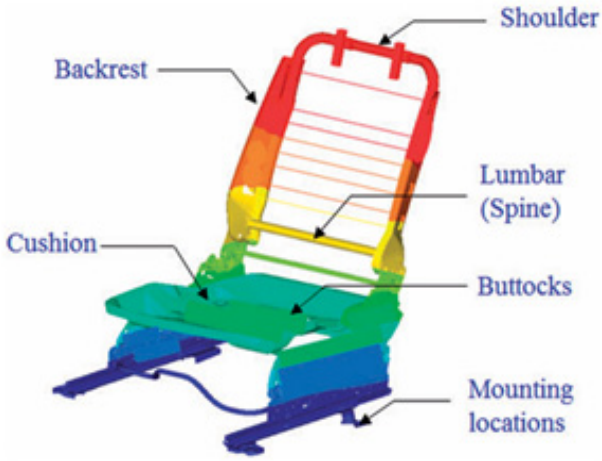


Figure 7. Critical zones of the seating system.

Such a type of critical natural frequency related to ride comfort is briefly described in ISO 2631. The degree of comfort varies from extremely comfortable to extremely uncomfortable in relation to the acceleration

Table 3. Ride comfort index defined by ISO 2631

Acceleration Level	Degree of comfort
< 0.315 m/s^2	Extremely comfortable
0.315 – 0.63 m/s^2	Comfortable
0.5 – 1 m/s^2	Fairly comfortable
0.8 – 1.6 m/s^2	Uncomfortable
1.25 – 2.5 m/s^2	Very uncomfortable
> 2 m/s^2	Extremely uncomfortable

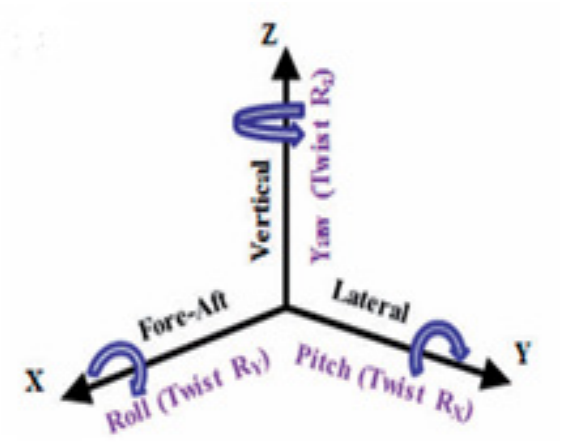


Figure 8. Vehicle Co-ordinate system related to mode shapes

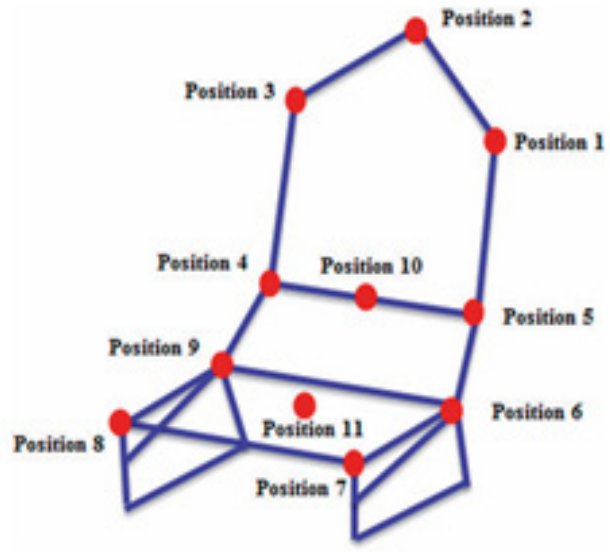


Figure 9. Responses measured positions

levels experienced by the rider[19]. The ride comfort index shows that acceleration levels of more than 2 m/s^2 are very uncomfortable for the rider of an automotive system. The vibration frequency ranging from 0.1Hz to 0.5Hz tends to provide an experience of motion sickness for the passenger. The time-dependent ride comfort index under ISO 2631 – 1: 1997 is shown in Table 3.

The models described in the case study consisted of 4 different seat models, analysed under FEA. Each model is tested under natural frequency with a dummy and without a dummy. The seat models were further analysed with the addition of stiffeners for 3 different mode shapes; Lateral mode, Fore-aft mode and Twisting mode as shown in Figure 8.

The case study concluded that, when automobile seating systems are excited with 1m/s^2 at 11 positions (as shown in Figure 9), the natural frequency and multiplication factors of the seating system are in agreement with the experimental results of the literature and the transmissibility of vibration from seat backrest to human body was found to be more than 1.

3.0 Types of Analysis Performed in Finite Element Regarding Bus Structure

Depending upon the concerns of design aspects and whether the design is confirming enough, various analyses are conducted to justify the conformity. Mainly, when a bus is subjected to a rollover incident, the bus structure deforms beyond the material's yield limit. In engineering terms, it is considered plastic deformation. In such cases, bus structure should be validated with industry-specified materials under rollover analysis. Hence, considering the safety of passengers, rollover analysis is very crucial, before finalising any bus structure.

Various types of analysis are conducted for bus structure, such as rollover incident, the centre of gravity of the bus, stability angle verification, natural frequency analysis for stiffness verification, passenger seat vibration analysis, etc. Such types of analysis are explored further in the following section.

3.1 Rollover Analysis

Taking reference to the literature, it is considered that, the bus structure should be strong enough to provide safety to its passengers when a bus is subjected to a rollover incident from the side. As per UN/ECE R 66, to certify any bus structure design, the manufacturer has to perform a rollover analysis under UN/ECE R 66, Annex 5 to 9[20]. A similar case is present in the Indian context i.e. to certify any Indian bus structure, the manufacturer has to perform rollover under AIS-031, Annex I to IV. The AIS are mandatory for certification in India and AIS-031 considers rollover analysis for type approval of bus structure. From Figures 10 and 11, the depiction of the bus rollover model in FEA could be understood with the structure deformation.

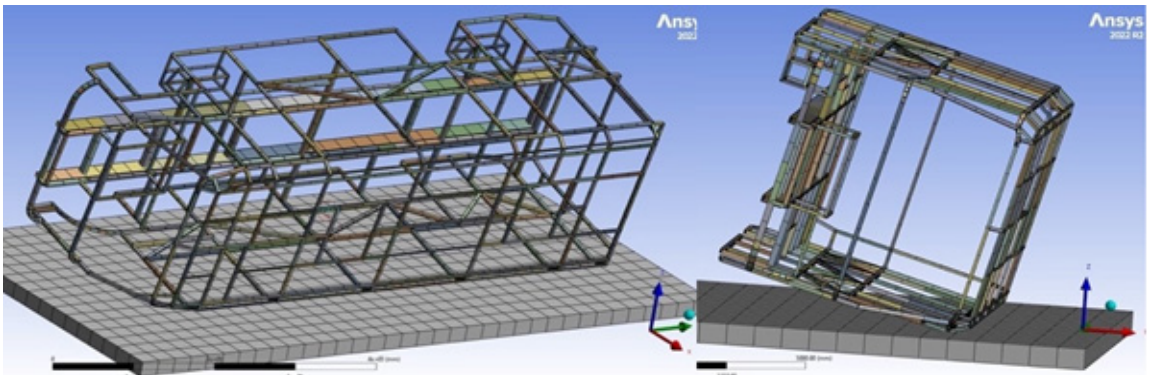


Figure 10. FEA model of bus structure depicting rollover incident as per UN/ECE R 66.

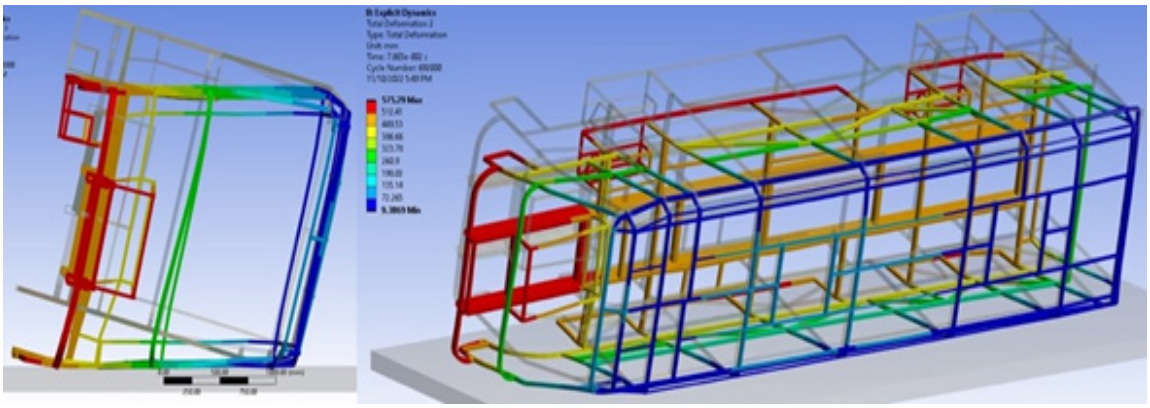


Figure 11. Structure deformation after rollover analysis.

3.2 Centre of Gravity Determination Using FEA

Figure 12 shows the Centre of Gravity (CG) measurement in buses. In this type of analysis, the FEM is prepared as per provided design and actual masses of bus components such as engine, transmission, fuel tank, axles, seats, panels, fitments, suspension systems, air conditioning system[21,22], etc. are applied in the model. The material data with density is added to the FEA model and a

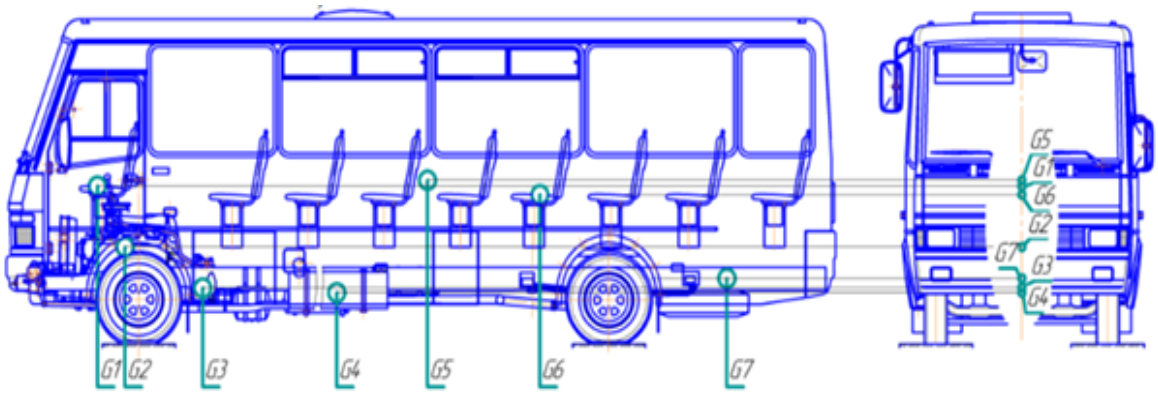


Figure 12. Determination of centre of mass or centre of gravity.

final assembly is created consisting of bus structure and masses. By using computer algorithms of the centre of gravity, FE software like Altair HyperMesh, ANSA, LS-Prepost, etc. can calculate the centre of gravity for the vehicle.

3.3 Determination of Instability Angle

When the centre of gravity is known for a vehicle, the FEA model can be used to determine the instability angle for the vehicle in empty condition (without passenger weight) or laden condition (with passenger weight). Considering the final weight of the bus, it is very necessary to determine the angle of instability as per AIS-052 for buses in India. Figure 13 shows the depiction of the instability angle determination of a complete bus[23,24].

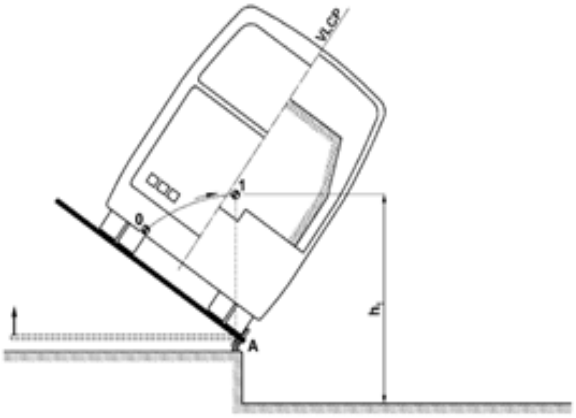


Figure 13. Instability angle determination depiction.

3.4 Natural Frequency of Bus Structure

The natural frequency analysis shows the stiffness of the bus structure when it is loaded with all masses except the axle and suspension. The axle and suspension system damps the vibration and frequency, resulting in poor accuracy regarding the bus structure. Hence, as per AIS-153 (Clause 2.3), the bus structure should be validated for natural frequency under bending mode and torsional mode. Figure 14 shows an example of vibration mode for a bus structure[25].

3.5 Passenger Seat Vibration Analysis

In general case, a passenger seat consists metallic structural frame, foam, bushes, recline mechanisms, etc,[26]. To determine the stiffness of the seat structure, finite element models are used to verify the overall natural vibrations in the seat structure frame. This type of analysis gives vibration mode concentration in the seat frame and the potential areas can be strengthened to remove the mode shape. In the physical aspect, vibration analysis is conducted by using the impact hammer method. Figure 15 shows the harmonic response of the passenger seat.

4.0 Technological Advancement

In recent years, the use of AI has increased in design generation and optimisation. Similarly, simulation methods are optimised and are based on Artificial Intelligence/ Machine Learning (AI/ML) algorithms. AI gives an upper hand in predictive analysis, as AI uses back

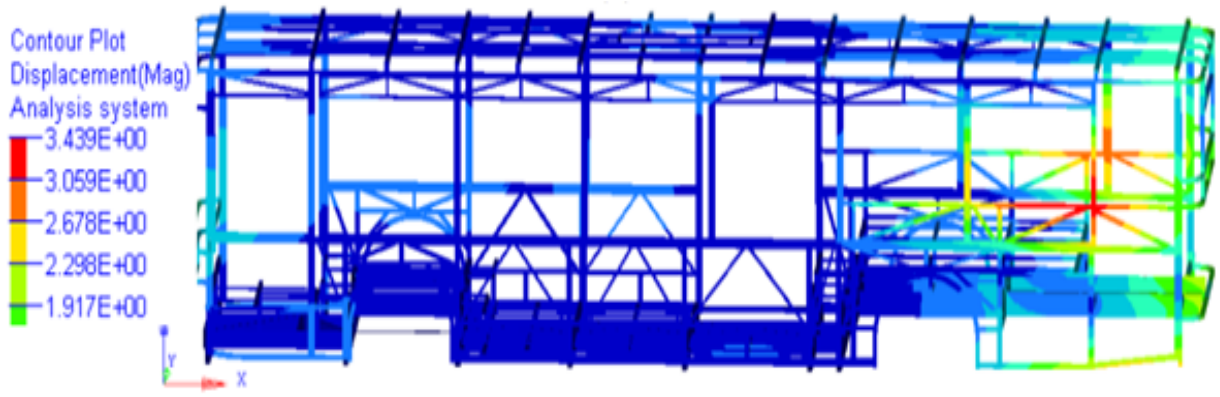


Figure 14. Vibration mode for bus structure.

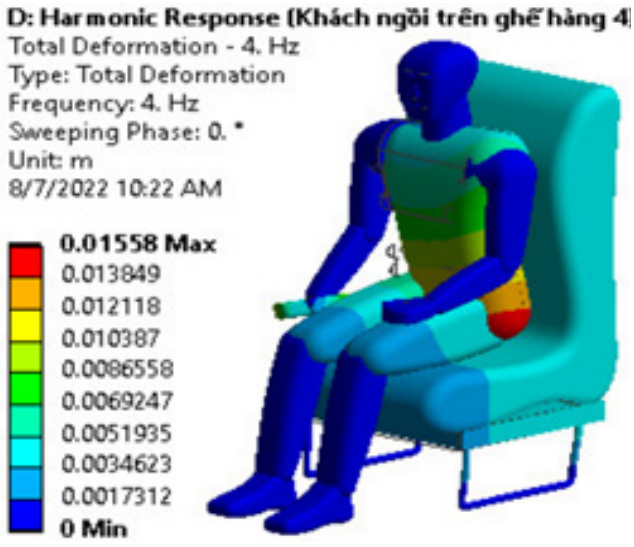


Figure 15. Passenger seat harmonic response[20].

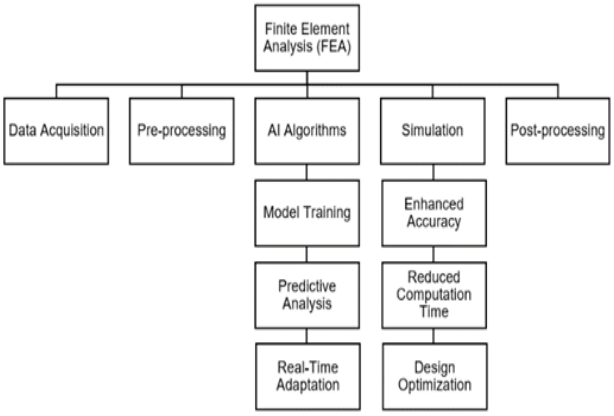


Figure 16. FEA process and AI element integration with FEA.

history data of previous simulations. In case of a rollover of a bus, various factors like centre of gravity, mass of model, etc. are very crucial,[27] Such data can be recorded to create history models for AI/ML application in FEA. The modes of failure are predicted by using AI algorithms and further optimisation can be done without solving the FEA simulation in the initial stages of product design.

Figure 16 shows the integration of AI elements with FEA processes. As AI can facilitate with generation of new ideas, finite element analysis methods can use the capability of adaptive learning from AI. This enhances the simulation process and optimised finite models are produced with fewer iterations. Initially, it may not produce acceptable models, however, accurate predictions can be achieved over time.

5. Future Scope

Considering bus safety, the use of lightweight and strong materials such as composites, alloys, etc. are considered to carry a significant potential. For more detailed clarification, correlation with the physical behaviour of the material with finite elements is very essential. This will generate an idea to give exponential growth in new designs and more safer buses can be manufactured with low costs. Material integration can be optimised in FEA by establishing a correlation in strain rates.

The modal frequency tests could be performed in ANSYS with Nonlinear Programming by Quadratic Lagrangian (NLPQL) optimisation algorithm. This could employ the automatic intelligent refinement for the finite element under global optima. This reduced the number of design points that are necessary for the optimisation of the bus superstructure can be efficiently found out.

The use of electric vehicles is increasing day-by-day and potential alternative fuels like hydrogen, bio-fuels, etc. are being considered with great application advantages. This will reduce the dependency on fossil fuels in buses as well. To provide safety with such fuels, new bus structure designs can be simulated in finite element analysis and comfort for passengers can also be optimised by reducing vibrations. In such types of buses, dynamic loading analysis conditions can be verified by using the FEM.

Since the finite element method is non-destructive, designers can perform various virtual prototypes and these prototypes can be tested without further expenditure and time. For this, Augmented Reality (AR) and Virtual Reality (VR) can be utilised to identify crucial stress concentrations and much more. Similarly, cloud-based computing can be explored for solving finite element problems.

6.0 Conclusion

In this review, we examined how vital natural frequencies are for typical bus structures and understood the need for improvements to avoid structural failures. The investigation of the necessary standards for bus bodies and focus on creating a new, improved bus superstructure

design could be possible by referring to such information. We understood how we can use composites and alloys in bus body construction with suitable cross-sections of load-carrying members. The optimisation of a lightweight structure will improve the efficiency of the vehicle as well and manufacturing costs will be reduced. The servicing of bus structures can be optimised by various methods that are studied above. These innovative designs should undergo thorough testing under both static and dynamic loads, including analysis of natural frequencies. An in-depth evaluation of the bus structure's natural frequencies should be carried out. This research analysis could pinpoint possible resonance points where vibrations from the engine, road conditions, or passenger activities could cause excessive strain. The required criteria for the bus body are reviewed in this paper. The study also aims to develop and validate the new enhanced design of bus superstructures under static and dynamic characteristics.

7.0 Author Contributions

Conceptualisation, S.M.J. and A.A.N.; Methodology, S.M.J.; Software, S.M.J. and A.A.N.; Validation, A.A.N.; Formal Analysis, S.M.J. and A.A.N.; Investigation, S.M.J. and A.A.N.; Resources, S.M.J. and A.A.N.; Data Curation, S.M.J.; Writing—Original draft preparation, S.M.J.; Writing—Review and Editing, A.A.N.; Visualisation, S.M.J. and A.A.N.; Supervision, A.A.N. All authors have read and agreed to the published version of the manuscript.

8.0 References

1. Houssami ME, Forsth M, Fredriksson H, Drean V, Guillaume E, Hofmann-Bollinghaus A. Fire safety of interior materials of buses. Fire Mater. 2023; 47:910-924. https://doi.org/10.1002/fam.3134
2. Dusane G, Gangrade K. Finite element analysis of bus structure as per ece r29. Int J Res P Rev. 2022; 3(3):1583-1585.
3. Madhuvanesan, Manimaran M, Praveen S, Sanjaikanth K, Sethupathy D. Design and analysis of bus body structure. Int J Res in Eng Sci. 2021; 9(1):72-80.
4. Purushottam RD, Ali SS, Srikanth D, Mohanachari JV. Design and analysis of bus body superstructure. IJRDO-J Mech Civ Eng. 2016; 2(2):1-8.
5. Rayakar RS, Bhat DS. Determination of strength analysis of bus body carline through fea. Int J Res Appl Sci Eng Tech. 2014; 2(7):260-268.
6. Kishan AH, Kondalarao P. Transient structural analysis of electric bus chassis frame. IOP Conf Ser Mater Sci Eng. 2021; 1185(1):1-10. https://doi.org/10.1088/1757-899X/1185/1/012038
7. Nayak J, Joshi A. Structural analysis of a bus body frame and its comparison on materials for I-section. Int Conf Rec Adv Interdiscip T Eng App; 2019. https://doi.org/10.2139/ssrn.3350940
8. Pravilonis T, Sokolovskij E, Analysis of composite material properties and their possibilities to use them in bus frame construction. Transport. VGTU Press. 2020; 35(4):368-378. https://doi.org/10.3846/transport.2020.13018
9. Padmavathi KR, Hariharan EKS, Muthu KV, Pranesh S, Raghul RR. Design and analysis of bus body frame. Int J Res in Eng Sci Mgmt. 2019; 2(3):882-888.
10. Guruprasad T, Babu S, Maruthi BH, Pramod KN, Manjunatha HG. Bus body structural strength analysis through fea, Int J Tech Res Eng. 2015; 2(11):2494-2498.
11. Anand SR, Ahamed SK, Rajath HG. Structural analysis of bus body frame using fea for static and dynamic analysis. IRJET. 2020; 7(8):2975-2979.
12. Haryanto I, Raharjo AF, 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. Int J Sust T Tech. 2018; 1(1):9-14. https://doi.org/10.31427/IJSTT.2018.1.1.2
13. Deng S, Han X, Yang L. Modal analysis and optimization of bus body structure, IOP Conf. Ser J Phy. 2018; 1074(1):1-8. https://doi.org/10.1088/1742-6596/1074/1/012048
14. Hu K, Dai Z, Zhang T. Modal analysis of the frame structure of a hybrid electric city bus. Int Conf Comp Eng Info Sci App Tech. 2016; 56(1):411-413. https://doi.org/10.2991/iccia-16.2016.74
15. Sokolovskij E, Pravilonis T, Kilikevicius A, Matijošius J, Kilikeviciene K. The usage of alternative materials to optimize bus frame structure. Sym MDPI. 2020; 12(6):1010. https://doi.org/10.3390/sym12061010
16. Xiu-Chun WU, Zheng WQ, Zhou P. Topology optimization design of bus body structure based on altair-optistruct, Int Conf Mech Civ Eng. 2014; 7(2):281-286. https://doi.org/10.2991/icmce-14.2014.50
17. Khatavkar S, Kumbhar SR, Wakode VV. Structural analysis of passenger bus body using fea by changing joint structure for improving strength. IJCESR. 2016; 3(6):28-32.

18. Ramalingam M, Jebaseelan DD. The effect of vibration characteristics of an automotive seating system on ride comfort - A finite element study. J Mech Eng Sci. 2019; 1-14. https://doi.org/10.1177/0954406219858172
19. Mechanical vibration and shock - Evaluation of human exposure to whole-body vibration - Part 1, International Standard - ISO 2631 - 1: 1997. [Internet] ISO Council; 2007. Available from: https://archive.org/details/gov.in.is.iso.2631.2.2003/mode/2up
20. Holenko K, Dykha A, Kalaczyński T, Matiukh S, Horbai O. Simulation of the TATA 079 suburban bus rollover tests in accordance with UN/ECE regulation no. 66. Matec Web Conf. 2023; 375:1-10. https://doi.org/10.1051/matecconf/202337502003
21. Min X, Pengyi B, Lin X, Chao L, Limin H, Mingyu A, Mingze F, Kaixuan L. Research on factors affecting bus fire based on numerical simulation. Clean Coal Tech.: Min Proc Safe and Eco. 2021; 303(01042):1-9. https://doi.org/10.1051/e3sconf/202130301042
22. United Nations Economic Commission for Europe. UN regulation no. 118. [Internet] UNECE; 2023. Available from: https://unece.org/transport/documents/2023/02/standards/un-regulation-no-118-rev3
23. United Nations Economic Commission for Europe. UN regulation no. 66. [Internet] UNECE; 2010. Available from: https://unece.org/transport/vehicle-regulations-wp29/standards/addenda-1958-agreement-regulations-61-80
24. ARAI. AIS-052: Code of practice for bus body design approval. [Internet] ARAI; 2017. Available from: https://hmr.araiindia.com/api/AISFiles/AIS-052_08236d19-d9a3-4ef8-a50e-08f917ea40dc.pdf
25. ARAI. AIS-153: Additional requirements for bus construction. [Internet] ARAI; 2020. Available from: https://hmr.araiindia.com/api/AISFiles/AIS-153%20with%20Amd%201%20to%206_a9470da4-0990-4634-8d51-c59b3e707e40.pdf
26. Ong LD, Quang NT, Quan LH, Hieu PM, Hieu LD. Analysis of vibration experienced by passengers and their seats in a bus vehicle. HaUI J Sci Tech. 2024; 60(5):219-221. https://doi.org/10.57001/huih5804.2024.186
27. ARAI. AIS-031: The Strength of Superstructure of Large Passenger Vehicles. [Internet] ARAI; 2017. Available from: https://hmr.araiindia.com/Control/AIS/320201824751PMAIS_031_n_Amd_4.pdf