JEATSA JOURNAL
JOURNAL OF ENGINEERING ADVANCES AND TECHNOLOGIES FOR SUSTAINABLE APPLICATIONS
Volume 1, Issue 1 (January, 2025)
Print ISSN: 3062-5629 Online ISSN: 3062-5637
Open Access
https://jeats.journals.ekb.eg/

Structural Analysis of an Electric Vehicle Chassis using Finite Element Analysis

Omar Zamzam[1,*], Mohamed Abdelaziz[2], Tamer Elnady[1], Aly A. Ramzy[1], Ayman A. Abd El-Wahab[1]
1 Department of Design and Production Engineering, Faculty of Engineering, Ain-Shams University, Cairo, Egypt
2 Department of Automotive Engineering, Faculty of Engineering Ain Shams University, Cairo, Egypt
omar.zamzam@eng.asu.edu.eg

doi: 10.21608/jeatsa.2025.427801
Received 22-7-2024
Revised 9-10-2024
Accepted: 1-11-2024
Published: Jan-2025

Print ISSN: 3062-5629
Online ISSN: 3062-5637

Abstract- As the world focuses on tackling climate change and finding sustainable energy solutions, electric vehicles are becoming a vital part of building an environmentally sustainable future. One of the critical tasks faced by vehicle designers is the structural analysis of the chassis. This ensures the structural integrity of the vehicle as well as the safety and comfort of passengers. This study focuses on assessing the performance of an electric vehicle chassis under static loading conditions using two finite element methods: the conventional mesh-based method using Autodesk Inventor software and the meshless method using SimSolid software. Stress distribution, displacement, and safety factor were analyzed to assess the structural integrity of the chassis. The results showed a maximum stress of 98.11 MPa and a safety factor of 2.46 in Inventor, with SimSolid producing closely aligned results and a maximum relative error of 4.96% compared to the Inventor results. The findings confirm that the chassis experiences only elastic deformation, validating its structural safety for practical use. Additionally, the study highlights the efficiency of the meshless method in reducing computational time by eliminating the time-consuming process of mesh generation while maintaining accuracy, making it suitable for the structural analysis of complex chassis designs.

Keywords- *Structural analysis; Electric vehicle; Chassis; Finite element analysis; SimSolid.*

I. INTRODUCTION

The chassis of an electric vehicle (EV) is considered the structural backbone that influences the performance and safety of the vehicle. It serves as a mounting platform that supports the suspension system, electric motor, and battery pack [1]. A robust electric vehicle chassis design provides a balance between strength, lightweight construction, and manufacturing feasibility, while meeting performance standards to ensure a safe driving experience.

To accurately design and analyze a vehicle chassis, engineers employ computer-aided engineering (CAE) tools, and experimental testing. Finite Element Analysis (FEA) is a computational tool used to simulate and analyse the behaviour of a part or assembly by Finite Element Method (FEM)[2]. It divides the model into small, finite elements that together form a mesh representing the geometry of the model being analysed. [3]. Ansys, Solidworks, and Inventor are popular modelling and simulation software that include different FEA tools serving diverse engineering applications. Engineers commonly use FEA to assess the structural integrity of a vehicle chassis under various loading conditions. These loads include static loads [1, 4–6], dynamic loads [7–9], and impact loads [10–12].

However, during stress analysis and simulation using FEA, stress concentration points on the model can lead to solution issues known as stress singularities. These singularities are nodes within the finite element mesh where certain design parameters in the simulation results tend to escalate toward infinity [13]. Among the strategies used to tackle singularities are mesh refinement around critical areas and simplification of the model geometry. However, these approaches can consume significant amounts of time during the simulation of complex components or assemblies and require the expertise of FEA specialists.

SimSolid is a computer-aided engineering (CAE) tool developed by Altair Engineering. It is designed for the simulation and structural analysis of complex structures without the need for geometry simplification and meshing, which are often the most time-intensive and skill-demanding tasks in conventional FEA. The core of the SimSolid computational engine is based on groundbreaking advancements in the theory of external approximations. SimSolid employs a mesh-free method that creates a mathematical representation based on the entire geometry of the CAD model. This method converts the model into nodes rather than dividing it into small finite elements, such as shell or solid elements [14]. This allows the model to adapt easily to complex geometry and deformation patterns. It then uses an adaptive solution technique that focuses computational resources on areas of interest in the model to achieve high accuracy results. The theoretical foundation of this mesh-free approach is detailed in Altair's whitepaper [15].

JEATSA JOURNAL
JOURNAL OF ENGINEERING ADVANCES AND TECHNOLOGIES FOR SUSTAINABLE APPLICATIONS
Volume 1, Issue 1 (January, 2025)
Print ISSN: 3062-5629 Online ISSN: 3062-5637
Open Access
https://jeats.journals.ekb.eg/

Recently, automotive manufacturers General Motors and Renault Group utilized SimSolid for structural analysis of vehicle components to verify compliance with safety and performance standards [16, 17]. The precise results, coupled with considerable time savings, have motivated these companies to integrate SimSolid extensively into their design workflows. SimSolid is quickly gaining recognition as an efficient structural analysis tool, with recent applications by researchers across various fields, including mechanical engineering [18, 19] and bioengineering [20, 21].
This research paper presents a study on the structural analysis of an electric vehicle chassis using SimSolid and Autodesk Inventor software. The simulation results were subsequently evaluated from a design perspective to confirm the safety and structural integrity safety of the chassis.

II. METHODOLOGY

The structural analysis and simulation procedures followed standard guidelines for analysing vehicle structures. A basic bending test was performed on the chassis by applying a static vertical load, with the assumption that it was simply supported at the wheels [22]. Figures Fig**.** and Fig**.** illustrate and compare the steps taken throughout the research with both software. During mesh-based FEA using Inventor, additional steps are required, such as model simplification, meshing, and mesh independence checks. On the other hand, SimSolid eliminates these steps, speeding up the process. The detailed explanation of these steps is provided in the following sections.

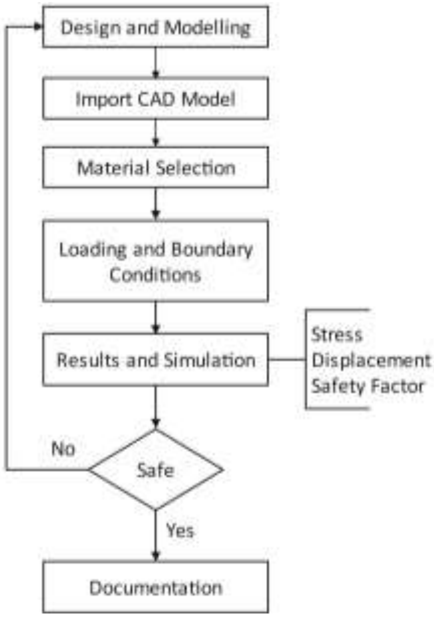

Fig. 1. Structural analysis and simulation flow chart using SimSolid

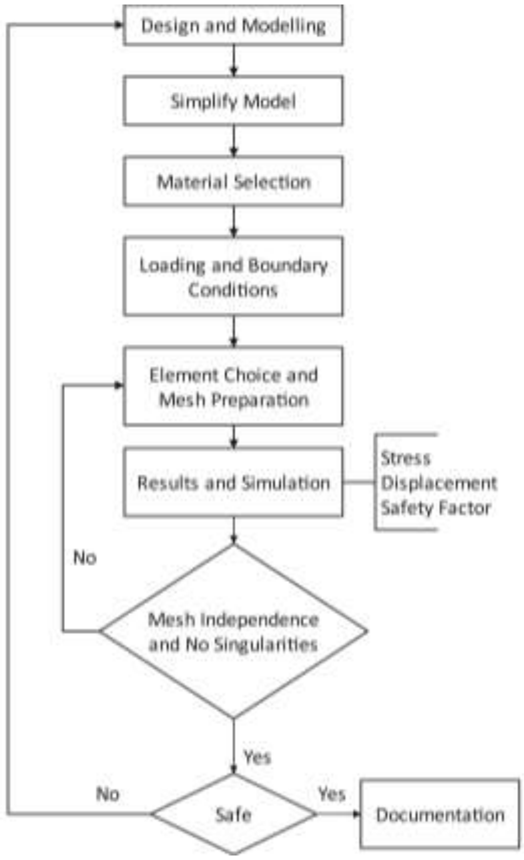

Fig. 2. Structural analysis and simulation flow chart using Inventor

Vehicle Model
The vehicle is a six-seat electric golf cart owned by Matgr Company in Egypt[23]. The proposed design of the vehicle skeleton, developed through extensive research and development efforts, is modelled using Inventor software, as shown in Figure 7. The chassis of the vehicle shown in Figure **Error! Reference source not found.**, is isolated from the skeleton for structural analysis, first using Inventor software, and second by exporting the model to SimSolid software. **Error! Reference source not found.** and Figure **Error! Reference source not found.** provide a description of the chassis dimensions.

Fig. 7. Model of the electric vehicle skeleton

JEATSA JOURNAL
JOURNAL OF ENGINEERING ADVANCES AND TECHNOLOGIES FOR SUSTAINABLE APPLICATIONS
Volume 1, Issue 1 (January, 2025)
Print ISSN: 3062-5629 Online ISSN: 3062-5637
Open Access
https://jeats.journals.ekb.eg/

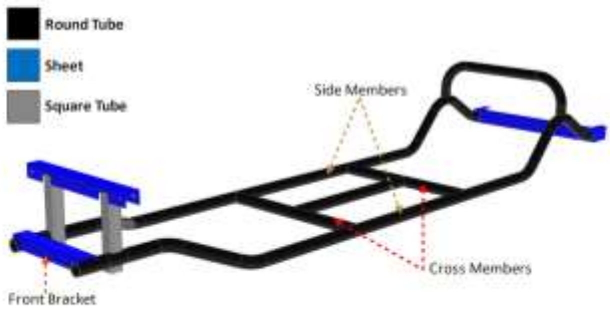

Fig. 1. Chassis assembly

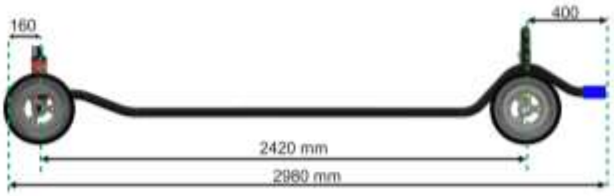

Fig. 2. Chassis dimensions

Table 1
Chassis dimensions

Parameter	Value (mm)
Length	2980
Width	980
Front overhang	160
Rear overhang	400
Wheelbase	2420

Material Selection
Steel was selected as the material for the chassis due to its favourable mechanical properties, local availability, and cost efficiency. The high strength and stiffness of steel will enable the chassis to withstand combined static and dynamic loads effectively. Furthermore, steel has the capacity to absorb and distribute impact energy, enhancing occupant safety by reducing the risk of injury in collision scenarios [22]. Additionally, steel is relatively inexpensive compared to other automotive materials like aluminium or carbon fibre composites. The chassis consists of steel tubes and sheet parts welded together as shown in Figure 1, and their properties are listed in Table 2.

Table 2
Properties of chassis materials [24]

Part	Round Tube	Square Tube	Sheet
Material	ASTM A106 gr B	ASTM A500 gr B	ASTM A572 gr 42
Yield Strength (MPa)	240	315	290
Tensile Strength (MPa)	415	425	415

Loading Conditions
Table 3 lists the main items that exert static loads on the chassis, including the weight of subassemblies, drivetrain components, the battery, passengers, cargo, and auxiliary equipment. Their total weight of 765 kg (7504 N) is applied as vertical loads distributed along the chassis, as shown in Figure 3.

Table 3
Weight distribution

Item	Weight (kg)	Quantity	Item Total Weight (kg)
Passenger	80	6	480
Subassemblies	110	1	110
Battery	70	1	70
Drivetrain Components	60	1	60
Cargo	25	1	25
Auxiliary Equipment	20	1	20
Total Weight			765 kg

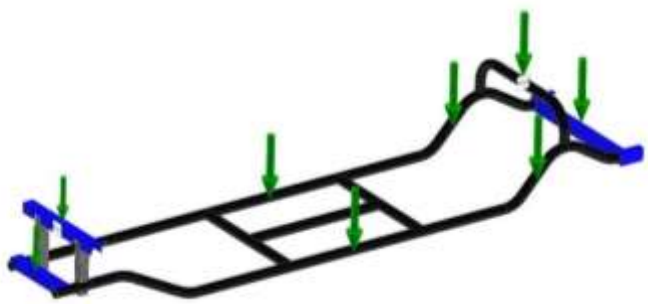

Fig. 3. Weight distribution on the chassis

Boundary Conditions
As previously stated, the chassis is being analysed as a simply supported structure. To accomplish this, fixed constraints as boundary conditions are applied to the suspension members to which the vehicle's shock absorbers are connected, as

JEATSA JOURNAL
JOURNAL OF ENGINEERING ADVANCES AND TECHNOLOGIES FOR SUSTAINABLE APPLICATIONS
Volume 1, Issue 1 (January, 2025)
Print ISSN: 3062-5629 Online ISSN: 3062-5637
Open Access
https://jeats.journals.ekb.eg/

indeterminate structure. Since the chassis is supported by four suspension points and subjected to various loads, the interactions between the chassis, suspension components, and the ground result in complex force distributions [25]. However, the analysis of internal stresses and reactions has become possible with CAE simulation tools, which will be discussed in the following section.

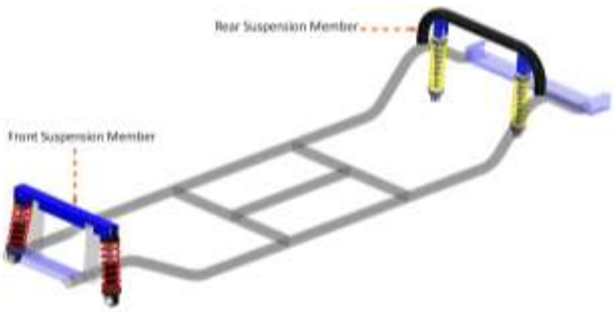

Fig. 1. Fixed suspension members

Meshing

Inventor software provides an automatic meshing tool that generates the appropriate element types based on the geometry of the model. For this study, Inventor selected linear tetrahedral solid elements, which provide a good balance between accuracy and computational efficiency for complex geometries like the chassis model. The software also offers manual control over the mesh generation process, allowing for mesh refinement, specification of element size, and control of element quality. Table 1 shows the mesh settings used during analysis and simulation. As discussed earlier, the meshing step is only performed when using Inventor software, whereas SimSolid software does not require meshing for stress analysis and simulation.

Table 1
Mesh settings

Parameter	Value
Element type	Linear tetrahedral solid element
Number of Elements	31261
Number of Nodes	63896
Average Element Size	0.1
Minimum Element Size	0.2
Grading Factor	1.5
Maximum Turn Angle	60 degrees
Curved Mesh Elements	No
Part-based measure for Assembly mesh	Yes

III. RESULTS AND DISCUSSION

Von Mises stress

Von Mises stress is a measure of equivalent stress in a material, used in FEA software to determine if a ductile material will fail under complex loading conditions. The von Mises stress value is calculated using the principal stresses experienced by the material, including the maximum and minimum normal stresses. [3]. The results of stress analysis show that the maximum von Mises stress values are concentrated on the side members of the chassis, with corresponding values of 98.11 MPa in Inventor and 93.24 MPa in SimSolid. Figures 2 and 3 show the stress distribution on the chassis by both software. The side members are considered the main longitudinal beams that extend along the entire length of the vehicle. The concentration of maximum von Mises stress on the side members is due to their primary role in bearing the bending moments and shear forces resulted from the distributed loads mentioned earlier. Notably, the results of the meshless method (SimSolid) and the traditional FEM (Inventor) for the maximum von Mises stress show strong consistency.

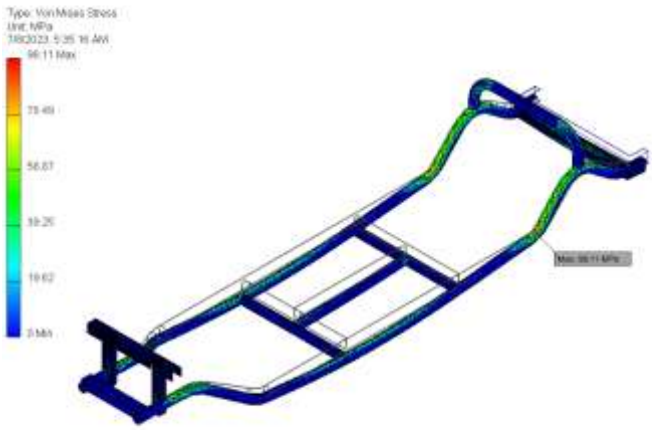

Fig. 2. Von Mises stress using Inventor

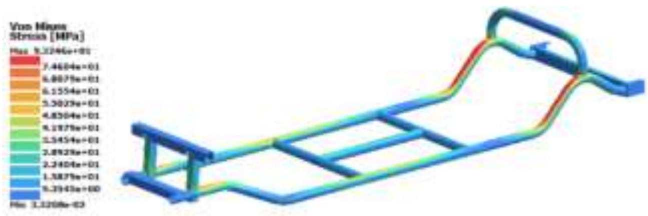

Fig. 3. Von Mises stress using SimSolid

Displacement

Displacement in FEA refers the movement of mesh nodes from their initial positions. It reflects the deformation experienced by a part under applied loads. The maximum displacement is 3.847 mm (using Inventor) and 3.826 mm (using SimSolid), which is the resultant of the displacements in the x, y, and z directions. The displacement distributions are shown in Figures **Error! Reference source not found.** and **Error! Reference source not found.** for each software.

JEATSA JOURNAL
JOURNAL OF ENGINEERING ADVANCES AND TECHNOLOGIES FOR SUSTAINABLE APPLICATIONS
Volume 1, Issue 1 (January, 2025)
Print ISSN: 3062-5629 Online ISSN: 3062-5637
Open Access
https://jeats.journals.ekb.eg/

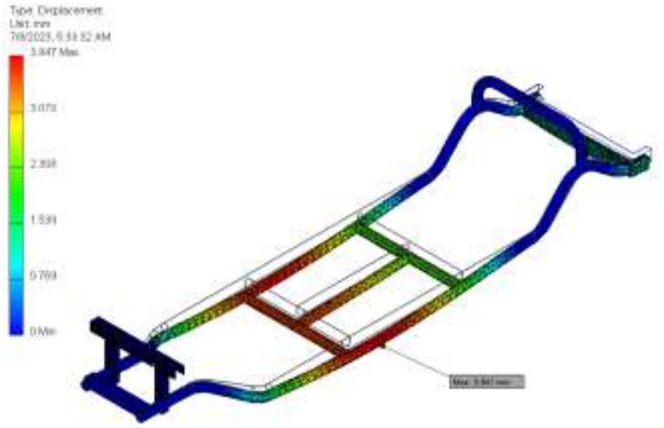

Fig. 1. Displacement distribution using Inventor

Fig. 2. Displacement distribution using SimSolid

Safety Factor

Safety factor is used to assess the ability of a structure to withstand the expected load without failure [26]. In this study, the safety factor is calculated as the ratio of the yield strength of the chassis material to the maximum von Mises stress acting on it, based on the von Mises stress failure criterion. .The minimum safety factor is 2.46 (using Inventor) and 2.5 (using SimSolid), as shown in Figures 3 and 4. It occurs on the chassis side members, which are made of ASTM A106 grade B material. Such an outcome was anticipated due to the high stress levels experienced by these side members.

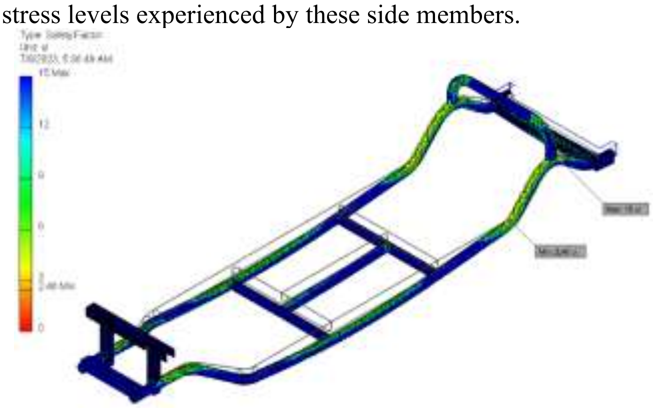

Fig. 3. Safety factor distribution using Inventor

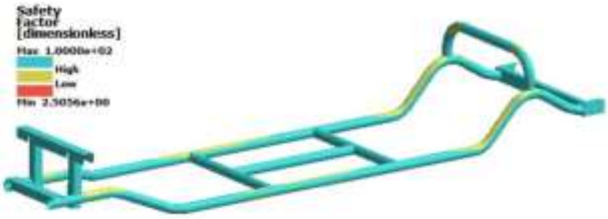

Fig. 4. Safety factor distribution using SimSolid

Error! Reference source not found. Table 1 summarizes the simulation results obtained from Inventor and SimSolid under the applied loading and boundary conditions, including a comparison of the relative error between the results from both software packages.

Table 1

Simulation results and the relative error between Inventor compared to SimSolid.

	von-Mises Stress (MPa)	Displacement (mm)	Safety Factor
Inventor	98.11	3.847	2.46
SimSolid	93.24	3.826	2.50
Relative Error (%)	4.96	0.54	1.8

Table 1, shows that the maximum relative error between software is 4.96%. Overall, the results demonstrate a consistent alignment between SimSolid and Inventor software, affirming the validity and accuracy of SimSolid software for structural analysis. In summary, these results show that the chassis is considered statistically safe with a good margin of safety factor when it is loaded with 765 kg and supported on four points.

VI. CONCLUSION

An electric vehicle chassis was designed and modelled using Autodesk Inventor. Structural analysis and simulations were conducted to determine the maximum stress, displacement, and safety factor. Two finite element methods were employed: a traditional mesh-based approach using Inventor and a meshless method using SimSolid.

The results confirmed that the chassis is statically safe, with a minimum safety factor of 2.46 in Inventor and 2.5 in SimSolid, indicating that the structure remains within the elastic deformation range. Both methods produced closely aligned results, with a maximum relative error of 4.96%, validating the accuracy of the meshless approach when compared to the conventional method. The use of SimSolid significantly reduced computational time by eliminating the time-intensive meshing process, demonstrating its potential as an efficient tool for rapid structural analysis of complex designs.

Further studies are needed to examine the effects of dynamic loading on the chassis. Additionally, more research is required to evaluate the quality of the welds and their behavior under fatigue loading, to ensure the long-term durability of the chassis.

VII. REFERENCES

[1] Nandhakumar, S., Seenivasan, S., Saalih, A. M., & Saifudheen, M. (2020). Weight optimization and structural analysis of an electric bus chassis frame. In *Materials Today: Proceedings* (Vol. 37, pp. 1824–1827). Elsevier Ltd. https://doi.org/10.1016/j.matpr.2020.07.404

[2] Majid, I. A., Laksono, F. B., Suryanto, H., & Prabowo, A. R. (2021). Structural assessment of ladder frame chassis using FE analysis: A designed construction referring to ford AC cobra. In *Procedia Structural Integrity* (Vol. 33, pp. 35–42). Elsevier B.V. https://doi.org/10.1016/j.prostr.2021.10.006

[3] Daryl L. Logan. (2017). *A First Course in the Finite Element Method* (6th ed.). Cengage.

[4] Kengkongan, A., Rio, A., & Imaduddin, F. (2020). Structural assessment of an energy-efficient urban vehicle chassis using finite element analysis - A case study. In *Procedia Structural Integrity* (Vol. 27, pp. 69–76). Elsevier B.V. https://doi.org/10.1016/j.prostr.2020.07.010

[5] Singh, R. P. (2010). *STRUCTURAL PERFORMANCE ANALYSIS OF FORMULA SAE CAR*.

[6] Widiyanto, I., Sutimin, S., Laksono, F. B., & Prabowo, A. R. (2021). Structural assessment of monocoque frame construction using finite element analysis: A study case on a designed vehicle chassis referring to ford GT40. In *Procedia Structural Integrity* (Vol. 33, pp. 27–34). Elsevier B.V. https://doi.org/10.1016/j.prostr.2021.10.005

[7] Santa Rao, K., Musalaiah, G., & Mohana Krishna Chowdary, K. (2016). Finite element analysis of a four wheeler automobile car chassis. *Indian Journal of Science and Technology*, *9*(2). https://doi.org/10.17485/ijst/2016/v9i2/83339

[8] Seelam, A. B., Aaqif Ahmed, A. K., & Sachidananda, K. H. (2020). Buggy role cage – Analysis and design. *International Journal of Safety and Security Engineering*, *10*(5), 589–599. https://doi.org/10.18280/ijsse.100502

[9] Muzakkir Ahamed, M., & Natrayan, L. (2022). Dynamic analysis and structural simulation of aluminum composite material based automobile chassis. *Materials Today: Proceedings*, *62*, 2244–2249. https://doi.org/10.1016/j.matpr.2022.03.466

[10] Tanik, E., & Parlaktaş, V. (2015). Design of a very light L7e electric vehicle prototype. *International Journal of Automotive Technology*, *16*(6), 997–1005. https://doi.org/10.1007/s12239-015-0102-6

[11] Bagawan, M., Vamsi Krishna, M., Rohit Kumar, P., Gautam Krishna, K., Fatima, T., & Haygriva, V. (2023). Design and analysis of electric motorcycle chassis frame. *Materials Today: Proceedings*. https://doi.org/10.1016/j.matpr.2023.05.072

[12] Parlaktaş, V., Tanık, E., Babaarslan, N., & Çalık, G. B. (2021). The Design and Manufacturing Process of an Electric Sport Car (EVT S1) Chassis. *Iranian Journal of Science and Technology - Transactions of Mechanical Engineering*, *45*(1), 103–113. https://doi.org/10.1007/s40997-019-00328-6

[13] Carpinteri, A., & Paggi, M. (2009). Asymptotic analysis in Linear Elasticity: From the pioneering studies by Wieghardt and Irwin until today. *Engineering Fracture Mechanics*, *76*(12), 1771–1784. https://doi.org/10.1016/J.ENGFRACMECH.2009.03.01

[14] Garg, S., & Pant, M. (2018). Meshfree Methods: A Comprehensive Review of Applications. *International Journal of Computational Methods*, *15*(4). https://doi.org/10.1142/S0219876218300015

[15] SimSolid Whitepaper. Altair Engineering, Inc. (2015). SIMSOLID technology overview.

[16] Speeding Structural Analysis at GM - Technology Center adopts Altair's Game-changing Simulation Tool. (n.d.). Retrieved from https://altair.com/resource/speeding-structural-analysis-at-gm-technology-center-adopts-altair-s-game-changing-simulation-tool

[17] Renault Nissan Mitsubishi Alliance Uses Altair SimSolid to Bring Vehicles Chassis Faster onto the Road. (n.d.). Retrieved from https://altair.com/resource/renault-nissan-mitsubishi-alliance-uses-altair-simsolid-to-bring-vehicles-chassis-faster-onto-the-road

[18] Cuciumita, C., Qin, N., & Shahpar, S. (2023). Adjoint based aero-structural design optimisation of a transonic fan blade. *Proceedings of the Institution of Mechanical Engineers, Part A: Journal of Power and Energy*. https://doi.org/10.1177/09576509231162169

[19] Cuciumita, C., John, A., Qin, N., & Shahpar, S. (2021). Structurally Constrained Aerodynamic Adjoint Optimisation of Highly Loaded Compressor Blades. https://doi.org/10.1115/GT2021-59717

[20] Fabra Rivera, A., Magalhães, F. de C., Moreno, A., & Campos Rubio, J. (2020). Assessment of the Highest Stress Concentration Area Generated on the Mandibular Structure Using Meshless Finite Elements Analysis. *Bioengineering*, *7*(4). https://doi.org/10.3390/bioengineering7040142

[21] Adugna, Y. W., & Lemu, H. G. (2023). Comparative Evaluation of Stress Concentration Generated on Human Femur Structure Using Meshfree Software. In *Lecture Notes in Electrical Engineering* (Vol. 994 LNEE, pp. 458–470). Springer Science and Business Media Deutschland GmbH. https://doi.org/10.1007/978-981-19-9338-1_56

[22] Barton, D. C., & Fieldhouse, J. D. (2018). Vehicle Structures and Materials. In *Automotive Chassis Engineering* (pp. 215–254). Springer International Publishing. https://doi.org/10.1007/978-3-319-72437-9_4

[23] MATGR. (2023). About Matgr. Retrieved August 24, 2023, from https://www.matgr.com/about/

[24] ASTM International - Standards Worldwide. (n.d.). Retrieved August 14, 2023, from https://www.astm.org/

[25] James L. Meriam, L. G. Kraige, & J. N. Bolton. (2018). *Engineering Mechanics Statics* (9th ed., Vol. 1). Wiley.

[26] Singh, D. K. (2021). *Strength of Materials* (4th ed.). Cham: Springer International Publishing. https://doi.org/10.1007/978-3-030-59667-5