UDC 539.3

ZONAL METHOD FOR CONTACT INTERACTION MODELING IN THE SEMI-ANALYTICAL FINITE ELEMENT METHOD

S.O. Pyskunov[1],
Doctor of Science (Engineering), Associate Professor

S.V. Mitsyuk[2],
Candidate of technical sciences, Associate Professor

D.V. Mitsyuk[2],
Ph.D Student

[1]*NTUU "KPI named after Igor Sikorskyi"*

[2]*Kyiv National University of Construction and Architecture*

DOI: 10.32347/2410-2547.2026.116.75-83

The article proposes a novel zonal method for modeling non-axisymmetric contact interaction within the framework of the Semi-Analytical Finite Element Method (SAFEM). The classical SAFEM, which employs Fourier series expansion of the solution in the circumferential direction, does not allow for accurate modeling of the non-axisymmetric contact behavior that arises under bending moment loading.

The developed method is based on partitioning the contact surface into longitudinal zones, each assigned its own elastic modulus. The modulus values are determined iteratively based on the analysis of gaps between the contacting surfaces: in contact zones, the elastic modulus of the material is assigned, while in opening zones, a significantly reduced value is prescribed. The iterative process continues until the distribution of moduli across the zones stabilizes. Verification of the method was performed on a benchmark problem involving contact modeling between two rectangular plates under bending moment loading. The computed results were compared with the solution obtained by the classical volumetric FEM in the LIRA-SAPR software package. Good agreement between the results was demonstrated, while the proposed method provides a substantial reduction in computational time compared to the classical FEM.

Keywords: semi-analytical finite element method, contact joint, non-axisymmetric contact, zonal method, iterative algorithm, contact zones, contact modeling.

Introduction

The application of the Semi-Analytical Finite Element Method (SAFEM) to contact interaction problems encounters significant challenges. SAFEM is widely used for the analysis of prismatic structures due to its high computational efficiency [1, 6]; however, under bending moment or concentrated force loading, the contact loses symmetry along the longitudinal axis: a compression zone forms on one part of the contact surface and an opening zone on the other. The classical SAFEM cannot accurately describe such a contact distribution, as it assumes material properties to be invariant along the longitudinal coordinate.

For modeling contact interaction in a three-dimensional formulation, special gap-type contact elements [3, 7, 8] are widely employed, implemented in modern software packages [5]. Semi-analytical methods of contact mechanics [2] also demonstrate effectiveness in solving such problems. These elements allow automatic detection of contact and opening zones, but require substantial computational resources due to the need for detailed discretization in all three directions. Studies of bolted connections accounting for bolt pretension and comparative analyses of different methods for calculating flange connections [4] highlight the importance of accurate contact interaction modeling for the precise prediction of the stress-strain state of structures. Numerical analysis of complex structures under thermo-mechanical loading and investigations of geometric nonlinearity confirm the necessity of developing efficient contact modeling methods for various types of connections.

This paper proposes a zonal method for contact modeling within the SAFEM framework, based on partitioning the contact surface into longitudinal zones with iterative determination of the elastic modulus in each zone. In contact zones, the actual elastic modulus of the material is assigned, while in opening zones a significantly reduced value is prescribed. This approach accounts for the asymmetric distribution of contact zones while preserving the linearity of the problem at each iteration, ensuring computational efficiency compared to a full three-dimensional formulation.

The objective of this work is the development and verification of the zonal method for contact interaction problems within the SAFEM framework, with comparison of results against calculations performed in the LIRA-SAPR software package [5].

1. Zonal Method

Contact Problem in the Semi-Analytical Finite Element Method.

In the Semi-Analytical Finite Element Method (SAFEM) [1, 6], for prismatic structures, the solution is represented as an expansion over a system of coordinate functions along the longitudinal coordinate, while retaining the finite element discretization in the cross-sectional plane.

The displacement field is represented in the form

$$u\left(x^1, x^2, x^3\right) = \sum u_l\left(x^1, x^2\right) \times \varphi^{(l)}\left(x^3\right)$$

where l – is the expansion term index, $u_l\left(x^1, x^2\right)$ – are the displacement amplitudes in the cross-sectional plane, $\varphi^{(l)}\left(x^3\right)$ – are the coordinate functions (Lagrange polynomials), x^3 – is the normalized longitudinal coordinate ($x^3 \in [-1,1]$), related to the physical coordinate y by the relation $x^3 = 2y/L - 1$, where L is the length of the structure

The Lagrange polynomials are defined as follows:

$$\varphi^{(0)} = \tfrac{1}{2}\left(1 - x^3\right), \quad \varphi^{(1)} = \tfrac{1}{2}\left(1 + x^3\right), \quad \varphi^{(l)} = f^{(l)} p^{(l)} - f^{(l-2)} p^{(l-2)}, l \geq 2,$$

where $f^{(l)} = \sqrt{\left[(7l^2 - 1)^{-1}\right]}$, $p^{(l)}$ – Lagrange polynomials.

For each expansion term l, a system of equations is formed:

$$[K]_l \{u\}_l = \{Q\}_l,$$

where $[K]_l$ – is the stiffness matrix, $\{u\}_l$ – is the nodal displacement vector, $\{Q\}_l$ – is the nodal load vector for the expansion term l.

A typical number of expansion terms $L = 16$ or 32 provides sufficient accuracy in representing the displacement field.

Under asymmetric loading (bending, shear forces), the contact surface is divided into contact zones and opening zones along the longitudinal axis. In contact zones, the bodies are in contact and transmit compressive stresses, while in opening zones a gap forms and stresses are absent. The boundary between the zones is not known a priori and depends on the deformed state.

The classical SAFEM formulation assumes that the material properties (in particular, the elastic modulus E) are independent of the longitudinal coordinate x^3. This leads to the appearance of non-physical tensile stresses in the opening zones and the inability to correctly model gap formation between the bodies.

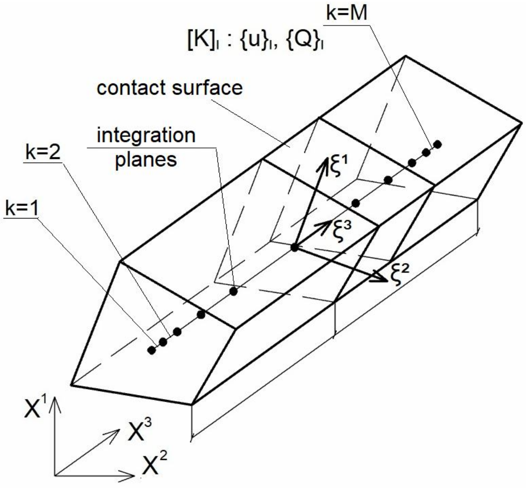


Fig. 1. Prismatic finite element and zone partitioning for the zonal method

2. Description of the Zonal Method

The proposed zonal method, in contrast to classical approaches [3, 7], is based on partitioning the longitudinal axis into zones and assigning to each zone an effective elastic modulus depending on the presence or absence of contact.

The longitudinal direction is divided into M zones in physical coordinates $y \in [0, L]$ (Fig. 1):

Zone k: $X^3 \in \left\lfloor X^3{}_k, X^3{}_{k+1} \right\rfloor$, $k = 1,2,\ldots,M$,

where $X^3{}_k$ - are the Gauss integration points for the Lagrange polynomials along the normalized coordinate $X^3 \in [-1,1]$. For $L = 16$ expansion terms, 17 integration points are used, forming M = 16 zones. In parametric studies M is varied from 10 to 20 to assess the convergence of the method.

3. Identification of Contact Elements

Contact elements are defined as finite elements in the cross-sectional mesh whose $\left(x^1, x^2\right)$, all nodes lie on the contact surface $x^2 = x^2\_contact$. During the pre-processing stage, a list of N_c contact elements is formed for the upper and lower plates.

For each zone *k* an elastic modulus is assigned, which defines the material properties of the contact elements in that zone:

$$E_k = E, \text{ if } gap_k \le 0 \text{ (contact)},$$
$$E_k = E_{soft}, \text{ if } gap_k > 0 \text{ (opening)},$$

where E – is the elastic modulus of the material; E_{soft} – is the reduced modulus in the opening zone; gap_k – is the gap between the contact surfaces at the center of zone *k*.

Physical interpretation: in the contact zone ($gap_k \le 0$) the material operates at full E and transmits compressive stresses. In the opening zone ($gap_k > 0$) the material becomes very soft ($E_k = E_{soft}$,), ($E_{soft} << E$), resulting in negligibly small stresses and simulating the absence of contact.

4. Assembly of the Stiffness Matrix

When assembling the stiffness matrix $[K]_l$ for expansion term l integration along the longitudinal coordinate is performed zonally. The contribution of contact elements is computed separately for each zone using the corresponding elastic modulus E_k:

$$[K]_l = [K]_l^{nekont} + \sum(elem \in N_c)\sum(k = 1..M)[K]_l^{elem`k}(E_k), \tag{1}$$

where $[K]_l^{nekont}$ – is the contribution of non-contact elements (with constant modulus E along the entire length L); $[K]_l^{elem`k}$ – is the contribution of contact element $elem$ in zone k.

The contribution of contact element in zone k is computed by integration over the interval $\left[x_k^3, x_{k+1}^3\right]$

$$[K]_l^{elem`k} = f\left[x_k^3 \ldots x_{k+1}^3\right][B]^T [D](E_k)[B]\left[\varphi^{(1)}\left(x^3\right)\right]^2 dx^3, \tag{2}$$

where $[B]$ – Gradient matrix, $[D](E_k)$ – is the elasticity matrix with modulus E_k.

Integral (1) is evaluated numerically using the Gauss quadrature method with integration points within zone k.

Thus, contact elements have variable stiffness along the length of the structure depending on the distribution of contact and opening zones, while non-contact elements retain constant material properties.

5. Iterative Algorithm

The distribution of elastic moduli E_k across the zones is determined iteratively, since the gap_k depend on the deformed state, which in turn depends on the distribution of moduli.

6. Solution Algorithm

Initialization (iteration 0):

The initial distribution of moduli across the zones is specified. A typical initial assumption: $E_k = E$ for all zones (full contact along the entire length is assumed).

Iteration $iter = 1,2,3,\ldots$:

Step 1. Solution of the problem with the current modulus distribution.

For each expansion term $l = 0,1,\ldots,L$:

а) Assemble the stiffness matrix $[K]_l$ according to (1)-(2) with the current values of E_k

б) Solve the system of linear algebraic equations:

$$[K]_l [u]_l = \{Q\}_l .$$

Step 2. Computation of gaps in the zones

For each zone $k=1,2,...,M$:

a) Determine the coordinate of the zone center:

$$y_k^c=\left(y_k+y_{k+1}\right)/2\,,$$

$$x^{3c}_{k}=2y_k^c/L-1\,;$$

b) Recover the vertical displacements of the contact surfaces by superposition of the expansion terms:

$$u_2^{top}\left(y_k^c\right)=\sum{}_l u_{21}^{top}\times\varphi^{(l)}\left(x^{3c}_{k}\right),$$

$$u_2^{bot}\left(y_k^c\right)=\sum{}_l u_{21}^{bot}\times\varphi^{(l)}\left(x^{3c}_{k}\right),$$

where $u_{2l}^{bot},u_{2l}^{top}$ – are the vertical displacements (in the x^2 direction) of a characteristic point on the contact surface for expansion term l ;

c) Compute the gap in zone k :

$$gap_k=u_2^{top}\left(y_k^c\right)-u_2^{bot}\left(y_k^c\right).$$

Step 3. Update of elastic moduli.

For each zone k:

If $gap_k\leq 0$:

$$E_k^{new}=E\quad\text{(contact)},$$

Otherwise:

$$E_k^{new}=E_{soft}\ \text{(opening)}.$$

Step 4. Convergence check.

If $E_k^{new}=E_k^{old}$ for all zones $k=1,...,M$:

STOP (solution obtained).

Otherwise:

$$E_k\leftarrow E_k^{new}\,.$$

Proceed to Step 1 (next iteration)

Convergence criterion.

The iterative process is considered converged when the distribution of elastic moduli E_k has stabilized, i.e., does not change between successive iterations. This means that the distribution of contact and opening zones is consistent with the deformed state of the structure.

7. Verification of Results

To verify the zonal method, consider a flange-type connection of two rectangular plates. Geometric parameters:

Length: L=500mm.

Width: B=500mm.

Thickness of the upper plate: h_{top}=6mm .

Thickness of the lower plate: h_{bot}=6mm .

Elastic modulus E=2.06×10^{5}MPa .

Poisson's ratio $\nu=0.3$.

The lower plate is rigidly fixed along its entire perimeter, modeling a stationary base of the structure.

The upper plate is fixed at 2 or 4 discrete points (see Fig. 3). At these points, displacements in all directions are restrained, simulating a bolted connection between the plates.

A tensile force N=250kN is applied to the end face of the upper plate in the longitudinal direction. The loading induces bending of the upper plate and opening of the contact between the bolted connections

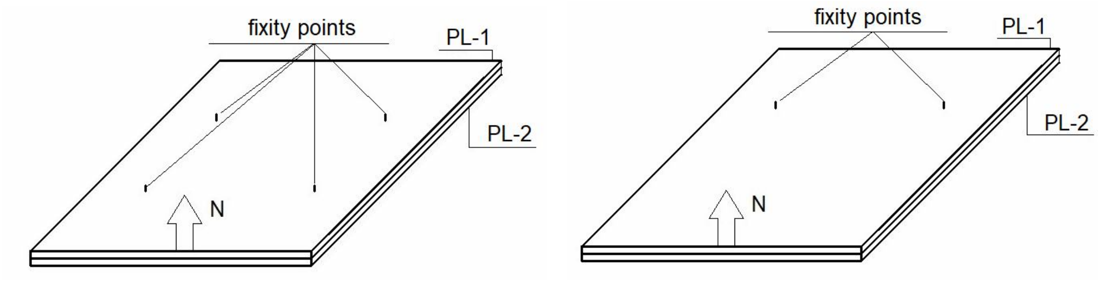


Fig. 2. General view of the benchmark problem

The benchmark problem was analyzed using the displacement-based Finite Element Method (FEM) with universal spatial eight-node isoparametric finite elements and a special contact element FE262 for contact modeling. This contact element transmits only compressive forces and is automatically deactivated upon the occurrence of tensile stresses, simulating contact opening.

The LIRA-SAPR software package [5] was used for the analysis. The general view of the structural model is shown in Fig. 3.

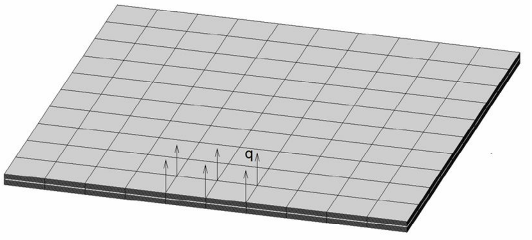


Fig. 3. General view of the structural model

8. Results and Analysis

The stress-strain state of the benchmark problem was analyzed using von Mises equivalent stresses. The study was carried out on the basis of numerical finite element modeling in a three-dimensional formulation, which allows for accurate representation of the actual behavior of the plates, loading conditions, and interaction between individual components. The numerical analysis was performed using the LIRA-SAPR software package [5].

The structural model in LIRA-SAPR consists of two rectangular plates connected through FE262 contact elements, which are distributed over the contact surface at a regular spacing. The load is applied as a uniformly distributed force on the end face of the upper plate. The use of unilateral FE262 contact elements allows automatic identification of active contact zones (transmission of compressive stresses) and opening zones (absence of interaction).

The lower plate is rigidly fixed along its entire surface and perimeter. The upper plate:

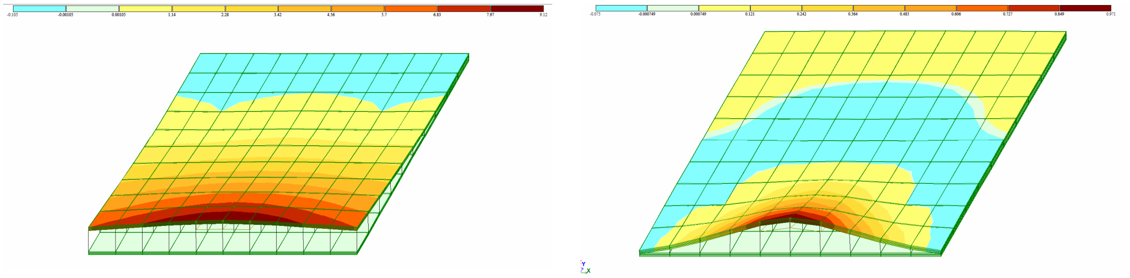
- **Case A:** pinned restraint at two discrete points, simulating a flange connection with two bolts (Fig. 2a);
- **Case B:** pinned restraint at four discrete points, simulating a flange connection with four bolts (Fig. 2b).

The general views of the structural models for both cases are presented in Fig. 4.

The obtained displacement distributions are shown in Fig. 4. Fig. 4a presents the results for Case A, where the maximum displacement is 9.12 mm; for Case B, the maximum displacement is 0.971 mm.

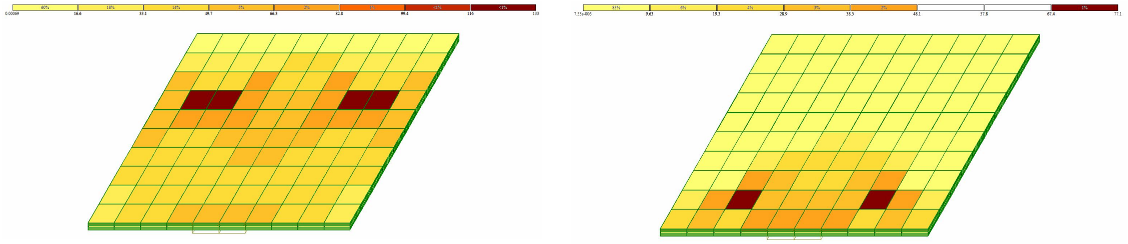
The numerical simulation also established that the maximum von Mises equivalent stresses reach:

- 133 MPa for the two-point restraint case (Fig. 5a);
- 77.1 MPa for the four-point restraint case (Fig. 5b).



(a) (b)

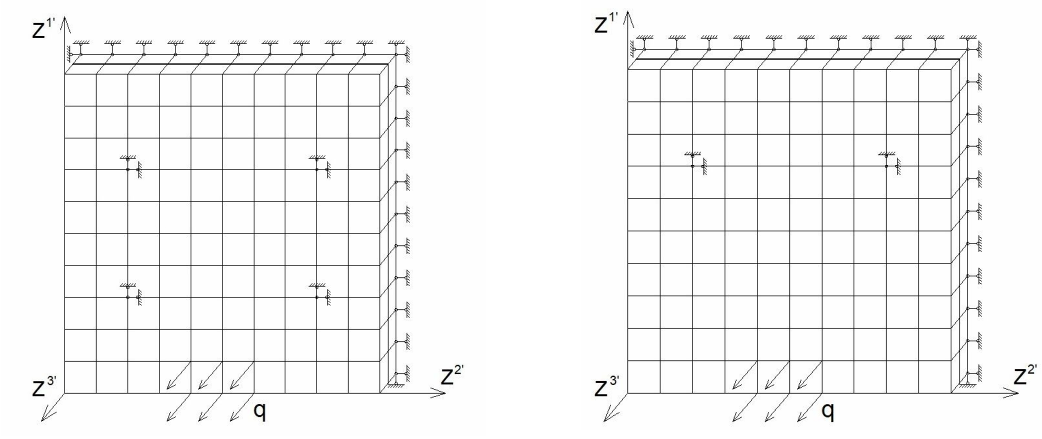
Fig. 4. Displacement distribution in the structural model



(a) (b)

Fig. 5. Stress distribution in the structural model

To verify the proposed zonal method, the same problem was analyzed using the Semi-Analytical Finite Element Method. The SAFEM structural model is shown in Fig. 6.



(a) (b)

Fig. 6. General view of the structural model depending on the number of restraint points

Based on the SAFEM analysis results, depending on the restraint configuration (Case A and Case B), the displacement values are:

- 8.99 mm for the two-point restraint case (Fig. 6a);
- 0.994 mm for the four-point restraint case (Fig. 6b).

The stress values are:

- 134 MPa for the two-point restraint case (Fig. 6a);
- 76.3 MPa for the four-point restraint case (Fig. 6b).

The obtained results are summarized in Table 1.

Table 1

Summary of analysis results

	FEM (Case A)	SAFEM (Case A)	%	FEM (Case B)	SAFEM (Case B)	%
Maximum displacements, mm	9,12	8,99	-1,44	0,971	0,994	2,37
Maximum stresses, MPa	133	134	0,75	77,1	76,3	-1,04

Comparative analysis of the results revealed high agreement between the three-dimensional FEM formulation and the SAFEM zonal method, with the maximum relative error not exceeding 2.4%. The obtained results verify the applicability of the proposed method for solving contact problems in prismatic structures.

Thus, the analysis results demonstrate high accuracy of the proposed zonal method in the Semi-Analytical FEM compared to the classical three-dimensional FEM in LIRA-SAPR. The maximum discrepancy in displacements between the two methods is 2.37% for Case B and −1.44% for Case A, confirming the correctness of the structural deformation modeling. The discrepancy in maximum von Mises equivalent stresses does not exceed 1.04%, indicating adequate reproduction of the stress-strain state in both contact and opening zones. The achieved agreement of results verifies the applicability of the zonal method for solving contact problems in prismatic structures at a significantly reduced computational cost compared to the full three-dimensional formulation.

Conclusion

This paper proposes and implements a zonal method for modeling contact interaction within the Semi-Analytical Finite Element Method for prismatic structures. The classical SAFEM formulation [1, 6] does not allow for accurate modeling of contact opening zones due to the assumption of material property invariance along the longitudinal coordinate, which leads to the appearance of non-physical tensile stresses. The developed method resolves this issue by partitioning the longitudinal axis into zones and iteratively determining the distribution of elastic moduli depending on the presence or absence of contact in each zone.

Verification of the method was performed on a benchmark problem of a flange connection of two plates under tensile loading, considering two bolt configurations. Comparison of the results with calculations in the LIRA-SAPR software package [5] demonstrated high accuracy of the proposed approach: the maximum discrepancy in displacements is 2.37%, and in stresses -1.04%. This agreement confirms that the zonal method adequately reproduces both the deformed state of the structure and the stress distribution in the contact and opening zones.

An important advantage of the method is its computational efficiency compared to the full three-dimensional problem formulation. In SAFEM, the number of degrees of freedom is determined solely by the cross-sectional discretization (approximately 300 equations per expansion term), whereas the three-dimensional FEM requires a detailed mesh in all three directions (approximately 30,000 equations). The iterative process of the zonal method demonstrates stable convergence, reaching the solution in 3–7 iterations, which makes the method suitable for engineering applications.

The proposed method can be applied to a broad class of contact problems [1, 6] in prismatic structures, including flange connections [3, 4, 7], multilayer composite structures, mechanically fastened joints, and other cases where opening zones may form under loading. Further development of the method may include the incorporation of friction on the contact surface [2], modeling of plastic deformations in stress concentration zones, and extension to dynamic loading cases.

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. 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. https://doi.org/10.1007/s11223-022-00412-4
3. 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

4. Mitsiuk S.V., Maksym'yuk Yu. V., Mitsiuk D.V., Comparative analysis of the results of flange connection calculation using the FEM and CFEM methods // Modern engineering and innovative technologies. – Issue №40, Part 2, June 2025. – С. 88-96. https://doi.org/10.30890/2567-5273.2025-40-02
5. 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 // Resistance of materials and theory of structures: scientific and technical collection - Kyiv: KNUBA, 2024. - Issue 113. - P. 44-49. https://doi.org/10.32347/2410-2547.2024.113.37-44
6. Prohramnyi kompleks LIRA-SAPR. Pryklady rozrakhunku i proektuvannia : navchalnyi posibny. (Software Package LIRA-SAPR. Examples of Analysis and Design: Study Guide). Kyiv: LIRA LAND, 2023. 586 p. URL: https://www.liraland.ua/download/private/lira/2023/lira_sapr_examples_ua.pd
7. 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) // Opir materialiv i teoriia sporud. – K.:KNUBA, Vyp.109, 2022, P. 109-119. https://doi.org/10.32347/2410-2547.2023.110.199-206
8. Vabishchevych M.O., Storchak D.A.. Solution of nonlinear contact problems of deformation of nodal connections of steel structures // Opir materialiv i teoriia sporud. –K.:KNUBA, Vyp.108, 2022, P. 178-179. https://doi.org/10.32347/2410-2547.2022.108.178-188

Стаття надійшла 19.03.2025

Пискунов С.О., Мицюк С.В., Мицюк Д.В

ЗОНАЛЬНИЙ МЕТОД КОНТАКТНОГО МОДЕЛЮВАННЯ ВЗАЄМОДІЇ У НАПІВАНАЛІТИЧНОМУ МЕТОДІ СКІНЧЕННИХ ЕЛЕМЕНТІВ

У даній роботі виконано комплексне дослідження можливостей моделювання контактної взаємодії у напіваналітичному методі скінченних елементів для призматичних конструкцій. Класична постановка НМСЕ передбачає незмінність властивостей матеріалу вздовж поздовжньої координати, що призводить до виникнення нефізичних розтягуючих напружень у зонах розкриття контакту. Для вирішення цієї проблеми запропоновано зональний метод, який базується на поділі поздовжньої осі на зони та ітераційному визначенні розподілу модулів пружності залежно від наявності або відсутності контакту у кожній зоні.

Особливу увагу приділено модифікації процедури формування матриці жорсткості, де інтегрування вздовж поздовжньої координати для контактних елементів виконується зонально з використанням різних модулів пружності. У зонах контакту використовується повний модуль матеріалу, у зонах розкриття - значно менший модуль, що імітує відсутність взаємодії. Розподіл модулів визначається ітераційно через розв'язання задачі, обчислення зазорів у центрах зон та оновлення властивостей матеріалу до досягнення збіжності.

Проведено верифікацію методу на тестовій задачі фланцевого з'єднання двох пластин з порівнянням результатів з розрахунками у програмному комплексі LIRA-SAPR. Результати показали високу збіжність між зональним методом та просторовою постановкою МСЕ: розбіжність у переміщеннях не перевищує 2,4%, у напруженнях — 1,04%. При цьому обчислювальна ефективність зонального методу значно вища завдяки меншій розмірності задачі та можливості паралелізації розрахунків для різних членів розкладання.

Отримані результати підтверджують доцільність та ефективність використання зонального методу для моделювання контактної взаємодії у призматичних конструкціях. Такий підхід дозволяє коректно враховувати зони розкриття при збереженні високої обчислювальної ефективності, що робить метод придатним для широкого класу інженерних задач, включаючи фланцеві з'єднання, багатошарові композити та інші випадки одностороннього контакту.

Ключові слова: напіваналітичний метод скінченних елементів, контактне з’єднання, неосесиметричний контакт, зональний метод, ітераційний алгоритм, зони контакту, розкриття контакту.

Pyskunov S.O., Mitsyuk S.V., Mitsyuk D.V.

ZONAL METHOD FOR CONTACT INTERACTION MODELING IN THE SEMI-ANALYTICAL FINITE ELEMENT METHOD

This paper presents a comprehensive investigation of contact interaction modeling capabilities within the Semi-Analytical Finite Element Method for prismatic structures. The classical SAFEM formulation assumes material property invariance along the longitudinal coordinate, which leads to the appearance of non-physical tensile stresses in contact opening zones. To address this issue, a zonal method is proposed, based on partitioning the longitudinal axis into zones and iteratively determining the distribution of elastic moduli depending on the presence or absence of contact in each zone.

Particular attention is given to the modification of the stiffness matrix assembly procedure, where integration along the longitudinal coordinate for contact elements is performed zonally using different elastic moduli. In contact zones, the full material modulus is applied, while in opening zones a significantly reduced modulus is used to simulate the absence of interaction. The modulus distribution is determined iteratively by solving the problem, computing gaps at zone centers, and updating material properties until convergence is achieved.

Verification of the method was carried out on a benchmark problem of a flange connection of two plates, with results compared against calculations performed in the LIRA-SAPR software package. The results demonstrated high agreement between the zonal method and the three-dimensional FEM formulation: the discrepancy in displacements does not exceed 2.4%, and in stresses — 1.04%. At the same time, the computational efficiency of the zonal method is significantly higher due to the reduced problem dimensionality and the possibility of parallelizing calculations for different expansion terms.

The obtained results confirm the feasibility and effectiveness of the zonal method for modeling contact interaction in prismatic structures. This approach allows for accurate representation of opening zones while maintaining high computational efficiency, making the method suitable for a broad class of engineering problems, including flange connections, multilayer composites, and other cases of unilateral contact.

Keywords: semi-analytical finite element method, contact joint, non-axisymmetric contact, zonal method, iterative algorithm, contact zones, contact modeling**.**

УДК 539.3
Пискунов С.О., Мицюк С.В., Мицюк Д.В. **Зональний метод контактного моделювання взаємодії у напіваналітичному методі скінченних елементів** // Опір матеріалів і теорія споруд: наук.-тех. збірн. – К.: КНУБА. 2026. – Вип. 116. – С. 75-83. – Англ.

Розроблено та верифіковано зональний метод моделювання неосесиметричної контактної взаємодії в рамках SAFEM, заснований на ітераційному визначенні модулів пружності по поздовжніх зонах, з підтвердженою точністю методу (похибка в межах 2,4%) шляхом порівняння з тривимірним методом скінченних елементів (МСЕ) в LIRA-SAPR на еталонній задачі фланцевого з'єднання.

Табл. 1. Іл. 6. Бібліогр. 7 назв.

UDC 539.3
Pyskunov S.O., Mitsyuk S.V., Mitsyuk D.V. **Zonal method for contact interaction modeling in the semi-analytical finite element method** // Strength of Materials and Theory of Structures: Scientific-and-technical collected articles. – K.: KNUBA. 2026. – Issue116. – P. 75-83.

A zonal method for modeling non-axisymmetric contact interaction within the SAFEM framework is developed and verified, based on iterative determination of elastic moduli across longitudinal zones, with method accuracy confirmed (error within 2.4%) through comparison with three-dimensional FEM in LIRA-SAPR on a benchmark flange connection problem.

Tabl. 1. Fig. 6. Ref. 7.

Автор (науковий ступінь, вчене звання, посада): професор, доктор технічних наук, завідувач кафедри динаміки і міцності машин та опору матеріалів НТУУ «КПІ ім. Ігоря Сікорського», Пискунов Сергій Олегович.
Адреса: 03056 Україна, м. Київ, просп. Берестрейський, 37, Національний технічний університет України “Київський політехнічний інститут імені Ігоря Сікорського”, кафедра динаміки і міцності машин та опору матеріалів., Пискунов Сергій Олегович
Тел. +38(044) 241-5555
E-mail: s_piskunov@ua.fm
ORCID ID: https://orcid.org/0000-0003-3987-0583

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 робоча: 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