UDC 539.3

ANALYSIS OF THE STRESS–STRAIN STATE OF A FLANGED CONNECTION BASED ON AN ANALYTICAL APPROACH USING THE FINITE ELEMENT METHOD AND COMPARISON WITH EXPERIMENTAL DATA

S.V. Mitsyuk,
Ph.D., Associate Professor

D.V. Mitsyuk,
Ph.D Student

Kyiv National University of Construction and Architecture, 31 Povitryanykh Syl Avenue, Kyiv, 03037, Ukraine

DOI: 10.32347/2410-2547.2025.115.157-163

The article presents a finite element analysis of a flanged bolted connection in a spatial formulation using three-dimensional eight-node isoparametric elements, along with a comparative analysis of the results obtained from experiments, as well as results from FEM and the Idea StatiCa software based on the component-based finite element method.
Ключові слова: finite element method, bolted connection, flanged bolted connection, contact stresses, experimental validation, thermal load.

Introduction

One of the most widely used and universal calculation methods is the Finite Element Method (FEM) [2]. This numerical method makes it possible to analyze the overall behavior of a structure, and by applying a three-dimensional problem formulation with eight-node isoparametric finite elements, it ensures an adequate reproduction of the spatial performance of the structure [1, 6, 7]. Local investigation of the stress–strain state is essential for improving prediction accuracy, optimizing parameters, and ensuring the reliability of the structure under operating conditions. Special attention should be paid to the connection nodes, particularly bolted connections, which determine the overall stiffness and load-bearing capacity of the system. Among the various types of bolted joints, flange connections are the most common in construction, industry, and mechanical engineering.

Flange joints can be subjected to both axial compression and tension, but quite often they are also influenced by bending moments, either due to externally applied loads or as a result of eccentricities arising from the action of axial forces. Such eccentricities may be caused by initial or acquired defects, which can directly affect the operational characteristics of both the joint element and the entire structure.

In this work, particular attention is given to the analysis of the stress–strain state of a pipe flange connection under the action of a bending moment, with a comparison to experimental data [9] as well as to results obtained using the finite element software packages LIRA-SAPR and Idea StatiCa Connection [8].

1. Problem Statement

The object of the study is a flanged bolted connection of two pipes:

- pipe diameter D_{mp}=762mm ;
- wall thickness of the pipes t_{tr}=6mm ;
- flange thickness is assumed to be t_f=40mm , the flange is considered to be of a ring type);
- outer radius of the flanges is assumed to be R_f=459mm ;
- the bolt circle radius is assumed to be R_b=423mm ;
- bolts M24 of strength class 10.9 are used;
- number of bolts - 24 pieces evenly spaced around the circle, Fig. 1.

The general view of the joint is shown in Fig. 2. A concentrated bending moment M=750kNm is applied to the joint, acting along the central axes of one pipe, while the other pipe is rigidly fixed. The material of the pipes and flanges is S355 steel, with modulus of elasticity E=2.06×10^{5}kPa and Poisson's ratio ν=0.3 specified.

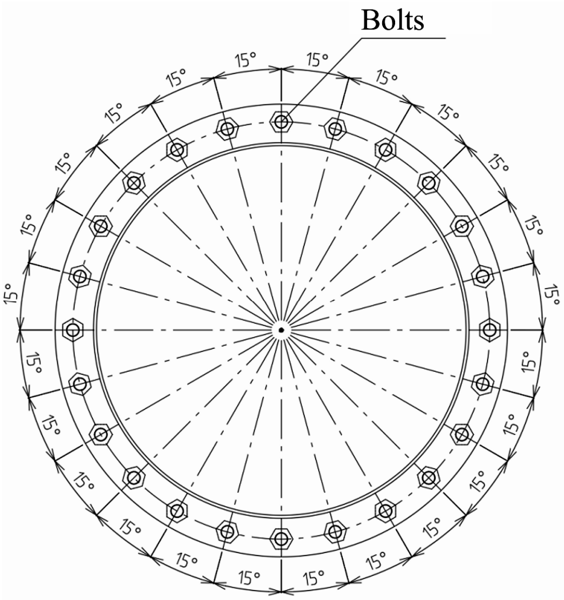

Fig. 1. Bolt arrangement in the flange connection

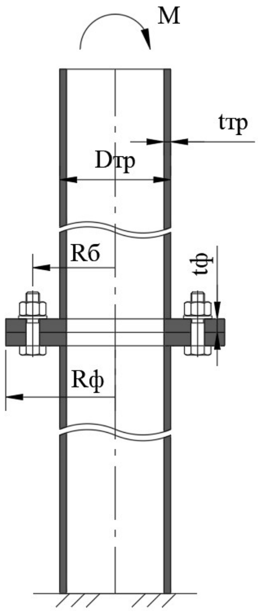

Fig. 2. General view of the joint

The joint was analyzed using the Finite Element Method (FEM) in displacement formulation, employing both universal three-dimensional eight-node isoparametric elements and universal four-node shell elements. The calculations were performed using the LIRA-SAPR software package [3]. The general view of the computational model is shown in Fig. 3.

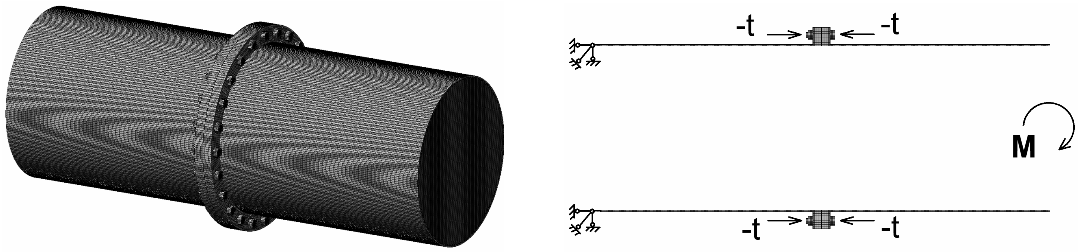

Fig. 3. General view of the computational model

The modulus of elasticity of the elements located in the zone between the bolt shank and the flange was assumed to be zero (E = 0) according to the previously proposed approach described in [4, 5]. This approach allows the stiffness transmission in the specified region to be excluded where necessary and ensures a correct representation of the stress-strain state of the joint.

Since the load was applied as a concentrated bending moment, compression zones develop in the joint, which are perceived by the flanges, and tension zones, which are carried by the bolts. To determine the boundaries of the compression and tension zones and to adequately model the behavior of the bolted connection, two-node finite elements with unilateral elastic links were used.

Bolt pretension was applied as a load resulting from uniform heating, with the load value assumed equal to the force causing the absolute elongation of the bolt, calculated based on the magnitude of thermal deformation ε_T

$$\varepsilon_T = \alpha \times \Delta T,$$

where α=0.000012 – is the coefficient of linear thermal expansion for steel, 1/°C; ΔT – temperature change, °C.

The stress and the corresponding forces are determined using the following formulas:

$$\sigma = E \times \varepsilon_T,$$

$$P = \sigma \times A,$$

where A=0.00045 m^2 – cross-sectional area of the bolt; P – bolt pretension force corresponding to $B_0 = 0.9 R_{bh} \times A_{bn} = 0.9 \times 700 \times 353 = 224.3\,\text{kN}$ where, according to DBN V.2.6-198_2014: $R_{bn} = 0.7 \times R_{bh} = 0.7 \times 1000 = 700\,\text{N/mm}^2$ – design tensile strength of the bolt; $A_{bn} = 353\,\text{mm}^2$ – cross-sectional area of the bolt at the threaded portion.

In accordance

$$\Delta T = \frac{P}{E \times \alpha \times A} = \frac{222.4}{2.06 \times 10^6 \times 0.000012 \times 0.00045} = 200^0\text{C}.$$

2. Calculation Results and Their Analysis

The analysis focused on the equivalent stresses in the connection elements and the forces occurring in the bolts, determined based on finite element method (FEM) calculations in a spatial formulation using universal quadrilateral shell elements and universal three-dimensional eight-node isoparametric finite elements (performed using the Lira SAPR software [3]).

The results of the calculations showed that the maximum equivalent stress in the pipe wall is 337 MPa when using universal quadrilateral shell elements, and 345 MPa when using universal three-dimensional eight-node isoparametric elements (Fig. 5, 6).

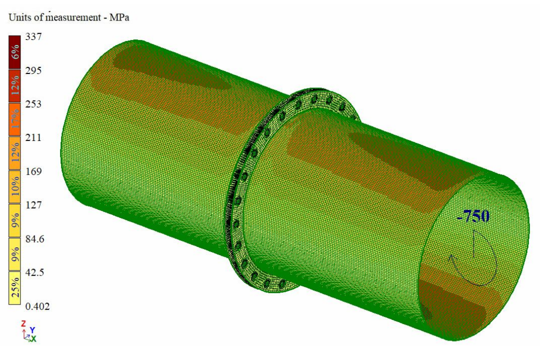

Fig. 5. FEM Equivalent Stresses Using Universal Quadrilateral Shell Elements

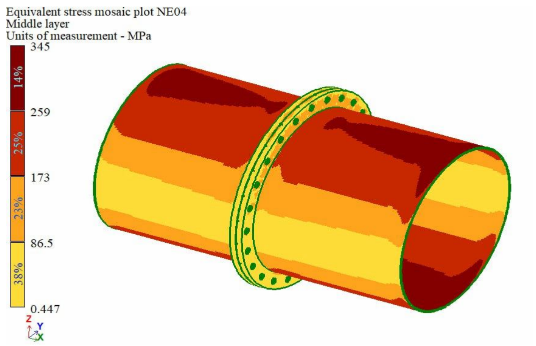

Fig. 6. FEM Equivalent Stresses Using Eight-Node Isoparametric Elements

At the same time, it is worth noting the differences in the distribution of forces in the bolts (see Figures 7–10). The maximum tensile force in the most stressed bolt is 217 kN when using universal quadrilateral shell elements, 293.2 kN when using three-dimensional eight-node isoparametric elements in Lira-SAPR, and 239.6 kN when using the IDEA StatiCa software. The distribution of forces in the finite elements modeling the bolts and the compression zones at the flange contacts is shown in Figures 7–10.

According to the experimental results [9], the maximum force in the bolt is 285 kN (Figure 7). Based on the finite element method (FEM) results, using quadrilateral shell elements, the maximum force is 217 kN (Figure 8); using eight-node isoparametric elements, the maximum tensile force reaches 293.2 kN (Figure 8) as the resultant force in the two-node unilateral elastic link elements (Figure 9); and using the component-based finite element method, it is 239.6 kN (Figure 10). The obtained calculation results are summarized in Table 1.

The calculation results were compared with the experimental data reported in [9], as well as with results obtained using the Lira-SAPR software based on FEM in a spatial formulation with universal quadrilateral shell elements, and with results from the IDEA StatiCa software using the component-based finite element method [8].

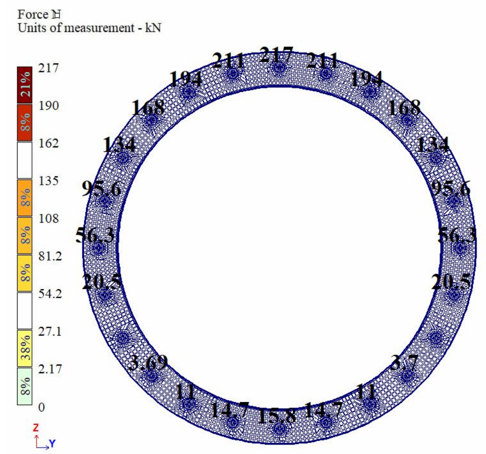

Fig. 7. Stress Distribution in the Flange Contact Zone Using Universal Quadrilateral Shell Elements

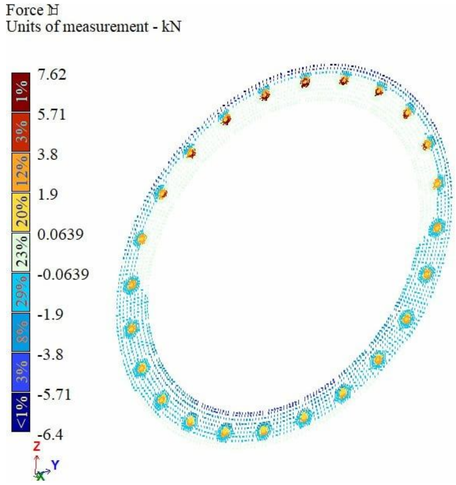

Fig. 8. Stress Distribution in the Flange Contact Zone Using Eight-Node Isoparametric Elements

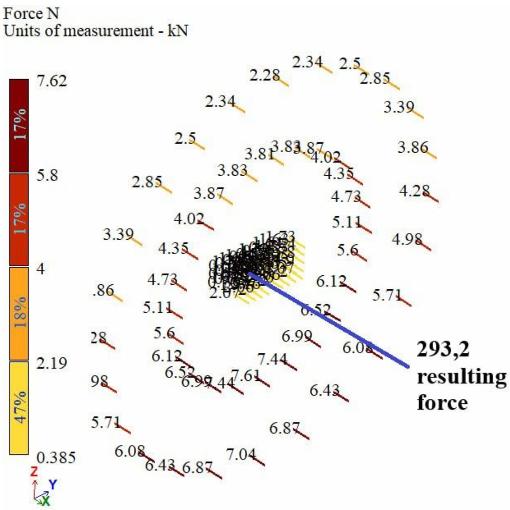

Fig. 9. Forces in Two-Node Unilateral Elastic Elements for the Most Stressed Bolt and Resulting Force

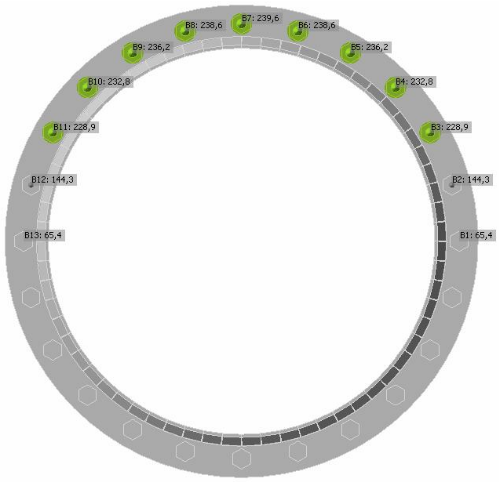

Fig. 10. Force distribution in the flange contact area using the IDEA StatiCa software

Table 1

Summary of the overall calculation results

Check	Experimental data [9]	FEM eight-node isoparametric elements	FEM four-node shell elements	IDEA StatiCa
Maximum tensile force in bolts, kN	285	293,2	217	239,6
% compared to experimental data	-	2,84	-27,09	-17,31

The nature of the local deformation according to the conducted numerical analysis is shown in Fig. 11, while the experimental data are presented in Fig. 12. It should be noted that there is a general agreement between the deformation distribution obtained numerically and the experimental observations, which indicates the adequacy of the applied model and the chosen analysis methods.

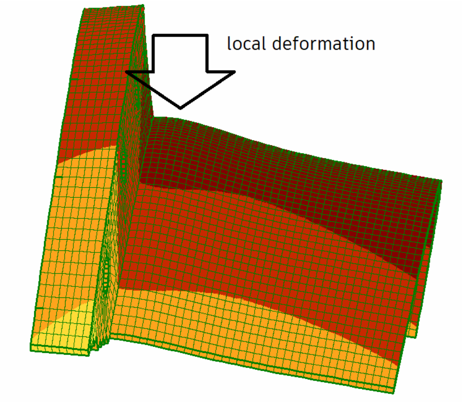

Fig. 11. Zone of local deformation of the pipe wall according to the numerical analysis

Fig. 12. Zone of local deformation of the pipe wall, experimental data [9]

Thus, the results of the finite element analysis of the flanged bolted joint using universal three-dimensional eight-node isoparametric elements show a high correlation with the experimental data [9]. At the same time, less accurate results are observed for finite element models using universal quadrilateral shell elements and the IDEA StatiCa software based on the component finite element method. Such deviations may be due to the simplification of geometry and the approximation of contact interactions in the models with quadrilateral shell elements, as well as differences in the approaches to modeling bolted joints in the component finite element method. The obtained data confirm the adequacy of using three-dimensional eight-node elements for accurate prediction of the stress-strain state of flanged bolted joints.

Conclusion. The results of the calculations and analysis indicate that modeling a flanged bolted joint, taking into account bolt preloading and loading from uniform heating using the finite element method, demonstrates a high correlation with experimental data [9]. In particular, a similar pattern of local deformations in the pipe wall and the forces in the bolts is observed. At the same time, further analysis of such joints should pay special attention to the area where the pipe is attached to the flange plate.

Calculations performed using universal quadrilateral shell elements and the IDEA StatiCa software based on the component finite element method showed lower agreement with experimental data and with the results obtained using three-dimensional eight-node isoparametric finite elements. This indicates the advantage of three-dimensional modeling with eight-node elements for accurate prediction of the stress-strain state of flanged bolted joints.

REFERENCES

1. Bazhenov V.A., Pyskunov S.O., Solodei I.I.. Chyselni doslidzhennia protsesiv neliniinoho statychnoho i dynamichnoho deformuvannia prostorovykh til (Numerical studies of the processes of nonlinear static and dynamic deformation of spatial bodies) // K.: Karavela, 2017. – 305 p.
2. Vabishchevych M.O., Storchak D.A.. Solution of nonlinear contact problems of deformation of nodal connections of steel structures // Strength of Materials and Theory of Structures: Scientific-and-technical collected articles. – K.: KNUBA. – Issue 108, 2022. - P. 178-179.
3. Prohramnyi kompleks LIRA-SAPR. Pryklady rozrakhunku i proektuvannia. (LIRA-SAPR software package. Examples of calculation and design) K. https://www.liraland.ua/download/private/lira/2023/lira_sapr_examples_ua.pdf.
4. Piskunov S.O., Mitsiuk S.V., Mitsiuk D.V., Repiakh Y. Analysis of the stress state of the bolted connection taking into account the bolt tension // Strength of Materials and Theory of Structures: Scientific-and-technical collected articles. – K.: KNUBA, 2024. - Issue 113. - P. 44-49.
5. Piskunov S.O., Mitsiuk S.V., Mitsiuk D.V., Repiakh Y. Comparative analysis of the stress state of a bolted joint based on an analytical approach using SFEM and experimental results// Strength of Materials and Theory of Structures: scientific and technical collection - Kyiv: KNUBA, 2025. - Issue 114. - P. 3-10..
6. Solodei I.I., Kozub Yu.H., Stryhun R.L., Shovkivska V.V. Chyselnyi analiz napruzheno-deformovanoho stanu trubchastoho elementa pry termosylovomu navantazhenni (Numerical analysis of the stress-strain state of a tubular element

under thermoforce loading) // Strength of Materials and Theory of Structures: scientific and technical collection - Kyiv: KNUBA, 2022. – Issue 109. - P. 109-119.
7. Bazhenov, V. A., Pyskunov, S.O., Maksym'yuk Yu. V., Mytsyuk S. V. Effect of geometric nonlinearity on the life of a herryingbone lock joint in creep // Opir materialiv, 2022, Vol. 54, No. 3, pp.372-376.
8. IDEA StatiCa Connection, software package designed. Bolted connection - Interaction of shear and tension https://www.ideastatica.com/support-center/bolted-connection-interaction-of-shear-and-tension
9. M. Couchaux, Mohammed Hjiaj, I. Ryan, A. Bureau. Bolted circular flange connections under static bending moment and axial force. Journal of Constructional Steel Research, 2019, 157, pp.314 - 336.

Стаття надійшла 15.09.2025

Мицюк С.В., Мицюк Д.В.

АНАЛІЗ НАПРУЖЕНО-ДЕФОРМОВАНОГО СТАНУ ФЛАНЦЕВОГО З'ЄДНАННЯ НА ОСНОВІ АНАЛІТИЧНОГО ПІДХОДУ З ВИКОРИСТАННЯМ МЕТОДУ СКІНЧЕННИХ ЕЛЕМЕНТІВ ТА ПОРІВНЯННЯ З ЕКСПЕРИМЕНТАЛЬНИМИ ДАНИМИ

У роботі проведено комплексний аналіз напружено-деформованого стану фланцевого болтового з'єднання труб під дією згинального моменту з використанням методу скінченних елементів (МСЕ) у просторовій постановці. Дослідження виконано із застосуванням тривимірних восьмивузлових ізопараметричних та чотирикутних оболонкових скінченних елементів у спеціалізованих програмних комплексах, а також із використанням компонентного методу скінченних елементів. Такий підхід дозволяє детально відтворювати просторову роботу конструкції та оцінювати локальні напруження і деформації, що виникають у критичних зонах з'єднання.

Особлива увага приділялася вузлам з'єднань, зокрема болтовим з'єднанням, які визначають загальну жорсткість та несучу здатність системи. У моделюванні враховано вплив натягу болтів, зон стиску та розтягу у фланцевому з'єднанні, а також можливий ексцентриситет прикладеного навантаження, що може виникати як через початкові, так і через набуті дефекти конструкції. Такий комплексний підхід дозволяє прогнозувати поведінку з'єднання під експлуатаційними навантаженнями та оцінювати надійність конструкції в цілому.

Результати чисельного моделювання порівнювалися з експериментальними даними та результатами розрахунків, виконаних у різних програмних комплексах. Порівняння показало високу збіжність розподілу напружень і локальних деформацій у випадку використання тривимірних восьмивузлових ізопараметричних елементів, тоді як застосування чотирикутних оболонкових елементів та компонентного методу скінченних елементів продемонструвало більші відхилення від експериментальних спостережень. Аналіз показав, що точність прогнозування напружено-деформованого стану значною мірою залежить від обраної геометрії елементів і адекватності моделювання контактних взаємодій між елементами.

Отримані результати підтверджують ефективність застосування тривимірних восьмивузлових ізопараметричних скінченних елементів для точного прогнозування напружено-деформованого стану фланцевих болтових з'єднань. Робота також підкреслює важливість уваги до зон кріплення труб до фланцевих пластин при проектуванні та експлуатації таких з'єднань, а також можливість використання методів МСЕ для оптимізації параметрів конструкцій і підвищення їхньої надійності.

Ключові слова: метод скінченних елементів, болтове з'єднання, фланцеве болтове з'єднання, контактні напруження, експеримент, навантаженням від рівномірного нагрівання.

Mitsyuk S.V., Mitsyuk D.V.

ANALYSIS OF THE STRESS–STRAIN STATE OF A FLANGED CONNECTION BASED ON AN ANALYTICAL APPROACH USING THE FINITE ELEMENT METHOD AND COMPARISON WITH EXPERIMENTAL DATA

This study presents a comprehensive analysis of the stress–strain state of a flanged bolted pipe connection under bending moment using the finite element method (FEM) in a spatial formulation. The investigation was conducted using three-dimensional eight-node isoparametric and quadrilateral shell finite elements in specialized software packages, as well as using the component-based finite element method. This approach allows detailed reproduction of the spatial behavior of the structure and enables the evaluation of local stresses and deformations occurring in critical areas of the connection.

Particular attention was given to the connection nodes, especially bolted joints, which determine the overall stiffness and load-bearing capacity of the system. The modeling accounted for bolt pre-tension, compression and tension zones in the flanged connection, as well as the potential eccentricity of the applied load, which can arise from both initial and acquired defects in the structure. Such a comprehensive approach enables prediction of the connection behavior under operational loads and assessment of the overall structural reliability.

The numerical modeling results were compared with experimental data and calculations performed using different software packages. The comparison showed high consistency in stress distribution and local deformations when using three-dimensional eight-node isoparametric elements, whereas the use of quadrilateral shell elements and the component-based finite element method exhibited greater deviations from experimental observations. The analysis demonstrated that the accuracy of predicting the stress–strain state largely depends on the chosen element geometry and the adequacy of modeling contact interactions between elements.

The results confirm the effectiveness of three-dimensional eight-node isoparametric finite elements for accurately predicting the stress–strain state of flanged bolted connections. The study also emphasizes the importance of considering the pipe-to-flange attachment zones during design and operation, as well as the potential of FEM-based methods for optimizing structural parameters and improving overall reliability.

Keywords: finite element method, bolted connection, flanged bolted connection, contact stresses, experimental validation, thermal load.

УДК 539.3
Мицюк С.В., Мицюк Д.В. **Аналіз напружено-деформованого стану фланцевого з'єднання на основі аналітичного підходу з використанням методу скінченних елементів та порівняння з експериментальними даними** // Опір матеріалів і теорія споруд: наук.-тех. збірн. – К.: КНУБА. 2025. – Вип. 115. – С. 157-163. – Англ.
The paper deals with the calculation of the bolted joint of two plates using an approach to modeling the initial tension of bolts in the joint and a comparative analysis of the results with experimental results.
Табл. 1. Іл. 12. Бібліогр. 9 назв.

UDC 539.3
Mitsyuk S.V., Mitsyuk D.V. **Analysis of the Stress–Strain State of a Flanged Connection Based on an Analytical Approach Using the Finite Element Method and Comparison with Experimental Data** // Strength of Materials and Theory of Structures: Scientific-and-technical collected articles. – K.: KNUBA. 2025. – Issue 115. – P. 157-163.
The article is devoted to the study of a bolted connection with different approaches to modeling bolts and their tension obtained on the basis of the semi-analytical finite element method (SFEM) and the finite element method (FEM).
Tabl. 1. Fig. 12. Ref. 9.

Автор (науковий ступінь, вчене звання, посада): кандидат технічних наук, доцент, докторант кафедри будівельної механіки КНУБА Мицюк Сергій Вікторович
Адреса: 03680 Україна, м. Київ, проспект Повітряних Сил, 31, Київський національний університет будівництва і архітектури, кафедра будівельної механіки, Мицюк Сергій Вікторович
Тел. +38(044) 241-5555
E-mail: serewka@ukr.net
ORCID ID: https://orcid.org/0000-0001-6481-4036

Автор (науковий ступінь, вчене звання, посада): аспірант кафедри будівельної механіки КНУБА Мицюк Дмитро Вікторович.
Адреса: 03680 Україна, м. Київ, проспект Повітряних Сил, 31, Київський національний університет будівництва і архітектури, кафедра будівельної механіки, Мицюк Дмитро Вікторович
Тел. +38(044) 235-1333
E-mail: mytsiuk.d.v@gmail.com
ORCID ID: https://orcid.org/0000-0002-3583-8052