BULETINUL INSTITUTULUI POLITEHNIC DIN IAŞI
Publicat de
Universitatea Tehnică „Gheorghe Asachi” din Iaşi
Volumul 71 (75), Numărul 3, 2025
Secţia
CONSTRUCŢII DE MAŞINI
DOI: 10.2478/bipcm-2025-0026

NVH INVESTIGATION OF VEHICLE CHASSIS

BY

ALEXANDRU MAFTEI[1], TONY PERNIN[2], COMAN BOGDAN[1], CARMEN BUJOREANU[1,*] and ADI-MIHĂIŢĂ VELNICIUC[1]

[1]“Gheorghe Asachi” Technical University of Iași,
Faculty of Mechanical Engineering, Iași, Romania
[2]Institut Supérieur de l'Automobile et des Transports, 49 Rue Mademoiselle Bourgeois,
58027 Nevers, France

Received: July 2, 2025
Accepted for publication: July 15, 2025

Abstract. This study presents a method for enhancing vehicle design through the integration of CAD-based topology optimization and finite element analysis (FEA) simulation. Specifically, the vehicle chassis is refined by incorporating design elements inspired by biomimicry, which are integrated to improve the vehicle’s NVH (Noise, Vibration, Harshness) performance. Noise and vibration have consistently played a critical role in vehicle design, as such, automakers devote substantial attention to minimizing the transfer of road and engine-induced disturbances into the cabin. This reduction is crucial, as excessive noise and vibration can lead to an unpleasant driving experience and may accelerate driver fatigue. Moreover, if the chassis does not effectively absorb these disturbances, the resulting stress on vehicle components can be significantly increased.

In this study, FEA is employed to establish a baseline chassis model, with simulations conducted to analyse its natural vibrational modal response. Once the initial performance thresholds are established, identical constraints are applied to assess the vibrational response of the modified chassis featuring the optimized

---

[*]Corresponding author; *e-mail*: carmen.bujoreanu@academic.tuiasi.ro

structure. The results underscore the critical value of integrating CAD and FEA methodologies in vehicle design, demonstrating a substantial positive impact by reducing development time and enhancing the overall quality of the resulting prototypes.

Keywords: Topology optimisation, NVH, FEA, vehicle, chassis.

1. Introduction

Vehicle design has evolved significantly over time in multiple aspects. Beginning with computer-aided design (CAD) and design validation using finite element analysis (FEA) and extending to the adoption of 3D printing technologies and modern manufacturing methods, fundamental changes have been introduced in the approach to design processes. Additionally, increased attention has been devoted to reducing research and development (R&D) costs in the design phase - an essential aspect, given its direct influence on the costs associated with design, manufacturing, and ultimately, the final price of the vehicle. The investigation in vehicle design of the phenomena of NVH, which is the acronym for Noise Vibration and Harshness, is a field that has attracted significant attention in the last years. The evolution of vehicle performance, comfort and reliability has led to a more important role of understanding and mitigating the effects of the negative aspects of the subject of NVH in vehicle use and design.

Understanding how a structure and assembly manifest and react to different outside and inside vibration and noise generators has become a leading direction for modern day research and development engineers that are working on vehicles. From the contact of the tire to the road, to the noise generated by seatbelt engagement, all noises in the modern-day vehicle undergo extensive investigation actions from the design engineers.

The classic methods of designing and improving a vehicle structure from the NVH aspect was to conduct a multitude of tests, but these were done after the vehicle itself was being built up as a prototype. In some cases, engineers were adding special insulating padding where they identify sensitive areas contributing to an increase of NVH phenomenon in the vehicle. Conducting investigations after the prototype had been built resulted in significant costs for vehicle manufacturers due to increased design lead time, extended testing and implementation periods, and the additional expenses associated with incorporating new materials to address identified issues. Usually, sources of noise in the vehicle are the motor system (Deryabin, 2020), exhaust system and drivetrain, but also the contact between road surface and the tyre are generating significant amount of NVH in the vehicle. Studies are investigating on how to mitigate this kind of effect on the vehicle (Deryabin, 2023).

The use of Computer-Aided Design (CAD) software and tools has allowed design teams working on vehicle components and complete vehicles to evaluate, from the early stages of development, how material properties, design choices, and component interactions influence the vehicle's NVH performance. This early insight enables informed decision-making before prototype production, thereby reducing overall research and development costs. From the use of CAD and FEA and understanding the testing requirements of a vehicle from NVH point of view, the Design of Experiments (DOE) methods can be applied to increase the quality of results from a from a specific analysis (Azadi, *et al.*, 2009).

2. NVH in the Vehicle

The phenomena of N.V.H. in the vehicle is analysed looking directly at the main contributors (Hardik, 2024) that come into play to define it. These are:

– Noise – the propagation of sound waves that are susceptible of being heard by human. This represents the most influential factor that may discourage a particular user. Sounds such as motor noise, car door closing, seatbelt buckle or working Heating, Ventilating and Air Conditioning (HVAC) system from a vehicle.

– Vibration – the oscillation that occurs and can be felt inside the vehicle, depending on the speed and source, they can occur in the steering wheel or the floor of the vehicle up to the seats and passengers.

– Harshness – is the correlation between noise and vibration and the influence to a certain individual, by knowing the references of human tolerance this is giving the limits for the other components that interact in the NVH investigation.

Depending on the source of the noise and vibration, several clusters can be identified, as we see in Fig. 1 (Bruel and Kjaer, 2025). These sources vary from aerodynamic source, mechanical source or electrical source, each of them appearing at different location in the vehicle and at certain operating speeds, depending on the system being analysed. The interior of the vehicle will receive certain amounts of noise and vibration from each individual source. The main interest is to reduce to a minimum the discomfort to the passengers and to reduce or eliminate the influence of the vibrations on the wear of the vehicle systems that are sensible to high frequency oscillations that may appear due to the source. In the image we can observe all the frequency that can interact. For this paper we only analysed the influence up to 200 Hz, meaning we did not pass in the audio spectrum of the vibration.

The classic solution for reducing the noise in the vehicle was to insulate the vehicle with a special padding material inside the big panels that were prone to vibrate and generate a noise discomfort. More investigations were done also by introducing a special foam material inside the vehicle structure.

Testing of the vehicle for NVH performance is done in a variety of methods which can apply on the entire vehicle assembly or on the individual subassemblies or even only at component level, depending on the scope.

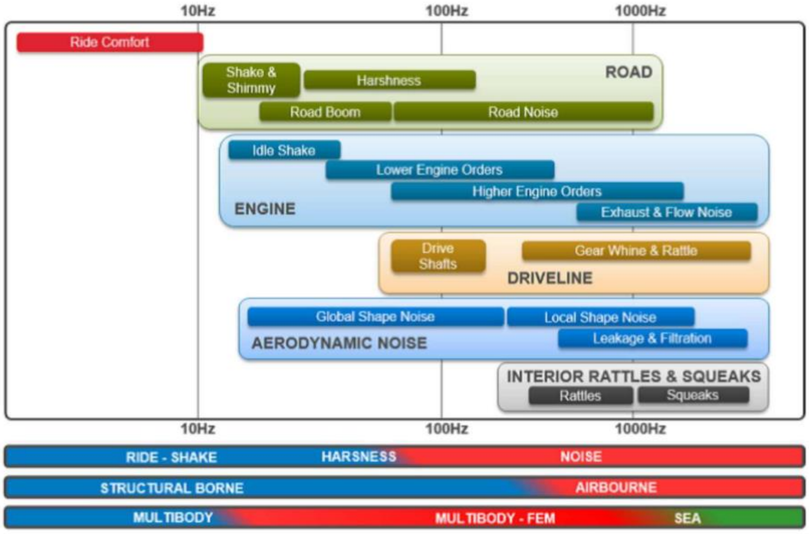


Fig. 1 – Different sources of noises and their correlated frequency.

When analysing the vehicle, attention is being given to the modal response (Fig. 2), vehicle vibration, noise (squeak and rattle), operational modal analysis, endurance test (Bruel and Kjaer, 2025). All the above should assure the design team that the response of the vehicle to certain vibration at certain frequency will satisfy the requirements. Special equipment is used to determine the modal response of the structure, from a simple vibration hammer to an entire vehicle structure vibration test apparatus. Special microphones and vibration sensors are usually mounted on the analysed product or vehicle and then is subjected to vibration, hammer hit or running condition, to collect data as a response from the system itself.

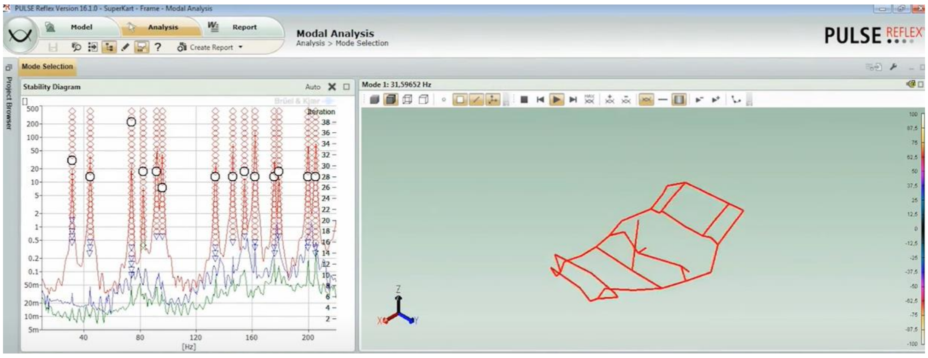


Fig. 2 – Modal response of cart chassis in a software test environment.

Using the Computer Aided Design method, the Finite Element Method and the Design of Experiment method, a configuration can be set up in order to subject the 3D model of the vehicle, system or component to the same frequency as in the test situation and observe the reaction of the tested system.

The current methods can depict a very precise result, almost like in the real situations. Differences can still be present due to small influences that cannot be totally controlled in real production and build up.

3. Modal Analysis on a Formula Student Chassis

Considering the former Formula Student Chassis from the Technical University from Iaşi (Fig. 3), a 3D model was created to replicate directly the real vehicle chassis.

The intent is to use a real vehicle for reference and consider this for the future investigations regarding DOE and test. The 3D model was refined and repaired for the correct simulation. Initial model had a lot of corrupt data, for the FEA set-up it is very important to have a closed model without any gaps and sharp edges, otherwise the simulation was corrupted.

Fig. 3 – Formula student kart from that was used as reference.

The used model was first modelled as a single component realised from multiple parts which replicated the steel pipes that are being used in the real vehicle (Fig. 4).

After the model of the vehicle was designed, another model of a special insert was created. This model is created as a honeycomb structure that fits inside the steel tubing of the chassis, an assembly of the steel tube chassis and the honeycomb inserts was created (Fig. 5) to the FEA analysis. The honeycomb inserts were mounted in the centre of the pipes around the driver compartment, they are not in the entire length of the tube section (Fig. 6). Dimensions considered for the honeycomb inserts are shown in (Fig. 7).

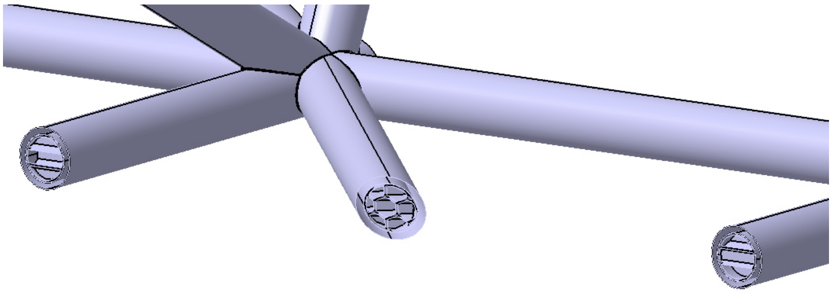


Fig. 4 – Formula student vehicle chassis - 3D model.

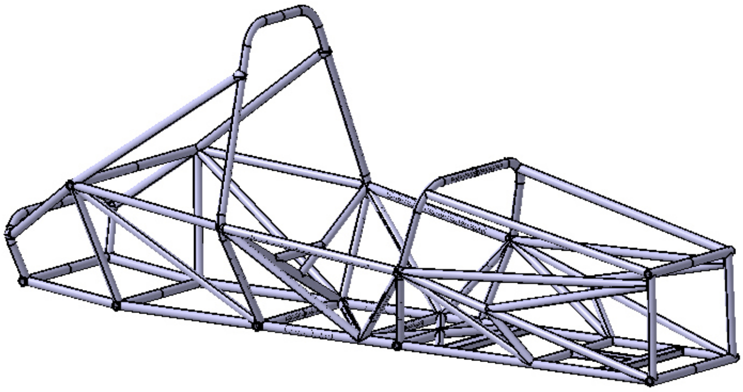


Fig. 5 – Honeycomb inserts assembled in the 3D of the tubular chassis.

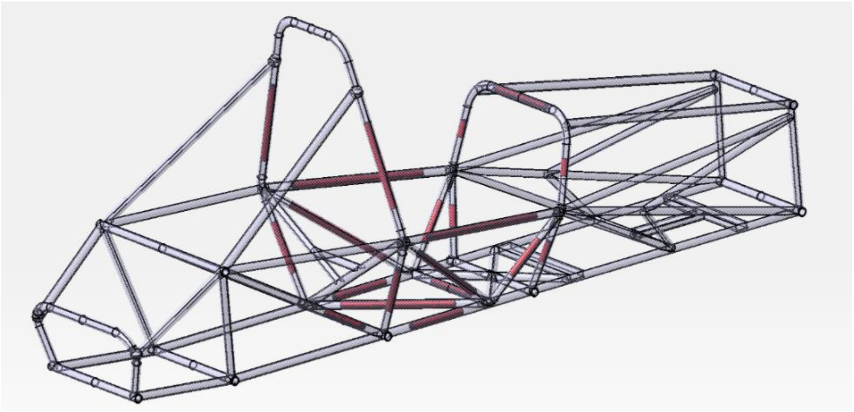


Fig. 6 – Honeycomb inserts considered only around driver area – shown with red.

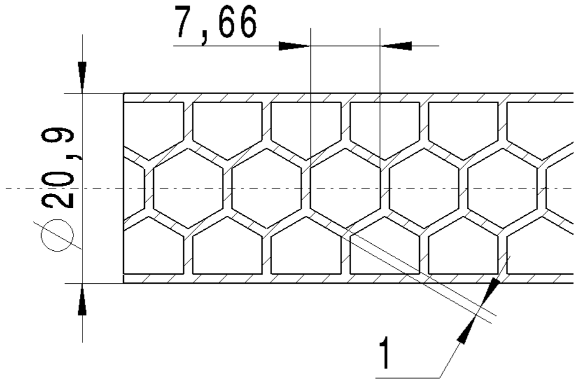


Fig. 7 – Honeycomb inserts dimensions considered for first investigation.

For the FEA analysis, different materials were considered for the vehicle tubular chassis as well as for the honeycomb inserts.

The materials considered are the ones which are available for the Formula Student team to use in the construction of the frame at a low cost. The honeycomb inserts were all considered as PBT material for all the simulations. The entire breakdown of considered materials and models used for the simulation is presented in Table 1.

Table 1

Materials considered in the simulation

Simulation No.	Material Considered for Frame	Insert yes/no	Chassis weight (kg)
1	*Steel*	*No*	*47.3*
2	*Steel*	*Yes*	*48.05*
3	*AlMgS0.5*	*No*	*16.2*
4	*PBT*	*No*	*9.2*

The simulation set-up was considered to be as close as possible to the source of principal vibration, these being the structure tubes closest to the wheel connections. All the fixing points for the input are presented in Fig. 8. On the front side there was a structure (the T one) and on the rear, the lowest connection tube on each side was considered. The location of the fixing points are considered with 0 degrees of freedom.

The harmonic response analysis has considered a 10G frequency range, between 0 – 200 Hz with excitation on the vertical direction. Frequency response of the system was considered on the top part of the chassis, the location where the steering wheel will be present and the driver will feel the entire vibrations transferred through the chassis (Fig. 9).

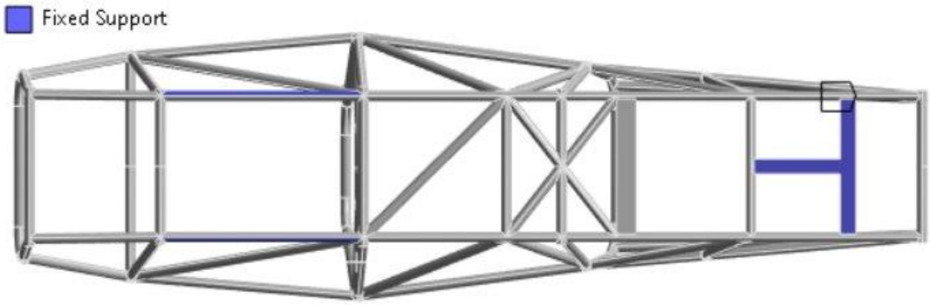


Fig. 8 – Fixed support considered for the FEA study.

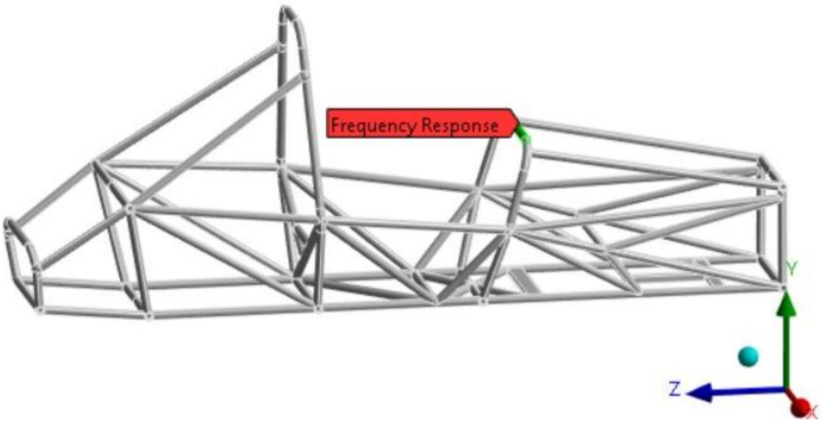
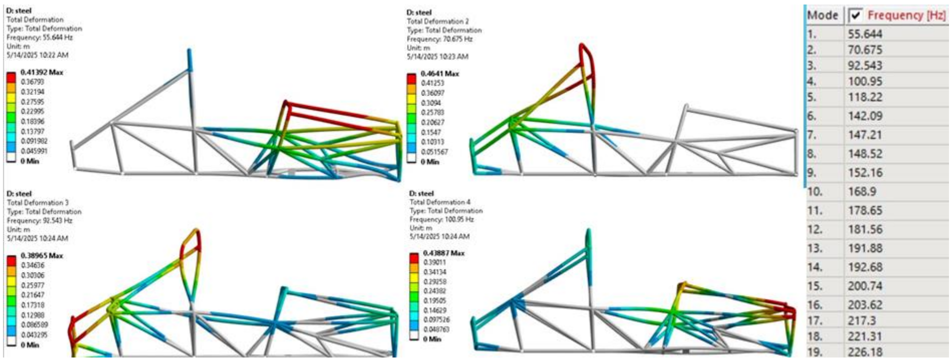


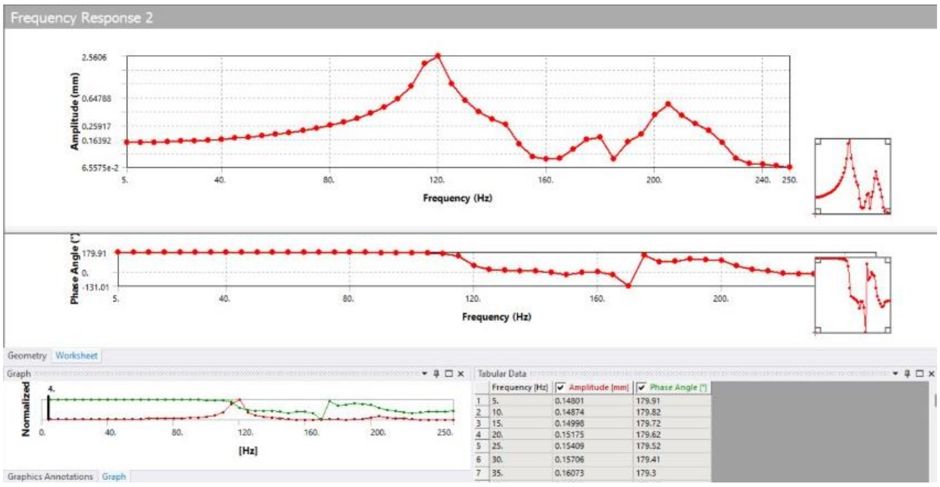
Fig. 9 – Considered frequency response.

Simulation 1.



Mode	☑ Frequency [Hz]
1.	55.644
2.	70.675
3.	92.543
4.	100.95
5.	118.22
6.	142.09
7.	147.21
8.	148.52
9.	152.16
10.	168.9
11.	178.65
12.	181.56
13.	191.88
14.	192.68
15.	200.74
16.	203.62
17.	217.3
18.	221.31
19.	226.18

Fig. 10 – Simulation model 1 displacement.

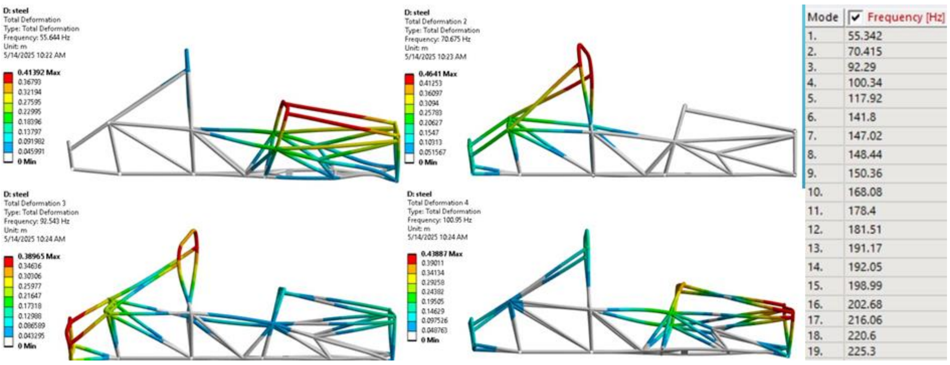


	Frequency [Hz]	☑ Amplitude [mm]	☑ Phase Angle [°]
1	5.	0.14801	179.91
2	10.	0.14874	179.82
3	15.	0.14998	179.72
4	20.	0.15175	179.62
5	25.	0.15409	179.52
6	30.	0.15706	179.41
7	35.	0.16073	179.3

Fig. 11 – Simulation model 1 modal response.

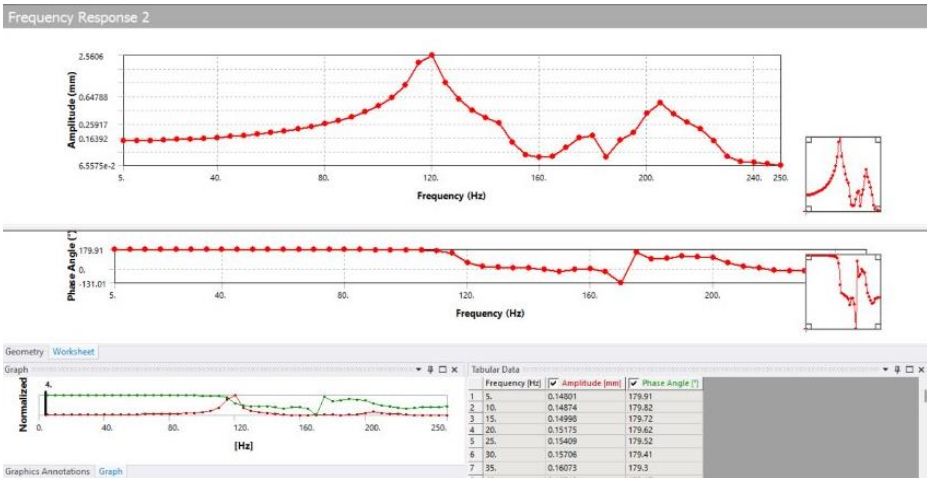
The simulation 1 uses the model 1 representing the normal chassis with no inserts and steel material, as the original vehicle is constructed. The displacement of the chassis can be seen in Fig. 10, the model response in Fig. 11.

Simulation 2. The model 2 for this simulation represents the normal chassis with the honeycomb inserts inside it, displacement (Fig. 12), seems to be similar with small deviations in the modal response (Fig. 13).



Mode	☑ Frequency [Hz]
1.	55.342
2.	70.415
3.	92.29
4.	100.34
5.	117.92
6.	141.8
7.	147.02
8.	148.44
9.	150.36
10.	168.08
11.	178.4
12.	181.51
13.	191.17
14.	192.05
15.	198.99
16.	202.68
17.	216.06
18.	220.6
19.	225.3

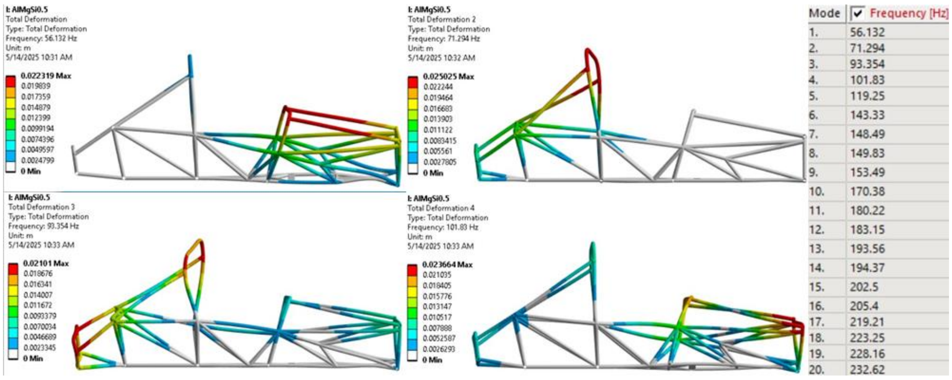
Fig. 12 – Simulation model 2 displacement.



	Frequency [Hz]	☑ Amplitude [mm]	☑ Phase Angle [°]
1	5.	0.14801	179.91
2	10.	0.14874	179.82
3	15.	0.14998	179.72
4	20.	0.15175	179.62
5	25.	0.15409	179.52
6	30.	0.15706	179.41
7	35.	0.16073	179.3

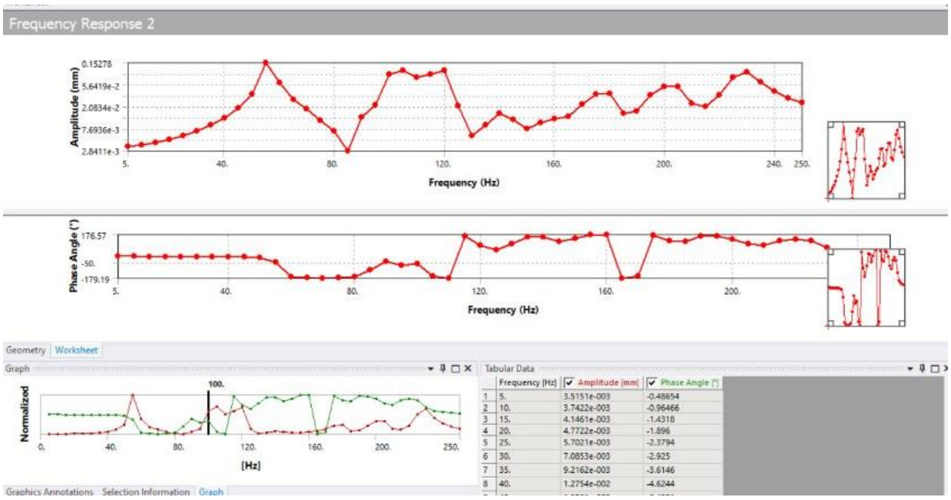
Fig. 13 – Simulation model 2 modal response.

Simulation 3. This simulation (on model 3) represents the chassis with no inserts and aluminium material. There is a big difference in weight, but the displacement of the chassis can be seen in Fig. 14, the model response in Fig. 15, with small deviation from the original steel one.



Mode	☑ Frequency [Hz]
1.	56.132
2.	71.294
3.	93.354
4.	101.83
5.	119.25
6.	143.33
7.	148.49
8.	149.83
9.	153.49
10.	170.38
11.	180.22
12.	183.15
13.	193.56
14.	194.37
15.	202.5
16.	205.4
17.	219.21
18.	223.25
19.	228.16
20.	232.62

Fig. 14 – Simulation model 3 displacement.

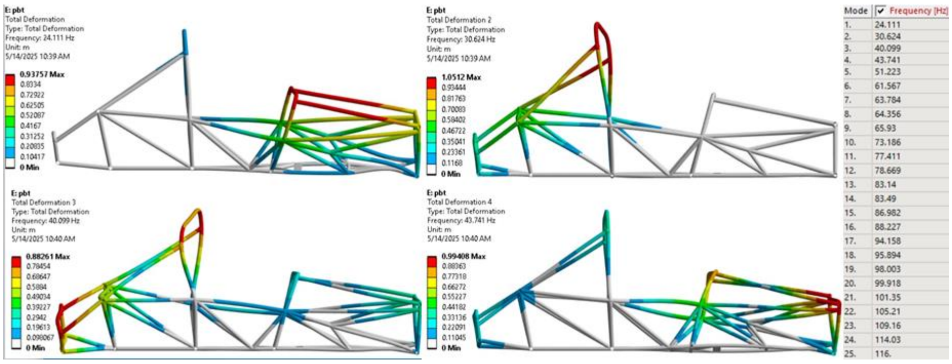


Tabular Data

	Frequency [Hz]	☑ Amplitude [mm]	☑ Phase Angle [°]
1	5.	3.5151e-003	-0.48654
2	10.	3.7422e-003	-0.96466
3	15.	4.1461e-003	-1.4318
4	20.	4.7722e-003	-1.896
5	25.	5.7021e-003	-2.3794
6	30.	7.0853e-003	-2.925
7	35.	9.2162e-003	-3.6146
8	40.	1.2754e-002	-4.6244

Fig. 15 – Simulation model 3 modal response.

Simulation 4. The model 4 from simulation represents the chassis made out of plastic (PBT) material and no honeycomb inserts inside it. Althgough there is a big difference in weight, the displacement of the material exhibits same reactions but with lower values. It can be seen in Fig. 16, the model response in Fig. 17, with high deviation from the original steel and aluminum solution.


E: pbt
Total Deformation
Type: Total Deformation
Frequency: 24.111 Hz
Unit: m
5/14/2025 10:39 AM

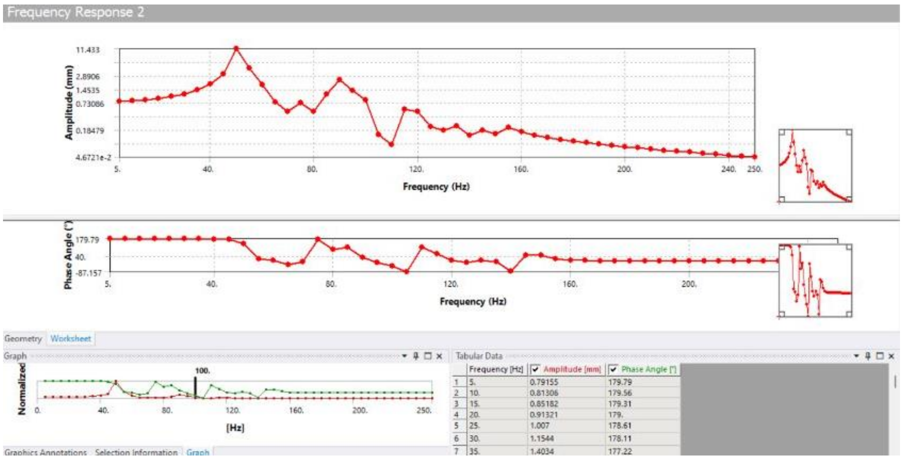
E: pbt
Total Deformation 2
Type: Total Deformation
Frequency: 30.624 Hz
Unit: m
5/14/2025 10:39 AM

E: pbt
Total Deformation 3
Type: Total Deformation
Frequency: 40.099 Hz
Unit: m
5/14/2025 10:40 AM

E: pbt
Total Deformation 4
Type: Total Deformation
Frequency: 43.741 Hz
Unit: m
5/14/2025 10:40 AM

Mode	☑ Frequency [Hz]
1.	24.111
2.	30.624
3.	40.099
4.	43.741
5.	51.223
6.	61.567
7.	63.784
8.	64.356
9.	65.93
10.	73.186
11.	77.411
12.	78.669
13.	83.14
14.	83.49
15.	86.982
16.	88.227
17.	94.158
18.	95.894
19.	98.003
20.	99.918
21.	101.35
22.	105.21
23.	109.16
24.	114.03
25.	116.

Fig. 16 – Simulation model 4 displacement.



	Frequency [Hz]	☑ Amplitude [mm]	☑ Phase Angle [°]
1	5.	0.79155	179.79
2	10.	0.81306	179.56
3	15.	0.85182	179.31
4	20.	0.91321	179.
5	25.	1.007	178.61
6	30.	1.1544	178.11
7	35.	1.4034	177.22

Fig. 17 – Simulation model 4 modal response.

4. Conclusions

Although small deviations from the initial values are observed, it can be concluded that the modal analysis of the steel chassis shows small decreases in the values obtained after considering the plastic inserts. Probably, the differences are given by the mass of the inserts themselves. The modal response in terms of deformation between the three materials presents the same behaviour of the analysed structure. The frequencies present differences between the metallic materials and the plastic one considered. The response frequencies between steel and aluminium are very similar. The PBT material presents lower natural frequencies, this shows a high potential for reducing the natural response of the vehicle and an improvement in NVH properties. Future investigation will consider to increase the frequency and observe what changes occur when entering the audio spectrum values.

REFERENCES

Azadi S., Azadi M., Zahedi F., *NVH Analysis and Improvement of a Vehicle Body Structure Using DOE Method*, Journal of Mechanical Science and Technology 23(11):2980-2989, 2009, DOI:10.1007/s12206-009-0909-z.

Bruel & Kjaer, *Automotive NVH testing on a SuperKart*, https://www.youtube.com/watch?v=XwcSio0RDlE, last accessed 2025/03/25.

Deryabin I., *On the issue of reducing the sound level emitted by the intake system of an internal combustion engine*, Journal of Physics: Conference Series 1679 052026, 2020, doi:10.1088/1742-6596/1679/5/052026.

Deryabin I., *Noise Reduction Method in Vehicle Interior*, Transportation Research Procedia: Volume 68, 2023, Pages 642-646, Online ISSN: 2352-1465.

Hardik S., *What is Automotive NVH*, https://www.ansys.com/blog/what-is-automotive-nvh, last accessed 2024/12/09.

INVESTIGAȚIA NVH A ȘASIULUI VEHICULULUI

(Rezumat)

Aceast studiu prezintă o metodă de îmbunătățire a proiectării vehiculului prin integrarea optimizării topologice bazate pe CAD și simularea analizei prin elemente finite (FEA). În mod specific, șasiul vehiculului este rafinat prin încorporarea elementelor de design inspirate de natură (biomorf), care sunt integrate pentru a îmbunătăți performanța NVH (Zgomot, Vibrații, Duritate) a vehiculului. În acest studiu, analiza cu element finit este utilizată pentru a stabili un model de bază al șasiului, cu simulări efectuate pentru a analiza răspunsul său modal vibrațional natural. Odată ce pragurile inițiale de performanță sunt stabilite, constrângerile identificate sunt aplicate pentru a evalua răspunsul vibrațional al șasiului modificat, care prezintă structura optimizată. Rezultatele subliniază valoarea critică a integrării metodologiilor CAD și FEA în proiectarea vehiculului, demonstrând un impact substantial și pozitiv prin reducerea timpului de dezvoltare și îmbunătățirea calității generale a prototipurilor rezultate.