SIMULATION OF CRASH AND DURABILITY ANALYSIS OF HEV TRUCK CHASSIS

MÔ PHỎNG VA CHẠM VÀ PHÂN TÍCH ĐỘ BỀN KHUNG XE TẢI HEV

Van Phuc Nguyen[1,2], Minh Duc Le[3]*, Dang Truong Nguyen[3], Minh Duc Nguyen[3]

[1]*PhD student in Transport Mechanical Engineering, UD - University of Science and Technology, Vietnam*
[2]*THACO College, Vietnam*
[3]*The University of Danang - University of Science and Technology, Vietnam*

*Corresponding author: minhducle@dut.udn.vn

(Received: April 14, 2025; Revised: June 14, 2025; Accepted: August 11, 2025)

DOI: 10.31130/ud-jst.2025.23(9A).208

Abstract – The article presents the process of simulating the crash and frame durability of a series hybrid electric vehicle (HEV) based on the Kia Frontier K200 light truck. A HEV model was built using Autodesk Inventor software, and the frame durability analysis was performed in a Stress Analysis environment under both static and maximum load conditions. By using the finite element analysis (FEA), parameters such as stress, displacement, and deformation of the HEV chassis were determined. The crash simulation of the frame was examined using the Radioss module in HyperWorks, with the frame fixed and the vehicle set at a velocity of 50 km/h. The results indicate that the frame meets durability requirements, with stress concentrated at load-bearing points, while the displacement does not significantly alter the shape of the chassis. These parameters are used as a basis for evaluating and improving the component layout to enhance durability, safety, and efficiency.

Key words – Hybrid truck; truck chassis; durability analysis simulation; crash simulation; Finite Element Analysis; battery pack.

Tóm tắt - Bài báo trình bày mô phỏng va chạm và độ bền khung của xe hybrid electric vehicle (HEV) kiểu nối tiếp được thiết kế dựa trên cơ sở xe tải nhẹ Kia Frontier K200. Mô hình HEV được xây dựng bằng phần mềm Autodesk Inventor và phân tích độ bền khung được thực hiện bởi Stress Analysis dưới điều kiện tải trọng tĩnh và tải trọng tối đa. Sử dụng phương pháp phân tích phần tử hữu hạn (FEA) để xác định ứng suất, chuyển vị và biến dạng của khung xe tải HEV. Mô phỏng va chạm của khung xe được phân tích bằng mô-đun Radioss trong Hyperwork, với giả thiết cố định khung và thiết lập vận tốc xe giả lập khi va chạm tại 50 km/h. Kết quả cho thấy, khung xe đáp ứng yêu cầu về độ bền, ứng suất tập trung tại các điểm chịu tải lớn, trong khi chuyển vị không làm thay đổi đáng kể hình dạng khung xe. Các thông số này được sử dụng làm cơ sở để đánh giá và cải tiến bố trí các chi tiết, nhằm tăng cường độ bền, độ an toàn và hiệu quả.

Từ khóa – Xe tải Hybrid; khung xe tải; mô phỏng bền; mô phỏng va chạm; phần tử hữu hạn; Pack pin.

1. Introduction

In the context of the current automotive industry shifting towards sustainable development and prioritizing clean energy solutions, research into converting traditional vehicles to hybrid power sources has become an inevitable requirement for progress [1-2]. Evaluating the durability of the Hybrid Electric Vehicle (HEV) chassis is therefore essential. Particularly in the light truck segment, there is a high demand for load-bearing capacity and structural durability. Thus, simulating safety and crash analysis of the HEV chassis based on the Kia Frontier light truck platform is proposed as a solution to build accurate computational models for the conversion of conventional trucks to hybrid trucks. These models not only facilitate comprehensive evaluation of influencing factors but also optimize the structural design of the chassis.

Previous studies have shown that durability analysis techniques are employed to optimize vehicle design processes. Research related to truck chassis mainly focuses on two directions: structural optimization and new material modeling. Both approaches have proven to be technically and economically effective in truck chassis design and manufacturing. For HEV/EVs, some notable studies include: evaluating the structural performance of electric vehicle chassis under static and dynamic loads [3]; optimizing the size and weight of EV batteries and thermal management systems [4]; structural optimization and development of a fully aluminum EV chassis platform with crashworthiness considerations [5]; and reliability optimization and analysis of multifunctional electric truck chassis under varying load positions [6]. In general, these studies focus on reducing the mass of HEV/EV chassis to optimize energy management for these vehicle types.

Applying CAE techniques combined with FEM solutions to optimize and analyze the structural durability of automotive chassis can be effectively addressed (stress, strain, displacement). This affirms that numerical analysis and simulation techniques are forecasted to bring orderly advancements in today's automotive design and manufacturing.

By applying finite element analysis (FEA) methods combined with advanced linear solutions, it is possible to visualize regions at risk of structural issues, thereby proposing design improvements to enhance operational efficiency, ensure maximum user safety, and extend vehicle lifespan. Crashworthiness and safety thresholds for HEV/EV chassis and battery packs are always critical factors in the design of these vehicles. Unlike previous studies that only focused on reducing chassis mass and energy management for HEV/EVs, this research applies finite element analysis (FEA) to analyze durability and simulate crash standards for an HEV chassis model designed based on the Kia Frontier K200 light truck. The research results not only open new directions for designers in developing light-duty HEV trucks but also provide a solid scientific basis to support strategic business decisions amid increasingly stringent market conditions and the prioritization of environmental protection [8-9].

2. Theoretical background

2.1. Linear problems

In structural analysis, the static linear problem plays a key role in assessing the load-bearing capacity and verifying the durability of engineering structures, especially in applications related to vehicle chassis, bridges, and modern constructions. Therefore, a clear understanding of the theoretical basis and accurate application of this method is crucial for ensuring safety and efficiency in design. The static linear method is built on the assumption that the relationship between stress and strain in materials is linear, represented by Hooke's law with the formula $\sigma = \varepsilon . E$ [9], where σ is the stress representing internal force per unit area, ε is the relative strain indicating the degree of dimensional change under load, and E is the modulus of elasticity, a material-specific constant representing stiffness and load-bearing capacity. The stress-strain graph in this case is a straight line through the origin, following the equation y = a.x, where a is the proportional coefficient, or tanα = σ/E, thus showing the direct proportionality between fundamental quantities when the material operates within the elastic range.

Durability analysis of the chassis using FEA with linear solutions is based on subdividing the entire chassis into finite elements, each described by specific material and geometric properties. The system of equations is constructed as $K = v.F$, where K is the stiffness matrix reflecting the load-bearing capacity of the chassis, v is the deformation vector, and F is the load vector acting on the structure. Simultaneously, the general dynamic equation [9-11] in static durability simulation is expressed as:

$$M.\ddot{v} + C.\dot{v} + K.v = F \quad (1)$$

where, M is the mass matrix, C is the damping matrix, and K is the stiffness matrix. In static analysis, the components related to acceleration $\ddot{v}$ and velocity $\dot{v}$ are omitted, yielding the fundamental equation $K = v.F$. The FEA simulation results provide an overview of the chassis behavior under load, helping to identify regions at risk of structural failure and supporting design improvements to enhance safety and usability. In material mechanics, the stress-strain-displacement equations are fundamental for describing and predicting material changes under loading. The basic relationships include:

Relationship between displacement and strain:

The strain component ε_{ij} is determined from displacement u by the formula:

$$\varepsilon_{\text{ij}} = \frac{1}{2}\left(\frac{\partial v_i}{\partial x_j} + \frac{\partial v_j}{\partial x_i}\right) \quad (2)$$

where, v_i is the displacement component in the x_i direction.

Stress-strain law (for linear elastic materials):

For homogeneous materials, Hooke's law describes the relationship between stress σ_{ij} and strain ε_{ij}

$$\sigma_{\text{ij}} = \lambda\delta_{\text{ij}}.\varepsilon_{kk} + 2\mu\varepsilon_{\text{ij}} \quad (3)$$

Here, λ and μ are Lamé constants, δ_{ij} is the Kronecker delta, and ε_{kk} is the trace of the strain tensor (sum of diagonal components).

Force equilibrium equation (static state):

The internal force equilibrium condition of the system is expressed as:

$$\frac{\partial \sigma_{\text{ij}}}{\partial x_j} + f_i = 0 \quad (4)$$

where, f_i is the component of the body force acting on the material. These equations collectively form the theoretical basis for analyzing structural behavior in FEA problems, thereby assessing strain and stress in materials under various loads.

2.2. Nonlinear problems

Nonlinear problems in crash simulation are a complex type of finite element analysis (FEA) used to simulate the behavior of structures (such as the chassis or battery pack) under strong impacts, where the relationship between load and system response (such as deformation, stress) does not follow linear rules, thus requiring iterative computational methods.

In crash simulation, nonlinear analysis arises when factors such as material, geometry, contact, and boundary conditions no longer adhere to linear assumptions (i.e., responses are not proportional to loads). The mathematical relationship of the nonlinear problem is $F = f(x)$, where $f(x)$ is a complex nonlinear function.

The stiffness matrix [K] in the FEA equation [K]{u} = {F} is no longer fixed but varies with displacement {u}. Crash simulations (such as chassis impacting a rigid wall or protecting battery packs in EVs) commonly encounter the following nonlinear factors:

Material Nonlinearity: The material exceeds the elastic region, entering plasticity or failure. The stress-strain relationship ($\sigma = f(\varepsilon)$) is no longer linear, requiring nonlinear material models such as Johnson-Cook or plasticity models.

Geometric Nonlinearity: Large deformations alter the structure's geometry, affecting force distribution. Geometry must be updated at each calculation step (large deformation analysis).

Contact Nonlinearity: Contact between parts changes during impact. Contact forces and boundary conditions vary, necessitating dynamic contact definitions in HyperMesh.

Boundary Condition Nonlinearity: Loads and constraints change over time or with system response. Load curves are used to describe load variation over time.

Dynamic Nonlinearity: Crash is a dynamic problem, with rapidly changing forces and accelerations.

Stress and strain in nonlinear problems

In crash simulation, especially in nonlinear problems, the relationship between stress and strain becomes complex due to the combination of deformation, nonlinear material behavior, and high strain rates (dynamic effects).

Stress in crash problems is often represented as the

Cauchy tensor, accounting for dynamic effects:

$$\sigma = \frac{F}{A} \tag{5}$$

where, F: Impact force (calculated from momentum or acceleration); A: Cross-sectional area (may change due to large deformation).

Strains in crash problems include elastic, plastic, and sometimes failure strains. Due to nonlinear properties, large strain tensors are typically used.

Effective Plastic Strain:

$$\varepsilon_p = \int \sqrt{\frac{2}{3} d\varepsilon_{ij}^p} \tag{6}$$

where, $d\varepsilon_{ij}^p$: Incremental plastic strain component.

In crashes, strain rate ($\dot{\varepsilon}$) greatly affects stress, especially for rate-sensitive materials (such as steel, aluminum):

Johnson-Cook model:

$$\sigma = \left[A + B(e_p)^n\right]\left(1 + C\ln\left(\frac{\dot{\varepsilon}}{\dot{\varepsilon}_0}\right)\right)(1 - T^{*m}) \tag{7}$$

where, A: Initial yield stress; B, n: Hardening constants; C: Strain rate sensitivity constant; $\dot{\varepsilon}_0$: Reference strain rate.

$T^* = \frac{T - T_0}{T_m - T_0}$: Normalized temperature (accounts for thermal effects in crashes).

3. Model setup

3.1. Material properties

The truck chassis is the component that bears the entire vehicle load, thus requiring high strength and rigidity to ensure safety and operational efficiency under harsh conditions. Additionally, the chassis design must be optimized for compactness to save space, and load and production costs must be carefully considered for economic viability. Based on these requirements, the selected materials for the designed vehicle are as follows:

- Chassis: AISI 317 steel (base chassis material);
- Mounting brackets: SS400 steel;
- Battery pack housing: 304 stainless steel.

3.2. HEV vehicle specifications

Table 1. *HEV truck specifications [12]*

No.	Parameter	Value
1	Allowable load (kg)	1900
2	Overall vehicle dimensions (mm)	5100x1750x2000
3	Wheelbase of the vehicle (mm)	2615
4	Ground clearance (mm)	185
5	Maximum power (kW)	80
6	Maximum torque (Nm)	240

Based on calculation results and the design process for HEV truck components derived from the Kia Frontier K200 light truck platform [12], this study has developed an overall architectural layout and technical specifications for the vehicle (Figure 1 and Table 1). In the designed HEV truck model, the battery pack is positioned at the rear of the chassis, encased in a protective metal box. The selected battery type for the HEV is Lithium-ion. The calculation process includes data analysis, technical evaluation, and application of road traffic regulations to optimize performance, durability, and safety under the support of FEA tools [13].

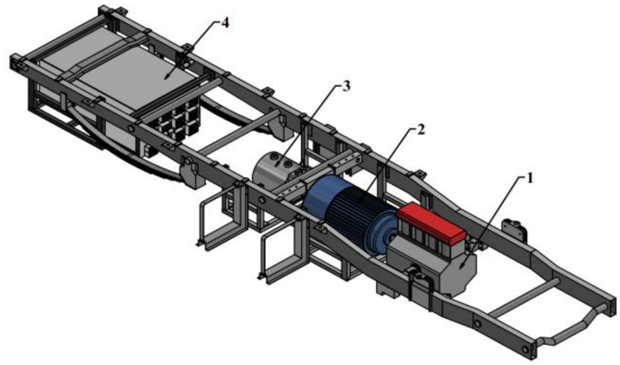

Figure 1. *Chassis of the HEV truck model*
1. ICE; 2. Generator; 3. Electric Motor; 4. Battery pack

The fundamental differences between the Kia Frontier K200 chassis and the HEV chassis are the additional mounting brackets for the electric motor, generator, and battery pack.

4. Chassis durability simulation using FEA

4.1. Boundary conditions

4.1.1. Fixed support boundary conditions

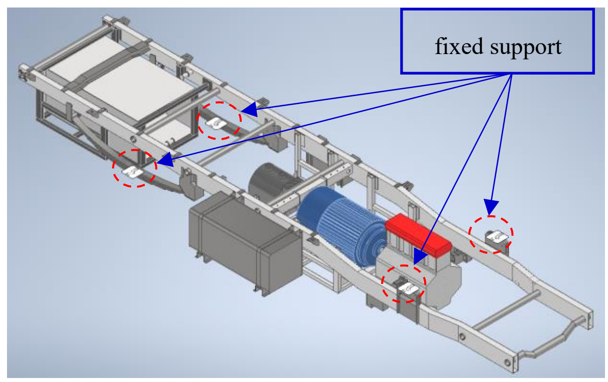

Figure 2. *Fixed support boundary conditions*

In chassis durability simulation, fixed supports are placed at the leaf spring and shock absorber mounts to accurately simulate the constraints between the chassis and the suspension system (Figure 2). These points bear significant loads and transmit forces from the chassis to the suspension. Applying fixed supports helps limit unwanted movement, ensures the model reflects real-world conditions, supports accurate stress and strain analysis, and thus optimizes the design to enhance durability and safety for the truck [14-15].

4.1.2. Load boundary conditions

In structural durability simulation, the static analysis method is applied to convert connected components into forces acting on the chassis [13]. Components directly mounted on the chassis are represented as forces, while elements such as the internal combustion engine, electric motor, generator, and battery pack, although not directly mounted, are represented by forces applied to their respective brackets (Figure 3). Each force vector is oriented vertically downward, with magnitudes varying depending on the load distribution characteristics of each component, ensuring accurate assessment of the chassis's load-bearing capacity [16-17]. Table 2 presents the load parameters set for the computational model.

Table 2. *Load parameters set up the HEV truck model*

No.	Part	Weight (N)
1	Engine	2900
2	Gear box	-
3	Tank	510
4	Battery	270
5	Spare tire	450
6	Full truck load	23000
7	Electric motor	650
8	Generator	2300
9	Pack Pin	1250

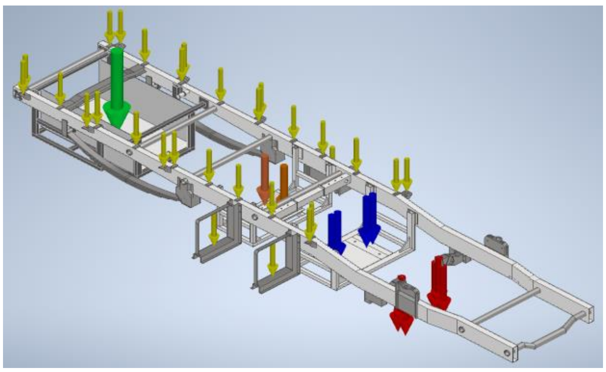

Figure 3. *Load boundary conditions*

4.1.3. Meshing boundary conditions

During meshing for FEA problems, meshing boundary conditions are the constraints and requirements applied at the boundaries of the computational domain to ensure the mesh is suitable for the geometry and physical characteristics of the problem. Mesh size is selected after performing convergence tests on computed values and must also ensure minimal solution time (Figure 4). The final chosen mesh parameters are:

- Minimum division: 0.01;
- Average division: 0.05;
- Mesh parameter: 1.5.

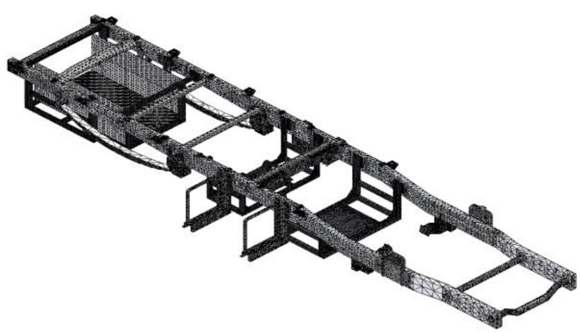

Figure 4. *HEV truck frame mesh*

4.2. Results and evaluation

4.2.1. Simulation results

a. Stress

Figure 5 shows the calculated stress distribution on the HEV truck chassis. The maximum measured stress is 295 MPa, located at the rear leaf spring mount. The primary cause is the battery pack being positioned at the end of the cargo area, significantly increasing the mass in this region. In other areas of the chassis, stresses are not excessively high, and the minimum stress occurs at the battery pack housing area. This result indicates that the current layout is reasonable, ensuring safety for the battery pack and contributing to the overall durability of the chassis.

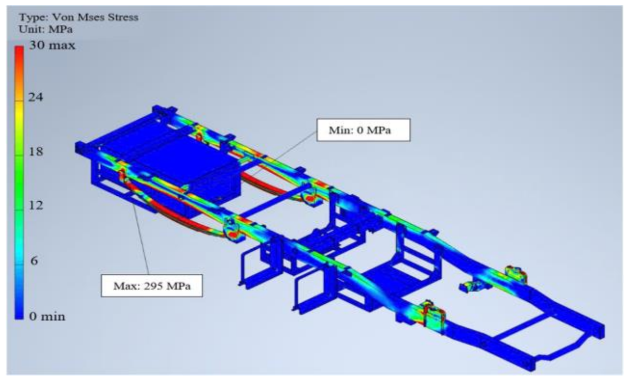

Figure 5. *Stress distribution diagram for the HEV truck chassis*

b. Displacement

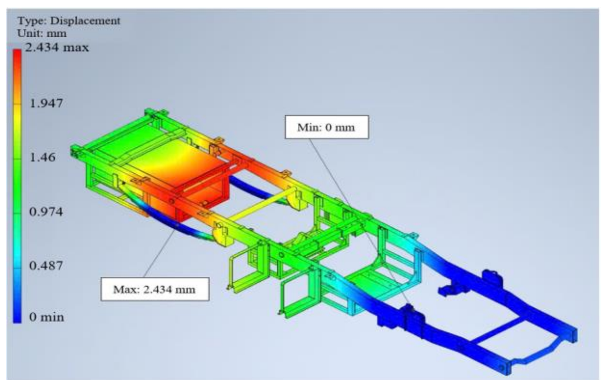

Figure 6. *Displacement distribution diagram for the HEV truck chassis*

Figure 6 shows the calculated displacement distribution on the HEV truck chassis. Displacements are concentrated mainly at locations with numerous chassis connections. The cargo area and battery pack housing, which have the largest masses, correspond to the highest displacement values (battery pack housing reaches 2.434 mm). In contrast, assemblies such as the generator and electric motor exhibit average displacements of about 1 mm, small enough to ensure accuracy and stability for rotating components.

4.2.2. Evaluation and discussion

The maximum stress value of 295 MPa at the rear leaf spring mount is lower than the material limit of 461 MPa, ensuring chassis durability at maximum load. Due to the truck's cargo-carrying nature, stress is concentrated at the rear leaf spring area. Additional brackets mounted on the original chassis have low stress values, indicating that the bracket connections are sufficiently strong and rigid when the truck is fully loaded.

Most of the cargo, battery pack, and electric motor mass is concentrated at the rear of the vehicle, which is why displacement in this area is highest. The relatively small displacement value of 2.432 mm ensures the chassis's rigidity.

5. Battery pack safety crash simulation

Positioning the battery pack at the rear of the HEV truck necessitates consideration of safety in the event of a crash. The HEV truck chassis structure is evaluated for side and rear-end durability during collisions at a speed of 50 km/h.

5.1.1. Crash simulation standards

Figure 7 shows the crash simulation model applying the ECE R95 side impact standard [18], specifically:

- Crash condition:

▪ The test vehicle is struck laterally by a simulated vehicle at the door;

▪ Impact object: A simulated vehicle (Mobile Deformable Barrier - MDB) with a mass of 950 kg;

▪ Impact speed: 50 [km/h];

▪ Impact angle: Perpendicular to the test vehicle;

▪ Object shape: Simulates the front of a vehicle, with a front energy-absorbing zone.

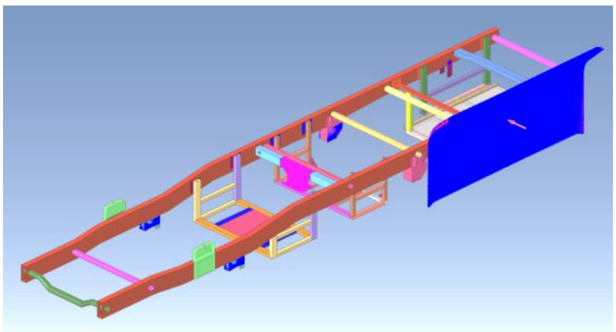

Figure 7. *Side impact crash simulation model for the HEV truck*

Figure 8 shows the crash simulation model applying the ECE R32 rear-end impact standard [19], specifically:

- Crash condition:

▪ The vehicle is tested for front or rear impact against a fixed barrier;

▪ Impact speed: Approximately 50 km/h;

▪ Test vehicle mass: Vehicle tested under nominal load conditions (may include simulated passengers).

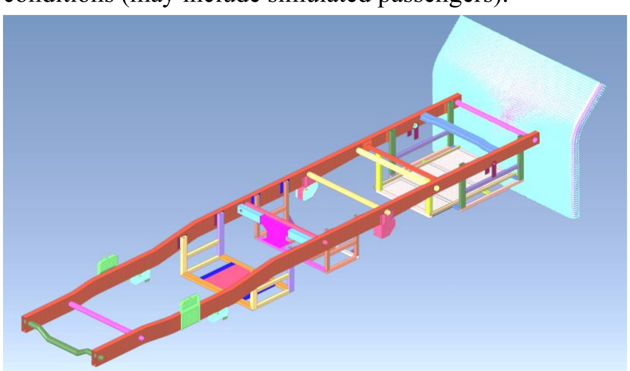

Figure 8. *Rear-end crash simulation model for the HEV truck*

5.2. Model setup for crash simulation analysis

5.2.1. Mô hình CAD

The CAD model of the chassis and the assumed front plane of the truck are designed using Inventor software (Figures 9 and 10). Inventor's Frame Generator feature allows rapid chassis design using a library of standard steel profiles.

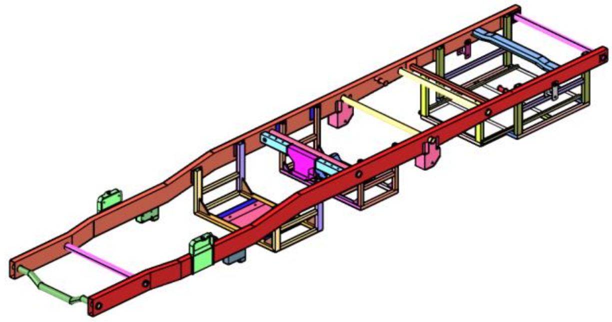

Figure 9. *CAD model of the HEV truck chassis*

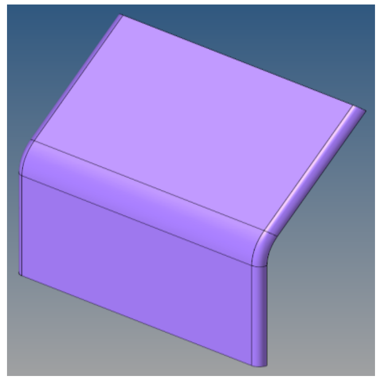

Figure 10. *Assumed front plane model of the truck created with Inventor software*

5.2.2. Finite element model

The finite element model of the chassis is built based on the CAD model using HyperMesh software [20]. The model is extracted in 2D for simulation to reduce complexity and computation time while maintaining analytical efficiency in FEA (Figures 11 and 12).

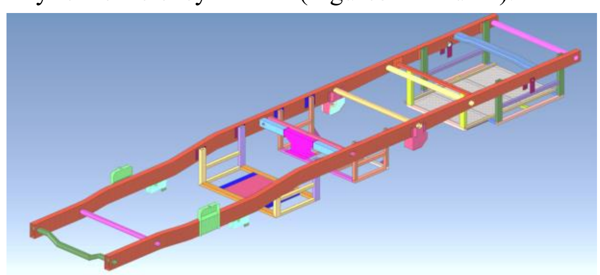

Figure 11. *Finite element model of the truck chassis after setup in HyperMesh*

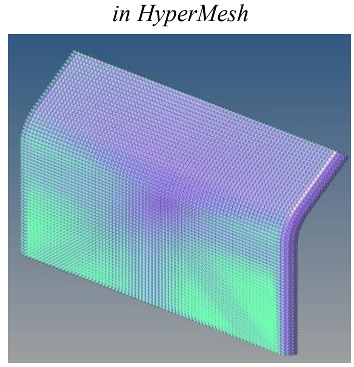

Figure 12. *Assumed front plane model of the truck after setup in HyperMesh*

Table 3. *Technical specifications of the chassis and assumed front plane*

Part	No. of elements	No. of Nodes
Chassis	90393	89624
Truck front	5142	5288

Table 4. *Material specifications for the entire model*

Part	Density (kg/m^3)	*E* (GPa)	Poission	Card Image
Chassis	7850	210	0.3	M36
Truck front	7850	210	0.3	M1

5.3. Boundary condition analysis in crash simulation

5.3.1. Meshing boundary conditions

To simulate battery pack safety in two crash scenarios (side and rear-end) in FEA, which involve large impact forces over short periods, the Automesh tool in HyperMesh is used to generate 2D meshes (shell elements). The complex geometry model is subdivided into smaller elements and nodes for thin plates, allowing the solution of mathematical equations describing the physical behavior of the structure (Figure 13) [21].

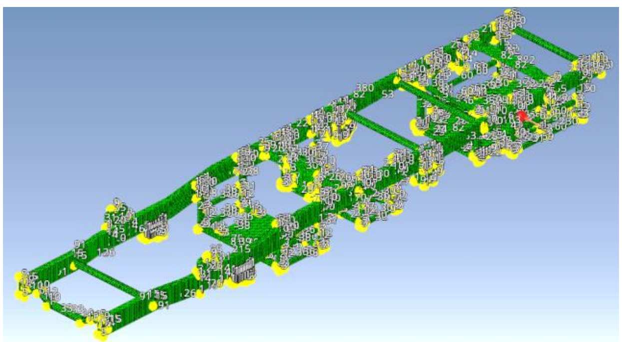

Figure 13. *Meshed model of the HEV truck chassis*

- Number of Nodes: 89624;
- Element size: 1 mm.

For the assumed truck model used to simulate chassis and battery pack crash, the mesh size is set similarly to the chassis to maximize and balance accuracy and computation time (Figure 14) [21].

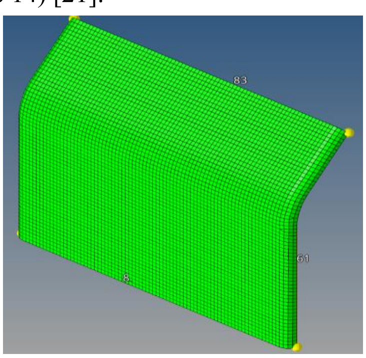

Figure 14. *Meshed model of the assumed truck front*

- Number of Nodes: 5142;
- Element size: 2 mm.

Mesh size statistics and material parameters for the chassis and assumed truck front are shown in Tables 3 and 4.

5.3.2. Fixed support boundary conditions

In a battery pack safety crash simulation, fixed supports are placed on shock absorber mounts, leaf spring mounts, and battery pack subframe beams. Fixed supports are placed at major load-bearing points to ensure the model complies with physical conditions (such as force and moment equilibrium) and technical standards (such as vehicle maximum load) and to identify regions of high stress or large deformation, thereby improving chassis design for crash scenarios. For crash simulation, fixing all six degrees of freedom at major load-bearing areas such as the battery pack location may require fixed supports to simulate anchoring when the vehicle is stationary, ensuring the model does not move freely when loads are applied. Placing fixed supports on parts of the chassis during crash simulation ensures anchoring during impact, thus enabling accurate stress and deformation analysis (Figure 15).

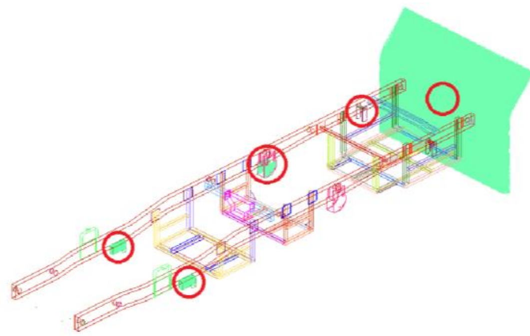

Figure 15. *Fixed supports placed on the chassis*

5.3.3. Load boundary conditions

In chassis crash simulation using HyperMesh, load boundary conditions play a crucial role in accurately reproducing structural behavior under strong impacts, typically including dynamic loads such as acceleration, initial velocity, concentrated impact force, and time-varying contact forces, characterized by nonlinearity due to plastic material deformation, large geometric changes, and unstable contact between surfaces. For battery pack safety crash simulation, focus is placed on loads applied to the chassis to adjust battery pack position, velocity, and model acceleration so that upon collision, the chassis maintains sufficient rigidity, and deformation does not affect the battery pack. Table 5 presents the load parameters set for the crash simulation model.

Table 5. *Load parameters for crash simulation*

Part	Weight (kg)	Velocity (km/h)	G (m/s^2)
Chassis	1000	0	9.81
Truck model	950	50	9.81

5.4. Results and evaluation

5.4.1. Simulation results

a. Side impact stress

Figure 16 shows the stress distribution on the chassis during a side impact. The maximum stress value reaches 806.4 MPa at the rear leaf spring mount. Chassis stress reaches 537.6 MPa. For the battery pack, the maximum stress is 179 MPa.

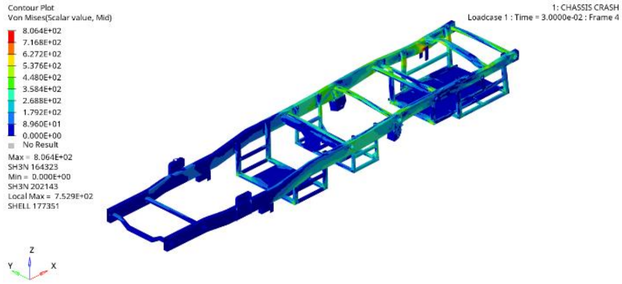

Figure 16. *Side impact stress distribution for the HEV truck chassis*

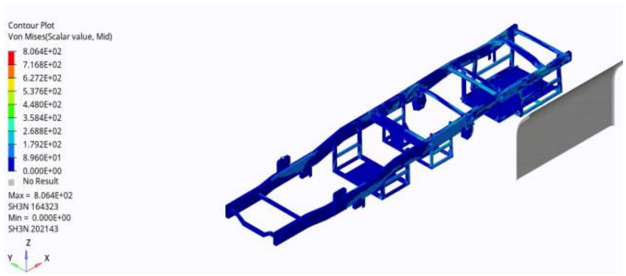

Figure 17. *Side impact stress distribution for the HEV truck chassis and battery pack*

b. Rear-end impact stress

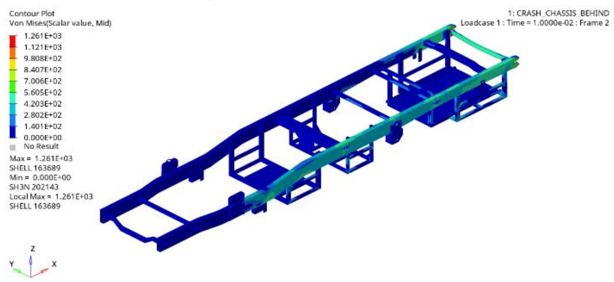

Figure 18. *Rear-end impact stress distribution for the HEV truck chassis*

(Maximum stress at the rear end of the longitudinal beam and chassis stress at the initial collision)

Figures 18 and 19 show the stress distribution on the chassis during a rear-end impact. The maximum stress value reaches 1,261 MPa at the rear end of the longitudinal chassis beam. Chassis stress reaches 560.5 MPa. For the battery pack area, the maximum stress is 280 MPa.

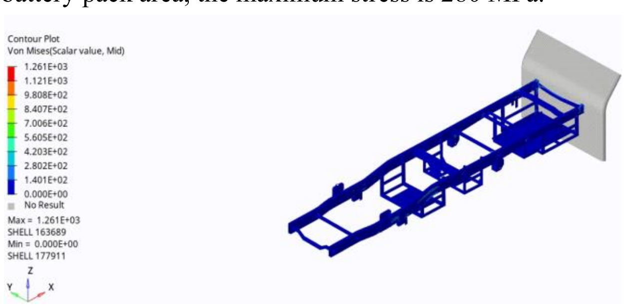

Figure 19. *Rear-end impact stress distribution for the HEV truck chassis and battery pack*

c. Side impact displacement

Figure 20 shows the displacement distribution on the chassis during a side impact. The maximum displacement value reaches 131 mm. For the battery pack area, the maximum displacement is 72 mm.

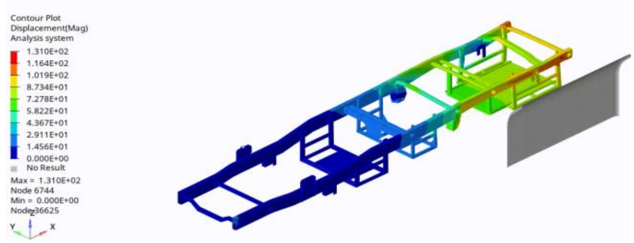

Figure 20. *Side impact displacement distribution for the HEV truck chassis*

d. Rear-end impact displacement

Figure 21 shows the displacement distribution on the chassis during a rear-end impact. The maximum displacement value reaches 80.2 mm. For the battery pack area, the maximum displacement is 17 mm.

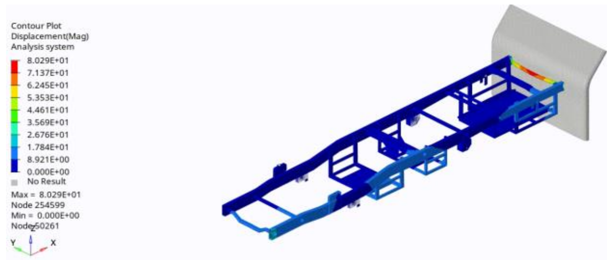

Figure 21. *Rear-end impact displacement distribution for the HEV truck chassis*

5.4.2. Evaluation

The crash simulation results at the battery pack installation location demonstrate the load-bearing capability of the chassis structures under various impact scenarios.

Side impact: The chassis endures a maximum stress of 806.4 MPa at the rear leaf spring mount, with chassis stress reaching 537.6 MPa, exceeding the material's allowable strength limit of 461 MPa. However, the maximum stress recorded for the battery pack is 179 MPa, which remains within the safe range. The maximum chassis displacement is 131 mm, and the battery pack frame displacement is 72 mm, indicating significant chassis movement, but the battery pack frame displacement still ensures safety for the battery cells. To improve safety during side impacts, it is proposed to increase the stiffness of the connection area between the leaf spring mount and the chassis. Specifically, using thicker reinforcement steel plates surrounding the area between the leaf spring mount and the chassis, and adding ribs aligned with the force transmission direction (from outside to inside the chassis) to effectively disperse impact forces and reduce the risk of chassis deformation. This approach ensures durability, is easy to implement, and optimizes production costs.

Rear-end impact: The chassis endures a maximum stress of 1,261 MPa, with chassis stress reaching 560.5 MPa at the rear end of the longitudinal chassis beam, exceeding the material's allowable strength limit of 461 MPa. However, the maximum stress recorded for the battery pack area is 280 MPa, which remains within the safe range. The maximum chassis displacement is 80.2 mm, and the battery pack displacement is 17 mm, indicating significant chassis movement, but the battery pack frame displacement still ensures safety for the battery cells. The fact that the chassis stress exceeds the allowable limit at the rearmost beam can be addressed by replacing the rearmost chassis beam with a box steel section of 100 x 50 x 3 mm.

6. Conclusion

This study presents crash and durability simulations for the KiA HEV chassis converted from the original KiA Frontier K200, utilizing CAD/FEA tools to analyze key parameters such as stress, displacement, and safety factors under maximum loads, thereby determining the optimal load limits for the chassis. Crash simulation results at a speed of 50 km/h show that the HEV chassis not only meets all safety and durability standards in crash scenarios but also provides adequate protection for the driver, assistant, and ensures the safety of the vehicle's battery pack system.

Additionally, safety for the battery pack, a crucial component of HEVs, is also ensured. Simulation results demonstrate that the battery pack layout is designed to prevent the risk of fire or damage in severe collision scenarios. Protective measures, such as robust enclosures and cooling systems, ensure that the battery system operates safely throughout its service life.

This research not only contributes to the development of safe hybrid vehicles but also lays the foundation for future improvements in chassis and safety system design for sustainable vehicles.

Acknowledgments: This research is funded by the Ministry of Education and Training through the project code B2024.DNA.14.

REFERENCES

[1] Office of Energy Efficiency and Renewable Energy (EERE), "Hybrid Electric Vehicles", *afdc.energy.gov*, April 10, 2025. [Online]. Available: https://afdc.energy.gov/vehicles/electric-basics-ev, [Accessed April 10, 2025].

[2] K. Ç. Bayindir, M. A. Gözüküçük, and A. Teke, "A comprehensive overview of hybrid electric vehicle: Powertrain configurations, powertrain control techniques and electronic control units", *Energy conversion and Management*, Vol. 52, No. 2, pp.1305-1313, 2011. https://doi.org/10.1016/j.enconman.2010.09.028

[3] O. Zamzam, A. A. Ramzy, M. Abdelaziz, T. Elnady, and A. A. A. El-Wahab, "Structural performance evaluation of electric vehicle chassis under static and dynamic loads", *Scientific Reports*, vol. 15, no. 1, pp. 5168, 2025. https://doi.org/10.1038/s41598-025-86924-w

[4] D. A. Dolla and R. B. Nallamothu, "Analytical Analysis of Electric Vehicle Chassis Frame and Battery Thermal Management System", In *Advances of Science and Technology: 8th EAI International Conference, ICAST 2020, Bahir Dar, Ethiopia, October 2-4, 2020, Proceedings*, Springer International Publishing, Part II 8, pp. 173-189. https://doi.org/10.1007/978-3-030-80618-7_11

[5] B. Liu, J. Yang, X. Zhang, and X. Li, "Topology optimization and lightweight platform development of pure electric vehicle frame-type aluminum body considering crash performance", *Journal of Materials Engineering and Performance*, vol. 34, pp. 2424–2434, 2025. https://doi.org/10.1007/s11665-024-09239-3

[6] T. Khansiriwong, K. Ruangjirakit, J. Srirat, S. Kongwat, T. Homsnit, and P. Jongpradist, "Optimization and reliability analysis of multi-purpose electric truck chassis under payload positioning variation", *Proceedings of the Institution of Mechanical Engineers, Part D: Journal of Automobile Engineering*, 09544070251328432, 2025. https://doi.org/10.1177/09544070251328432

[7] Z. Lin, Z. Lin, F. Wang, and B. Xu, "A series electric hybrid wheel loader powertrain with independent electric load-sensing system", *Energy*, Vol. 286, pp. 129497, 2024. https://doi.org/10.1016/j.energy.2023.129497

[8] Z. Zhang, B. Yang, Y. Zhang, L. Li, B. Zhao, and T. Zhang, "Powertrain modeling and performance simulation of a novel flywheel hybrid electric vehicle", *Energy Reports*, Vol. 9, pp. 4401-4412, 2023. https://doi.org/10.1016/j.egyr.2023.03.098

[9] S. K. Gupta, *Textbook of Automobile Engineering*. S. Chand Publishing, 2020.

[10] G. Rill, *Road Vehicle Dynamics: Fundamentals and Modeling*, 1st edition. Boca Raton: CRC Press, 2011. https://doi.org/10.1201/9781439897447

[11] A. H. Wickens, *The Dynamics of Vehicles on Roads*, 1st edition. London: Routledge, 1982. https://doi.org/10.1201/9780203736876

[12] THACO, "KIA FRONTIER K200", *thacotai.vn*, March 09, 2024. [Online]. Available: https://thacotai.vn/kia-frontier-k250, [Accessed April 10, 2025]

[13] L.S. Hansen, *Autodesk Inventor 2025: A Tutorial Introduction*, SDC Publications, 2024.

[14] A. Babamiri, S. Hadian, G. Wheatley, "Fatigue and Stress Analysis of a Load Carrying Car Chassis with Reinforced Joint Using Finite Element Method", *Revista GEINTEC: gestao, inovacao e tecnologias*, Vol. 11 No. 4. pp. 2156-2176, 2021. http://dx.doi.org/10.47059/revistageintec.v11i4.2262

[15] M. Gurjar, S. Deshmukh, S. Goswami, V. Mathankar, S. Shrivastava, "Design and Durability Analysis of Ladder Chassis Frame", *SSRN*, 2018. https://dx.doi.org/10.2139/ssrn.3372318

[16] D. Ruban, L. Kraynyk, H. Ruban, M. Zakharova, V. Metelap, and V. Khotunov, "Devising an approach to assessing the durability of bus body on a frame chassis", *Eastern-European Journal of Enterprise Technologies*, Vol. 122, no. 1, 2023. http://dx.doi.org/10.15587/1729-4061.2023.275656

[17] C. T. Le and M. D. Le, "An application of finite element analysis in durability analysis of multi-steel leaf spring for light trucks", *UD-JST*, Vol. 20, no. 7, pp. 64-70, 2022. https://jst-ud.vn/jst-ud/article/view/7784

[18] Official Journal of the European Union, "Regulation No 95 of the Economic Commission for Europe of the United Nations (UNECE) - Uniform provisions concerning the approval of vehicles with regard to the protection of the occupants in the event of a lateral collision [2015/1093]", *op.europa.eu*, July 10, 2015. [Online]. Available: http://data.europa.eu/eli/reg/2015/95/oj, [Accessed April 10, 2025].

[19] Official Journal of the European Union, "Regulation No 32. Uniform provisions concerning the approval of vehicles with regard to the behaviour of the structure of the impacted vehicle in a rear-end collision", *op.europa.eu*, July 10, 1975. [Online]. Available: https://treaties.un.org/Pages/showDetails.aspx?objid=08000002800123c6&clang=_en, [Accessed April 10, 2025].

[20] Hyperwork, "Altair HyperWorks 2025", *altair.com/altair-hyperworks*, September 1, 2025. [Online]. Available: https://help.altair.com/simulation/pdfs/relnotes/altairhyperworks_2025_releasenotes.pdf, [Accessed April 10, 2025].

[21] M.J.A.E. Gohlke, *Practical aspects of finite element simulation.* Michigan, USA: Altair, 2015.