The 11th International Congress
Motor Vehicles & Motors 2026
Kragujevac, Serbia
October 8th - 9th, 2026

Motor Vehicles & Motors

MVM2026-053
https://doi.org/10.24874/CMVM2026.053
Technical paper

Mihajlo Jevtić[1]
Aleksandra Obradović[2]
Bogdan Todorović[3]
Nebojša Hristov[4]
Damir Jerković[5]

FINITE ELEMENT ANALYSIS OF STRUCTURAL SUPPORT REACTIONS IN WHEELED MOBILE PLATFORM UNDER HIGH-INTENSITY IMPULSIVE LOADING

ABSTRACT: The structural behaviour of support systems for mobile platforms subjected to high-intensity impulsive forces represents an important aspect in the assessment of their load-carrying capacity, stability, and operational reliability. This paper presents preliminary numerical analysis of the dynamic response of a three-dimensional support structure mounted on a military cargo vehicle for weapon system integration. The analysis was performed using the Finite Element Method (FEM) under short-duration impulsive force. A three-dimensional finite element model was developed, incorporating the relevant geometric characteristics, material properties, and boundary conditions of the structure. The impulsive force was applied in different spatial directions, and the resulting stress distribution, structural displacements, and support reactions were evaluated. Particular attention was given to the influence of load direction on the magnitude and distribution of support reactions in order to examine the load transfer mechanism within the structure. The study represents a preliminary assessment of the proposed structural solution based on a representative military cargo vehicle and weapon system configuration. The analysis enabled the identification of critical loading cases and highly stressed regions, providing a basis for further structural optimization and more detailed investigations of structural performance under impulsive loading.

KEYWORDS: FEM, mobile platform, impulsive loading, support reactions, dynamic response

[1] *Mihajlo Jevtić, University of Kragujevac, Faculty of Engineering, Sestre Janić 6, Kragujevac, Serbia, mixxerskate@gmail.com, https://orcid.org/0009-0004-4176-4936*
[2] *Aleksandra Obradović, Military Technical Institute, Ratka Resanovića 1, Belgrade, Serbia, aleksandraobradovic.va@gmail.com, https://orcid.org/0000-0001-7973-8590*
[3] *Bogdan Todorović, Matikon GmbH & Co. KG, Milana Mijalkovića 21, Jagodina, Serbia, bogdanto2000@gmail.com, https://orcid.org/0009-0008-4572-1318*
[4] *Nebojša Hristov, University of Defence in Belgrade, Military Academy, Veljka Lukića Kurjaka 33, Belgrade, Serbia, nebojsahristov@gmail.com, https://orcid.org/0000-0002-8163-2223*
[5] *Damir Jerković, University of Defence in Belgrade, Military Academy, Veljka Lukića Kurjaka 33, Belgrade, Serbia, damir.jerkovic@va.mod.gov.rs, https://orcid.org/0000-0001-5182-7057*

ANALIZA REAKCIJA OSLONACA MOBILNE PLATFORME NA TOČKOVIMA PRIMENOM METODE KONAČNIH ELEMENATA PRI IMPULSNOM OPTEREĆENJU VISOKOG INTENZITETA

REZIME: Strukturno ponašanje sistema oslonaca mobilnih platformi izloženih impulsnim opterećenjima visokog intenziteta predstavlja važan aspekt pri oceni njihove nosivosti, stabilnosti i operativne pouzdanosti. U ovom radu prikazana je preliminarna numerička analiza dinamičkog ponašanja noseće konstrukcije vojnog teretnog vozila, koja je namenjena integraciji sistema naoružanja. Analiza je sprovedena primenom metode konačnih elemenata (MKE) za slučaj kratkotrajnog impulsnog opterećenja. Razvijen je trodimenzionalni model konačnih elemenata koji obuhvata relevantne geometrijske karakteristike, mehaničke osobine materijala i granične uslove konstrukcije. Impulsno opterećenje primenjeno je u različitim prostornim pravcima, a analizirani su raspodela napona, pomeranja konstrukcije i reakcije u osloncima. Posebna pažnja posvećena je uticaju pravca opterećenja na reakcije u osloncima, u cilju sagledavanja mehanizma prenosa opterećenja kroz konstrukciju. Istraživanje predstavlja inicijalnu procenu predloženog konstruktivnog rešenja zasnovanog na reprezentativnom primeru vojnog teretnog vozila i sistema naoružanja. Analiza je omogućila identifikaciju kritičnih slučajeva opterećenja i oblasti sa povišenim naprezanjima, što predstavlja osnovu za dalju optimizaciju konstrukcije i detaljnija ispitivanja njenog ponašanja u uslovima impulsnog opterećenja.

KLJUČNE REČI: MKE, mobilna platforma, impulsno opterećenje, reakcija u osloncima, dinamičko ponašanje

INTRODUCTION

Vehicle stability is a fundamental consideration in the design and operation of heavy-duty vehicles and mobile platforms. The stability of such systems is determined not only by their geometric configuration and loading conditions but also by the mechanical characteristics of the supporting structure and the distribution of external loads. Therefore, the development of mathematical and mechanical models for the determination of support reactions and assessment of structural stability is essential for predicting the mechanical response of the vehicle and ensuring its safe and reliable operation. The objective of this study is to develop a preliminary mathematical and mechanical model for evaluating the stability of a mobile wheel-supported vehicle platform as part of a conceptual engineering study on the integration of a weapon system onto an existing vehicle platform. The study is intended to assess the technical feasibility of the proposed integration and to identify the critical structural and stability conditions resulting from the additional loads imposed by the integrated weapon system. The structural response of the vehicle chassis to an impulsive external load is investigated using the Finite Element Method (FEM), (Ghazaly 2014) emphasizes the importance of finite element analysis in evaluating the stress distribution within vehicle chassis structures. Numerical analysis enables the determination of stress and deformation fields within structural components, thereby providing a basis for evaluating their load-bearing capacity, structural integrity, and the overall structural performance of the proposed vehicle configuration. In this regard, the FEM represents a robust numerical framework for assessing the response of structural systems subjected to various loading conditions, while also enabling the consideration of both linear and nonlinear material and structural behaviour (Kojić et al. 2010; Dunica & Kolundžija, 1985).The analysis of vehicle chassis structures should account for their response to both static and dynamic loading conditions, as each loading regime may have a significant influence on the structural integrity of the chassis (Han Fui & Abd. Rahman, 2007).

From the standpoint of internal ballistics, the firing process generates a transient impulse that is transmitted from the weapon system to its supporting structure. The resulting recoil force represents the mechanical response to the rapid momentum change associated with the acceleration of the projectile and the movement of the recoiling masses. The magnitude and time history of this impulse depend on the ballistic characteristics of the weapon system and the associated firing conditions. Previous studies have demonstrated that the determination of recoil impulse and its transmission to the supporting structure is an important factor in assessing structural loads, support reactions, and overall firing stability. The design of weapon systems requires consideration of internal ballistic phenomena and the interaction between the gunpowder gases, projectile, and weapon barrel, as these processes directly influence the overall performance of the system (Tančić, 2014).

The mechanical model of the analysed vehicle, together with the acting external forces and support reactions, is presented in Figure 1. The vehicle is supported by four contact points, designated as A, B, C, and D. The four supports form the complete support configuration through which the external loads are transferred from the vehicle structure to the supporting surface.

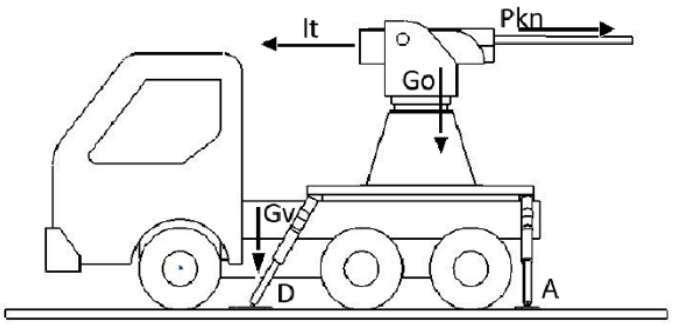


Figure 1 Vehicle with acting forces and support resistance

The external forces considered in the mechanical model are defined as follows:

- Pkn denotes the tractive force;
- It represents the inertial force associated with the moving or recoiling mass;
- Go denotes the gravitational force corresponding to the mass of the tractive-force source;
- Gv represents the gravitational force due to the vehicle mass.

The tractive force Pkn acts in the longitudinal direction of the vehicle and represents the load transmitted during acceleration. The inertial recoil force It results from the acceleration of the moving mass and acts in the opposite direction to the corresponding acceleration. The forces Go and Gv act vertically downward through the respective centres of gravity. The reactions at supports A, B, C, and D represent the resultant forces transmitted between the vehicle structure and the supporting surface. Their distribution is determined by the magnitude and direction of the applied forces, the positions of their points of application, the locations of the centres of gravity, and the geometric configuration of the support points. Determination of these reactions is therefore necessary for establishing the equilibrium conditions and evaluating the stability of the vehicle under the considered loading scenario.

DEVELOPMENT OF THE FINITE ELEMENT MODEL AND ANALYSIS

The finite element model of the supporting platform was developed from the three-dimensional CAD geometry. The geometry was simplified by removing non-load-bearing details and features that do not significantly affect the global stiffness and load distribution of the structure. The simplified geometry retained the principal load-bearing members, connection regions, and hydraulic outriggers, which are critical for the transfer of loads from the weapon system to the ground. The model was formulated as a shell finite element model. Four-node shell elements without midside nodes were used for the discretization of the structural surfaces. The FEM model is shown in Figure 2.

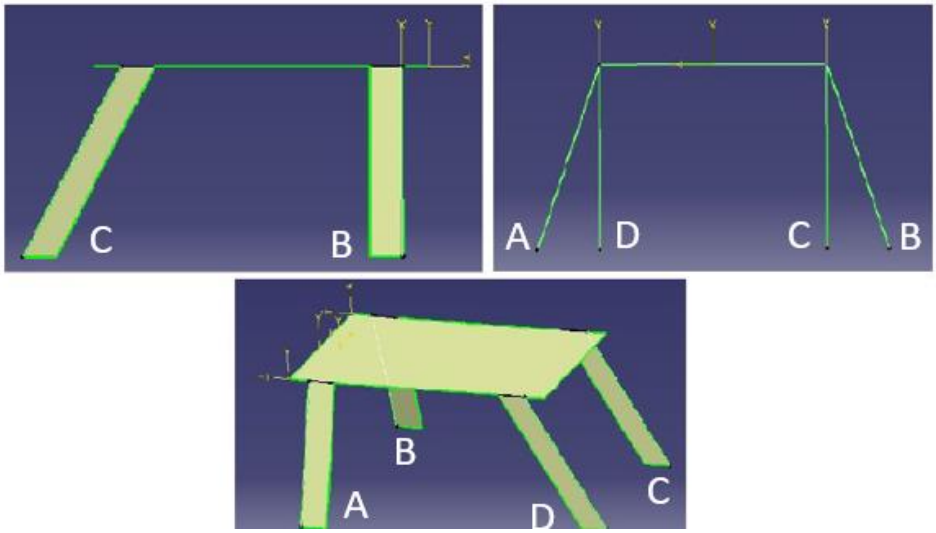


Figure 2 The FEM model

The finite element mesh was generated using four-node shell elements without midside nodes, with an appropriate mesh density to adequately represent the structural behaviour, particularly in regions where higher stresses and support reactions were expected. The resulting finite element mesh and boundary conditions are presented in Figure 3.

The applied loads consist of the inertial recoil force It, the gravitational force of the mounted system Go, and the gravitational force of the vehicle Gv. The forces were applied at the corresponding nodes defined in Table 1. The gravitational forces Go and Gv act vertically downward along the negative (Z)-axis, while the inertial recoil force It acts in the spatial direction defined by the elevation angle θ and direction angle ψ. Therefore, the direction and magnitude of the inertial recoil force It vary according to the considered loading configuration.

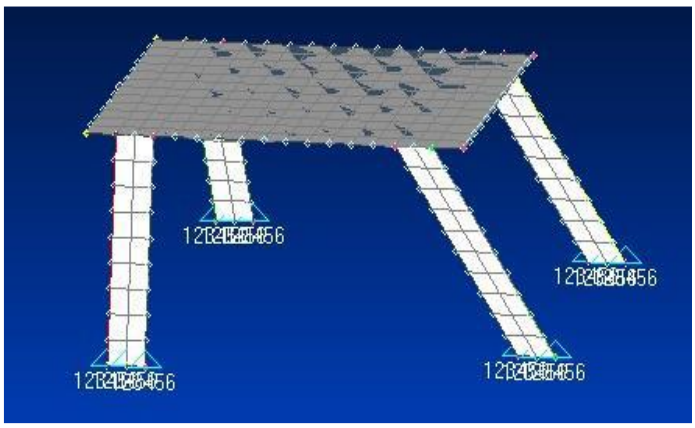


Figure 3 The finite element mesh

Table 1 Locations of force application node

Force	**Position (description / mm)**
Weight of the mounted system (Go)	Center node of the supporting plate
Vehicle weight (Gv)	(0, 1900, 0)
Inertial recoil force (It)	Center node of the supporting plate (0, 1280, 1200)

The stress analysis was performed for different combinations of elevation and direction angles. For each loading case, the resulting stress state, deformation, and support reactions were determined. The locations of the supports at which the reactions were evaluated, and 3D model are shown in Figure 4. For each support, the node with the maximum resultant reaction was selected as the representative value for further analysis. Numerical analysis based on the finite element method can be used to assess the structural behaviour of a vehicle chassis, including the distribution of stresses and deformations, as well as the identification of critical areas within the structure. In addition to static loading conditions, the dynamic behaviour of a vehicle can significantly affect the loads transmitted through the structure; therefore, the effects of dynamic loading should also be considered when evaluating the structural response of vehicle chassis (Kengkongan Ary et al. 2020; Kwasniewski et al. 2006).

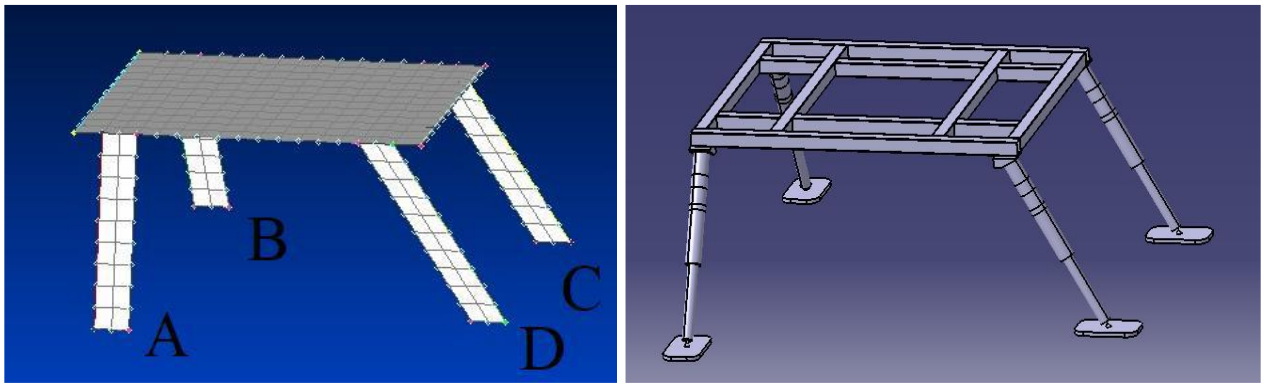


Figure 4 The locations of the supports and 3D model of platform

Boundary conditions and loading applied in the FEM model:

- the lower surfaces of the hydraulic supports were defined as fixed supports;
- all six degrees of freedom were constrained at the corresponding support nodes;
- translational displacements in the (Ox), (Oy), and (Oz) directions were constrained;
- rotations about the (Ox), (Oy), and (Oz) axes were constrained;
- forces Go and Gv were applied vertically downward along the negative (Oz) axis;
- inertial recoil force It was applied at the defined load application nodes according to the considered angles values of θ and ψ;
- the magnitude and spatial orientation of inertial recoil force It were varied according to the analysed loading cases;
- the hydraulic supports were assumed to remain stationary relative to the ground throughout the analysis.

Determining the unit vector of Inertial recoil force

The inertial recoil force It has a spatial orientation defined by the elevation angle θ and the direction angle ψ. Its line of action and spatial orientation are presented in Figure 5. These angular parameters define the orientation of inertial recoil force It with respect to the vehicle coordinate system and are used in determining its corresponding force components and moments. The resulting force components and moments are subsequently considered in the formulation of the equilibrium equations and in the evaluation of the support reactions.

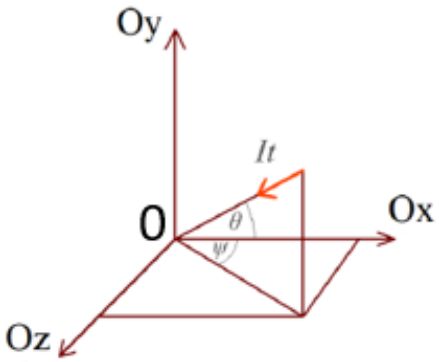


Figure 5 The spatial orientation of force It

Accurate determination of the magnitude and direction of the acting forces is essential for evaluating the stability and load-bearing capacity of the vehicle support structure. Under impulsive loading conditions, the external and inertial forces affect the distribution of support reactions and the resulting stresses in the vehicle structure. FEM analysis enables the structural response to these loads to be evaluated and provides a basis for assessing the stability and structural integrity of the vehicle. The allowable azimuth and elevation ranges are defined by the operational requirements and mechanical constraints of the weapon system intended to be integrated onto the vehicle. The azimuth angle is limited to the range ±60°, while the elevation angle is limited to -5° to 20°. These angular limits define the operational envelope of the weapon system considered for integration and are used to establish the loading conditions for the structural analysis of the supporting platform.

From the standpoint of overall vehicle stability, the theoretically most critical elevation condition corresponds to an elevation angle of 0°. At this configuration, the resulting moment generated by the inertial recoil force It reaches its maximum value for a direction angle of 0°, which represents the most critical condition with respect to potential failure of the supporting structure. For azimuth angles of ±60° the corresponding loading configuration becomes critical with respect to the overturning stability of the vehicle. In contrast, from the structural load-bearing standpoint, the most critical condition for the supporting structure corresponds to the maximum permissible elevation angle. At this configuration, the force transmitted to the supporting structure reaches its highest vertical component, resulting in the greatest structural loading. Therefore, both the minimum-elevation condition and the maximum-elevation condition must be considered when assessing the stability and structural integrity of the vehicle.

The components of the inertial recoil force It are defined by the following equations:

$$xIt = I_t \cdot cos\theta \cdot sin\psi \tag{1}$$

$$yIt = I_t \cdot cos\theta \cdot cos\psi \tag{2}$$

$$z_{It} = I_t \cdot sin\theta \tag{3}$$

Since the considered problem exhibits axial symmetry, the analysis of the azimuth angle can be limited to one direction without loss of generality. This approach reduces the number of required load cases and consequently improves the computational efficiency of the numerical analysis. The corresponding components of the recoil inertial force It are presented in the following Table 2.

Table 2 The corresponding components of the recoil inertial force It

Angles		**Components of the recoil inertial force It**		
θ [°]	ψ [°]	It_X [KN]	It_Y [KN]	It_Z [KN]
-5	60	137.6	79.4	-13.9
-5	30	79.4	137.6	-13.9
-5	0	0	158.9	-13.9
0	60	138.1	79.7	0
0	30	79.7	138.1	0
0	0	0	159.5	0

Table 2 (continuation)

Angles		Components of the recoil inertial force It		
θ [°]	ψ [°]	It_X [KN]	It_Y [KN]	It_Z [KN]
5	60	137.6	79.4	13.9
5	30	79.4	137.6	13.9
5	0	0	158.9	13.9
10	60	136	78.5	27.7
10	30	78.5	136	27.7
10	0	0	157	27.7
15	60	133.4	77	41.2
15	30	77	133.4	41.2
15	0	0	154	41.2
20	60	129.8	74.9	54.5
20	30	74.9	129.8	54.5
20	0	0	149.9	54.5

FEM ANALYSIS

The analysis was conducted as a preliminary structural assessment aimed at evaluating the feasibility of integrating the considered weapon system onto a mobile vehicle platform. Accordingly, generalized mechanical properties representative of conventional structural steel were adopted for the numerical model, including the corresponding Young's modulus, Poisson's ratio, and density. Material-specific properties associated with a particular steel grade or final production material were not considered at this stage. This approach was adopted because the objective of the analysis is to assess the general structural feasibility and load-transfer characteristics of the proposed integration concept rather than to provide a final design verification of the structure.

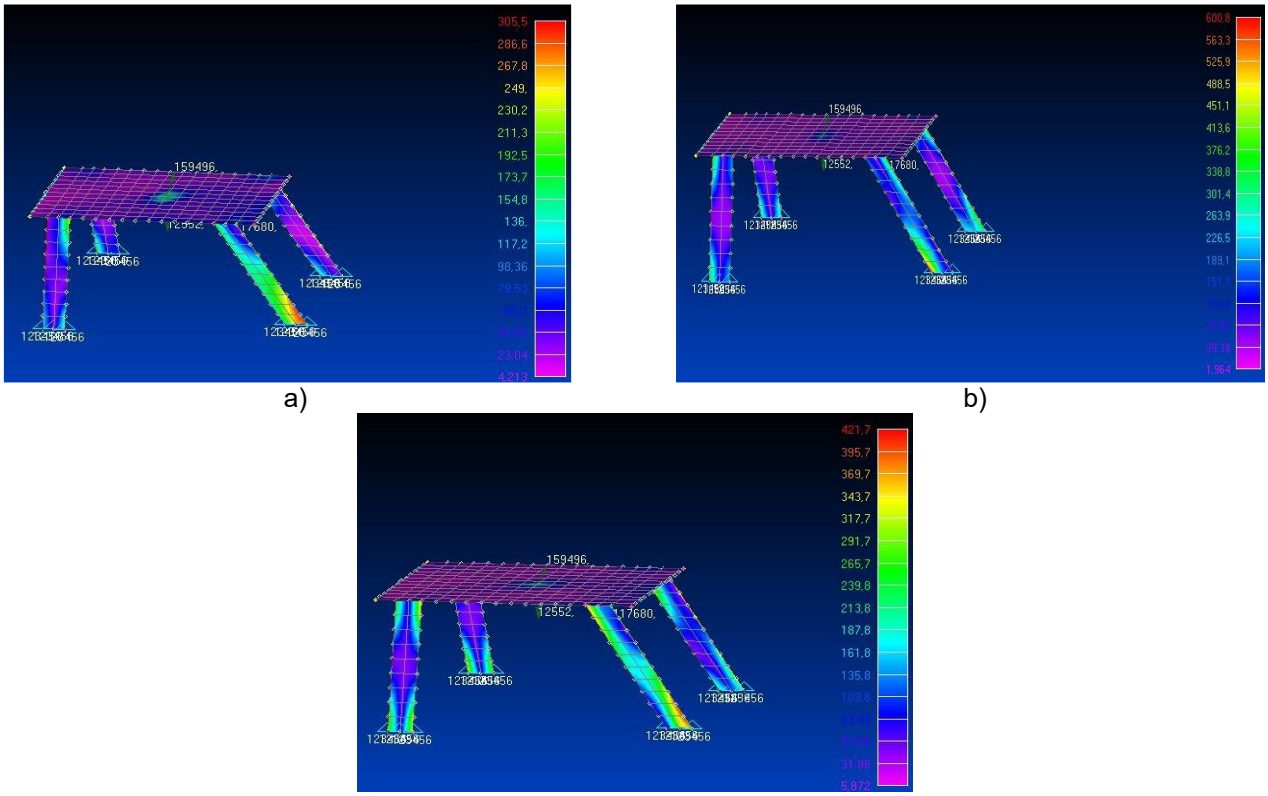
The FEM model of the vehicle supporting platform was developed from the three-dimensional geometric model of the supporting structure and hydraulic outriggers. The supporting structure consists of welded rectangular hollow sections with dimensions of 100 x 100 mm and a wall thickness of 4 mm. The hydraulic outriggers were modelled as integral load-bearing components of the platform. To reduce computational complexity while preserving the relevant structural characteristics, the geometry was simplified and discretized using four-node shell elements without midside nodes.

The FEM analysis was conducted for different combinations of elevation and azimuth angles, representing the considered loading conditions of the vehicle. For each configuration, the stress and deformation state of the supporting structure and the resultant reactions at supports A, B, C, and D were determined. The obtained numerical results were used to identify the critical loading conditions and to assess the structural integrity, load-bearing capacity, and stability of the supporting platform under impulsive loading. The magnitudes and positions of the forces acting on the supporting platform are defined according to the previously established mechanical model. The corresponding force values applied to the platform are presented in Table 3.

Table 3 Force values

Force	Value
Weight of the mounted system (Go)	12.552 N
Vehicle weight (Gv)	117.680 N
Inertial recoil force (It)	Table 2

The following section presents the FEM analysis of the supporting platform for three representative loading configurations defined by elevation and direction angles. These loading cases were selected to evaluate the structural response of the supporting platform for different orientations of the inertial recoil force It. The analysis considers the resulting stress distribution and support reactions, with the objective of determining the most critical loading condition and verifying the structural integrity of the platform. Static and modal analyses based on the finite element method provide useful information about the structural behaviour and dynamic characteristics of vehicle chassis structures (Rajappan & Vivekanandhan, 2013).



a) b)

c)

Figure 6 Support reactions and stress distribution for the investigated loading conditions a) θ=20°and ψ=30° with a maximum reaction of 34 kN at support A, b) θ=-5°and ψ=0° with a maximum reaction of 49 kN at support B, c) θ=-5°and ψ=60° with a maximum reaction of 86 kN at supports C and D

THE RESULTS

The FEM analysis was performed as a preliminary assessment of the proposed concept for integrating the weapon system onto a mobile vehicle platform. The analysis was conducted to evaluate the load-transfer characteristics of the supporting structure and to determine the support reactions resulting from the applied recoil force It for different combinations of elevation angle θ and azimuth angle ψ. The results were used to identify the critical loading configurations and the regions of the structure subjected to the highest loads. The results presented below correspond to the conceptual integration of the weapon system onto the mobile platform. For each analysed combination of θ and ψ, the reactions at supports A, B, C, and D were determined.

For θ=20° and ψ=30° the maximum reaction occurs at support A and is approximately 34 kN, as shown in Figure 6 a). For θ=-5° and ψ=0° the maximum reaction is obtained at support B, with a value of approximately 49 kN. The maximum overall reactions occur for θ=-5° and ψ=60° with approximately 87 kN at support C, and 86 kN at support D. As shown in Figure 6, the maximum stress concentrations occur at the lower sections of the hydraulic outriggers and in the central region of the supporting plate. The latter is primarily a consequence of the gravitational load of the weapon system. Based on the obtained results, the most critical loading condition is θ=-5° and ψ=60°, with the maximum support reaction. The support reactions resulting from the applied loading are summarized in Table 4. The table presents the maximum reaction obtained at each support, together with the corresponding values of θ and ψ at which the maximum reaction occurs.

Table 4 Results of support reactions A, B, C and D

Angles [°]		**Support A [N]**		
θ	**ψ**	**Oz**	**Ox**	**Oy**
20	60	7116	-11867	-13464
20	30	2986	-17580	-13746
20	0	-2860	-18491	-10134
Angles [°]		**Support B [N]**		
θ	**ψ**	**Oz**	**Ox**	**Oy**
-5	60	5640	-9718	6160
-5	30	826	-18276	14335
-5	0	-5530	-22660	21267

Table 4 (continuation)

Angles [°]		**Support C [N]**		
θ	**ψ**	**Oz**	**Ox**	**Oy**
-5	60	-27178	-9052	50833
-5	30	-18296	-14882	31997
-5	0	-6197	-16809	6395
Angles [°]		**Support D [N]**		
θ	**ψ**	**Oz**	**Oz**	**Oz**
-5	60	-27272	-7443	51189
-5	30	-18459	-13688	32612
-5	0	-6385	-16181	7107

The adopted horizontal angular range is consistent with the operating envelopes commonly encountered in existing concepts of self-propelled artillery systems and weapon system mounted on a two-support carriage. The considered azimuthal range therefore represents a realistic design assumption for the proposed integration concept. In contrast, the vertical field of operation is primarily determined by the elevation capabilities and mechanical characteristics of the weapon system intended for integration. The adopted elevation range is therefore defined in accordance with the operational requirements and intended application of the system. These angular limits provide an appropriate basis for evaluating the load transfer and structural response of the mobile platform within the intended operating envelope.

CONCLUSIONS

The present study investigated the structural response of a supporting platform mounted on a chassis using the FEM, with the primary objective of assessing the feasibility of integrating the considered weapon system onto an existing mobile platform. The analysis represents a conceptual assessment of the proposed integration and was performed to evaluate the general load-transfer behaviour of the supporting structure under the action of recoil forces. The FEM analysis was conducted for selected characteristic combinations of elevation angle θ and azimuth angle ψ, corresponding to loading configurations considered potentially critical for the supporting structure and vehicle stability. For each configuration, the resulting support reactions and structural response were determined. The numerical results show that the recoil load is predominantly transferred to the ground through the hydraulic outriggers. For θ=20° and ψ=30° the maximum reaction at support A is approximately 34 kN, while for θ=-5° and ψ=0°, the maximum reaction at support B is approximately 49 kN. The maximum support reactions are obtained for θ=-5° and ψ=60°, with approximately 87 kN at support C and 86 kN at support D. This configuration represents the governing load case among those considered. The calculated stress distribution indicates that the most critical regions are located in the vicinity of the hydraulic outriggers and in the central portion of the supporting plate. The results demonstrate that the elevation and azimuth angles have a significant influence on the load-transfer mechanism and the resulting support reactions. Due to the longitudinal symmetry of the structural model, the azimuth angle was analysed in only one direction, while the opposite direction would produce an equivalent response with the reactions exchanged between the corresponding symmetric supports.

It should be emphasized that the presented FEM analysis represents a preliminary engineering assessment intended to investigate the feasibility of the proposed integration concept. Standardized and generalized material parameters were adopted for the numerical model, without using material-specific properties corresponding to a particular production material or final structural design.

The adopted angular operating envelope provides a realistic basis for the proposed integration concept. The horizontal operating range is consistent with the characteristics of existing self-propelled artillery systems and weapon system mounted on a two-support carriage. The vertical operating range is primarily determined by the operational requirements and performance and mechanical characteristics of the weapon system intended for integration, while also being consistent with its intended purpose. The defined operating envelope therefore provides an appropriate basis for evaluating the structural response and load-bearing capability of the proposed mobile platform.

Consequently, the obtained results should be considered indicative of the structural response and load-transfer characteristics rather than as a final verification of the structural strength or serviceability of the platform. The results indicate that the proposed concept provides a basis for further investigation of the integration of the weapon system onto the selected mobile platform. For complete structural verification, subsequent analyses should incorporate the actual material properties and detailed structural geometry, together with more comprehensive static, dynamic, stability, and fatigue analyses. Experimental validation would additionally be required to confirm the numerical results and assess the structural behaviour of the final configuration under realistic loading conditions.

ACKNOWLEDGMENTS

The paper is a part of the research done within the project of the University of Defence in Belgrade, No. VA-TT/1/24-26.

REFERENCES

Dunica, Š., & Kolundžija, B. (1985). *Nonlinear analysis of structures*. University of Belgrade, Faculty of Civil Engineering. (In Serbian).

Ghazaly, N. M. (2014). Applications of finite element stress analysis of heavy truck chassis: Survey and recent development. *Journal of Mechanical Design and Vibration*, 2(3), 69–73. https://doi.org/10.12691/jmdv-2-3-3

Han Fui, T., & Abd. Rahman, R. (2007). Statics and dynamics structural analysis of a 4.5 ton truck chassis. *Jurnal Mekanikal*, 24, 56–67.

Kengkongan Ary, A., Rio Prabowo, A., & Imaduddin, F. (2020). Structural assessment of an energy-efficient urban vehicle chassis using finite element analysis: A case study. *Procedia Structural Integrity*, 27, 69–76. https://doi.org/10.1016/j.prostr.2020.07.010

Kojić, M., Slavković, R., Živković, M., & Grujović, N. (2010). *Finite element method I—Linear analysis*. University of Kragujevac, Faculty of Mechanical Engineering. (In Serbian).

Kwasniewski, L., Li, H., Wekezer, J., & Malachowski, J. (2006). Finite element analysis of vehicle–bridge interaction. *Finite Elements in Analysis and Design*, 42(11), 950–959. https://doi.org/10.1016/j.finel.2006.01.014

Rajappan, R., & Vivekanandhan, M. (2013). Static and modal analysis of chassis by using FEA. *The International Journal of Engineering and Science*, 2(2), 63–73.

Tančić, Lj. (2014). *Internal ballistics design*. University of Defence, Military Academy, Media Center Defence. (In Serbian).