HOLOS
ISSN 1807 - 1600

STRUCTURAL ANALYSIS OF BAJA SAE VEHICLE CHASSIS USING FINITE ELEMENTS METHOD

R. H. OLIVEIRA FILHO[1*], P. P. R. NISHIDA[1], T. A. GUIMARÃES[2], G. H. T. SOUSA[1]
Universidade Federal de Uberlândia (UFU) [1]
Universidade Federal do Triângulo Mineiro (UFTM) [2]
ORCID ID: https://orcid.org/0000-0002-6077-9410*
ricardo.humberto@ufu.br[*]

Submetido 23/07/2025 - Aceito 18/11/2025

DOI: 10.15628/holos.2025.18855

ABSTRACT

The Baja SAE program constitutes an international engineering competition in which undergraduate students, organized into teams, design and construct off-road prototypes to compete against one another. Within this framework, the principal parameters evaluated by the program are vehicle safety and performance. One of the primary areas for ensuring performance in these parameters is chassis development, which serves as the principal pilot protection system, in addition to being responsible for supporting the other vehicle components. In this context, the present study had as its primary objective the validation of the new chassis of the Cerrado Baja Team through the evaluation of stresses and displacements in situations involving lateral, frontal, and rollover impacts utilizing the finite element method (FEM). Based on the results obtained from the analyses, it was possible to conclude that, in the event of real occurrence of the scenarios analyzed, the chassis would satisfy the technical requirements specified in the literature, with displacements smaller than the minimum distance to the pilot, thereby ensuring the structural validation of the chassis.

KEYWORDS: structural analysis, finite elements, Baja SAE.

ANÁLISE ESTRUTURAL DE CHASSI DE VEÍCULO BAJA SAE UTILIZANDO MÉTODO DOS ELEMENTOS FINITOS

RESUMO

O programa Baja SAE é uma competição internacional de engenharia em que alunos de graduação, organizados em equipes, projetam e constroem um protótipo *off-road* para competir entre si. Nesse âmbito, os principais parâmetros avaliados pelo programa são a segurança e o desempenho do veículo. Uma das áreas principais para garantia de performance nesses parâmetros é o desenvolvimento do chassi, que é o principal sistema de proteção ao piloto, além de ser responsável pelo suporte aos outros componentes do veículo. Nesse contexto, o presente trabalho teve como principal objetivo a validação do novo chassi da Equipe Cerrado Baja por meio da avaliação das tensões e deslocamentos em situações de impactos laterais, frontais e de capotamento utilizando o método dos elementos finitos (MEF). A partir dos resultados obtidos pelas análises foi possível concluir que, em caso de acontecimento real das análises realizadas, o chassi atende aos requisitos técnicos especificados pela literatura, com deslocamentos menores que a distância mínima ao piloto, garantindo a validação estrutural do chassi.

PALAVRAS-CHAVE: análise estrutural, elementos finitos, Baja SAE.

1 INTRODUCTION

The Baja program, facilitated by SAE (Society of Automotive Engineers), aims to foster the application of knowledge acquired in the academic environment by engineering students through the realization of competitive events. In the competition, the off-road vehicle prototype is evaluated for safety, design, acceleration, speed, and endurance through various tests (SAE, Programas estudantis, 2024).

Among the tests performed, the endurance event occurs as the main one. In this test, vehicles must complete laps on a rough terrain track, off-road, with obstacles, for up to 4 hours (SAE, Programas estudantis, 2024). Due to the long duration of the event, with a high number of vehicles simultaneously on the track, the occurrence of accidents is common. Thus, ensuring pilot safety is a fundamental aspect in project conception.

The manner and regularity of accidents vary according to track conditions at the time of the test; however, it is notable that most accidents occur at curve exits and ramp-type obstacles, generating rollover, frontal impact, and lateral impact accidents.

The chassis is the structure used in an automotive vehicle to support all present components, in addition to being responsible for accommodating and protecting the pilot. Therefore, it must present adequate capacity to withstand loads and have appropriate stiffness (Seward, 2014). The chassis is considered the main pilot protection system when impact and rollover situations occur (Lottermann, 2014).

Within the context of validation and analysis of vehicle chassis for competitions using the finite element method, some works can be highlighted. Lottermann (2014) sought to propose a mathematical formulation for the case of rollover and overturning of a Baja SAE type chassis. For this, he used international standards for ROPS (Roll Over Protection Structure) testing to perform numerical and experimental validation of the chassis in the described cases. The numerical validation was made from an FEM model based on shell elements and the application of point loads on the chassis according to Australian protocol determinations. The same loads were applied in the testing bench with the real chassis utilizing hydraulic pistons. The results between the numerical and experimental tests showed significant discrepancies regarding displacements, nevertheless, the author argues that the discrepancies were expected, since the computational model did not account for imperfections, homogeneity, and microcracks in the material. Despite the differences in the results, the author concluded that the chassis was safe, since there were no fractures in chassis members during the experimental test.

In his work, Cézar (2017) aimed to propose a numerical model with the intention of simulating static and dynamic situations in a mini-baja type chassis. The author used the finite element method with beam-type elements to acquire the first 5 natural frequencies of the chassis under various boundary conditions, in addition to performing tests with static loads to obtain stress levels. To validate his model, he used accelerometers and strain gauges to instrument the car and collect experimental results. At the end of his work, the author concluded that the proposed FEM model was valid after presenting good coherence with the experimental results in both the dynamic and static analyses.

Andrade (2022) intended to validate the SamaBaja team chassis regarding a frontal impact situation and the torsional stiffness provided by it. The author used the finite element method with beam elements for discretization. In the frontal impact simulation, static load application was also used to represent the collision, and these forces were calculated through the impulse-momentum theorem, starting from mass, velocity, and impact time information. Based on the results of the initial analyses, the author concluded that the presented chassis needed upgrades and implemented them. Following the optimization of the initial geometry, the author ran all simulations again and verified that the chassis achieved sufficient torsional stiffness and proved to be safe in frontal impact situations.

Following this line of reasoning, this study intends to perform general structural validation of the Cerrado Baja team chassis through static analyses that simulate critical collision situations against the chassis. The analyses will be performed using the finite element method, with beam elements for discretization, applying the ANSYS® software solver. The simulated collision scenarios will be frontal impact, lateral impact, and rollover, and similarly to the work written by Andrade (2022), they will be represented by the application of static loads, calculated by the impulse-momentum theorem.

With the results acquired by the simulations, it is expected to draw pertinent conclusions about the safety of the chassis, the need for modifications to the geometry of the structure and the safety of the driver in general.

2 MATERIALS AND METHODS

Initially, to perform the analyses, it is necessary to define the geometric form of the chassis through a CAD (Computer Aided Design) program. For this purpose, the 3D chassis model was elaborated using SolidWorks® software, following the minimum requirements proposed by RATBSB (Administrative and Technical Regulation Baja SAE Brazil) Amendment 6. The 3D model can be observed in Figure 1.

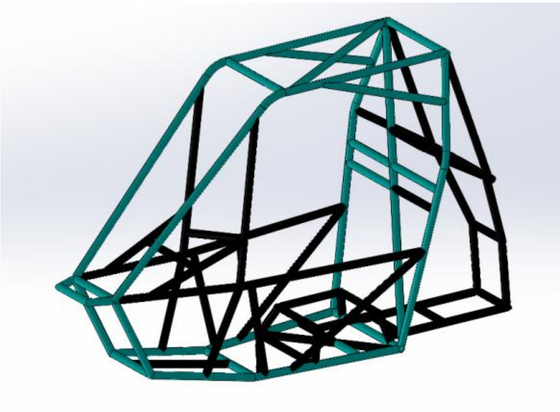


Figure 1. 3D chassis model.

The model was designed with two different tube profiles of the same material, SAE 4130 steel. In Figure 1, primary tubes are represented by cyan-colored tubes, and secondary tubes are black-colored tubes.

With the 3D model developed, logical stages were defined to conduct the necessary analyses for chassis validation, namely: pre-processing, processing, and post-processing. In the pre-processing stage, the material properties of the chassis will be defined in the software, the mesh generation process will be performed on the geometry, and the boundary conditions that will guide the simulations will be established. Processing refers to the stage of utilizing the ANSYS® solver for model solutions. Finally, post-processing refers to the evaluation of the results obtained to ensure correspondence between numerical analyses and the expected results from the real-world situation.

2.1. Pre-processing

Following the established methodological framework, the initial phase of pre-processing involved configuring the material properties of SAE 4130 steel, the alloy selected for automotive chassis fabrication, within the computational environment. This configuration process utilized the metallurgical specifications detailed in Table 1.

Table 1. SAE 4130 steel mechanical properties.

	SAE 4130 Steel	
Mechanical Property	**Magnitude**	**Unit**
Yield Strength	460.00	MPa
Ultimate Tensile Strength	560.00	MPa
Density	7850.00	Kg/m^3
Elastic Modulus	205.00	GPa
Poisson's Ratio	0.30	N/A
Elongation	21.00	%

Source: ASM (1983).

Subsequently, geometry preparation is necessary for conducting the proposed analyses. In this stage, the 3D model made in SolidWorks® is imported into ANSYS® and must undergo a process of repairs and manipulation within the SpaceClaim module present in the software.

In this module, beam elements were extracted from the initial geometry. In a structural simulation based on a complete 3D model, the required computational cost is quite high. Therefore, the use of beam elements is employed, which provides results that closely approximate real load conditions, in addition to demanding less computational processing capacity compared to three-dimensional elements (Alves, 2012).

Furthermore, beam elements, illustrated by Figure 2, can be understood as a line with associated cross-sectional properties. They are one-dimensional elements without physical dimensions in the directions normal to their length and are the natural choice for structures composed of welded elements where the length is significantly greater than the other dimensions

(Kurowski, 2015). This type of element can represent situations involving tensile, compressive, bending and torsional stresses. Based on the Timoshenko beam formulation, they also include the effects of shear deformation. Additionally, the element has six degrees of freedom at each node: rotations around the x, y and z axes and translations in x, y and z directions. (ANSYS®, 2025).

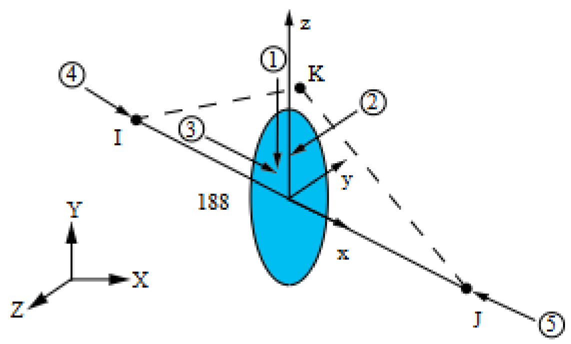


Figure 2. Beam element illustration.
Source: ANSYS® (2025).

Thus, the geometric properties of the tubes presented in Table 2 were defined for each of the equivalent beam elements within the SpaceClaim workspace.

Table 2. Cross-section properties of the tubes

Tube Type	outer diameter [mm]	Inner diameter [mm]	area moment of inertia [mm^4]	polar moment of inertia [mm^4]
Primary	31.75	28.55	17268.84	34537.68
Secondary	25.40	23.60	5204.59	10409.17

The mesh generation process was initiated with the extraction of beam elements, a step that consists of converting the 3D modeled geometry into equivalent structural elements. In this phase, each chassis component is transformed into a beam with proportions identical to those of the original model, ensuring that all connections between elements are preserved. However, for the software to finalize the mesh implementation, it is necessary to define the size of each element that will represent the extracted beams. This step is essential to guarantee the approximation of theoretical results with the real behavior of the structure, which led to performing the mesh convergence analysis process. For this purpose, successive comparisons were carried out between the von Mises equivalent stress and the number of elements present in the mesh, until the stress magnitude converged, as presented in Figure 3.

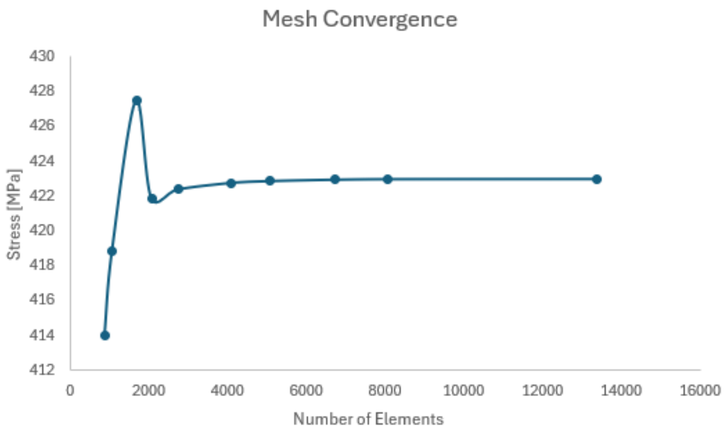


Figure 3. Mesh convergence analysis.

The analysis of Figure 3 reveals that, starting from approximately 4000 elements, the von Mises equivalent stress converges to 423 MPa, such that additional refinement of the element size would not improve the error presented by the model. Consequently, mesh generation with approximately 4300 elements was defined for the performed static analyses.

The final step of preprocessing involves defining the boundary conditions, constraints, and loads that will govern the simulation of each proposed static analysis scenario.

In the context of frontal and lateral impacts, the determination of the load magnitudes to be applied to the model was performed based on estimates of impact duration, vehicle velocity now of collision, and the prototype's mass, in accordance with the impulse-momentum theorem, as exemplified by Equation (1).

$$F = \frac{m * v}{\Delta t} \tag{1}$$

where:

- F – Impact force in N;
- m – Vehicle mass in kg;
- v – Vehicle speed in m/s;
- Δt – Impact duration in s.

The magnitude of the force applied in the model for the simulation of the rollover situation will be defined using the model proposed by Shah (2021), in which the acting force in a rollover situation corresponds to 25% of the force calculated for a frontal impact.

The current prototype of the Cerrado Baja team has a maximum speed of 45 km/h. However, during the Baja competition events, the vehicles maintain an average speed close to 30 km/h (Andrade, 2022). Programs that provide certification regarding vehicle safety adopt different speed values in their tests. Latin NCAP, in its frontal collision tests, adopted an impact at 64 km/h against an offset deformable barrier, to simulate a collision between two vehicles at 56 km/h (Latin NCAP, 2021). The test protocol suggested by Euro NCAP for the certification of European cars proposes an impact speed of 50 km/h with a tolerance of 1 km/h above or below this value (Euro NCAP, 2024).

For the side impact test scenario, Latin NCAP uses a deformable barrier mounted on a trolley, which reaches a speed of 50 km/h before impacting the test vehicle at a right angle (Latin NCAP, 2025).

Thus, seeking a balance between the impact speed used in traditional vehicle tests and the speed achieved by Baja SAE-type prototypes, a speed of 40 km/h was adopted to estimate the force generated by the impacts.

The estimated mass of the Cerrado Baja Team prototype is approximately 250 kg, including the driver. Thus, this will be the mass used for the force magnitude calculations.

Therefore, the last parameter to be defined for the calculation of the magnitude of the forces used in the lateral and frontal impact simulations is the duration of these impacts. In this regard, Agaram et al. (2000), when analyzing Crash-Pulse data, the acceleration curve recorded in a vehicle during a crash test, available in the NHTSA (National Highway Traffic Safety Administration) database for frontal impact tests at 48 km/h, concluded that the average pulse duration is 116 ms.

During the certification of the T-Cross vehicle, manufactured by Volkswagen do Brasil, by Latin NCAP, the durations of the lateral and frontal impacts against deformable barriers were measured by the car manufacturer. In this test, the duration of the frontal impact measured by the manufacturer was 115 ms, and the duration of the side impact was 60 ms (Online, 2019).

Wilson and Haight (2012) conducted nine side impact tests between two cars to analyze the dynamics of vehicle impacts and compare their results with those obtained in mobile deformable barrier tests conducted by NHTSA. In this study, it was found that the duration of the crash pulses between vehicles was on average more than twice as long as those presented in the tests conducted by NHTSA.

Considering that the simulations to be performed aim to represent impact situations between two vehicles, the findings by Wilson and Haight (2012) will be applied, seeking to correct the impact duration values derived from deformable barrier tests found in the literature. Thus, taking the duration values measured in the tests conducted on the Volkswagen T-Cross and applying a correction factor of two and a half times for the side impact situation, the duration time in each impact case is presented in Table 3.

Table 3. Duration time for each impact case.

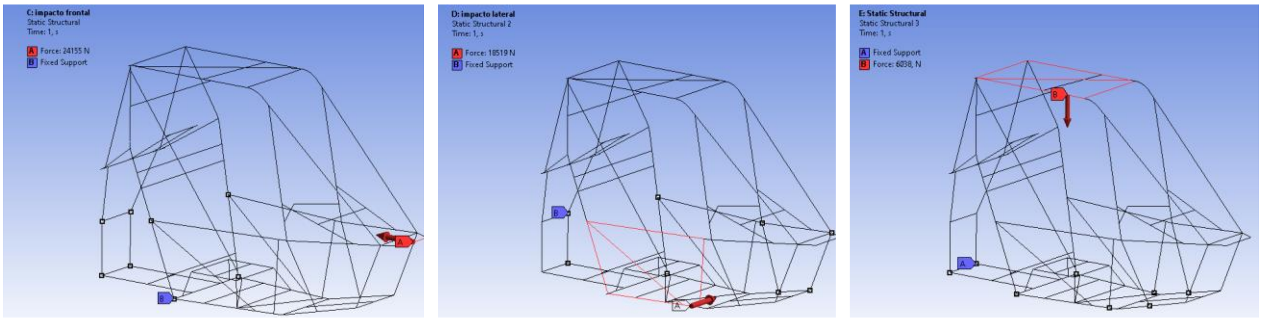
Impact Type	Δt [ms]
Frontal	115
Lateral	150

Therefore, the force parameters used in the simulations, calculated by Equation (1), are presented in Table 4.

Table 4. Force magnitudes applied to the model.

Impact Situation	Force magnitude [N]
Frontal Impact	24154.6
Lateral Impact	18518.5
Rollover	6038.5

With the force parameters defined, the next step is the definition of the boundary conditions for the analyses, in the form of enumerating the nodal constraints and the elements to which the forces will be applied in the model. In the context of frontal impact, the chassis was considered to suffer impact on the front tube, with the rear suspension attachment points restricted regarding displacements in the 3 directions – Figure 4 (a). In the lateral impact condition, the suspension attachment points on one side of the chassis had their translational degrees of freedom restricted, with the load applied to the tubes on the opposite side of the structure – Figure 4 (b). In the rollover condition, the loading was applied to the tubes that compose the chassis roof, and the fixed supports considered were the exterior nodes of the tubes that define the vehicle floor plane – Figure 4 (c). In the Figure 4, points highlighted in black and white represent where the nodal constraints were applied, and members in red represent the location of the applied force in the chassis.



(a) (b) (c)

Figure 4. Boundary conditions for the analyses performed: a) conditions for frontal impact; (b) conditions for lateral impact and c) conditions for rollover.

2.2. Model validation

The nature of the finite element method evidences the need to compare the results obtained by simulation with some demonstrative parameters of the real situation. In this way, a certain degree of reliability of the simulation results can be established through the analysis of the errors between these two parameters.

In this regard, the validation of the numerical model was performed through the comparison between theoretical and computational results in the bumper during frontal impact simulations. This component was chosen for being the main responsible for the absorption of the initial mechanical load in frontal collisions, being affected almost exclusively by this type of impact. Thus, this situation proved to be ideal for comparison with a theoretical beam model.

For the analysis, a simply supported beam with length equivalent to the bumper member of the chassis (270 mm) was used. A distributed load of the same magnitude as calculated for the impact was applied, and the displacements obtained computationally were compared with the displacements generated theoretically. Figure 5 illustrates the beam in question.

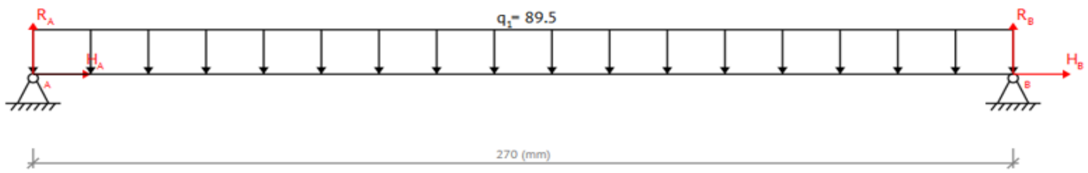


Figure 5. Theoretical Beam Model

The maximum displacement for simply supported beams under distributed loading conditions is presented by Equation (2) (Hibbeler, 2010).

$$\delta = \frac{-5 * q * L^4}{384 * E * I} \quad (2)$$

where:

- δ – Displacement caused by the loading;
- q – Distributed loading;
- L – Beam length;
- E – Elastic modulus of the material;
- I – Area moment of inertia of the cross section.

Figure 6 shows the shape of the elastic curve for this type of loading and support condition. In it, it is observed that the maximum displacement found for this type of beam is located at its midpoint, in the same direction as the load application.

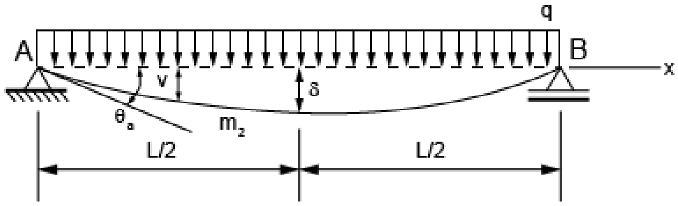


Figure 6. Elastic curve for a simply supported beam under distributed loading.
Source: Adapted from Hibbeler (2010).

3. RESULTS AND DISCUSSIONS

3.1. Model validation results

As stated previously, the model validation for the static analyses will be based on the comparison of the maximum displacement results in the bumper, calculated computationally and theoretically. The numerical result for the displacement in the analyzed member can be observed in Figure 7.

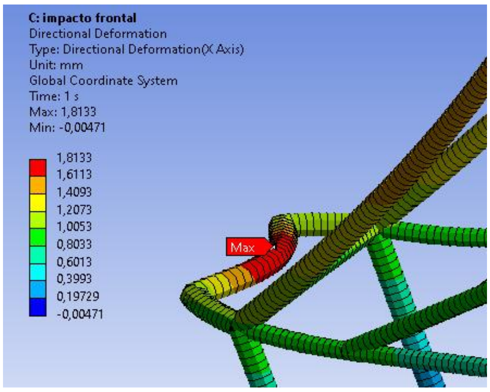


Figure 7. Directional displacement in the bumper

The analysis of Figure 7 reveals that the maximum directional displacement was 1.81 mm, located at the midpoint of the front tube, and in the same direction as the force application, corroborating with the theoretical model and with the elastic curve.

In that regard, Table 5 presents the results for displacement calculated theoretically applying the previously mentioned parameters in Equation (2), and its comparison with the results exposed in Figure 7.

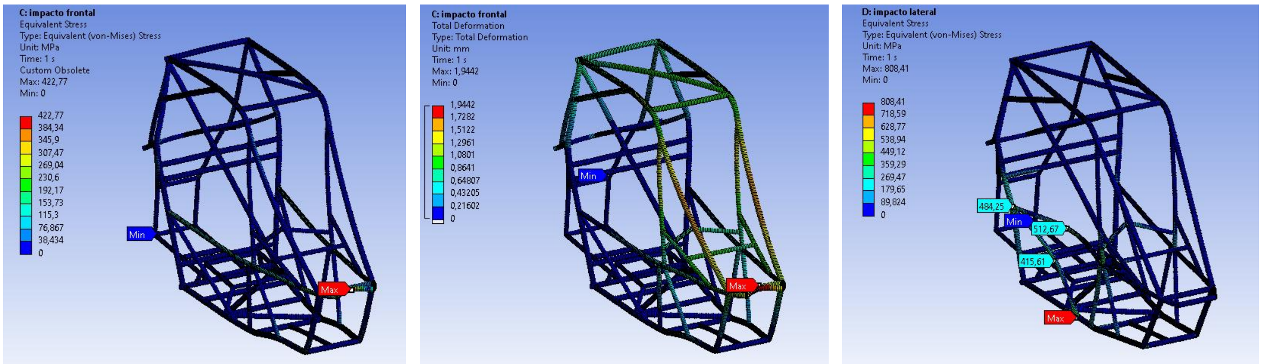
Table 5. Displacement results.

Computational Displacement [mm]	Theoretical Displacement [mm]	Relative Error
1.81	1.75	3.4 %

Observing the results shown in Table 5, the theoretical displacement, calculated by Equation (2), was 1.75 mm located halfway along the beam's length, with a difference of 0.06 mm in absolute value from the result presented in Figure 7. Furthermore, Table 5 shows that the displacement calculated by the computational model differs from that calculated by the theoretical model by 3.4%, characterizing low relative variation between the results. The conformity presented by the results demonstrates the quality of the proposed computational model, since it not only presents compatible displacement results but also presents similarity in the form of beam deformation with the theoretical elastic curve.

3.2. Static analysis of frontal impact, lateral impact and rollover

The frontal impact analysis was taken as the starting point, since it represents the highest occurrence situation when dealing with accidents in the automotive field. Furthermore, it represents the situation in which the most fatalities resulting from impacts occur. Figure 8a presents the results of the von Mises stresses acting on the chassis structure due to the frontal impact situation, Figure 8b shows the maximum displacement for this simulation. Figure 8c presents the result of the lateral impact stresses.



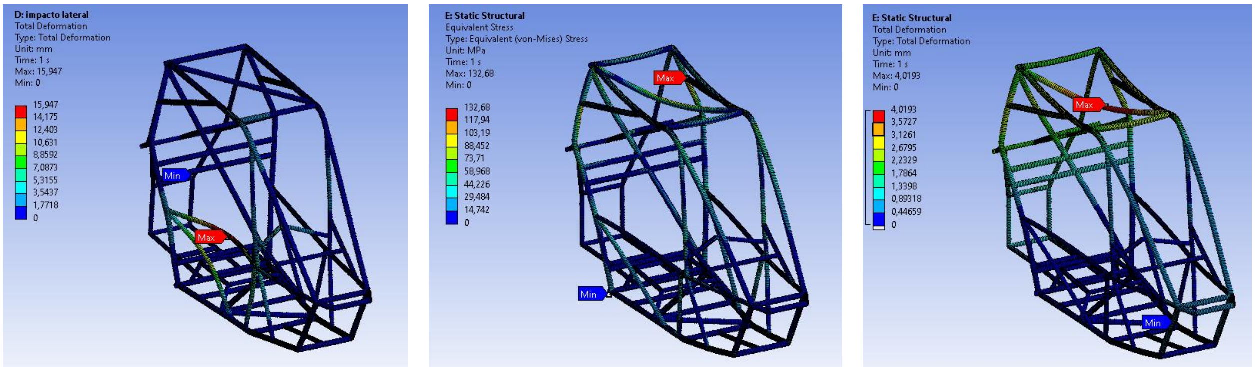
(a) (b) (c)

Figure 8. Simulation results: a) stresses for frontal impact; b) displacements for frontal impact and c) stresses for lateral impact.

As can be observed in Figure 8a, the maximum von Mises equivalent stress occurs in the bumper where the force was applied in the model. The stress magnitude of 422.77 MPa does not exceed the yield stress limit of the material, with no plastic deformations occurring at any point in the structure. The analysis of the displacements shown in Figure 8b shows that the maximum displacement calculated by the computational method was 1.94 mm, also in the bumper. Thus, it is possible to affirm that, in the occurrence of this type of impact, the driver would be safe, since no plastic deformations or ruptures occur in the chassis members, in addition to the displacements being small in magnitude and in directions opposite to the cockpit area, as suggested by the regulations.

The analysis of Figure 8c evidences several points where the equivalent stress is greater than the yield stress and the tensile strength. The maximum stress found (808.41 MPa) occurs at the lower junction of the connection between the SIM (Side Impact Member) tube and the LFS (Lower Frame Side Member) tube, a region subjected to the welding process. Therefore, there is a great chance that this impact situation will cause the structure to rupture at the weld point. Furthermore, the stress resulting from the impact in the central region of the SIM tube exceeds the yield limit of SAE 4130 steel, characterizing permanent deformation in the tube.

Despite the imminent rupture in the connection between the SIM and the LFS and the plastic deformation in the SIM for this situation, when evaluating cases such as these, for the guarantee of driver safety, it is important to analyze the size of the displacements caused by the impact, and whether these are sufficient to reach the driver inside the cockpit. Figures 9a, 9b and 9c present the result of the maximum displacements for lateral impact, stresses for rollover and displacements for rollover respectively.



(a) (b) (c)

Figure 9. Simulation results: a) displacements for lateral impact; b) stresses for rollover and c) displacements for rollover.

Upon examining Figure 9a, it is demonstrated that the maximum displacement occurring in the geometry is 15.95 mm in the central region of the SIM tube. According to item B6.1.4.1 of the RATBSB, any member of the driver's body must have a minimum clearance of 76.0 mm to the roll cage, thus the largest displacement occurring in the simulated geometry would not come to touch the driver, indicating the driver's safety in case of lateral impact.

It is demonstrated by the examination of Figure 9b that the maximum stress resulting from the rollover scenario is 132.68 MPa, a value below the yield stress of the material. However, this value occurs at a welded joint point in the structure, which leads to the need for additional analyses, should this rollover situation come to happen.

Based on Figure 9c, it is possible to observe that the maximum displacement, of approximately 4.01 mm, found in the upper region of the structure, is smaller than the clearance required by item B6.1.3.1 of the RATBSB which mentions that the helmet must be at least 152.00 mm away from a straight line applied to any two points of the roll cage members. Thus, it can be guaranteed that, in a rollover situation, the displacement of the upper tubes would not cause risks to the driver's safety.

4. FINAL CONSIDERATIONS

This work had as its main objective to conduct the structural validation of the chassis developed for the Cerrado Baja Team, employing a methodology based on the Finite Element Method (FEM). The simulations focused on the evaluation of stresses and displacements resulting from critical scenarios, including frontal impacts, side collisions, and rollover situations. The proposed model was validated through the comparison of the computational results with those obtained from the relevant theory for the frontal impact scenario. The validation process showed good agreement between theoretical and computational results, demonstrating the model's reliability.

The static analysis results showed differentiated structural behavior in each situation. In the frontal impact scenario, the maximum stress remained below the yield limit of SAE 4130 steel, and the computed displacements were small, showing no risk of plastic deformation or contact with the driver in the event of impact. In the side impact scenario, a probable failure point was identified at the junction of the SIM and LFS members, with equivalent stress exceeding the material's tensile strength; however, the analysis of maximum displacements did not indicate any risk of intrusion into the cockpit, respecting RATBSB standards. For the rollover circumstance, both stress and displacement values were below the critical limits established by the literature and RATBSB standards, demonstrating the project's robustness in protecting the driver's integrity even under adverse conditions.

Therefore, it can be concluded that the studied chassis meets the normative and functional requirements for Baja SAE vehicles, ensuring structural integrity in frontal impact and rollover scenarios and driver safety in the side impact scenario.

5. REFERENCES

Agaram, V., Xu, L., Wu, J., Kostyniuk, G., & Nusholtz, G. (2000). Comparison of frontal crashes in terms of average acceleration [Apresentação de artigo]. SAE 2000 World Congress, Detroit, MI, Estados Unidos. Sae Publications Group.

Alves, A. F. (2012). Elementos finitos: A base da tecnologia CAE - Análise não linear. Érica.

American Society for Metals. (1983). Asm metals reference book: A handbook of data about metals and metalworking (2. ed.). Asm International.

Andrade, A. G. (2022). Análise estrutural de chassi tipo spaceframe veículo off-road tipo Baja Sae [Trabalho de Conclusão de Curso]. Instituto Federal do Espírito Santo.

Ansys, Inc. (s.d.). BEAM 188. https://ansyshelp.ansys.com/account/secured?returnurl=/Views/Secured/corp/v251/en/ans_elem/Hlp_E_BEAM188.html

Barroso, D. A., Bessa, J. A., Bezerra, C. A. D., de Moraes, J. L., & Alexandria, A. R. de. (2015). Análise do balancim da suspensão do protótipo Fórmula SAE por método de elementos finitos. HOLOS, 5, 170–181. https://doi.org/10.15628/holos.2015.2189

Cézar, É. S. (2017). Análise dinâmica e estrutural de um chassi tubular através de elementos finitos com validação experimental [Dissertação de Mestrado]. Universidade Federal do Pampa.

Euro NCAP. (2024). Crash protection frontal impact (9. ed.). https://www.euroncap.com/media/85407/euro-ncap-protocol-crash-protection-frontal-impact-v09.pdf

Hibbeler, R. C. (2010). Resistência dos materiais (7. ed.). Pearson Prentice Hall.

Kurowski, P. M. (2015). Finite element analysis for design engineers. Society of Automotive Engineers International, Inc.

Latin NCAP. (2021). Impacto frontal. https://www.latinncap.com/po/nossos-testes/passageiro-adulto/impacto-frontal

Latin NCAP. (s.d.). Side impact test. https://www.latinncap.com/en/our-tests/adult-occupant/side-impact-test

Lottermann, J. P. T. (2014). Projeto estrutural de chassi tipo gaiola para veículos minibaja: Análise numérica e experimental [Dissertação de Mestrado]. Universidade de Brasília.

Mecânica Online. (2019, 20 de abril). Quanto tempo dura um acidente? Entenda como acontecem crash test frontal, lateral e contra um poste [Vídeo]. YouTube. https://www.youtube.com/watch?v=7RDIqoZ10VE

SAE Brasil. (2024). Programas estudantis. http://www.saebrasil.org.br

Seward, D. (2014). Race car design. Palgrave Macmillan.

Shah, S. (2021). Static structural analysis of SAE Baja chassis. International Journal of Advance Research, Ideas and Innovations in Technology, 7(3), 1281–1285.

Wilson, L., & Haight, S. H. (2012). Dynamics of vehicle-to-vehicle side impact crash tests. Collision: The International Compendium for Crash Research, 7(2), 10–21.

COMO CITAR ESTE ARTIGO:

Oliveira Filho, R. H. de, Nishida, P. P. R., Guimarães, T. A., & Sousa, G. H. T. ANÁLISE ESTRUTURAL DE CHASSI DE VEÍCULO BAJA SAE UTILIZANDO MÉTODO DOS ELEMENTOS FINITOS. *HOLOS*, *4*(41). https://doi.org/10.15628/holos.2025.18855

SOBRE OS AUTORES

R. H. DE OLIVEIRA FILHO
Doutor em Engenharia Mecânica pela Universidade Federal de Uberlandia/UFU; Professor do Departamento de Engenharia Mecânica da Universidade Federal do Triângulo Mineiro/UFTM (Brasil).
E-mail: rhofilho@gmail.com
ORCID ID: https://orcid.org/0000-0002-6077-9410

P. P. R. NISHIDA
Professor da Universidade Federal de Uberlândia (Faculdade de Engenharia Mecânica).
E-mail: pedronishida@ufu.br
ORCID ID: https://orcid.org/0000-0002-1640-5980

T. A. GUIMARÃES
Professor do Departamento de Engenharia Mecânica da Universidade Federal do Triângulo Mineiro.
E-mail: tobias.guimaraes@uftm.edu.br
ORCID ID: https://orcid.org/0000-0002-6350-7867

G. H. T. SOUSA
Graduação em Engenharia Mecânica na Universidade Federal de Uberlândia (Faculdade de Engenharia Mecânica).
E-mail: gabrielteodoro@ufu.br
ORCID ID: https://orcid.org/0009-0001-8182-8744

Editora Responsável: Gustavo Fontoura Souza

Pareceristas *Ad Hoc*: Neil de Medeiros e Ricardo Breganon

Recebido 23 de julho de 2025

Aceito: 18 de novembro de 2025

Publicado: 23 de dezembro de 2025