BULETINUL INSTITUTULUI POLITEHNIC DIN IAŞI
Publicat de
Universitatea Tehnică „Gheorghe Asachi” din Iaşi
Volumul 71 (75), Numărul 4, 2025
Secţia
CONSTRUCŢII DE MAŞINI
DOI: 10.2478/bipcm-2025-0036

FINITE ELEMENT ANALYSIS OF A SCALED BRIDGE SPAN: INFLUENCE OF VEHICLE MASS ON DISPLACEMENTS

BY

ADI-MIHĂIȚĂ VELNICIUC, CARMEN BUJOREANU* **and ANA TUFESCU**

"Gheorghe Asachi" Technical University of Iași,
Faculty of Mechanical Engineering, Iași, Romania

Received: August 20, 2025
Accepted for publication: August 31, 2025

Abstract. This paper presents the design and finite element analysis (FEA) of a 1:25 scaled span of the Mircești Bridge over the Siret River, to investigate the influence of vehicle mass on structural displacements. A 60-meter span was modeled in SolidWorks, resulting in a 2.4-meter physical prototype. FEM simulations in CATIA V5 R19 incorporated realistic material properties for steel and concrete and used a detailed tetrahedral mesh. Static load cases between 2–12 kg was applied at mid-span to simulate various vehicle masses. The analysis targeted vertical displacements induced by varying vehicle masses. Results revealed a linear relationship between mass and displacement, confirming the model's ability to reflect elastic behavior under load. Despite the reduced scale, the structure demonstrated high stiffness and maintained displacement values within safe limits. The findings support the upcoming experimental validation phase and highlight the relevance of scaled modeling and FEM in assessing bridge performance under traffic loading.

Keywords: vehicle mass influence, bridge modelling, finite element analysis (FEA), displacement, scaled bridge model.

---

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

1. Introduction

The structural integrity of bridges under vehicular loading is a fundamental concern in civil and mechanical engineering. Accurate assessment of displacements caused by moving or stationary vehicles is crucial for ensuring safety, serviceability, and long-term durability. Traditionally, bridge behaviour under load has been studied through analytical models and field experiments, but these methods often require significant time and resources. Recent advances in computational tools, particularly finite element analysis (FEA), have allowed engineers to simulate complex load interactions with high precision.

This paper focuses on the design and simulation of a scaled span model of the Mircești Bridge, located over the Siret River in Romania. A 60-meter span of the bridge was selected and reduced to a 1:25 scale to create a 2.4-meter model, which was designed using SolidWorks software. This scaled prototype enables detailed structural analysis before physical fabrication and testing.

The core objective is to evaluate how different vehicle masses influence the displacements of the bridge structure. To achieve this, the model was subjected to static FEA using CATIA V5 R19. The analysis includes three loading scenarios corresponding to vehicle masses ranging from 2 to 12 kilograms, applied at the mid-span, a critical point for structural response (Velniciuc, 2025).

The study contributes both methodologically and practically by combining design precision with simulation accuracy. The results serve as a baseline for future dynamic testing and validation with the physical model. Ultimately, this work supports the development of efficient design verification strategies for bridges through scale modelling and virtual experimentation.

2. Bridge Model Design and Development

Based on the dimensional plans of the Mircești Bridge, located over the Siret River in Romania (Fig. 1), a decision was made to develop a metallic scale model representing one of its spans.

Fig. 1 – Real life view of the Mircești Bridge over Siret River.

From a longitudinal perspective, the bridge consists of four spans of varying lengths: 50.35 m, 60.00 m, 60.00 m, and 50.35 m (Fig. 2).

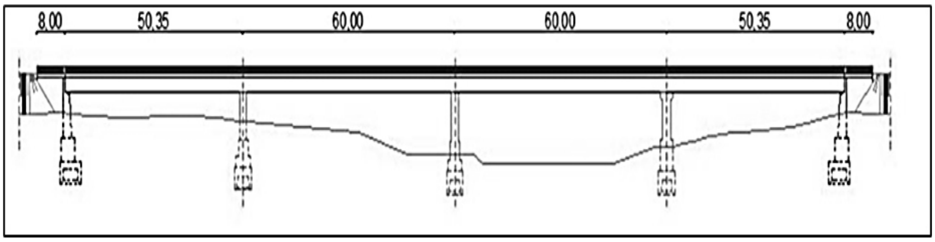

Fig. 2 − Spans of the Mircești Bridge (CNAIR, 2019).

Structurally, the bridge is a composite steel-concrete construction, based on a statically continuous girder system. The 60-meter span was selected for this study (Fig. 3).

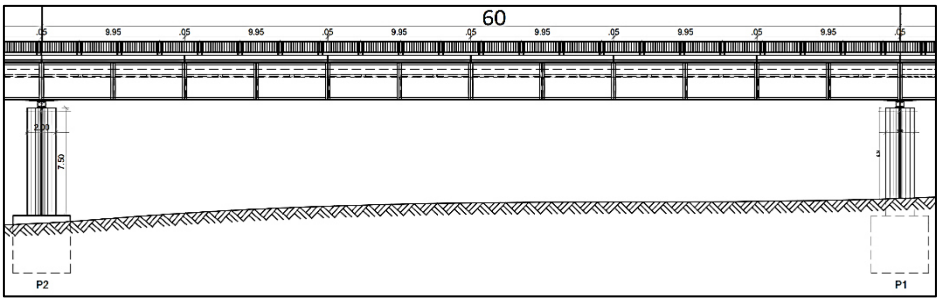

Fig. 3 – The 60 m span of the Mircești Bridge (CNAIR, 2019).

To physically replicate the selected bridge span, a scale ratio of 1:25 was adopted, leading to the development of a scaled model with a total length of 2.4 meters. This scaling approach enabled the preservation of geometric proportions while ensuring the model remained suitable for both digital analysis and future physical fabrication. The entire design process was carried out using SolidWorks 2023, a professional computer-aided design (CAD) software that allowed for high-precision modeling and the management of complex assemblies. All dimensions from the original bridge structure were carefully converted to the selected scale, ensuring fidelity to the real-life configuration. Each structural element of the bridge, such as longitudinal beams (Fig. 4), cross members, bracing elements, and connection details-was individually modeled in separate design files (Velniciuc, 2025).

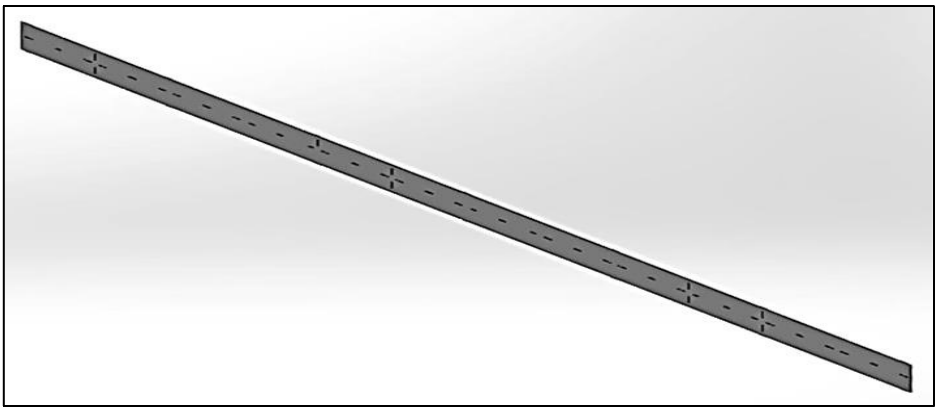

Fig. 4 – Design of bridge lateral beams.

To ensure dimensional precision and to facilitate real-world assembly, design enhancements were implemented by incorporating an interlocking nut-and-feder system. As part of this approach, technological cutouts were introduced to achieve accurate alignment and secure fitting of all components.

The model was designed with the assumption that all parts would be manufactured from DC01 steel sheets, processed through laser cutting. These components were then assembled into the overall structure through the systematic application of dimensional constraints, such as coincident, concentric, and distance relations. This procedure ensured proper alignment and connectivity between parts, reflecting the intended structural behavior of the original system (Velniciuc, 2025).

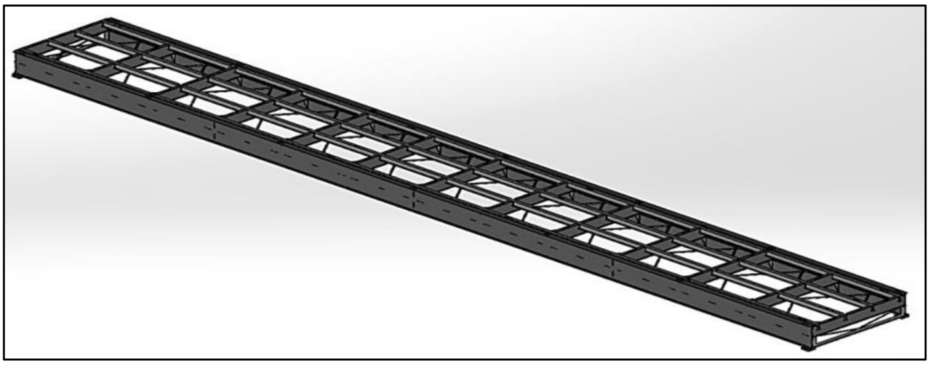

Fig. 5 – Isometric view of the upper part of the bridge span steel structure model.

In order to accommodate manufacturing and assembly tolerances, all interlocking features were designed with an additional tolerance of +0.2 mm. The final model consists of 320 metallic parts with thicknesses ranging from 0.8 mm to 2.5 mm, specifically tailored to meet the structural and assembly requirements of the scaled representation (Figs. 5-6).

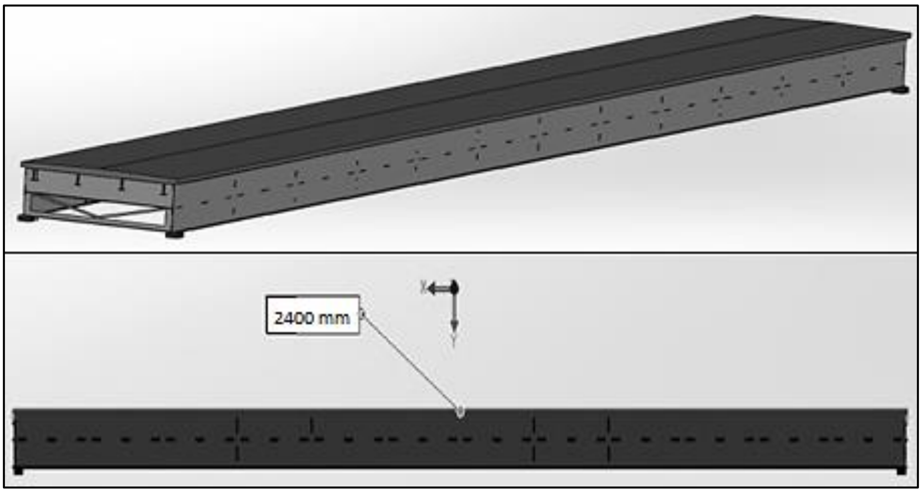

Fig. 6 – Complete assembled bridge span model (steel structure and bridge deck).

The parametric nature of the CAD model also allowed for easy updates and design modifications, supporting both manufacturability and structural analysis considerations. This step-by-step assembly process contributed to the development of a geometrically accurate and functionally coherent digital representation of the bridge span, forming a reliable basis for subsequent finite element simulations and prototype fabrication.

3. Finite Element Analysis (FEA)

Finite Element Analysis (FEA) has become an essential methodology in the field of structural engineering, enabling detailed simulation and evaluation of complex structural systems operating under various load conditions. Its widespread adoption is attributed to its ability to provide precise and comprehensive insights into structural behaviour, thereby supporting the refinement of design strategies and minimizing the need for extensive experimental testing (Qassim and Ali, 2020; Ma and Zhang, 2018; Hino *et al*., 1984; Zienkiewicz *et al*., 2005; Bathe, 1996). The technique involves segmenting a continuous structure into a finite set of smaller elements, each described by mathematical shape functions that facilitate accurate numerical representation of physical responses (Hutton, 2004).

One of the key strengths of finite element analysis lies in its adaptability to complex geometrical configurations and its capacity to handle a wide range of material characteristics and loading scenarios. It can be effectively applied to simulate both static and dynamic structural behaviours, making it a versatile tool in engineering investigations (Cook *et al*., 2001; Brownjohn *et al*., 2001; Ding *et al*., 2012; Kwasniewski *et al*., 2006).

Additionally, FEA allows for the detailed modelling of different support conditions and loading cases, which facilitates a comprehensive evaluation of the factors influencing structural performance.

In this article, the finite element method was employed to assess the structural behaviour of a scaled metallic representation of the Mircești Bridge, reduced to a 1:25 ratio (Velniciuc, 2025).

The simulations were carried out using CATIA V5R19 (Nader, 2008; Ottosen and Petersson, 1992; González, 2010), a software environment known for its ability to perform detailed static analyses with high geometric accuracy. The digital model of the bridge span, initially developed in SolidWorks, was imported into CATIA to enable numerical evaluation.

The main goal of this analysis was to quantify the vertical displacements resulting from different vehicular mass scenarios, with loads positioned at the central section of the bridge model.

3.1. Model definition

The 3D model of the metallic bridge span from Mircești already developed using SolidWorks (Fig. 7) was imported into CATIA V5R19, where the finite element analysis (FEA) was performed under static loading conditions (Velniciuc, 2025).

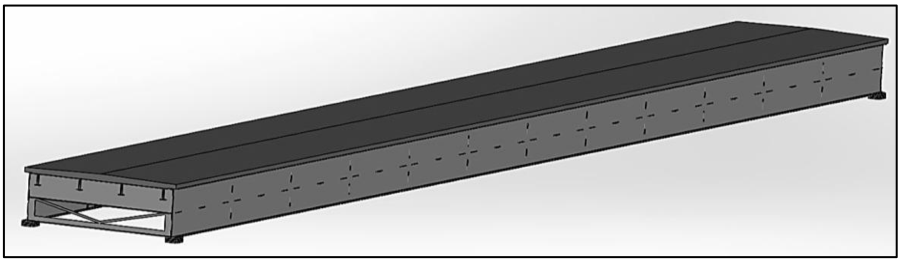

Fig. 7 – 3D model of the scaled Mircești Bridge span, created in SolidWorks.

In order to accurately simulate the impact of vehicle mass on the bridge's displacements, the model was configured by defining several critical parameters. These included the mechanical properties of the metallic structural elements and the materials used for the bridge deck, the selection of finite elements employed in the mesh discretization, the application of boundary conditions to represent the bridge's support system, as well as the positioning and orientation of the applied loads.

The bridge is modeled as a composite structure composed of steel and reinforced concrete, with the interaction between the steel framework and the concrete slab explicitly taken into account. In accordance with current engineering standards, the steel elements were assigned a material grade of S355, while the concrete slab was represented using high-performance concrete of class C30/37. Both materials were assumed to exhibit isotropic behavior, and their corresponding mechanical properties are detailed (Table 1), (Velniciuc, 2025).

Table 1

Material properties employed in the definition of the FEM model

Material	Steel S355	Material	Reinforced concrete C30/37
Young's modulus	*2.1x10^5 [MPa]*	*Young's modulus*	*3.2x10^4 [MPa]*
Poisson's ratio	*0.3*	*Poisson's ratio*	*0.2*
Density	*7860 [kg/m^3]*	*Density*	*2500 [kg/m^3]*
Yield strength	*355 [MPa]*	*Yield strength*	*30 [MPa]*

The finite element mesh (Fig. 8) was created by dividing the entire geometry into smaller, discrete elements, allowing for a distributed calculation of the structure's mechanical properties.

The mesh was primarily composed of TE10 parabolic tetrahedral solid elements, 10-node isoparametric elements known for their high accuracy in capturing complex geometries, though they require greater computational resources (Velniciuc, 2025).

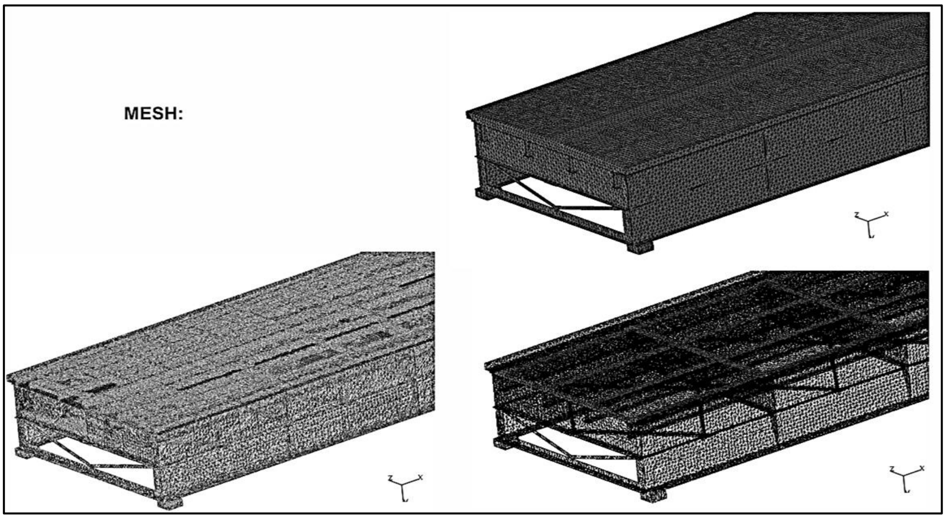

Fig. 8 – The mesh of the analysis model.

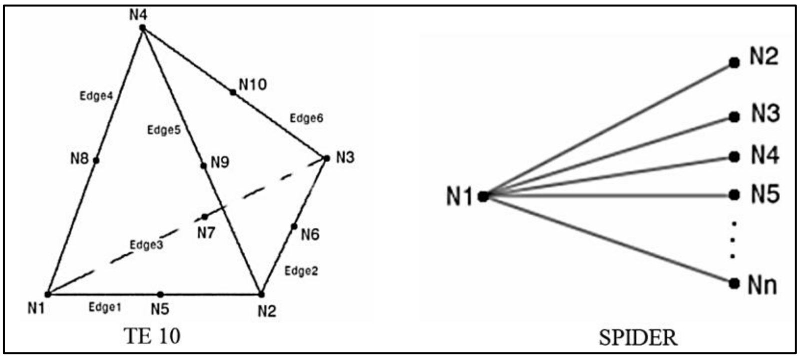

Fig. 9 – Type of elements used in the discretization mesh (Catiadoc.free, 2024).

The final mesh included approximately 1.98 million nodes and 1.1 million elements. Among these, 96.32% were TE10 elements, while the remaining 3.68% consisted of SPIDER-type elements, which were specifically used in transitional regions (Fig. 9).

Boundary conditions play a crucial role in the accuracy of FEA simulations. For this model, fixed supports were applied at both ends of the bridge span (Fig. 10), simulating a rigid anchorage into the infrastructure.

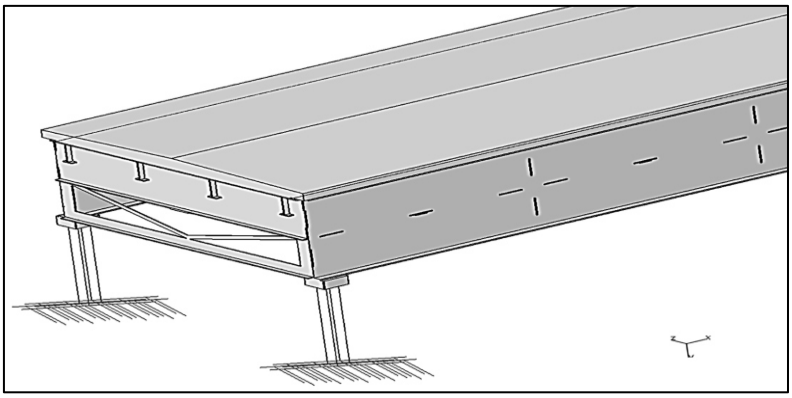

Fig. 10 – Boundary conditions applied to the analysis model.

Load positioning is another decisive factor in structural analysis. In this study, the loads-representing vehicle masses-were centrally applied, as this location typically generates the maximum bending moment and displacement in beam-like structures. The loads were defined as concentrated forces acting along the vertical (Z) axis, perpendicular to the surface of the bridge deck (Fig. 11), (Velniciuc, 2025).

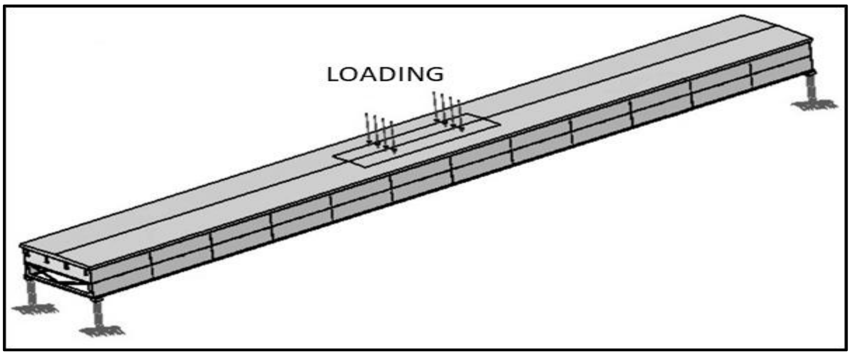

Fig. 11 – Positioning of loads on the model used in the finite element analysis.

This strategy enabled a focused evaluation of structural behavior under critical loading scenarios, supporting the identification of stress concentrations and zones potentially requiring reinforcement.

3.2. Loading scenarios

After defining the model, three distinct loading scenarios were established. In each case, the force was applied at the mid-span of the bridge to simulate different vehicle mass configurations: 2 kg, 6 kg, and 12 kg, respectively. These specific values were selected to match those planned for an upcoming physical experiment, which will be carried out once the physical model is completed. Table 2 presents a detailed summary of the loading conditions used in the FEM analysis to assess the structural response under varying mass loads (Velniciuc, 2025).

Table 2
Loading cases for the finite element analysis of the 3D bridge model

Case	Load placement	Load mass [kg]	Load force [N]
C1	*Bridge mid-span: 1210 mm*	*2*	*19.61*
C2	*Bridge mid-span: 1210 mm*	*6*	*58.84*
C3	*Bridge mid-span: 1210 mm*	*12*	*117.68*

3.3. Results and discussions

Following the execution of simulations in CATIA, displacement distributions were obtained for each of the three loading scenarios (Figs. 12–14). The analysis focused on vertical displacements along the bridge span, with special emphasis on the maximum deflections occurring along the Z-axis.

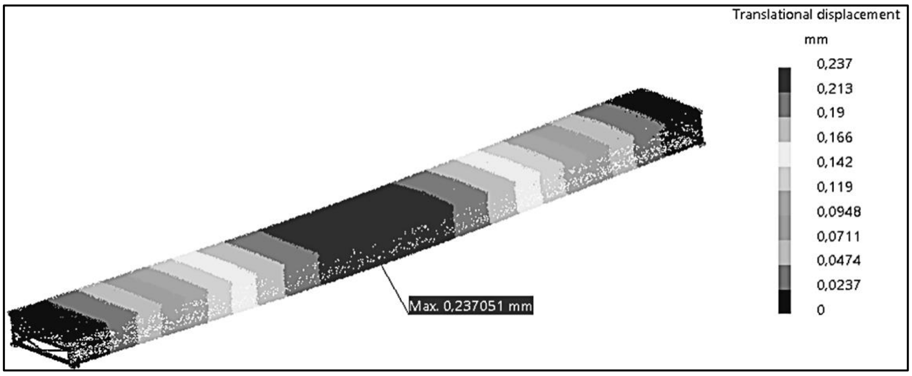

Fig. 12 – Displacements obtained for Case C1.

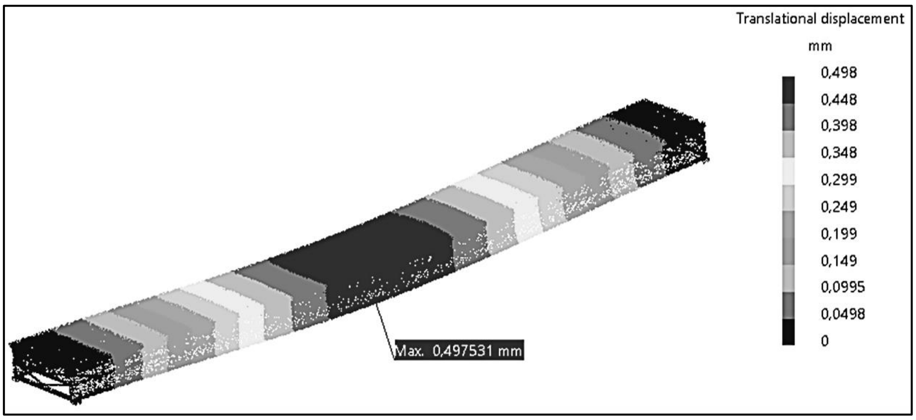

Fig. 13 – Displacements obtained for Case C2.

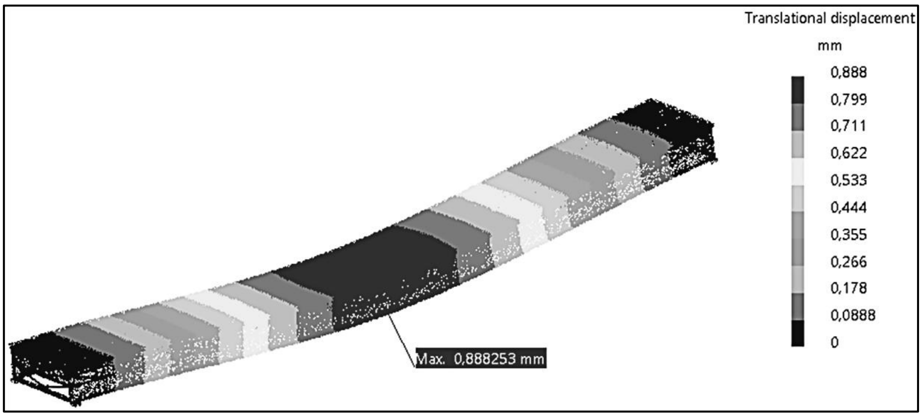

Fig. 14 – Displacements obtained for Case C3.

All displacement values were reported as negative, indicating downward movement relative to the initial, undeformed state of the structure. Table 3 summarizes the peak displacements recorded for each case (Velniciuc, 2025).

Table 3
Displacement summary for the Mircești bridge – FEM results for the three cases

Case	Load placement	Load mass [kg]	Displacement [mm]
C1	*Bridge mid-span: 1210 mm*	*2*	*-0.2*
C2	*Bridge mid-span: 1210 mm*	*6*	*-0.5*
C3	*Bridge mid-span: 1210 mm*	*12*	*-0.9*

In Case C1, a vehicle mass of 2 kg (equivalent to 19.61 N) was applied, resulting in a vertical displacement of -0.2 mm, indicating minimal deformation under light loading. For Case C2, the applied mass increased to 6 kg (58.84 N), leading to a displacement of -0.5 mm, which represents a 150% increase compared to Case C1 and demonstrates the structure's sensitivity to increased loads. In Case C3, a mass of 12 kg (117.68 N) was applied, resulting in a displacement of -0.9 mm - 4.5 times greater than in Case C1 and 80% higher than in Case C2 - confirming a clear and consistent increase in deformation with higher loading conditions (Velniciuc, 2025).

These results reveal a linear correlation between the applied mass and the observed vertical displacement, which aligns with the expected elastic response of materials within their operational limits. While the structural deformation is perceptible, the actual values remain small in absolute terms relative to the total length of the bridge span (Velniciuc, 2025).

The limited displacements observed across all three scenarios suggest a high degree of stiffness within the structural model, confirming its capacity to endure significant loads without exhibiting critical deformation. The FEA results will serve as a benchmark for comparison against experimental measurements obtained from the physical prototype. This dual approach ensures consistency between theoretical predictions and real-world observations.

4. Conclusions

The conducted research demonstrates the effectiveness of using a scaled physical model and finite element analysis (FEA) for investigating the structural response of bridge spans under varying vehicular loads. The study focused on a 60-meter span of the Mircești Bridge, modeled at a 1:25 scale, which enabled a detailed and geometrically faithful replication of the original structure. The digital model, designed in SolidWorks and analyzed in CATIA V5 R19, incorporated realistic material properties and accurate boundary conditions to simulate actual support and loading conditions.

The finite element simulations conducted for different vehicle masses revealed a linear relationship between applied load and vertical displacement. This outcome confirms the elastic behavior of the structure within the analyzed load range and validates the structural stiffness and load-bearing capacity of the scaled model. The displacements recorded, although perceptible, remained well within the permissible limits, indicating the adequacy of the design and the precision of the mesh and material definitions used in the simulations.

These findings provide a reliable computational foundation for the upcoming experimental phase, where the physical prototype will be tested dynamically. The close correlation between numerical predictions and anticipated experimental outcomes enhances confidence in the adopted modeling methodology and supports its applicability in future studies of bridge dynamics under vehicular influence.

Overall, this work substantiates the utility of combining CAD-based design, high-fidelity finite element modeling, and physical prototyping in the structural analysis of bridges. It also underscores the importance of scaled modeling in reducing development costs and facilitating controlled testing environments for validating structural performance.

REFERENCES

Bathe K.J., *Finite Element Procedures*, Prentice Hall, 1996.

Brownjohn J.M.W., Moyo P., Omenzetter P., Lu Y., *Assessment of highway bridge upgrading by dynamic testing and finite-element model updating*, Journal of Bridge Engineering, 6(6), 2001, pp. 400-406.

CNAIR, (2019), Regional Directorate of Roads and Bridges Iasi, Technical execution documentation 118/2019 - "Periodic maintenance of the bridge located on DN 28 at km 6+957, over the Siret River, in Scheia".

Cook R.D., Malkus D.S., Plesha M.E., Witt R.J., *Concepts and Applications of Finite Element Analysis*, 4th ed., Wiley, 2001.

Ding Y., Chen B., Peng Y., *Testing and numerical analysis on a highway steel bridge under vehicle loads*, Advanced Steel Construction, 8(2), 2012, pp. 158-170.

González A., *Vehicle–bridge dynamic interaction using finite element and modal superposition methods*, Journal of Sound and Vibration, 329(14), 2010, pp. 2817-2836.

Hino J., Yasuda T., Naito T., *Numerical study on the interaction of moving vehicle and bridge structure*, Journal of Sound and Vibration, 97(2), 1984, pp. 195-206.

Hutton D.V., *Fundamentals of Finite Element Analysis*, McGraw-Hill, 2004.

Kwasniewski L., Jones N., *Finite element analysis of bridge deck response to moving loads*, Engineering Structures, 28(3), 2006, pp. 457-465.

Ma H., Zhang Y., *Effects of vehicle mass on dynamic behavior of long-span bridges*, Journal of Bridge Engineering, 23(2), 2018.

Nader G., *Finite Element Modeling in Engineering Practice*, Alpha Science, 2008.

Ottosen N.S., Petersson H., *Introduction to the Finite Element Method*, Prentice Hall, 1992.

Qassim R., Ali A., *Application of FEM for the prediction of bridge response under vehicular load*, Procedia Engineering, 172, 2020, pp. 771-778.

Velniciuc A.M., *Cercetări privind influența masei și vitezei vehiculelor asupra deplasărilor podurilor*, doctoral thesis, in press.

Zienkiewicz O.C., Taylor R.L., Zhu J.Z., *The Finite Element Method: Its Basis and Fundamentals*, 6th ed., Elsevier Butterworth-Heinemann, 2005.

ANALIZA CU ELEMENTE FINITE A UNEI DESCHIDERI SCALATE DE POD: INFLUENȚA MASEI VEHICULELOR ASUPRA DEPLASĂRILOR

(Rezumat)

Acest articol prezintă proiectarea și analiza cu elemente finite (FEM) a unei deschideri scalate a podului de la Mircești, amplasat peste râul Siret, având ca obiectiv investigarea influenței masei vehiculelor asupra deplasărilor structurale. A fost selectată o deschidere de 60 de metri, modelată la scara 1:25 în SolidWorks, rezultând o machetă cu lungimea de 2,4 metri. Simulările FEM au fost realizate în CATIA V5 R19, modelul integrând proprietăți realiste ale materialelor pentru componentele din oțel și beton și fiind discretizat cu o rețea detaliată de elemente tetraedrice. Au fost aplicate cazuri de încărcare statică în zona mediană, corespunzătoare unor mase variind între 2 și 12 kg, pentru a simula sarcini generate de vehicule. Analiza s-a concentrat asupra deplasărilor verticale generate de variația masei vehiculelor, evidențiind regiunile structurale cele mai solicitate. Rezultatele au evidențiat o relație liniară între masă și deplasare, confirmând capacitatea modelului de a reproduce comportamentul elastic al structurii sub acțiunea sarcinilor. În ciuda scării reduse, structura a demonstrat o rigiditate ridicată, cu valori ale deplasărilor menținute în limite sigure. Concluziile sprijină etapa viitoare de validare experimentală și subliniază relevanța utilizării machetelor scalate și a analizei FEM în evaluarea comportamentului podurilor supuse încărcărilor din trafic rutier.