An educational computational program for nonlinear geometric analysis of truss structures using the Finite Element Method based on Positions

Danilo Silva Bomfim[1], Luiz Fernando Martha[2], Rodrigo Ribeiro Paccola[1]

[1]*Dept. of Structural Engineering, São Carlos School of Engineering, University of São Paulo*
Av. Trabalhador São Carlense, 400, Parque Arnold Schimidt, 13566-590, São Paulo/São Carlos, Brazil
dsbomfim2@hotmail.com, rpaccola@sc.usp.br
[2]*Dept. of Civil and Environmental Engineering, Pontifical Catholic University of Rio de Janeiro*
Rua Marques de São Vicente, 225, 22451-900, 22451-900, Gávea, Rio de Janeiro/Rio de Janeiro, Brazil
lfm@tecgraf.puc-rio.br

Abstract. This study presents an educational software called the Truss Positional Finite Element Method Program (TPFEM), designed for analyzing truss structures using the Finite Element Method based on Positions (FEMP). Unlike the conventional Finite Element Method (FEM), which uses displacements as degrees of freedom, FEMP employs the nodal positions of discretized elements. This positional approach naturally and effectively incorporates geometric nonlinearity into its formulation, simplifying the nonlinear geometric analysis of structures. The developed application aims to enhance the teaching and learning process of FEMP for truss structures, making the experience more dynamic, user-friendly, and engaging. The educational program offers a range of functionalities, allowing users to model multiple truss structures simultaneously through an intuitive multi-window interface and conduct both static and dynamic analyses of truss structures.

Keywords: Educational Computational Program, Finite Element Method, Nonlinear geometric analysis, Truss structures.

1 Introduction

Educational software plays a crucial role in the teaching and learning process, providing interactive tools and dynamic resources that complement traditional education. These programs facilitate the understanding of complex concepts, promote autonomous and personalized learning, and increase student engagement. In the context of teaching numerical methods and engineering, Finite Element Method (FEM) programs are especially important.

FEM is a fundamental technique for analyzing complex problems in engineering and physics, allowing the solution of partial differential equations that describe phenomena such as stress distribution in structures, heat transfer, and fluid dynamics. Educational software that incorporates FEM offers students the opportunity to visualize and interact with these analyses, enhancing their theoretical and practical understanding. They also facilitate detailed simulations and experimentation with different parameters, preparing students to face real-world challenges in a professional environment. Thus, the use of FEM software in education not only enriches learning but also promotes a more robust and practical training, aligned with the current demands of industry and scientific research.

This work presents an educational program for the static and dynamic analysis of truss structures using the Finite Element Method based on Positions (FEMP), also known as the Positional Finite Element Method. The software, named the Truss Positional Finite Element Method Program (TPFEM), integrates a hybrid code. The graphical interface and data structure for creating geometric models were developed in Python using the PyQt [1], PyOpenGL [2], and HETOOL [3] libraries. The static and dynamic analyses are then performed by a code developed in FORTRAN, utilizing the FEMP method.

The integration of FORTRAN code for conducting static and dynamic analyses ensures precise and robust calculations. Furthermore, using FEMP as a methodological foundation simplifies the understanding of complex concepts, promoting deeper and more practical learning. This approach also innovatively facilitates the natural and efficient integration of geometric nonlinearities. TPFEM enriches the educational experience by increasing interactivity and engagement, while also preparing students to grasp the intricate nonlinear geometric behavior of truss structures.

2 HETOOL library

This study employs the HETOOL library, developed in Python by Bomfim et al. [3], to construct the educational program TPFEM developed in this work. HETOOL represents an innovative object-oriented framework designed for interactive 2-D geometric modeling, built on the well-established Half-Edge Data Structure developed by Mäntylä [4]. The Half-Edge data structure is based on the concept of a half-edge, which stores significant topological information. Conceptually, a half-edge originates from the subdivision of an edge into two oriented semi-edges within incident loops having opposite directions. Most topological operations rely on information stored in half-edges.

The Half-Edge data structure consists of several key topological elements: solids (S), faces (F), loops (L), edges (E), half-edges (HE), and vertices (V). Specific operators are employed to modify it in a topologically consistent manner, guided by the Euler equation: $V - E + 2F - L - 2 = 0$. This equation balances the counts of each topological element within a solid. Operators based on this equation are known as Euler Operators, as described by Hoffman [5], allowing for the creation or modification of closed surfaces by adding or removing faces, edges, and vertices.

HETOOL is designed to efficiently manage general 2-D models and planar subdivisions commonly used in scientific and engineering applications. Featuring a dynamic data structure, this library automates the seamless integration of geometric elements and provides powerful functionalities that enable users to harness its capabilities without needing an extensive grasp of underlying topological principles. HETOOL has been applied in several studies by Bomfim et al. [6], Soares et al. [7], Peixoto et al. [8], Peixoto and Rangel [9], and Peixoto [10]. For more detailed information about HETOOL, refer to Bomfim [11].

3 Finite Element Method based on Positions

The FEMP, originally described by Bonet et al. [12] Coda and Greco [13], is a numerical method used to discretize a domain to find approximate solutions for nonlinear equations derived from the variation of mechanical potential energy. Unlike the traditional FEM, which uses displacements as degrees of freedom, FEMP uses the nodal positions of the discretized elements. This method is based on a total Lagrangian formulation, meaning that the current configuration of the bodies is defined from the initial configuration. The following sections provide a summary of FEMP, with further details available in the works of Coda [14], Paccola and Coda [15], Ramos et al. [16], Paccola et al. [17], Sampaio et al. [18].

3.1 Kinematics and positional mapping

The change in the shape of a generic body typically occurs from an initial configuration B^0 to a current configuration B, described by a function $\vec{f}$, known as the configuration change function. This function $\vec{f}$ is invertible, with its inverse denoted as $\vec{g}$. The initial configuration (B^0) is defined by the coordinates x_1, x_2, x_3, while the current configuration (B) is characterized by the coordinates y_1, y_2, y_3. By considering two infinitesimal vectors $d\vec{x}$ and $d\vec{y}$, within B^0 and B, one can express the configuration change function $\vec{f}$, evaluated at a point (x_1, x_2, x_3) and in the neighborhood of (x_1^0, x_2^0, x_3^0), using differential calculus as follows:

$$\vec{f}\,(x_1, x_2, x_3) = \vec{f}\left(x_1^0, x_2^0, x_3^0\right) + \nabla\,\vec{f} \cdot d\vec{x}. \tag{1}$$

By taking the difference between $\vec{f}(x_1, x_2, x_3)$ and $\vec{f}(x_1^0, x_2^0, x_3^0)$, one arrives at the following expression:

$$d\vec{y} = d\vec{f} = \vec{f}\,(x_1, x_2, x_3) - \vec{f}\left(x_1^0, x_2^0, x_3^0\right) = \nabla\,\vec{f} \cdot d\vec{x} = \mathbf{A} \cdot d\vec{x} \tag{2}$$

where the gradient of the change of the configuration function is defined as $\mathbf{A}$.

3.2 Saint-Venant-Kirchhoff constitutive law

To describe a geometrically nonlinear formulation, an objective strain measure is necessary, one that can register zero strain for rigid body translations and rotations. The Green-Lagrange strain is such an objective and Lagrangian measure, defined by the following expression:

$$\mathbb{E} = \frac{1}{2}\left(\mathbf{A}^T \cdot \mathbf{A} - \mathbf{I}\right) = \frac{1}{2}\left(\mathbf{C} - \mathbf{I}\right) \tag{3}$$

where $\mathbf{C}$ is the second-order symmetric tensor defined as the right Cauchy-Green stretch tensor, and $\mathbf{I}$ is the second-order identity tensor.

In this study, the Saint-Venant-Kirchhoff (SVK) constitutive model is utilized for the finite elements, which is appropriate for large displacements and moderate strains. This model defines a linear relationship between the second Piola-Kirchhoff stress tensor and the Green-Lagrange strain tensor. Consequently, the specific strain energy is expressed as:

$$u_e(\mathbb{E}) = \frac{1}{2}\mathbb{E} : \mathfrak{C} : \mathbb{E} \tag{4}$$

where $\mathfrak{C}$ is the elastic constitutive tensor.

3.3 Principle of Stationarity of Mechanical Energy

According to the Principle of Stationarity of Mechanical Energy, the mechanical equilibrium of a body occurs when the variation of its mechanical potential energy is zero. The functional form of the mechanical energy of a solid can be expressed as:

$$\Pi = \mathbb{P} + \mathbb{U} + \mathbb{K} \tag{5}$$

where $\mathbb{P}$ is the work of external forces, $\mathbb{U}$ is the internal strain energy, and $\mathbb{K}$ is the kinetic energy.

In the FEMP, positions are considered as unknowns, which are utilized to calculate variations. Hence, for any given body, one can express the variation of the mechanical energy functional concerning positions as:

$$\delta\Pi = \int_{V_0} \rho_0 \ddot{\vec{Y}} \cdot \delta\vec{Y}\, dV_0 - \int_{V_0} \vec{b}_0 \cdot \delta\vec{Y}\, dV_0 - \int_{A_0} \vec{p}_0 \cdot \delta\vec{Y}\, dA_0 + \int_{V_0} \mathbf{S} : \delta\mathbb{E}\, dV_0 = 0 \tag{6}$$

where $\vec{b}_0$ is the volume force vector in the initial configuration, $\vec{p}_0$ is the surface force vector in the initial configuration, V_0 is the initial volume of the solid, A_0 is the initial surface area of the solid, and ρ_0 is the initial mass density.

3.4 Solution strategy

The FEMP employs a geometrically exact approach centered on positions, resulting in a set of nonlinear equations. These equations are solved using the Newton-Raphson method [19], where the current position serves as an initial guess, generating a imbalance among the internal force vectors F_i^{int}, external forces F_i^{ext}, and inertial forces F_i^{iner}. This imbalance is represented by the residual vector g_i, which is derived from eq. (5). Thus, one can obtain:

$$g_i = \frac{\partial \Pi}{\partial Y_i} = \frac{\partial \mathbb{U}}{\partial Y_i} + \frac{\partial \mathbb{P}}{\partial Y_i} + \frac{\partial \mathbb{K}}{\partial Y_i} = F_i^{int} - F_i^{ext} + F_i^{iner} \neq 0_i. \tag{7}$$

The Newton-Raphson method is employed to determine the current configuration of the structure, aiming to minimize the mechanical imbalance vector towards zero. Assuming the components of g_i are continuous functions near the solution and starting with a provisional solution $\vec{Y}^0$ close to the exact solution, the vector g_i can be approximated by a truncated Taylor series up to the first order as:

$$g_i(\vec{Y}) = g_i(\vec{Y}^0) + \left.\frac{\partial g_i}{\partial Y_j}\right|_{\vec{Y}^0} \Delta Y_j \approx 0 \tag{8}$$

where ΔY_j denotes the correction of the current positions. Equation (8) can be rewritten as:

$$\Delta Y_j = \left(\left.\frac{\partial g_i}{\partial Y_j}\right|_{\vec{Y}^0} \right)^{-1} g_i(\vec{Y}^0) = -\left(\mathbf{H}_{ij}\right)^{-1} g_i(\vec{Y}^0) \tag{9}$$

where $\mathbf{H}_{ij}$ is called the Hessian matrix or tangent stiffness of the problem for the trial position.

The trial solution is then refined at each iteration by $Y_j = Y_j^0 + \Delta Y_j$. Subsequently, the current position values and eq. (9) are utilized to compute a new adjustment. This iterative process continues until the correction becomes smaller than a predefined error tolerance (tol). The stopping criterion employed in this study compares the magnitude of the current position correction ($\Delta\vec{Y}$) to the magnitude of the initial position vector ($\vec{X}$), expressed as $\frac{\|\Delta\vec{Y}\|}{\|\vec{X}\|} < tol$.

4 Truss Positional Finite Element Method Program (TPFEM)

The developed application aims to enhance the teaching-learning process of the FEMP for truss structures, making the learning experience more dynamic, user-friendly, and engaging. The TPFEM allows users to model multiple truss structures simultaneously through an intuitive multi-window interface, where windows can be easily added or removed. Users can conveniently save model information in JSON file format.

The program's interface, as shown in Fig. 1, showcases all the application's functionalities, including managing window visualization limits, creating and deleting elements, and saving and loading models. Additionally, the program allows for the application of boundary conditions, such as constraints on the nodal displacements of the structure and the application of concentrated loads. The subsequent figures demonstrate the main functionalities and results achievable with TPFEM. For more detailed information regarding the parameters and dimensions of the represented structures, refer to Coda [14].

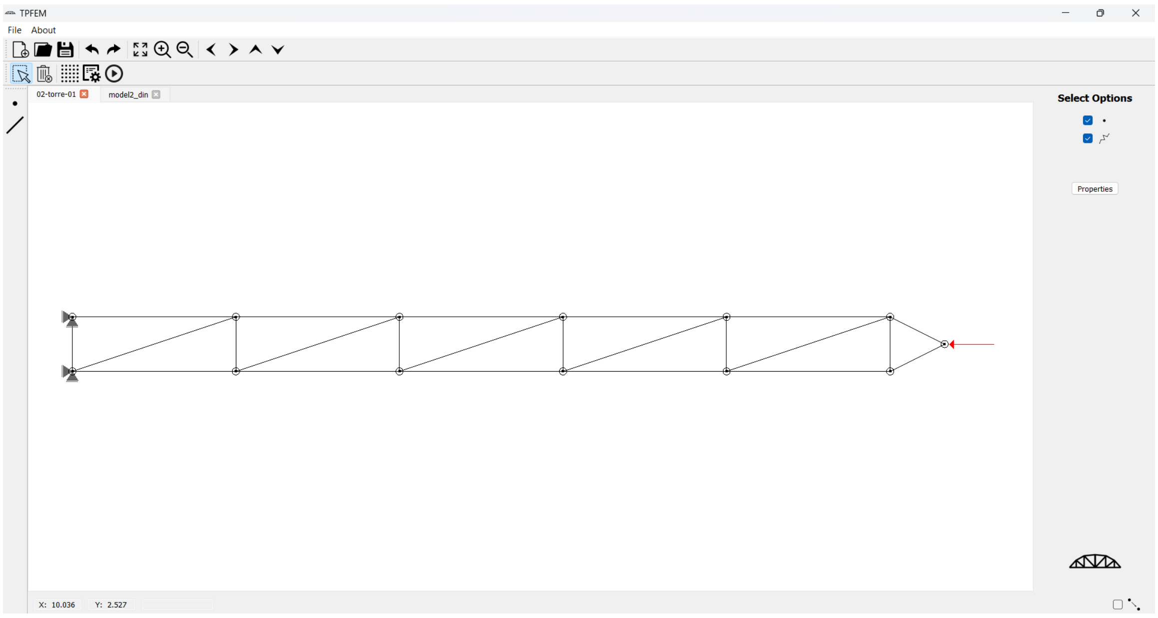

Figure 1. Initial interface

Figure 2 presents the static analysis of the structure shown in Fig. 1 using the FEMP. This figure features a color scale that correlates the normal stress values for each bar of the structure. Additionally, the response at each load step can be visualized, with the result for load step 94 provided as an example.

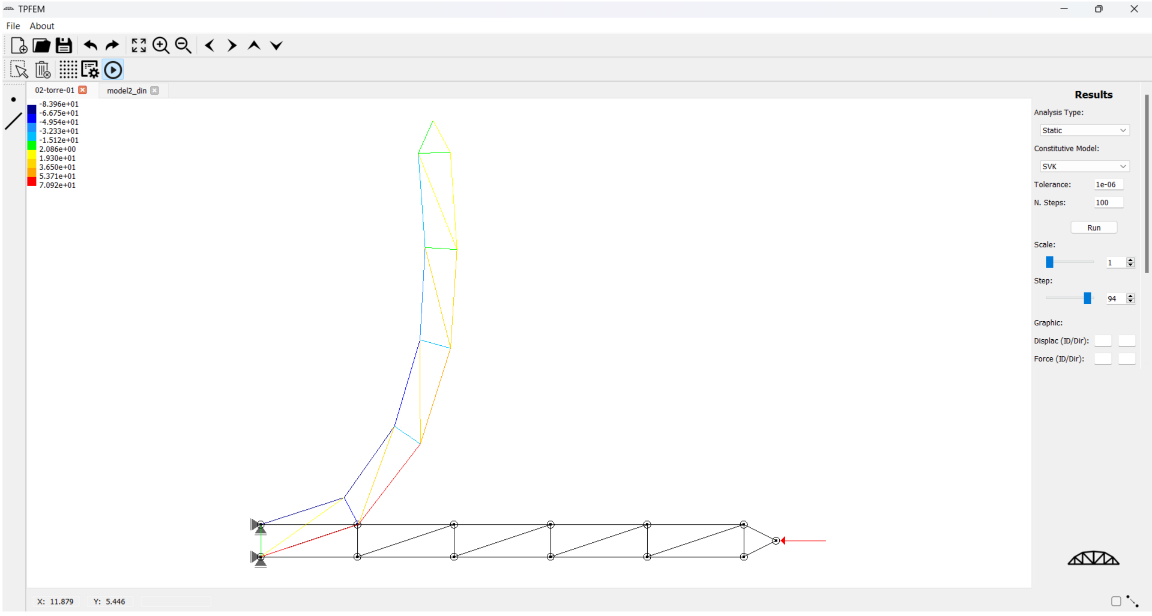

Figure 2. Static geometric nonlinear analysis of a truss structure

Figure 3 presents a dynamic analysis of free vibration (i.e., considering that the structure is not subjected to any loads) to obtain the natural frequencies of vibration and the time step Δt that will be used in the dynamic analysis of the truss structure.

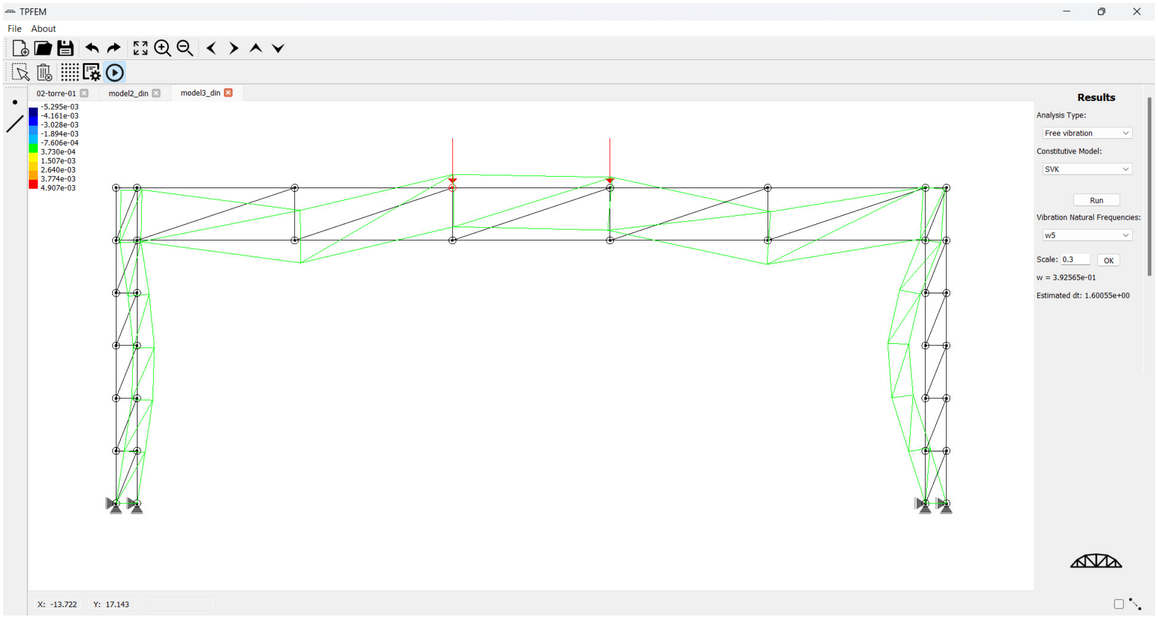

Figure 3. Geometric nonlinear analysis of free vibration of a truss structure

Figure 4 shows the dynamic analysis of the structure previously analyzed in Fig. 3, using a time step corresponding to the fifth natural vibration of $\Delta t = 1.6$ seconds and analyzing 1000 time steps as illustrated. The deformed configuration presented corresponds to time step 34.

Furthermore, in Fig. 4, positioned on the right side of the program interface, there is a ”Plot” button enabling users to select a specific node and displacement direction. This feature generates a graph illustrating displacement over time t during dynamic analysis. For static analyses, the program generates force versus displacement graphs based on user-defined nodes and directions.

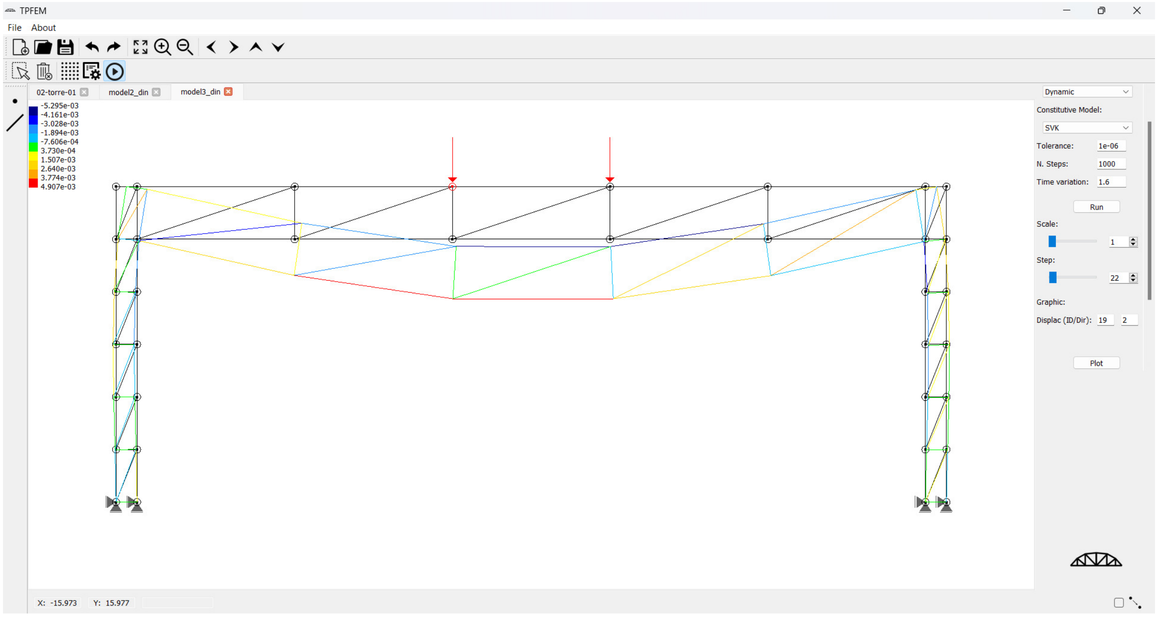

Figure 4. Dynamic geometric nonlinear analysis of a truss structure

Figure 5 presents the displacement graph in the vertical direction (direction 2) of node 19, highlighted in red in Fig. 4 following selection via mouse click, plotted against time t. This node corresponds to the location where the concentrated load, positioned farthest to the left, was applied.

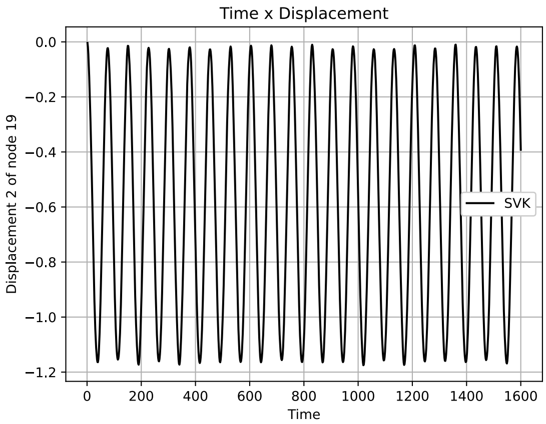

Figure 5. Structural response of the dynamic geometric nonlinear analysis of a truss structure

5 Conclusions

This study introduces the Truss Positional Finite Element Method Program (TPFEM), a software designed to enhance the teaching and learning of structural analysis using the Finite Element Method based on Positions (FEMP). By offering an intuitive interface and robust analytical capabilities, TPFEM not only facilitates the exploration of geometric nonlinearity in truss structures but also prepares students with practical skills essential for tackling real-world engineering challenges. This integration of advanced educational software underscores its pivotal role in bridging theoretical knowledge with hands-on application, thereby ensuring a more comprehensive educational experience aligned with industry needs and scientific advancements.

Acknowledgements. This work was supported by the Conselho Nacional de Desenvolvimento Científico e Tecnológico (CNPq) [grant numbers 140275/2022-3] and the Fundação Coordenação de Aperfeiçoamento de Pessoal de Nível Superior (CAPES).

Authorship statement. The authors hereby confirm that they are the sole liable persons responsible for the authorship of this work, and that all material that has been herein included as part of the present paper is either the property (and authorship) of the authors, or has the permission of the owners to be included here.

References

[1] R. C. Limited. Pyqt: Python bindings for the qt application framework. `https://www.riverbankcomputing.com/software/pyqt/`. Accessed: 2024-07-09, 2024.

[2] P. Developers. Pyopengl: The python opengl binding. `http://pyopengl.sourceforge.net/`. Accessed: 2024-07-09, 2024.

[3] D. S. Bomfim, R. L. Soares, L. F. Bez, P. C. Lopes, A. M. Pereira, and L. F. Martha. Hetool: A half-edge topological object-oriented library for generic 2-d geometric modeling. *SoftwareX*, vol. 21, pp. 101307, 2023.

[4] M. Mäntylä. *An introduction to solid modeling*. Computer Science Press, Inc., 1987.

[5] C. M. Hoffmann. *Geometric and solid modeling: an introduction*. Morgan Kaufmann Publishers Inc., 1989.

[6] D. S. Bomfim, R. L. Soares, L. F. Bez, P. C. Lopes, A. M. Pereira, and L. F. Martha. Development of a python application aiming at the teaching-learning process of the half-edge data structure. In *XLII Ibero-Latin American Congress on Computational Methods in Engineering*, volume 3, 2021.

[7] R. Soares, D. Bomfim, L. Bez, P. Lopes, A. Pereira, and L. Martha. A collaborative web computer-aided design application. In *XLII Ibero-Latin American Congress on Computational Methods in Engineering*, volume 3, 2021.

[8] J. C. L. Peixoto, D. S. Bomfim, R. L. Soares, L. F. Bez, P. C. Lopes, A. M. Pereira, R. L. Rangel, and L. F. Martha. Some modeling features for two-dimensional isoparametric and isogeometric finite element analysis. In *XLIII Ibero-Latin American Congress on Computational Methods in Engineering*, volume 4, 2022.

[9] J. C. L. Peixoto and R. L. Rangel. Isogeometric analysis with interactive modeling of multi-patches nurbs. In *XLIV Ibero-Latin American Congress on Computational Methods in Engineering*, volume 5, 2023.

[10] J. C. L. Peixoto. Análise isogeométrica com modelagem interativa de múltiplas regiões nurbs e t-splines. Master's thesis, PUC-Rio, 2024.

[11] D. S. Bomfim. Uma estratégia de modelagem aberta e extensível para a criação de modelos de subdivisões planares para mecânica computacional. Master's thesis, PUC-Rio, 2022.

[12] J. Bonet, R. Wood, J. Mahaney, and P. Heywood. Finite element analysis of air supported membrane structures. *Computer methods in applied mechanics and engineering*, vol. 190, n. 5-7, pp. 579–595, 2000.

[13] H. B. Coda and M. Greco. A simple fem formulation for large deflection 2d frame analysis based on position description. *Computer methods in applied mechanics and engineering*, vol. 193, n. 33-35, pp. 3541–3557, 2004.

[14] H. B. Coda. *O Método dos Elementos Finitos Posicional: sólidos e Estruturas - Não Lineariedade Geométrica e Dinâmica*. EESC–USP, 2018.

[15] R. R. Paccola and H. B. Coda. A direct fem approach for particulate reinforced elastic solids. *Composite Structures*, vol. 141, pp. 282–291, 2016.

[16] É. S. Ramos, R. Carrazedo, and R. R. Paccola. Modeling particles elements in damaged reinforced concrete structures. *Latin American Journal of Solids and Structures*, vol. 18, pp. e345, 2021.

[17] R. R. Paccola, M. d. S. M. Sampaio, and H. B. Coda. Fiber-matrix contact stress analysis for elastic 2d composite solids. *Latin American Journal of Solids and Structures*, vol. 12, pp. 583–611, 2015.

[18] M. S. Sampaio, R. R. Paccola, and H. B. Coda. Fully adherent fiber–matrix fem formulation for geometrically nonlinear 2d solid analysis. *Finite Elements in Analysis and Design*, vol. 66, pp. 12–25, 2013.

[19] R. De Borst, M. A. Crisfield, J. J. Remmers, and C. V. Verhoosel. *Nonlinear finite element analysis of solids and structures*. John Wiley & Sons, 2012.