

BULETINUL INSTITUTULUI POLITEHNIC DIN IAȘI
Publicat de
Universitatea Tehnică „Gheorghe Asachi” din Iași
Volumul 72 (76), Numărul 1, 2026
Secția
CONSTRUCȚII DE MAȘINI
DOI: 10.2478/bipcm-2026-0010

DEVELOPMENT OF A SPECIALIZED FIXTURE FOR TENSILE TESTING OF LIFTING EYES USED IN DOCK LEVELER HANDLING

BY

ANDREI-EMANUEL ANCUȚA* and BOGDAN-DOREL CIOROAGĂ

University Politehnica Timișoara,
Faculty of Engineering in Hunedoara

Received: May 31, 2026

Accepted for publication: June 9, 2026

Abstract. This study presents a specialized fixture designed for tensile testing machines to evaluate the mechanical behavior of lifting eyes used in dock leveler handling. The device overcomes the limitations of conventional tensile setups, which cannot reproduce the complex loading conditions generated during forklift operations. It consists of upper and lower subassemblies that enable realistic load transfer and accurate boundary condition simulation. The design process included geometric optimization and structural validation to ensure rigidity, durability, and minimal influence on specimen behavior, supporting loads up to 40 kN. Proper alignment and repeatable positioning were prioritized to improve testing consistency, while the fixture was engineered to reduce parasitic stresses and ensure uniform load distribution. Experimental results confirm that the fixture allows the acquisition of experimentally relevant structural response data under realistic loading conditions and enables more representative loading conditions compared to conventional direct gripping configurations. Overall, the solution enhances safety evaluation and supports

*Corresponding author; *e-mail*: andrei_ancuta@yahoo.com

© 2026 Andrei-Emanuel Ancuța and Bogdan-Dorel Cioroagă

This is an open access article licensed under the Creative Commons Attribution-NonCommercial-NoDerivatives 4.0 International License (CC BY-NC-ND 4.0).

refined design of lifting components, offering a methodology adaptable to other mechanically loaded systems.

Keywords: design, simulation, device, tensile, handling.

1. Introduction

Ensuring the safety and structural integrity of handling equipment used in industrial handling systems requires reliable mechanical testing that replicates real operational conditions. In applications such as dock levelers and heavy equipment handling, lifting eyes transfer complex loads during lifting, positioning, and manipulation operations. However, conventional tensile testing machines are typically designed for standard specimens and do not inherently support the geometric and loading complexities of lifting devices. As a result, there is a critical need for custom fixtures that adapt standard tensile testing equipment to evaluate lifting eyes accurately and reproducibly under service-representative conditions.

The design and development of adapters and fixtures for tensile testing machines have been the subject of extensive research in material testing literature. A foundational example is the work by Zhang et al., who present the design of a biaxial tensile fixture that enables biaxial stress states to be applied using a uniaxial tensile machine, demonstrating that existing testing infrastructure can be modified to replicate more complex loading conditions with high accuracy and low error (load error < 0.4%) (Zhang *et al.*, 2024). Similarly, Zhou et al. describe the practical design and application of a biaxial tensile fixture based on a uniaxial tensile machine, showing that motion conversion mechanisms can accurately produce biaxial deformation for cruciform specimen testing, highlighting the feasibility of adapting widely available testing systems without requiring specialized equipment (Zhou *et al.*, 2020). These works underscore the importance of integrating tailored mechanical fixtures with standard testing machines to expand their experimental capabilities.

While these studies focus on fixtures primarily for material characterization under multiaxial load states, the mechanical behavior of lifting eyes and lugs under complex service loads has also been the focus of advanced research. Recent work by Chen et al. investigates the dynamic mechanical behavior of welded-plate lifting lugs subjected to eccentric and dynamic loads, revealing significant stress concentrations and non-uniform stress distributions that are influenced by loading angle and dynamic effects (Chen *et al.*, 2026). This research emphasizes that mechanical testing of lifting components should consider non-standard load paths and dynamic amplifications that occur during real lifting operations, rather than relying solely on simple uniaxial tensile conditions. Complementing this, Choi and Lee provide an analysis of fracture

behavior in lifting lugs, using finite element analysis (FEA) combined with strength calculations to identify stress concentrations at the pin-hole interface and to assess fatigue life predictions, demonstrating the importance of numerical modeling in understanding failure mechanisms (Choi and Lee, 2026).

Finite element analysis has become a vital tool both for design optimization of lifting devices and for validating custom test fixtures. For instance, Lee et al. conducted a structural stability evaluation of lifting lugs for marine engines using FEA, illustrating how numerical approaches inform safe loading conditions and boundary constraints during design, particularly for heavy industrial components with complex geometries (Lee *et al.*, 2007). Likewise, international fatigue studies, such as tension failure assessments at lug hole edges carried out using FEA and fracture mechanics models, provide insight into crack growth and failure paths relevant for lifting eye testing and highlight the need for fixtures capable of replicating realistic stress states (Fisher, 2019).

Beyond handling tools, research on fixture design for tensile testing itself continues to evolve. For example, Ru et al. developed an equal-biaxial test device for large deformation in cruciform specimens where the test apparatus ensures equal stress states and minimizes installation deviation, demonstrating how fixture design can directly impact experimental repeatability and accuracy (Ru *et al.*, 2022). Another example in fixture design is ASTM-standard research on biaxial tensile fixtures for fatigue testing of uniaxial machines, which validates that such adaptations can achieve reliable results with minimal fixture-induced error, reinforcing the suitability of retrofit fixtures for advanced mechanical testing (Zhang *et al.*, 2024).

Collectively, this body of literature indicates that accurate mechanical assessment of lifting eyes requires not only consideration of geometric and loading complexities but also a robust methodology for fixture design and validation. Custom adapters must account for load multiaxiality, dynamic effects, and boundary conditions representative of operational contexts, while numerical simulations such as FEA play a crucial role in both optimizing fixture geometry and ensuring that the test conditions match service requirements. In the context of lifting eyes for dock leveler applications, these considerations motivate the development of a novel adaptive fixture that integrates with standard tensile testing machines and provides a platform capable of replicating realistic load paths. By doing so, the research contributes to improved experimental fidelity, enhanced understanding of lifting eye behavior under representative conditions, and support for safer design practices in industrial applications.

The primary contribution of this work is the development and validation of a specialized adaptation fixture intended to enable tensile testing of lifting eyes under service-representative loading conditions using a conventional uniaxial tensile testing machine. The lifting eye specimen is employed as a

demonstrative application case for validating the structural functionality, rigidity, and load transfer capability of the proposed fixture concept.

2. Materials and Methods

This study primarily addresses the design and validation of a specialized adaptation fixture intended to facilitate tensile testing of lifting eyes under loading conditions representative of real handling operations.

The lifting eyes investigated in this study were manufactured from structural steels of grades S235 and S355, in accordance with the European standard EN 10025-2 (European Committee for Standardization, 2019). These materials are widely used in load-bearing applications due to their well-characterized mechanical properties, including adequate yield strength, good weldability, and availability in plate form.

The lifting eye was designed as a welded assembly composed of multiple components. The main structural element consists of a ring fabricated from laser-cut steel plate. In industrial practice, the thickness of these plates typically varies between 6 mm and 14 mm. In the present study, a conservative approach was adopted by selecting the minimum thickness of 6 mm, representing the most unfavorable configuration in terms of load-bearing capacity. This assumption allows for the evaluation of the structural integrity of the lifting eye under critical tensile loading conditions encountered during service.

Additional components include cylindrical bushings, which close the ring geometry and provide a guiding interface for an M16 bolt. This bolt ensures the fixation of the lifting eye into threaded holes provided on the handled components. The entire assembly is realized through welded joints, resulting in a monolithic test specimen.

The tested specimen, together with its technical execution details, is presented in Fig. 1. Detail A illustrates the engineering drawing of the welded assembly, while Detail B shows the physically manufactured specimen prepared for testing.

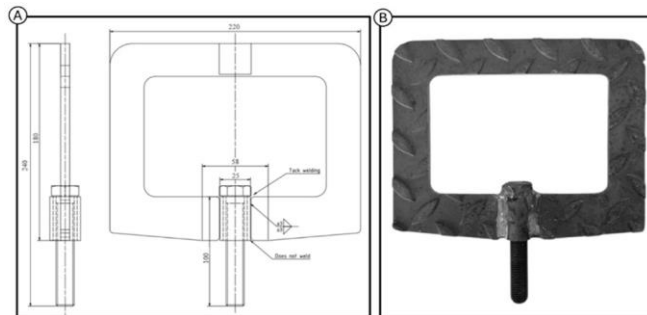


Fig. 1 – Lifting eye specimen: (A) execution drawing of the welded assembly; (B) manufactured specimen prepared for tensile testing.

The experimental investigation was carried out using uniaxial tensile tests. The validation criteria for lifting eye performance were defined based on two limiting conditions: (i) reaching a tensile load of 40 kN, which represents the target service load, and (ii) premature failure of the specimen prior to reaching this load level. In both cases, the test was terminated once either condition was met.

To ensure that the applied energy is predominantly absorbed by the specimen rather than dissipated through deformation of the testing fixture, a safety factor of $k = 3$ was introduced in the fixture design. Consequently, the fixture was dimensioned for a load of 120 kN. This value was further used as the reference load in the numerical simulations performed via finite element analysis.

During the conceptual development stage, the fixture was designed for a conservative load level of 120 kN to ensure adequate structural capacity independently of the testing equipment subsequently employed for the experimental investigations. This approach also allowed the fixture to maintain compatibility with tensile testing machines of different capacities and configurations.

The tensile tests were conducted using a Lab Test 6.5 universal testing machine, with a nominal capacity of 65 kN. The specimen was clamped using standard grips equipped with serrated jaws, operating on a vice-type mechanism actuated by two lateral screws. The gripping system also includes a central hole for attachment to the testing machine, secured by a locking pin. The experimental setup, including the testing machine (Detail A) and the gripping system (Detail B), is shown in Fig. 2.

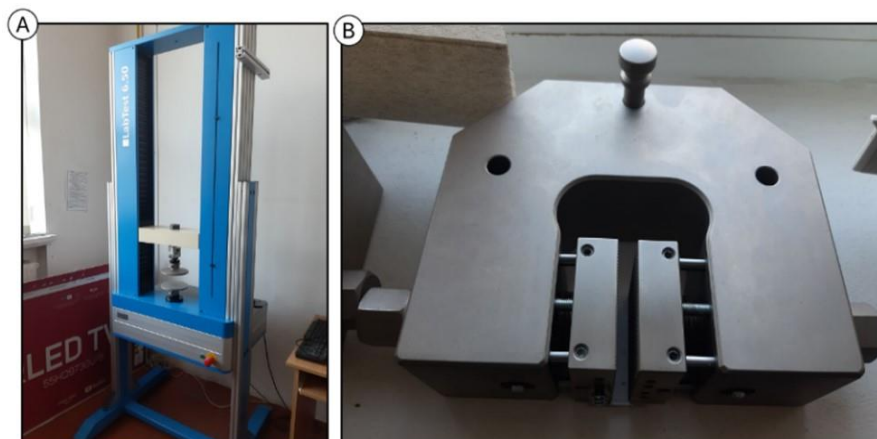


Fig. 2 – Experimental setup: (A) tensile testing machine (Lab Test 6.5); (B) standard gripping system with serrated jaws used for specimen clamping.

Therefore, the experimental campaign was intended to validate the operational functionality and load transfer capability of the fixture rather than its structural behavior under the full numerical verification load.

A finite element analysis (FEA) was performed to simulate the experimental conditions and to evaluate the stress distribution within the lifting eye assembly and the testing fixture. The boundary conditions were defined to replicate the real test configuration.

The lower grip of the testing machine was modeled as a fixed constraint, while a tensile force was applied to the upper grip, oriented vertically upward and acting from the geometric center of the grip. All components of the fixture, illustrated in Fig. 3, were defined as being in direct contact with each other using bonded contact conditions, ensuring no relative motion between contact surfaces.

Bonded contact conditions were adopted as a simplifying assumption in order to evaluate the global structural response and load transfer mechanism of the assembly while maintaining numerical stability and reducing computational complexity.

All parts of the assembly were arranged coaxially along the vertical axis, consistent with the physical testing setup. This modeling approach ensures an accurate representation of load transfer and stress distribution within the system during tensile loading.

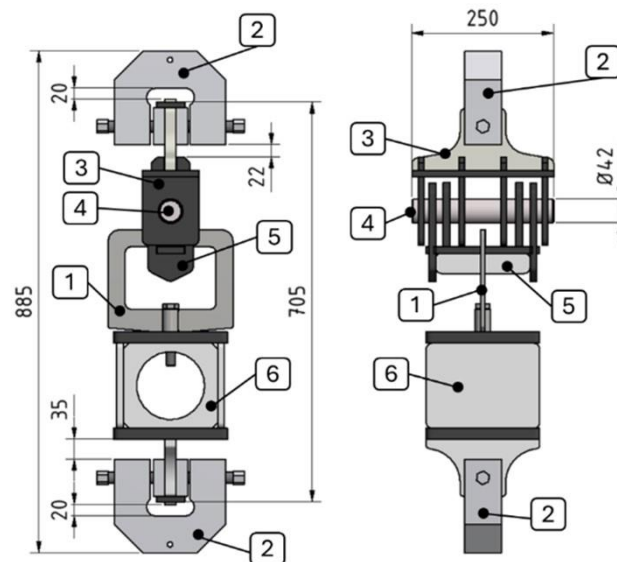


Fig. 3 – Finite element model of the testing assembly, showing the lifting eye specimen and fixture components with bonded contact conditions and coaxial alignment.

As illustrated in Fig. 3, the finite element model of the testing assembly comprises several key components that replicate the experimental configuration and the real operating conditions of the lifting system. The lifting eye itself (1), representing the test specimen, is mounted within the fixture and constitutes the primary load-bearing element under investigation. The assembly is interfaced with the tensile testing machine through the standard gripping jaws (2), which ensure proper transmission of the applied load. The upper part of the fixture (3) is connected to the movable grip of the testing machine and serves as the load introduction element. The connection between this upper component and the specimen holding element is achieved by means of a $\varnothing 42$ mm bolt (4), which also plays a functional role in simulating the contact with the forklift fork typically used during handling operations. The element responsible for holding the lifting eye within the fixture (5) is specifically designed to reproduce the geometry and loading conditions associated with the forklift fork. At the lower end of the assembly, the base component (6) simulates the platform of the handled product, into which the lifting eye is fastened via an M16 threaded connection. This base element is fixed to the lower grip of the testing machine, ensuring the overall stability of the system and enabling accurate application of tensile loading conditions throughout the test.

Fig. 4 presents a detailed view of the three main components that constitute the adaptation fixture designed for the tensile testing machine. As previously introduced, components (3) and (6) are provided with specially designed gripping zones that allow secure fixation within the machine jaws.

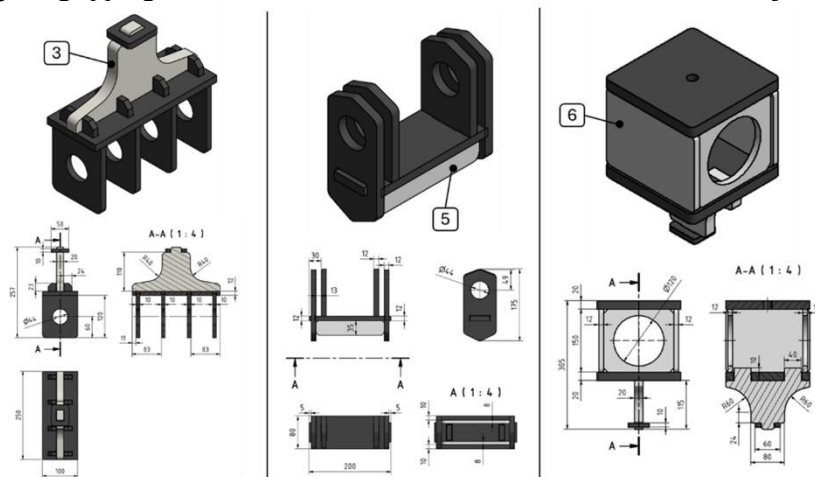


Fig. 4 – Detailed representation of the tensile testing adaptation fixture and principal structural components.

These areas also incorporate a wing-tip type geometry at the extremities of the clamping regions, intended to prevent slippage during loading.

Components (3) and (5) are mechanically interlocked, ensuring proper alignment of the connection holes through which the fixture is secured using the Ø42 mm bolt. This assembly step is performed after positioning the lifting eye specimen, with its ring placed onto the support surface of the holding element.

All subassemblies are manufactured using a combination of poka-yoke type mechanical interlocking features and welded joints, ensuring both correct positioning and structural integrity. The plates used in the gripping regions are fabricated from 20 mm thick steel sheets, while the plate in which the threaded rod of the test specimen is mounted within component (6) also has a thickness of 20 mm. Component (6) is further provided with a central hole of 120 mm diameter, designed to facilitate access for tightening the nut used to secure the lifting eye within the fixture.

Additional constructive features include reinforcing plates and vertical interlocking tabs (for components (3) and (5)), which are manufactured from 10 mm thick steel plates. These elements contribute to the increased rigidity and load transfer capacity of the overall fixture, ensuring reliable performance under tensile loading conditions.

All structural components of the testing fixture were manufactured from S355 structural steel in order to ensure sufficient rigidity and load-bearing capacity under the imposed loading conditions.

3. Results and Discussion

Finite element analyses were performed on the complete assembly of the tensile testing fixture in its operational configuration.

The finite element simulations were performed assuming linear elastic isotropic material behavior. The material properties introduced in the numerical model were based on standard values corresponding to structural steels S235 and S355, including Young's modulus $E=210$ GPa, Poisson's ratio $\nu=0.3$ and the corresponding yield strengths according to EN 10025-2 (European Committee for Standardization, 2019).

The finite element mesh used in the analysis is also visible in all subfigures. The average element size was defined as a fraction of the global bounding box dimension, with a value of 0.100, ensuring a balanced compromise between numerical accuracy and computational cost. For shell-type elements, applied in regions characterized by reduced thickness or plate-like behavior, a finer average element size of 0.050 was used in order to capture stress gradients more accurately in thin-walled structures.

A minimum element size of 0.200 (relative scaling parameter) was imposed to prevent excessive local refinement and to avoid the formation of highly distorted elements or uncontrolled mesh density increases in localized regions. Mesh gradation was controlled using a grading factor of 1.5, ensuring a

smooth transition between fine and coarse mesh regions and improving numerical stability.

Additionally, a maximum curvature angle of 60° was applied to enable automatic mesh refinement in geometrically curved regions. This criterion improves the representation of curved surfaces and enhances the accuracy of stress prediction in areas where stress concentrations are expected due to geometric discontinuities.

Fig. 5 presents the results of the simulation in the form of contour plots illustrating the von Mises stress distribution throughout the system.

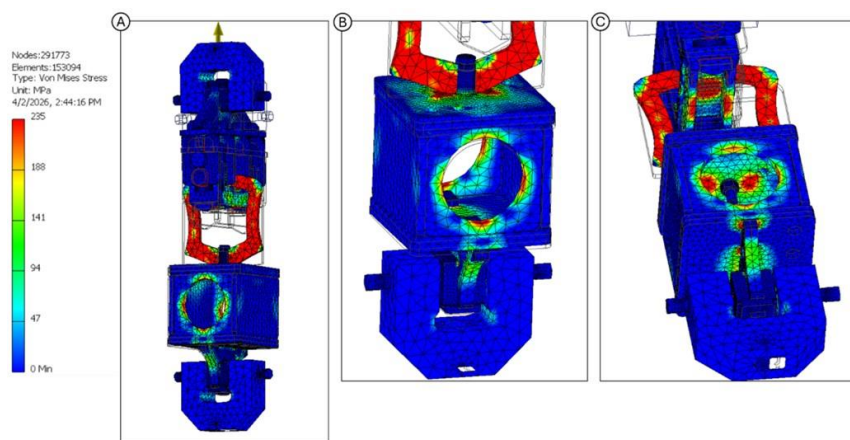


Fig. 5 – Finite element analysis results showing the von Mises stress distribution in the complete testing assembly under operational loading conditions.

Detail A in Fig. 5 shows the overall assembly configuration, including the direction of the applied load. It can be observed that the maximum stress level corresponds to approximately 235 MPa, which is associated with the yield strength of the material used for both the fixture and the test specimen. The stress distribution indicates that the lifting eye specimen is subjected to stress levels exceeding this value, as evidenced by the red zones shown in the contour scale. The entire body of the specimen is predominantly covered by high-stress regions, indicating a critical loading condition close to or beyond yielding.

Details B and C provide a more localized view of stress concentration areas within the fixture. These regions are primarily located along the circular contour of the central hole in the lower component of the fixture, as well as in the vicinity of the contact interface between the specimen and the fixture gripping elements. These zones act as stress concentrators due to geometric discontinuities and load transfer conditions.

Fig. 6 presents the von Mises stress distribution in the same finite element model as the previous figure, but with additional probing points used for local stress evaluation. The results indicate a maximum von Mises stress of

approximately 1084 MPa in the body of the lifting eye specimen. This value significantly exceeds the yield strength of the material (235 MPa), corresponding to an exceedance factor of approximately 4.6 times. This clearly indicates that the specimen undergoes plastic deformation and structural failure well before reaching the design load of 120 kN used in the numerical simulation. Since the numerical analysis was performed under linear elastic assumptions, stress values exceeding the material yield strength should be interpreted qualitatively as indicators of severe overstressing and onset of plastic deformation rather than exact physical stress magnitudes beyond yielding.

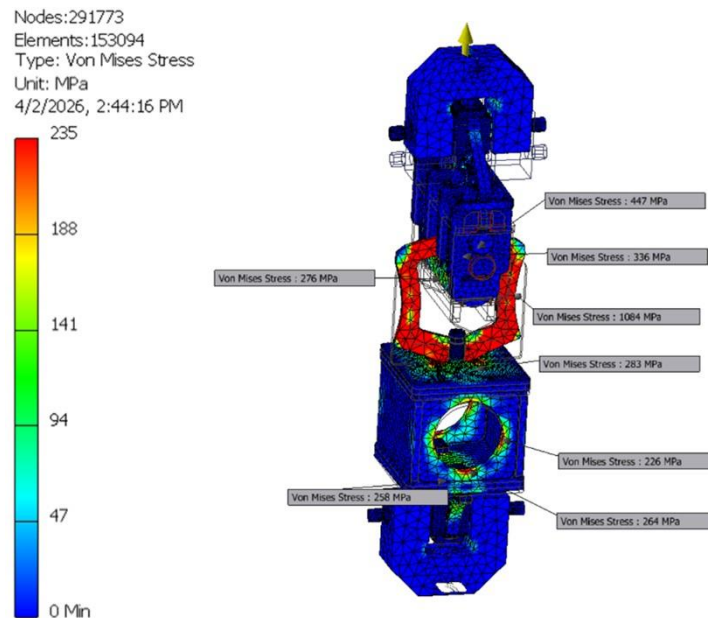


Fig. 6 – Von Mises stress distribution with probe points in the testing assembly.

Within the components of the testing fixture, stress levels are considerably lower, ranging between approximately 226 MPa and 447 MPa in localized regions. The maximum value of 447 MPa appears as a small, highly localized stress concentration, associated with localized contact concentration effects, at the contact interface between the lifting eye and the fixture element that simulates the forklift fork action. Despite these localized peaks, the overall structural response of the fixture remains within acceptable limits.

The results suggest that the testing assembly is structurally adequate for its intended purpose, as no widespread yielding or global failure is observed within the fixture components. The lifting eye specimen fails significantly earlier than the fixture, ensuring that the experimental setup is capable of

transmitting the required loads without structural degradation. Consequently, it can be concluded that the fixture design is sufficiently robust to withstand operational loading conditions, while the specimen reaches failure prior to the maximum test load of 40 kN, or well before any damage could occur to the testing device.

Fig. 7 presents the finite element results in terms of total displacement distribution across the entire testing assembly under tensile loading conditions. It can be observed that the lower part of the fixture exhibits very limited deformation, with displacement values ranging from approximately 0.241 mm to 0.664 mm, indicating a high degree of rigidity in this region.

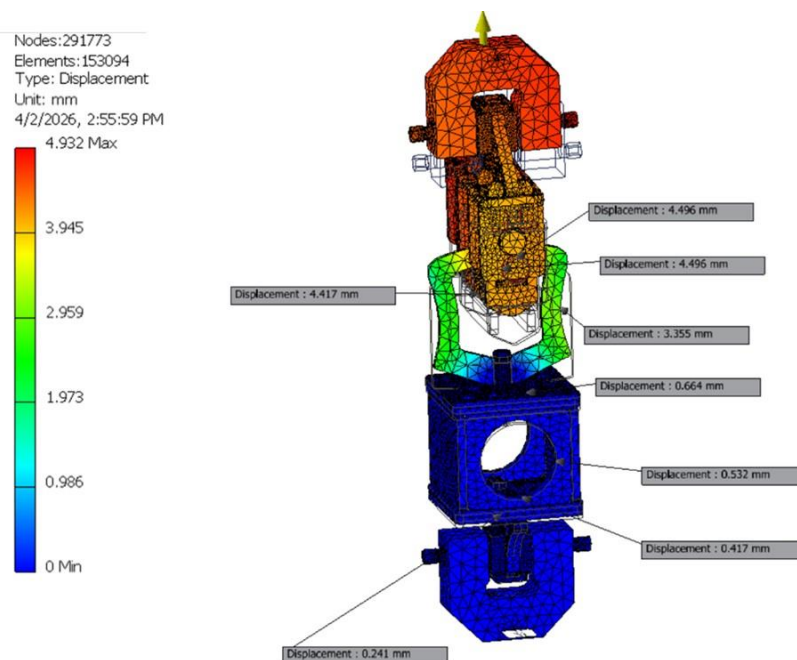


Fig. 7 – Total displacement distribution of the testing assembly under tensile loading conditions.

In contrast, the lifting eye specimen shows significantly higher deformation levels, with maximum displacements reaching approximately 4 mm. This behavior is consistent with the role of the specimen as the primary load-bearing and deformable element within the system.

The highest displacement value is observed in the upper gripping jaw of the testing machine, where a maximum deformation of approximately 4.932 mm is recorded in the region where the tensile load is applied. This localized deformation reflects the load transfer zone and the compliance of the gripping interface under high tensile loading.

Overall, the results indicate a clear stiffness gradient within the assembly, with the fixture maintaining structural rigidity while the specimen undergoes dominant deformation, as expected under tensile loading conditions.

To validate the numerical results and confirm the correct functioning of the proposed testing setup, experimental observations were also considered.

Experimental validation of the proposed fixture was performed through tensile testing of lifting-eye specimens on a universal testing machine under a maximum load of 40 kN. The tests revealed visible deformation only in the lifting-eye specimens, whereas the fixture remained structurally unaffected throughout the testing procedure. Owing to its initial design capacity of 120 kN, the fixture provides sufficient stiffness and strength for the safe transfer of loads to the tested components, while also enabling a realistic simulation of their service loading conditions.

Fig. 8 presents the real-world implementation of the testing assembly mounted in the tensile testing machine under operational conditions. Detail A illustrates the manufactured device installed in the machine, with the lifting eye specimen properly positioned and prepared for testing, representing the initial, unloaded configuration of the system. Detail B shows the specimen during tensile loading, where progressive deformation is clearly observed as a result of the applied force.

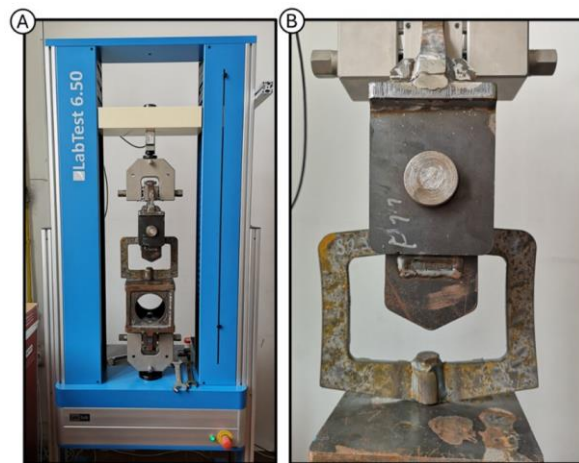


Fig. 8 – Experimental validation of the tensile testing setup: installed configuration and specimen deformation during testing.

The experimental evidence confirms that the fixture ensures correct alignment and reliable load transmission to the specimen throughout the test. No instability or unintended displacement of the assembly components was observed during operation. Therefore, it can be concluded that the developed device is fully functional and suitable for its intended purpose of safely and effectively testing lifting eyes under tensile loading conditions.

The numerical and experimental investigations confirm that the proposed fixture fulfills its intended role as an adaptation system for tensile testing machines, ensuring stable specimen positioning, correct load transmission, and sufficient structural rigidity throughout the testing procedure.

4. Conclusions

The finite element analysis confirmed that the lifting eye geometry governs the overall structural response, with plastic deformation initiating in the specimen well before any critical stress state is reached in the testing fixture components.

The designed adaptation fixture ensures an efficient and stable load transfer mechanism, with stress levels remaining predominantly below the material yield limit, except for highly localized stress concentration regions generated by geometric discontinuities and contact interactions throughout the structural elements of the device.

Localized stress concentrations were identified in specific contact and geometric discontinuity regions; however, these do not compromise the global integrity or functional safety of the testing assembly.

The displacement field distribution demonstrates a clear separation of roles between components, with the specimen acting as the primary deformable element and the fixture maintaining overall rigidity under load.

Experimental observations validate numerical predictions and confirm that the developed testing system is suitable for controlled and stable tensile testing under service-representative loading conditions.

The present study should be regarded primarily as the conceptual and structural validation of a specialized experimental fixture intended for future investigations of lifting components subjected to service-representative loading conditions.

Further research will include experimental testing of lifting eyes with different geometric configuration, combined with varying material grades and plate thicknesses, allowing a comparative assessment of their mechanical behavior under tensile loading conditions.

Acknowledgements. The authors gratefully acknowledge the support provided by the academic staff of the Faculty of Engineering in Hunedoara during the experimental activities conducted for this study.

REFERENCES

- Chen X., Li Y., Wang Z. Experimental Investigation of the Dynamic Behavior of Welded Plate Lifting Lugs for Hoisting Large Span Steel Cap Beams, *Buildings*, **16**, 2, 370 (2026).

- Choi Y. J., Lee J. Analysis of Fracture Behavior of a Lifting Lug based on Strength Calculations and Finite Element Analysis, *Trans. Korean Soc. Mech. Eng. A*, **50**, 3, 167–177 (2026).
- European Committee for Standardization EN 10025-2:2019, *Hot rolled products of structural steels – Part 2: Technical delivery conditions for non-alloy structural steels*, Brussels (2019).
- Fisher R. F. M. Tension failure assessment at lug hole edges, *Int. J. Fatigue*, **121**, 293–300 (2019).
- Lee W. C., Kim S. H., Park J. Y. Structural Stability Evaluation for Lifting Lug Design of Large Marine Engine by Finite Element Analysis, *Key Eng. Mater.*, **353–358**, 2855–2859 (2007).
- Ru M., Lei X. Q., Liu X. M. An Equal Biaxial Test Device for Large Deformation in Cruciform Specimens, *Exp. Mech.*, **62**, 677–683 (2022).
- Zhang Z., Pan Y., Lei H., Wu Z. R. Design of Biaxial Tensile Fixture for Fatigue Testing on Uniaxial Tensile Testing Machine, *J. Test. Eval.*, ASTM, DOI: 10.1520/JTE20240537 (2024).
- Zhou L., Liu J., Liu H. Design and Application of Biaxial Tensile Fixture Based on Uniaxial Tensile Testing Machine, *Exp. Sci. Technol.*, **18**, 1, 1–5 (2020).

DEZVOLTAREA UNUI DISPOZITIV SPECIALIZAT PENTRU ÎNCERCAREA LA TRACȚIUNE A OCHIURILOR DE RIDICARE UTILIZATE ÎN MANIPULAREA RAMPelor DE EGALIZARE

(Rezumat)

Acest studiu prezintă un dispozitiv specializat destinat mașinilor de încercare la tracțiune pentru evaluarea comportamentului mecanic al ochiurilor de ridicare utilizate în manipularea rampelor de egalizare. Dispozitivul elimină limitările configurațiilor convenționale de încercare la tracțiune, care nu pot reproduce condițiile complexe de încărcare generate în timpul operațiilor cu stivuitorul. Acesta este alcătuit din subansamble superioare și inferioare care permit transferul realist al sarcinii și simularea precisă a condițiilor la limită. Procesul de proiectare a inclus optimizarea geometrică și validarea structurală pentru a asigura rigiditate, durabilitate și o influență minimă asupra comportamentului specimenului, susținând sarcini de până la 40 kN. Alinierea corectă și poziționarea repetabilă au constituit priorități pentru îmbunătățirea consistenței încercărilor, iar dispozitivul a fost proiectat pentru reducerea tensiunilor și asigurarea unei distribuții uniforme a sarcinii. Rezultatele experimentale confirmă faptul că dispozitivul permite obținerea unor date experimentale relevante privind răspunsul structural în condiții realiste de încărcare și oferă condiții de solicitare mai reprezentative comparativ cu configurațiile convenționale de prindere directă. În concluzie, soluția contribuie la îmbunătățirea evaluării siguranței și susține optimizarea proiectării componentelor de ridicare, oferind o metodologie adaptabilă și altor sisteme solicitate mecanic.