UDC 539.3:004.94:519.6
DOI: 10.15587/1729-4061.2025.345896

DEVELOPMENT AND POST-PROCESSING OF MATHEMATICAL AND COMPUTER MODELS OF CONTACT PROBLEMS

Tetyana Zaytseva
PhD, Associate Professor*
Ganna Shyshkanova
PhD, Associate Professor
Department of Mathematics
National University «Zaporizhzhia Polytechnic»
Universytetska str., 64, Zaporizhzhia, Ukraine, 69063
Yaroslav Honcharov
Corresponding author
PhD Student*
E-mail: goncharov.yar.an@gmail.com
*Department of Computer Technologies
Oles Honchar Dnipro National University
Nauky ave., 72, Dnipro, Ukraine, 49010

This study investigates the process of contact interaction between a rigid punch and an elastic half-space under the action of a centrally applied clamping load. The task addressed is automating the processing of results for data analysis.

This paper considers a problem for a punch with complex geometry. The contact zone has a doubly connected shape bounded by concentric ellipses, which complicates the analytical description of the stressed-strained state.

To solve the problem analytically, a perturbation method was applied, using the previously found expansion of the simple layer potential with a small parameter. This makes it possible to reduce the problem to a configuration with a circular ring whose exact solutions are known in the form of a series with recurrent coefficients. The obtained analytical results serve as a reference basis for assessing the accuracy of numerical modeling, for example, when constructing a finite element model. Such mathematical formalization makes it possible to effectively assess the reliability of the results.

Consequently, calculation models were built in the Ansys software environment taking into account features of the punch shape. Special software was developed that enables exporting calculation data to the MATLAB software package with subsequent post-processing to automate data processing. The proposed approach reduced the complexity of post-processing by approximately 45–55%.

It was found that with an increase in the eccentricity of the ellipse, the pressure under the punch increases. In the central zone, a lower pressure is recorded, which increases along the edges of the contact area. The numerical results correlate well with the analytical ones.

The results reported here could be used for strength calculations in engineering practice under conditions of contact interaction between structural elements of mechanical engineering where the contact zone has the shape of an elliptical ring

Keywords: contact zone, punch, software tools, finite element method, analytical solution

Received 25.09.2025
Received in revised form 01.12.2025
Accepted 08.12.2025
Published 30.12.2025

***How to Cite:** Zaytseva, T., Shyshkanova, G., Honcharov, Y. (2025). Development and post-processing of mathematical and computer models of contact problems. Eastern-European Journal of Enterprise Technologies, 6 (7 (138)), 6–15. https://doi.org/10.15587/1729-4061.2025.345896*

1. Introduction

Areas of stress concentration are most often contact zones where local damage, wear, or destruction occurs. This is especially relevant for parts working under significant loads, under conditions of friction, aggressive environment, or variable temperature [1]. Solving the normal problem without frictional contact is the first step to the analysis of more complex models. The problems of adhesive contact, as well as contact with elastomers, are often reduced to its non-adhesive form [2, 3]. Thus, the problem without frictional contact forms the conceptual basis of all contact mechanics. Failure to take geometric features into account when calculating strength can lead to fatal consequences. However, each new form usually requires a new analytical approach and solution method. Therefore, the mechanics of contact interaction continues to actively evolve, which is reflected in many current studies based on the materials from earlier classical research [4, 5].

It should be noted that the application of modeling of such processes is complicated by a number of factors. Therefore, numerical methods are a tool for solving such problems. In particular, in the Ansys software environment, the finite element method (FEM) makes it possible to take into account complex geometry, various boundary conditions, and physical properties of materials. However, functional limitations in the field of post-processing do not always allow for a simple analysis of the achieved results.

Therefore, the issue of the need to develop specialized software that increases the efficiency of processing results in Ansys is important. It should provide automated data processing, statistical analysis, construction of plots, comparison with reference results, as well as adjustment of model parameters. Thus, the combination of Ansys with MATLAB makes it possible to create an effective environment for the analysis of contact interactions. This is especially important when studying bodies with complex geometry. It is the visualization, interpretation, and possibly additional adjustment of the results in such post-processing that become more effective, which is a necessary modern condition for successful engineering research.

Thus, the relevance of the scientific area related to the mechanics of contact interaction is predetermined not only by the need to model contact processes but also by the necessity to devise adaptive computing solutions that integrate various methodologies – analytical, numerical, statistical, and meet the modern requirements of engineering practice. This provides additional opportunities for devising and analyzing effective methodologies for assessing strength, optimizing structures, as well as increasing their operational reliability.

2. Literature review and problem statement

An important step in the mechanics of contact interaction was the development of analytical methods for perturbing the shape of the boundary and perturbing elastic properties for spatial problems in the theory of elasticity for solving problems of piecewise homogeneous non-canonical regions with interfaces close to canonical ones [6]. But the issues of two-connected areas remained unresolved. Despite the high theoretical and practical significance of that work, it should be recognized that a complete study of all possible non-canonical cases is impossible because of their infinite number. The statement and method of solving problems about the stressed-strained state of elastic bodies with initial stresses during their contact interaction with rigid and elastic punches was proposed in [7]. Using this method, it is possible to construct exact solutions to the problem for an elastic half-space with initial stresses when pressing an elastic punch, for which solutions of corresponding problems without initial stresses are constructed [7, 8]. But issues close to the ring regions remained unresolved. Therefore, the analytical solution of the problem with an elliptical contact ring can serve as the first step to solving the problem with initial stresses.

Papers [9, 10] report the advancement of the Wiener-Hopf method and its application to the solution of contact problems in the theory of elasticity. But despite the theoretical elegance, for many, even relatively simple, functions, the analytical (in closed form) implementation of factorization is extremely difficult. Work [11] gives an overview of stress concentration studies in elastic bodies. The asymptotic approach to the study of stress distribution near curved openings and cutouts with non-smooth contours is especially highlighted. But, despite the large number of applied problems, the consideration of doubly connected contours is beyond the scope of that work.

Paper [12] reports the latest mathematical models and methods in the mechanics of contact interaction. In the work, the interaction of axisymmetric bodies with circular and annular regions is considered in detail, taking into account the adhesive component. The effect of adhesion and frictional sliding was studied. Contact problems and bodies with surface notches were solved. But not enough attention was paid to research for non-canonical contact areas.

With the evolution of technologies and materials, the tasks of contact interaction take on new forms and challenges. It is the contact interaction of punches of different shapes that remains an urgent task, including those with an elliptical shape. The adhesive contact of a rigid elliptical punch with an elastic half-space was numerically investigated in [13]. The simulation was carried out using the boundary element method applying the fast Fourier transform. But the issues related to the bivalent contact area were not considered. The likely reason is difficulties associated with the analysis of cases of contact with doubly connected regions, which requires taking into account additional topological conditions and is an important step in understanding contacts of arbitrary complexity, which are not taken into account in classical solutions.

In work [14], a new approach was devised and applied to study steady-state vibrations of a ring punch on an elastic half-space without taking into account frictional forces. The solution to the contact problem is reduced to a singular integral equation, which allows it to be solved by the method of orthogonal polynomials. Other statements for such a problem, for example for regions close to rings, have not been solved. The next stage would be interesting to consider a non-axisymmetric problem taking into account friction. However, this would greatly complicate the reduction to an integral equation with the subsequent solution of orthogonal polynomials.

In the classic work [15], an analytical solution to the problem for a ring punch was obtained in the form of a double series, in which the coefficients are calculated exactly from simple recurrence relations. Owing to this approach, simple closed-form formulae can be obtained in each approximation, convenient for analysis and engineering practice. An analytical solution in the form of a series often preserves the dependence on the parameters of the problem in symbolic form. This makes it possible to analyze the impact of changing these parameters without the need for recalculation. Once the coefficients of the series are determined, the value of the function at any point is calculated much faster compared to numerical methods that require processing a large grid or matrix. Another advantage is the reduction of computing resources. To obtain a solution, only the coefficients of the series and the calculation of the sum are required, which significantly saves memory and processor time compared to the solution of complex systems of equations by FEM with a significant number of variables. Therefore, analytical solutions in the form of series are a benchmark for checking numerical methods and are indispensable for problems where high accuracy and deep theoretical analysis are required.

The advancement of the method proposed in [15] is the analytical method devised in [16]. Problems with areas close to a circular ring are reduced to problems with areas in the form of a circular ring, which makes it possible to use already known solutions. The basis of this method is the development of the potential of a simple layer. It was applied in [16] to the problems of the propagation of cracks, which are close in shape to a circular ring. But paper [16] did not consider the issues of contact interaction. This fact shows the possibility of applying the proposed method for different classes of problems. The development of this method with respect to the family of contact problems for dies with a flat base with a cross-sectional shape close to a circular ring is reported in [17, 18]. But the use of this approach is limited by the complexity of the mathematical apparatus.

Most modern contact problems with complicated conditions require a numerical solution. Inverse contact problems are very complex (often ill-stated) because they require the determination of unknown boundary conditions (e.g., contact shape, pressure distribution) based on limited measurements (e.g., surface displacements). Because of their complexity and instability, most of them do not have analytical solutions and therefore require the use of numerical methods, for example, FEM, in combination with regularization methods to ensure the stability of the solution. The contact problem in the theory of elasticity, taking into account friction and separation, can be solved as an inverse, in which the role of an unknown function is played by the deviation of tangential movements from their values in the case of complete adhesion [19]. But that work did not consider issues of three-dimensional statement of the problem. The likely reason is difficulties associated with the mathematical solution of spatial problems.

Analytical methods remain important for the construction of reference solutions. Comparison with classical solutions is a critically important engineering procedure (verification). It ensures that the tool (Computer-aided engineering (CAE) system) and the model built by the user give physically reasonable and close to accurate results before they are used to solve real, complex engineering problems. In [20, 21], a static analysis of the stressed-strained state of rolling ball bearing was implemented by FEM using the Ansys software package. Within this approach, the three-dimensional contact interaction between the bearing elements is reproduced, which includes a complex type of contact, and material characteristics are selected and refined directly in Ansys to ensure the correctness of the simulation. Works [20, 22] compare the results obtained by the finite element method using Ansys and classical solutions to elasticity problems using the Hertz method, which are used to verify numerical models, for examples with a circular contact area. But those papers lack automatic post-processing, which makes it difficult to analyze the results for specific tasks.

The open MATLAB code with a volume of 528 lines, as well as a detailed mathematical interpretation of the finite element analysis algorithms of contact interaction, is reported in [23]. It is proposed to solve nonlinear equations iteratively, using the Newton-Raphson method. Numerical examples are given and a comparison is made with the results obtained using the FEBio (Finite Elements for Biomechanics) open software package, which served as a validation of the approach.

In [24, 25], a modeling methodology combining Ansys software environments (Maxwell and Twin Builder) with MATLAB (Simulink) is proposed. The results of numerical experiments confirm the effectiveness of the proposed approach. The developed computational prototype provides a correct engineering assessment of the structure and creates the basis for its further optimization. But this approach can be successfully used in practice, first of all, by specialists with deep knowledge of both mathematics and mechanics, as well as programming.

Our review of the literature demonstrates that most current studies focus on single-connected contact zones and classical statements, while problems with doubly-connected regions remain underexplored. Also, the coordination of analytical and numerical approaches is critically important for the verification of models and ensuring their engineering reliability. The problems of contact interaction for bodies with a complex, in particular multi-connected, contact area remain insufficiently researched. A possible reason is the complexity of the mathematical apparatus required for solving such a class of problems. The lack of specialized software for post-processing automation complicates the analysis of the results. The combination of using ready-made software packages with the development of proprietary software creates the basis for a more flexible and reliable analysis of contact problems. It is the integration of these approaches that determines the promising direction of development, ensuring the optimization of engineering solutions and the expansion of modeling capabilities.

The above allows us to argue that it is reasonable to conduct a study on FEM, using reference analytical solutions, with post-processing for problems with contact areas close to circular ones.

3. The aim and objectives of the study

The purpose of our research is to devise an approach to analyzing the contact interaction between bodies of complex geometry by combining analytical solutions, numerical modeling, and developing specialized software for post-processing automation. This will make it possible to improve the process of analyzing complex contact problems, due to greater complete reproducibility and automation for different forms of dies and loads. It will also create conditions for integration with, for example, Excel, LaTeX, Word, Git, if necessary Python, Julia, through external interfaces.

To achieve this aim, the following objectives were accomplished:

– on the basis of the existing general analytical solution, perform calculations for the characteristic variants of the problem of contact interaction between an elliptical ring punch and an elastic half-space;

– to build a FEM model of the contact problem in the Ansys environment, conduct numerical experiments;

– to develop software for post-processing the problem in MATLAB;

– to represent the results of modeling and post-processing for specific numerical examples.

4. The study materials and methods

The object of our research is the process of contact interaction between a rigid punch and an elastic half-space under the action of a centrally applied compressive load. Within the limits of the model, we take the punch to be absolutely rigid with a flat base occupying a two-connected elliptic region in the plane, and we consider the elastic half-space to be homogeneous and isotropic.

The principal hypothesis assumes that the synergistic use of Ansys and MATLAB in solving contact problems of this class enables an increase in the accuracy and efficiency of modeling. Within the scope of the study, mathematical and computer modeling of the stressed-strained state of the system was implemented, which makes it possible to investigate the influence of geometrical and physical-mechanical parameters on the distribution of contact stresses.

As a mathematical model, such a statement of the problem will be chosen that is based on reducing the problem of embossing a flat punch of a non-circular ring shape to a sequence of equivalent problems for a punch in the form of a circular ring [15–18]. This approach will be used to verify the results obtained by the numerical method.

The finite element model will be implemented using the Ansys software environment. Calculations will be performed in version 2025 R2 (free Student Software) [26].

Processing of numerical results and visualization was performed in MATLAB in the MATLAB language [27] (we applied MATLAB and Simulink Student Suite R2025b personal license) based on data exported from Ansys. The starting points (x, y coordinates and z stress) are read from tabular files and transferred to a uniform regular grid using cubic interpolation.

It is planned to develop a software app for exporting numerical calculations from Ansys to MATLAB in tabular format, as well as develop software for post-processing and analysis of results.

5. Results of the simulation study of the contact interaction between a punch and an elastic half-space

5. 1. Analytical solution to the problem of indentation of a two-link punch with a base in the form of an elliptical ring

Indentation on a homogeneous and isotropic elastic half-space by a rigid punch with a flat absolutely smooth base, bounded in plan by an elliptical ring, is considered. The origin of the coordinate system coincides with the center of symmetry of the punch. The elastic half-space fills the entire part of the half-space $x_3 \le 0$ (Fig. 1). On the Ox_1x_2 plane, a doubly connected region Ω bounded by closed lines Γ_1 and Γ_2 is considered. It is assumed that there are no tangential stresses throughout the Ox_1x_2 plane.

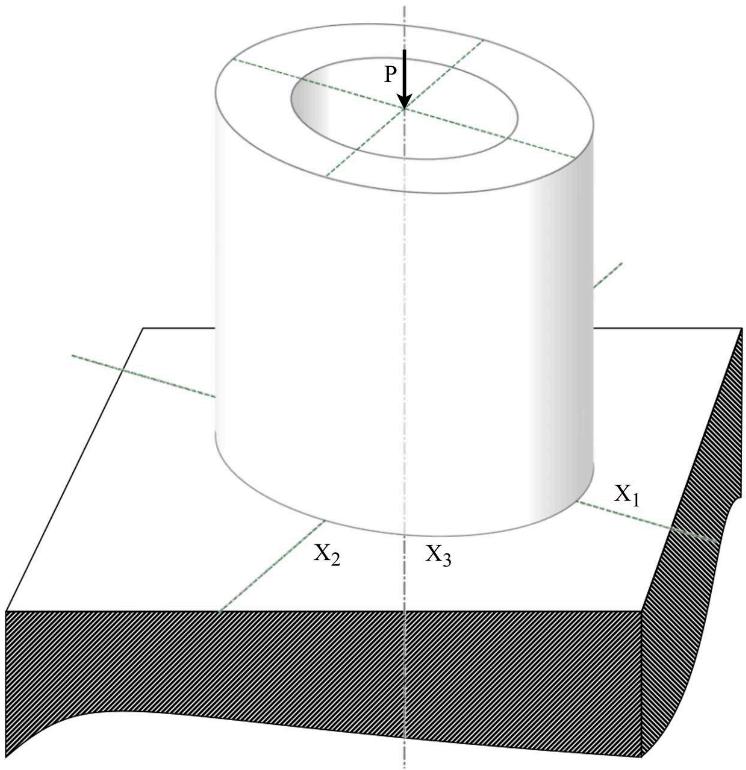

Fig. 1. Punch loading scheme

The problem considers only isotropic and homogeneous materials operating within the limits of elasticity. Given the generalized nature of the problem statement, averaged physical and mechanical characteristics of steel, representative of most structural brands, were used in the calculations. Calculations were performed for various material characteristics using Ansys. The choice of a specific brand of material was not carried out, which does not affect the correctness and universality of our results within the limits of the formulated model.

For the analytical model, solutions were obtained in dimensionless form. The resulting dimensionless dependences enable the further transition to dimensional values for any specific material through the reverse transition from dimensionless parameters to dimensional ones. This makes it possible to generalize the result and apply it to a wide range of elastic materials, which is proposed in many studies on mechanics [28].

It is assumed that there are no normal stresses on the Ox_1x_2 plane outside the region Ω of the punch contact with the elastic half-space. At the points of the region Ω, the elastic medium is subjected to a compressive load $p(x_1, x_2)$ whose function characterizes the pressure distribution under the punch and is not predetermined. Apart from the centrally applied force, there is no other load. Under the action of the load, the punch will move gradually without rotation. The equilibrium equations are written in the following form [2]:

$$P = \iint_{\Omega} p(x_1, x_2)\,\mathrm{d}x_1 \mathrm{d}x_2, \qquad (1)$$

$$M_1 = 0, \quad M_2 = 0, \qquad (2)$$

where P, M_1, M_2 are the main vector and main moments of forces applied to the punch.

In the case of a flat punch [2, 17], the condition for the vertical movement of the points of area Ω is reduced to a two-dimensional integral equation of the first kind for the desired normal pressure distribution $p(x_1, x_2)$

$$\delta = \frac{1-\nu^2}{\pi E} \iint_{\Omega} \frac{p(x_1, x_2)\,\mathrm{d}x_1 \mathrm{d}x_2}{\sqrt{(x_{10}-x_1)^2 + (x_{20}-x_2)^2}}, \qquad (3)$$

where E is the modulus of elasticity, δ is the gradual movement parallel to the vertical axis x_3. The δ value is unknown in advance; equations (1), (2) are used to determine it.

The coordinate system is transformed to a polar (ρ, θ) one in which the position of a point is given by polar radius ρ and angle θ:

$$x_1 = \rho \cos\theta,$$

$$x_2 = \rho \sin\theta. \qquad (4)$$

The perturbation method was used to find a solution to the problem [6, 16]. For this purpose, it is assumed that the equations of lines Γ_1 and Γ_2 bounding the contact area are concentric and they can be represented in the form of power series of a small dimensionless parameter ε:

$$\Gamma_1 : \rho_1 = a \cdot \left(1 + f(\varepsilon, \theta)\right);$$

$$\Gamma_2 : \rho_2 = b \cdot \left(1 + f(\varepsilon, \theta)\right), \qquad (5)$$

where $a < b$, ε<1, f(ε, θ) is a continuous and single-valued function, such that it can be represented by power series of ε

$$f(\varepsilon, \theta) = \varepsilon f_1(\theta) + \varepsilon^2 f_2(\theta) + \varepsilon^3 f_3(\theta) + \varepsilon^4 f_4(\theta) + \ldots \qquad (6)$$

A small parameter quantitatively characterizes the deviation of the shape and dimensions of the contact area from some known (for example, circular) configuration. Let's choose ε as the eccentricity of the ellipses

$$\varepsilon^2 = 1 - \frac{a_1^2}{a^2} = 1 - \frac{b_1^2}{b^2}, \qquad (7)$$

where $a < b$, a, b are focal, a_1, b_1 are minor semi-axes of ellipses.

Then the equations of the boundaries of the contact zone (4) take the form:

$$\Gamma_1 : \rho_1 = a \frac{\sqrt{1-\varepsilon^2}}{\sqrt{1-\varepsilon^2 \cos^2\theta}};$$

$$\Gamma_2 : \rho_2 = b \frac{\sqrt{1-\varepsilon^2}}{\sqrt{1-\varepsilon^2 \cos^2\theta}}. \qquad (8)$$

And equation (5) is in the following form

$$f\left(\varepsilon,\theta\right)=-\frac{1}{2}\varepsilon^2\sin^2\theta-\frac{1}{8}\varepsilon^4\left(4-3\sin^2\theta\right)\sin^2\theta-.... \qquad (9)$$

An assumption was adopted, similarly to [16–18], that the desired distribution of normal forcing $p(\rho, \theta)$ and the gradual displacement δ can also be represented in the form of expansions in powers of ε. Taking this into account, we look for a solution to equation (3) in the form of a power series ε, whose coefficients in the middle of the contact area Ω are continuous functions with continuous derivatives

$$p\left(\rho,\theta\right)=\sum_{k=0}^{\infty}p_k\left(\rho,\theta\right)\varepsilon^k. \qquad (10)$$

The solution to equations (1)–(3) will be stable and converge when $\varepsilon < 1$ [29, 30].

To obtain a dependence for $p(\rho, \theta)$ suitable for practical calculations, a fixed point with coordinates $(\rho_{\Gamma 2}, \theta_*)$ is taken on the outer contour of the punch in the plan. We draw a beam from the origin of coordinates through this point. It will cross the inner contour of the contact area Ω at the point with coordinates $(\rho_{\Gamma 1}, \theta_*)$. When determining the contact pressure $p(\rho, \theta)$ (10) at point (ρ, θ_*) of this beam, it was found that the expression $p(\rho, \theta_*)$ [17] for the first two approximations takes the form

$$p(\rho,\theta_*)=P_0+\varepsilon^2 P_1, \qquad (11)$$

where

$$P_0=\frac{Q}{2\pi b^2}\sigma_0,$$

$$P_1=\frac{Q}{2\pi b^2}\left[0.5\sigma_0+\left(0.004741\sigma_0-0.1692\sigma_1\right)\cos 2\theta_*\right]. \qquad (12)$$

Coefficients σ_i are determined from recurrence relations similar to those found in [15, 17]. Unlike other approaches, our analytical solution in the form of a series provides the possibility of accurate calculation in each approximation. This fundamental property significantly increases the reliability and accuracy of the results.

5. 2. Numerical modeling of interaction between a punch and an elastic half-space based on the finite element method

In Fig. 2, a sequence of actions for the numerical solution to the contact problem is proposed, covering four interrelated stages. First, a three-dimensional geometric model of the punch and elastic half-space is built in the Ansys software package [26].

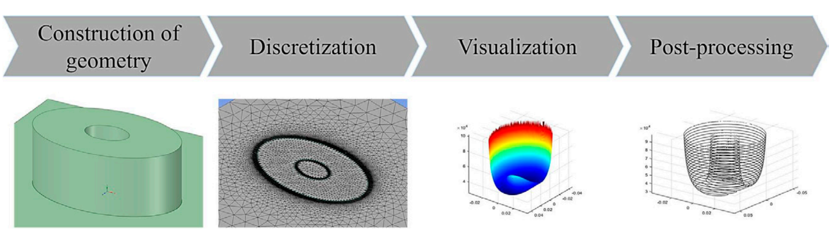

Fig. 2. Scheme of algorithm implementation for solving the contact problem

Next, discretization of the area is performed, namely, a grid of finite elements is built with local thickening in the studied zones and mandatory control of element quality metrics. For which Ansys Meshing with Mechanical profile and Quadratic element order is applied. SOLID187 (Tet10) tetrahedral elements with intermediate nodes provide the most adequate reproduction of the curvilinear boundaries of the ring contours of the punch (Fig. 3). We enabled Adaptive Sizing for the background (half-space volume), which makes it possible to automatically adjust the sampling step depending on the local geometric complexity and curvature, as well as the global High smoothing, to remove sharp quality fluctuations in transition zones.

The discretization strategy is based on the idea of local thickening at the boundaries of the punch's contact with the half-space with a transition to a coarser grid. To do this, Face Sizing with Behavior with the value Hard (the step is fixed) and the option Smooth Transition (controlled growth of the size of the elements outwards) is imposed on both ring bands. Discretization quality is evaluated by the Element Quality metric under the Aggressive Mechanical test mode. Owing to Face Meshing at the edges of the contact zone, the correct division of the half-space surface into finite elements is obtained. After that, a numerical calculation is performed.

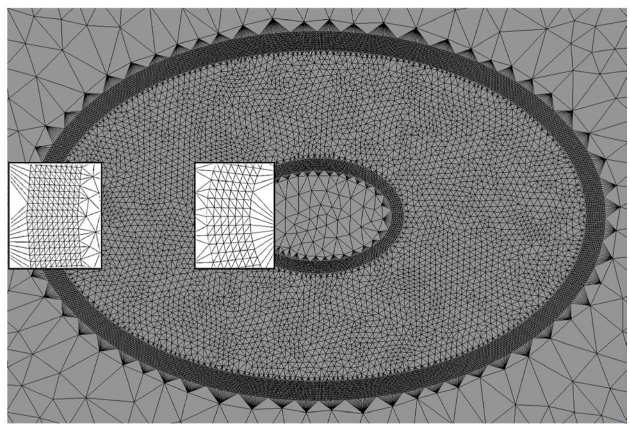

Fig. 3. An example of finite-element discretization of the half-space taking into account the elliptic two-link geometry of the punch

Our calculated data are exported to MATLAB [27]. At the current stage, data exchange between Ansys and MATLAB is implemented with the help of a specially developed software module in the APDL (user-defined macro) language, which automates the construction of tabular files of results. The macro refers to the results of calculations (coordinates of nodes, stresses, displacements), forms ordered arrays of data from them and writes them into an Excel spreadsheet file.

The final stage is postprocessing in MATLAB. Quantitative analysis is performed (search for extreme values, comparison of calculation results for cases of different initial conditions), as well as construction of final plots and comparative tables.

5. 3. Development of software for post-processing of the problem in MATLAB

The developed module in the MATLAB environment is intended for automated post-processing of calculation results and construction of visualizations for further analysis. The module forms a sequence of operations: import of tabular data, evaluation of discretization parameters, construction of a regular grid, interpolation on it based on the initial nodal values, and further detection of areas with a predetermined pressure level. Using standard MATLAB functions requires repeating many manual operations by hand, which slows down the analysis. In contrast, the proposed software implements complex, fully automated processing of a series of calculations and obtaining a single system of graphic materials.

In the fragment in Fig. 4, the Excel file with the results of the numerical experiment is shown; the first three columns of the range *A:C* are read, which are interpreted as the coordinates of the nodes *X*, *Y* and the corresponding stress values *Z*. The resulting vectors are used as data for further processing, namely interpolation on a regular grid. This approach avoids loss of accuracy as it does not require prior data aggregation or format changes. In this case, MATLAB works directly with the original tabular data.

First, the sampling step *h* is estimated using the *k-NN* method (distance to the second neighbor), which adequately characterizes the local density of points in an unevenly discretized area (Fig. 5). Next, the working step of the regular grid is set: $\Delta = 0.7h$ was chosen, which provides a smoother reconstruction without excessive overdensity. Node vectors are formed and, with the help of meshgrid, a rectangular regular grid (X_q, Y_q) for further data transfer.

```
filePath = '...\Data\Elips03.xlsx';
data = readmatrix(filePath, 'Range', 'A:C');
X = data(:, 1);
Y = data(:, 2);
Z = data(:, 3);
```

Fig. 4. Reading of raw data and formation of *X, Y, Z* arrays

```
Delta = 0.7 * h;
nx = max(500, min(1000, ceil(scaleX / Delta)));
ny = max(500, min(1000, ceil(scaleY / Delta)));
xq = linspace(min(X) - 0.1*scaleX, max(X) + 0.1*scaleX, nx);
yq = linspace(min(Y) - 0.1*scaleY, max(Y) + 0.1*scaleY, ny);
[Xq, Yq] = meshgrid(xq, yq);
```

Fig. 5. Construction of a regular grid with a step of $\Delta = 0.7h$ and safety fields

The stress values are interpolated onto a regular grid. Matrix $Z_q = F(X_q, Y_q)$ is calculated on the resulting grid, which is used further (Fig. 6).

```
interpMethod = "natural";
extrapMethod = "none";
F = scatteredInterpolant(X, Y, Z, ...
    interpMethod, extrapMethod);
Zq = F(Xq, Yq);
```

Fig. 6. $Z_q = F(X_q, Y_q)$ matrix reconstruction

To analyze our results, characteristic zones of the given pressure were found (Fig. 7).

A mask is formed where Z_q is below the selected percentile relative to the valid values (Fig. 8). At the output, a binary mask is obtained for building the desired area of pressure distribution.

Second derivatives Z_{xx}, Z_{yy}, Z_{xy} are calculated. Based on them, matrix $H = \begin{bmatrix} Z_{xx} & Z_{xy} \\ Z_{xy} & Z_{yy} \end{bmatrix}$ is formed and eigenvalues $\lambda_{1,2}$ are found. Borders are zones where the minimum eigenvalue $\min(\lambda_1, \lambda_2)$ is negative (Fig. 9).

```
regions = find_interesting_zones(Zq, ...
'Sigma', 2.5, ...
'LowPct', 5, 'LowMinArea', 30, 'LowCloseDisk', 2, ...
'UseMarkerWS', true, 'WS_Grad', true, 'WS_Pct', 25, ...
'WSMinArea', 150, 'WSCloseDisk', 2, ...
'HessianKappa', 0.01, 'ValleyMinArea', 150, ...
'PlateauGradPct', 30, 'PlateauZPct', 20, 'PlateauMinArea', 150);

f3=figure('Color','w');
tiledlayout(2,2,'Padding','compact','TileSpacing','compact');
nexttile; imshow(regions.low_pressure.mask,[]);
nexttile; imshow(regions.basins.mask,[]);
nexttile; imshow(regions.valleys.mask,[]);
nexttile; imshow(regions.plateaus_low.mask,[]);
```

Fig. 7. Zone selection diagnostic panel and key parameters of the algorithm

```
thr = prctile(Zvalid, S.LowPct);
BW  = (Zsm <= thr) & M;
if S.LowOpenDisk>0,  BW = imopen (BW, strel('disk',S.LowOpenDisk)); end
if S.LowCloseDisk>0, BW = imclose(BW, strel('disk',S.LowCloseDisk)); end
BW  = bwareaopen(BW, S.LowMinArea);
if S.CleanBorder, BW = imclearborder(BW); end
regions.low_pressure.mask = BW;
```

Fig. 8. Search for zones of a given pressure

```
if S.UseValleys
    hx  = [1 -2 1]; hy = hx';
    Zxx = imfilter(Zsm, hx, 'replicate','conv');
    Zyy = imfilter(Zsm, hy, 'replicate','conv');
    Zxy = imfilter(imfilter(Zsm,[1 0 -1]/2,'replicate'), ...
        [1;0;-1]/2,'replicate');
    tr   = Zxx + Zyy;|     detH = Zxx.*Zyy - Zxy.^2;
    disc = max(tr.^2/4 - detH, 0);
    lambda1 = tr/2 + sqrt(disc);   lambda2 = tr/2 - sqrt(disc);
    kappa = S.HessianKappa * zRange;
    valley = (min(lambda1, lambda2) < -kappa) & M;
    valley = bwareaopen(valley, S.ValleyMinArea);
    if S.CleanBorder, valley = imclearborder(valley); end
    regions.valleys.mask = valley;
end
```

Fig. 9. Determining the boundaries of the specified pressure zones

The gradient $\nabla Z_q = (G_x, G_y)$ field and its modulus $|\nabla Z_q|$ are calculated. Low pressure zones are defined as the intersection of two criteria: $|\nabla Z_q|$ below the percentile threshold (PlateauGradPct), Z_q does not exceed the specified level (Fig. 10).

```
if S.UsePlateaus
    [Gy, Gx] = gradient(Zsm); Gmod = hypot(Gx,Gy);
    gthr = prctile(Gmod(M), S.PlateauGradPct);
    zthr = prctile(Zvalid,    S.PlateauZPct);
    plateau = (Gmod <= gthr) & (Zsm <= zthr) & M;
    plateau = bwareaopen(plateau, S.PlateauMinArea);
    if S.CleanBorder, plateau = imclearborder(plateau); end
    regions.plateaus_low.mask = plateau;
end
```

Fig. 10. Determining low pressure zones

The proposed software, implemented by means of MATLAB, provides a complete sequence of data processing and visualization generation operations, in particular:

– automated calculation of the dimensions of the stress matrix taking into account the results of numerical modeling;

– interpolation of stress values on a regular grid;

– selection of zones of extreme or specified pressure;

– saving processing results in the required formats.

Standard MATLAB functions (readmatrix, scatteredInterpolant, gradient, eig, etc.) act as basic building blocks in the developed software.

5. 4. Results of modeling and postprocessing and comparing analytical and numerical results

Applying the algorithm described above, a specific example of forcing into the elastic half-space by a cylindrical absolutely rigid flat punch was considered, the cross-section of which has the area of an elliptical ring in the plan (Fig. 1). The reference analytical solution was found using formulae (11), (12) in dimensionless parameters. The normal dimensionless pressure was determined as p / p^*, $p^* = Q / (2\pi b^2)$, and the geometric dimensions of the punch were determined in relation to b. This makes it possible to universalize the result and carry out analysis for different classes of materials by substituting specific physical and mechanical characteristics. The calculations in Ansys use the average values of density and modulus of elasticity, which are typical for most structural grades of steel. The following parameters are set: half-space – Structural Steel (isotropic linear elastic) with parameters: density $\rho = 7850$ kg/m^3, Young's modulus $E = 2.0 \times 10^{11}$ Pa (200 GPa), Poisson's ratio $\nu = 0.3$, shear modulus $G = 7.6923 \times 10^{10}$ Pa ($\approx$ 76.9 GPa). To simplify the model, the punch is assigned the same material, but it is declared absolutely solid (Rigid). The outer diameter of the punch is chosen to be 100 mm, and the inner diameter is 40 mm. Centrally applied load, $P = 1000$ N.

Analytical and numerical calculations have been compared. The dependence of pressure distribution under the elliptical ring punch is shown in Fig. 11 for the case $\varepsilon = 0.6$: solid lines – analytical solution using equations (11), (12); dashed – finite elements; lines 1 correspond to sections $\theta = \pi / 2$, lines 2 – $\theta = \pi / 4$, lines 3 – $\theta = 0$.

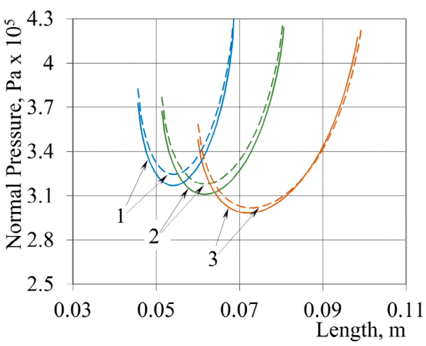

Fig. 11. Comparing the methods

The dependence of pressure distribution on the eccentricity in the cross section along the x_1 axis when $\theta = 0$ is illustrated by the plots shown in Fig. 12.

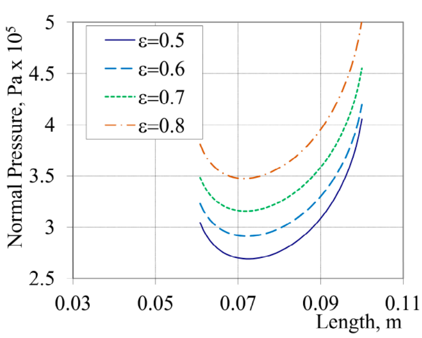

Fig. 12. Influence of eccentricity on normal pressure

Fig. 13 shows the pressure distribution surface over the contact area for values $\varepsilon = 0.6$.

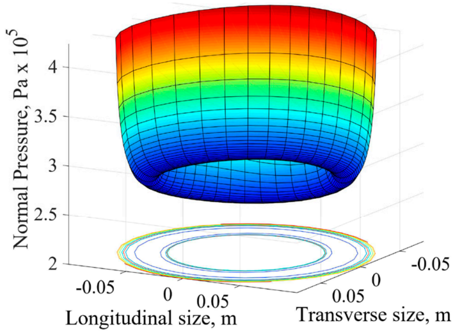

Fig. 13. Distribution of normal pressure under the base at $\varepsilon = 0.6$

A similar surface is depicted in Fig. 14 for values $\varepsilon = 0.8$.

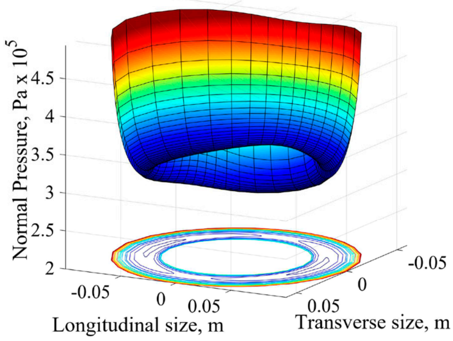

Fig. 14. Distribution of normal pressure under the base at $\varepsilon = 0.8$

The corresponding picture of equal pressure is depicted in the Ox_1x_2 plane.

6. Discussion of results of devising an approach to analyzing the contact interaction between bodies of complex geometry

The devised approach to analyzing the contact interaction between bodies of complex geometry is explained by the need to improve the work of engineers at the stage of designing parts, structures, and mechanisms. Our approach includes three successive stages. The first, calculations according to formulae (11), (12), is an analytical benchmark for the distribution of contact pressure. Next, a highly detailed FEM-modeling is carried out in Ansys based on the finite-element discretization of the half-space, taking into account the elliptic two-link geometry of the punch (Fig. 3). At the last stage, postprocessing is performed in MATLAB (Fig. 4–10).

Each stage not only provides its data but also serves as a check on the previous one. For example, for interpretation and verification, a comparison of analytical and numerical solutions for the typical case $\varepsilon = 0.6$ (Fig. 11) demonstrates practically sufficient convergence of the pressure distribution curves, which confirms the adequacy of the constructed model. That is, analytics sets a physically based benchmark. FEM replaces the infinitesimal nature of the differential equations with a finite, discrete grid. The discrepancy between the analytical solution and the solution obtained by FEM is greatest in narrow sections or zones of sharp change in the geometry or gradient of the pressure function (for example, stress concentration), since the accuracy of FEM depends on the density (fineness) of the mesh. In narrow sections, physical parameters (such as stresses or gradients) change very quickly. If the grid step in these critical zones is not small enough, FEM cannot adequately approximate the rapid change of these parameters. Such a case can lead to a significant discretization error, which decreases with a decrease in the size of the elements (dense grids) and, accordingly, a greater discrepancy with the exact analytical result. In addition, FEM requires solving a large system of linear algebraic equations. This process, especially when using iterative solvers, always contains a numerical round-off error, which also contributes to the divergence.

Ansys calculates stresses for real 3D geometry. MATLAB performs a quantitative mapping and highlights areas of potential congestion. As a result, calculated data in tabular and graphical forms were obtained, which allow us to analyze contact pressure distribution to identify stress concentrations and rational design.

Owing to the analysis of the distribution of contact pressure under the base of the two-link elliptical punch, characteristic regularities were revealed. With an increase in the eccentricity of the ellipse (which corresponds to a decrease in the area of the contact area), the value of the average pressure under the punch increases (under the condition of the same load) (Fig. 12). When comparing Fig. 13, 14 it can be seen that the minimum values are located at the points on the rays $\theta = 0$ and $\theta = \pi$. Lines of equal pressure form closed contours around the areas with minimum values, and approaching the contact boundaries, they gradually transform into curves similar in shape to the boundaries of the contact area itself. At $\varepsilon = 0.6$, the pressure is more evenly distributed than at $\varepsilon = 0.8$, there are no clearly expressed minima, this homogenization is due to the fact that the contact area takes on a shape closer to a circular ring. A lower pressure is recorded in the central zone, which increases at the edges of the contact area. The identified numerical dependences are consistent with the analytical calculation according to formula (11), confirming the correctness of our results.

The analytical module is based on classical solutions for ring punches [15, 17, 18]. In contrast to typical applied works, where the results of FEM are not verified [21], we use the analytical model as an independent reference for validating the pressure field and checking the quality of the mesh. This fundamentally increases the reproducibility of the resulting pressure distributions.

At the numerical stage, Ansys is not used as a black box but as a controllable toolbox. A three-dimensional model of an absolutely rigid punch and an elastic half-space was constructed, contact pairs were introduced, and local thickening of the mesh along the inner and outer boundaries of the elliptical ring was performed. Similar strategies for grid construction are used in [20, 21] but, in this case, the calculation results are compared with the analytical reference ones.

The MATLAB module performs full-fledged post-processing: import of Ansys nodal data, interpolation, automatic selection of zones of specified pressure. Similarly to the well-known approaches of simultaneous application of Ansys-MATLAB, reported in [24, 25], such an approach, for example, helps diagnose potentially dangerous areas of contact.

Therefore, the proposed approach makes it possible to combine the strengths of analytical methods and numerical modeling. The analytical component performs the role of a physically based standard to control the correctness of the results. The Ansys finite element analysis environment provides a geometrically adequate three-dimensional contact interaction statement and a robust computational solution. The MATLAB-based software that we have developed implements formalized quantitative post-processing, which includes interpolation, identification of characteristic load zones, and comparison of different geometry and load variants. Previous approaches used Ansys and MATLAB in isolation [20–23, 31, 32] or integrated mainly as tools for the simulation of controlled processes [24, 27]. In contrast to already existing approaches, an integrated scheme together with our proprietary software module is proposed. The scheme built functions as a reproducible engineering procedure for quantitative assessment and interpretation of contact pressure distribution in spatially complex two-connected contact zones.

Conventional use of standard functions in MATLAB requires monotonous repetition of dozens of operations for each calculation, manual selection of computational grid parameters, and separate construction of final visual and tabular data for each case. Instead, the proposed software offers comprehensive automation of computational processing and immediately generates a single package of graphical materials (surfaces, isolines, cross-section diagrams) and tabular data necessary for comprehensive analysis of results. This approach significantly increases the analytical efficiency.

Despite these advantages, our study has some limitations and drawbacks. The work assumes (for simplification) an absolutely rigid punch and no friction (non-adhesive contact) with linear-elastic behavior of the half-space material. Only static loading was taken into account, and dynamic effects, thermoelasticity, or plasticity of the material were not taken into account.

Further research should eliminate the specified restrictions and expand the scope of application of our methodology. Future studies may aim to implement our approach to problems that take into account the roughness of the half-space, friction forces, and adhesion in contact [2, 13, 17, 29], which will require the improvement of the mathematical apparatus for solving more complex contact problems. It is important to apply the proposed algorithm to other types of contact interactions – in particular, for punches of arbitrary shape or contacts with anisotropic and layered materials. Analysis of dynamic contact processes [4, 14, 28] and thermomechanical contact conditions [1] is a relevant area, although its implementation will require significant computing resources and may be accompanied by additional difficulties.

7. Conclusions

1. We have analytically solved the problem of contact interaction between a rigid punch, which has a base in the form of an elliptical ring, and an elastic half-space; it serves a reference basis for the verification of numerical modeling. The resulting formulae are convenient for engineering practice, which is explained by their closed form. The formulae serve as a direct engineering toolkit for calculating structural elements with elliptical annular contact areas to ensure their reliability and durability.

2. Based on the construction of FEM models for solving the contact problem in the Ansys environment, reasonable variants of discretization of the contact zone were obtained. The most acceptable discretization of the contact zone was chosen. Such FEM models have a thickened mesh at the edges of the contact zone and a sparser mesh where the normal

pressure reaches its minimum values. The FEM models built allow us to solve individual problems in the general task of modeling the process of interaction of bodies with a two-link contact area in the form of an elliptical ring by varying the properties of materials and boundary conditions.

3. Based on the developed APDL macro, an automated process of data transfer from Ansys to MATLAB was obtained. Specially developed software in MATLAB implemented the task of postprocessing. Such a combination partially eliminates the problem associated with insufficient automation of standard post-processing for solving problems of contact mechanics. The features and differences are the elimination of the stage of manual export and formatting of data, minimizing the risk of errors related to the human factor, improving the quality of results, building visualizations.

4. A comparative analysis between analytical and numerical results was conducted. In the central zone of the elliptical ring, which is the contact zone, the maximum discrepancy does not exceed 5–10%. In numerical modeling, the largest, albeit finite, pressure values are located on the boundaries of the contact zone. In the case of an analytical solution, the value of the pressure at the boundaries goes to infinity. This difference is caused by the discreteness of the FEM model and simplifications of the analytical statement of the boundary conditions.

Conflicts of interest

The authors declare that they have no conflicts of interest in relation to the current study, including financial, personal, authorship, or any other, that could affect the study, as well as the results reported in this paper.

Funding

The study was conducted without financial support.

Data availability

All data are available, either in numerical or graphical form, in the main text of the manuscript.

Use of artificial intelligence

The authors confirm that they did not use artificial intelligence technologies when creating the current work.

Authors' contributions

Tetyana Zaytseva: Conceptualization, Methodology, Software, Formal analysis, Resources, Writing – original draft, Writing – review & editing, Supervision; **Ganna Shyshkanova**: Methodology, Software, Investigation, Writing – original draft, Writing – review & editing, Visualization; **Yaroslav Honcharov**: Software, Validation, Formal analysis, Investigation, Resources, Data Curation, Writing – original draft, Writing – review & editing, Visualization.

References

1. Martynyak, R. M., Mykytyn, M. M., Malanchuk, N. I. (2025). Thermomechanical contact of an elastic half-space and a rigid base under the action of a circular heat sink. Materials Science, 61 (1), 129–136. https://doi.org/10.1007/s11003-025-00970-3
2. Popov, V. L., Heß, M., Willert, E. (2019). Handbook of Contact Mechanics. Springer Berlin Heidelberg. https://doi.org/10.1007/978-3-662-58709-6
3. Barber, J. R., Ciavarella, M. (2000). Contact mechanics. International Journal of Solids and Structures, 37 (1-2), 29–43. https://doi.org/10.1016/s0020-7683(99)00075-x
4. Barber, J. R., Ciavarella, M. (2023). Approximate methods for contact problems involving beams. The Journal of Strain Analysis for Engineering Design, 59 (2), 133–138. https://doi.org/10.1177/03093247231200911
5. Saha, S., Kumar, V., Bhushan, A., Das, A. N. (2023). Mechanics of Indentation for an Elastic Half-Space by Punches. Wseas Transactions On Applied And Theoretical Mechanics, 18, 310–317. https://doi.org/10.37394/232011.2023.18.29
6. Guz', A. N., Nemish, Yu. N. (1987). Perturbation of boundary shape in continuum mechanics (review). Soviet Applied Mechanics, 23 (9), 799–822. https://doi.org/10.1007/bf00887782
7. Guz', A. N., Babich, S. Yu., Rudnitsky, V. B. (1998). Contact Problems for Elastic Bodies With Initial Stresses: Focus on Ukrainian Research. Applied Mechanics Reviews, 51 (5), 343–371. https://doi.org/10.1115/1.3099009
8. Babych, S. Yu., Yarets'ka, N. O. (2021). Contact Problem for an Elastic Ring Punch and a Half-Space with Initial (Residual) Stresses*. International Applied Mechanics, 57 (3), 297–305. https://doi.org/10.1007/s10778-021-01081-7
9. Ostryk, V. I., Ulitko, A. F. (2019). Circular interface crack with frictional contact of faces. Matematychni Metody ta Fizyko-Mekhanichni Polya, 47 (1), 84–94. Available at: http://journals.iapmm.lviv.ua/ojs/index.php/MMPMF/article/view/2644
10. Ostryk, V. I. (2024). Contact problem for interfacial exfoliated inclusion. Acta Mechanica, 235 (9), 5863–5884. https://doi.org/10.1007/s00707-024-04004-y
11. Savruk, M. P. (2021). Stress concentration near curvilinear holes and notches with unsmoothed contours. Materials Science (Physicochemical mechanics of materials), 57 (3), 41–52. Available at: http://jnas.nbuv.gov.ua/article/UJRN-0001260776
12. Martyniak, R. M., Malanchuk, N. I., Kuzmenko, V. I., Mykhalchuk, H. Y., Chumak, K. A., Tkachuk, M. M. et al. (2022). Kontaktna mekhanika. Fryktsiyna i adheziyna vzaiemodiya poverkhon. Lviv: Vydavets Viktoriya Kundelska, 254. Available at: https://www.researchgate.net/publication/357335537
13. Li, Q., Wilhayn, J., Lyashenko, I. A., Popov, V. L. (2022). Adhesive contacts of rough elliptical punches. Mechanics Research Communications, 122, 103880. https://doi.org/10.1016/j.mechrescom.2022.103880

14. Vaisfeld, N. D., Protserov, Yu. S., Tolkachov, A. V. (2024). Axisymmetric Problem of Vibrations of Circular or Ring Punch on Elastic Half-Space*. International Applied Mechanics, 60 (3), 313–322. https://doi.org/10.1007/s10778-024-01284-8
15. Roitman, A. B., Shishkanova, S. F. (1973). The solution of the annular punch problem with the aid of recursion relations. Soviet Applied Mechanics, 9 (7), 725–729. https://doi.org/10.1007/bf00882996
16. Shishkanova, S. F. (1990). Stress state of an elastic half-space weakened by a plane crack which is close to being annular. Soviet Applied Mechanics, 26 (5), 430–435. https://doi.org/10.1007/bf00887257
17. Shyshkanova, G. A., Zaytseva, T. A., Frydman, A. D. (2015). The analysis of manufacturing errors effect on contact stresses distribution under the ring parts deformed asymmetrically. Metallurgical and Mining Industry, 7, 352–357. Available at: https://www.metaljournal.com.ua/assets/Journal/english-edition/MMI_2015_7/055Shyshkanova-352-357.pdf
18. Shyshkanova, G., Zaytseva, T., Zhushman, V., Levchenko, N., Korotunova, O. (2023). Solving three-dimensional contact problems for foundation design in green building. Journal of Physics: Conference Series, 2609 (1), 012001. https://doi.org/10.1088/1742-6596/2609/1/012001
19. Obodan, N. I., Zaitseva, T. A., Fridman, O. D. (2019). Contact Problem for a Rigid Punch and an Elastic Half Space as an Inverse Problem. Journal of Mathematical Sciences, 240 (2), 184–193. https://doi.org/10.1007/s10958-019-04346-2
20. Šulka, P., Sapietová, A., Dekýš, V., Sapieta, M. (2018). Static structural analysis of rolling ball bearing. MATEC Web of Conferences, 244, 01023. https://doi.org/10.1051/matecconf/201824401023
21. Bhandari, S., Jotautienė, E., Juostas, A., Ganta, A. (2021). Finite Element Analysis of Deep Groove Ball Bearing to obtain stress and contact pressure. Žmogaus ir gamtos sauga 2021. Available at: https://www.vdu.lt/cris/bitstreams/b992eb0a-68ca-45e4-b2a7-b2524e4df313/download
22. Verma, A., Sahoo, V. (2025). Analytical and numerical analysis of load distribution in a thin walled flexible race ball bearing. The Journal of Strain Analysis for Engineering Design, 60 (7), 526–539. https://doi.org/10.1177/03093247251320838
23. Wang, B., Bai, J., Lu, S., Zuo, W. (2025). An open source MATLAB solver for contact finite element analysis. Advances in Engineering Software, 199, 103798. https://doi.org/10.1016/j.advengsoft.2024.103798
24. Mersha, T. K., Du, C. (2021). Co-Simulation and Modeling of PMSM Based on Ansys Software and Simulink for EVs. World Electric Vehicle Journal, 13 (1), 4. https://doi.org/10.3390/wevj13010004
25. Schemmel, R., Krieger, V., Hemsel, T., Sextro, W. (2021). Co-simulation of MATLAB and ANSYS for ultrasonic wire bonding process optimization. Microelectronics Reliability, 119, 114077. https://doi.org/10.1016/j.microrel.2021.114077
26. Ansys for Students. Available at: https://www.ansys.com/academic/students
27. MATLAB for Students. Available at: https://uk.mathworks.com/products/matlab/student.html
28. Korotunova, O., Mastynovsky, Y., Shyshkanova, G., Mizerna, O., Nechyporenko, N., Zaytseva, T. et al. (2022). Multilayer Structures under Non-Stationary Loading: Prediction of Damages and Further Operation Possibility. Journal of Physics: Conference Series, 2346 (1), 012006. https://doi.org/10.1088/1742-6596/2346/1/012006
29. Shyshkanova, G. (2018). About Stability of First Kind Equation Solving. 2018 IEEE First International Conference on System Analysis & Intelligent Computing (SAIC), 1–5. https://doi.org/10.1109/saic.2018.8516805
30. Kress, R. (1989). Tikhonov Regularization. Linear Integral Equations, 243–258. https://doi.org/10.1007/978-3-642-97146-4_16
31. Sun, P., Wang, J., Bai, J., Zuo, W. (2024). An Open Source Matlab Solver for Finite Element Analysis of Continuous Fiber Reinforced Composites Using Membrane-Embedded Model. https://doi.org/10.2139/ssrn.5005014
32. Mose, B. R., Shin, D.-K., Nam, J.-H. (2024). Numerical analysis of stresses on angular contact ball bearing under the static loading with respect to race thickness and housing stiffness. Scientific Reports, 14 (1). https://doi.org/10.1038/s41598-024-66479-y