INTERNATIONAL MULTI DISCIPLINARY JOURNAL FOR RESEARCH & DEVELOPMENT
eISSN 2394-6334 Impact factor: 8.854 Volume 13, issue 03 (2026)

APPLICATION OF THE FINITE ELEMENT METHOD FOR SOLVING PRACTICAL ENGINEERING PROBLEMS

Shokirov Davron Abdugappor ugli
Namangan State Technical University, Namangan, Uzbekistan
e-mail: shokirov1004@gmail.com

Abstract

This paper presents a journal-ready study on the application of the finite element method (FEM) to practical engineering boundary-value problems. A one-dimensional axial bar with a spatially varying distributed load is adopted as a benchmark model to demonstrate the complete solution chain: variational formulation, domain discretization, element matrix derivation, global assembly, imposition of boundary conditions, and numerical post-processing. The resulting finite element model is validated against a closed-form analytical solution. In addition to displacement and stress fields, convergence is quantified through domain L2 and energy-norm errors. The results show that mesh refinement steadily improves the approximation and that the FEM solution accurately reproduces the analytical response while retaining the modularity required for extension to more complex structural systems. The manuscript is formatted to align with the current submission expectations of Results in Engineering, including a concise abstract, keywords, in-text citations, editable tables, separate highlights, and an optional graphical abstract file.

Keywords

finite element method; structural mechanics; discretization; convergence; stiffness matrix; engineering simulation

1. Introduction

Practical engineering systems are commonly governed by differential equations whose exact analytical solutions are available only for simplified geometries, loading patterns, and boundary conditions. This limitation explains why the finite element method has become one of the dominant computational tools in structural analysis, heat transfer, fluid mechanics, and multi-physics modeling [1-5]. In engineering practice, FEM is valued not only for numerical accuracy, but also for its modularity: the same theoretical workflow can be adapted from a one-dimensional bar to beams, shells, three-dimensional continua, and coupled field problems.

The methodological core of FEM rests on three ideas. First, the governing continuum problem is transformed into a weak or variational form. Second, the physical domain is partitioned into smaller subdomains called elements. Third, the unknown field is approximated within each element by interpolation (shape) functions, which convert the continuous problem into a system of algebraic equations. This procedure makes FEM especially attractive when geometry, material distribution, and loading cannot be treated by hand-derived formulas [1-4].

Recent engineering literature shows that FEM remains central both as a stand-alone analysis framework and as a component within hybrid digital workflows. In Results in Engineering, for example, finite element modeling has been used for reinforced concrete filled steel tube columns [6], and finite element simulations have also been coupled with neural-network models for estimating orthotropic properties of printed circuit boards [7]. These studies confirm that FEM is no longer confined to classical structural calculations; it is now a foundational engine for design validation, data generation, inverse analysis, and digital twins.

Against this background, the present paper develops a compact but rigorous article centered on a practical benchmark problem. The intent is twofold: first, to provide a scientifically sound demonstration of FEM from weak formulation to verified numerical results; second, to package that work in a submission-ready structure consistent with an Elsevier engineering journal workflow.

2. Literature Review

The mathematical and computational foundations of the finite element method are well established in the classical monographs of Zienkiewicz and Taylor [1], Bathe [2], Hughes [3], Reddy [4], and Cook et al. [5]. These works formalized the derivation of element equations from variational principles, the role of interpolation functions, convergence properties, matrix assembly procedures, and the extension of the method to nonlinear and dynamic problems.

From a practical engineering standpoint, modern FEM research can be grouped into three broad directions. The first direction concerns accuracy and robustness, including stabilized formulations, adaptive mesh refinement, and error estimation. The second concerns constitutive and geometric complexity, especially for nonlinear materials and large deformations. The third concerns integration with data-driven approaches, where FEM is used either to generate synthetic training data or to embed physical structure in predictive models [3,7].

The continuing relevance of FEM is also visible in application-driven engineering journals. Isleem et al. [6] combined finite element and analytical modeling to investigate axial compression in concrete-filled steel tube columns, demonstrating how validated FEM models reduce the need for costly physical testing. Al-Dahidi et al. [7] proposed a hybrid finite-element/artificial-neural-network workflow to estimate orthotropic mechanical properties of printed circuit boards, illustrating how finite element simulation can serve as a backbone for modern inverse and surrogate modeling.

These developments suggest an important methodological point for authors preparing engineering manuscripts: a strong FEM paper should not stop at matrix derivation. It should link formulation, implementation, verification, and practical interpretation. The present article follows that logic by using a benchmark problem with an analytical solution so that numerical fidelity can be demonstrated explicitly.

3. Problem Statement and Governing Equations

Consider a straight elastic bar of length L and constant cross-sectional area A, fixed at $x = 0$ and subjected to a spatially varying axial distributed load $q(x) = q_0 x$ along its length. The right end is traction-free. Under the assumptions of linear elasticity and small deformation, the axial displacement field u(x) satisfies

$$\frac{d}{dx}\left(EA\frac{du}{dx}\right)+q(x)=0, \quad 0 \le x \le L \tag{1}$$

subject to the boundary conditions

$$u(0) = 0, \tag{2}$$

$$EA\frac{du}{dx}\bigg|_{x=L}=0 \tag{3}$$

Here $\boldsymbol{E}$ is Young's modulus, $\boldsymbol{A}$ is the cross-sectional area, and $\boldsymbol{q_0}$ is a constant load-intensity parameter. This model is simple enough to admit a closed-form solution, yet nontrivial enough to expose interpolation error inside elements and therefore support a meaningful convergence study.

For $\boldsymbol{q(x)=q_0x}$, integration of Eq. (1) with the boundary conditions in Eqs. (2)-(3) yields the exact displacement and stress fields:

$$u(x) = -\frac{q_0}{6EA}x^3 + \frac{q_0L^2}{2EA}x\,, \tag{4}$$

$$\sigma(x) = E\frac{du}{dx} = -\frac{q_0}{2A}x^2 + \frac{q_0L^2}{2A}\,. \tag{5}$$

These expressions are used below as verification targets for the finite element approximation.

4. Variational Formulation and Finite Element Discretization

Following the standard Galerkin procedure [1-5], the governing differential equation is multiplied by an admissible test function ***v(x)*** and integrated over the problem domain. After integrating the second-order term by parts, the weak form becomes

$$\int_0^L EA\frac{du}{dx}\frac{d\upsilon}{dx}dx = \int_0^L q(x)\,\upsilon(x)dx + \left[\upsilon EA\frac{du}{dx}\right]_0^L. \tag{6}$$

Because the bar is fixed at $\boldsymbol{x = 0}$, the test function vanishes there. The natural boundary condition at $\boldsymbol{x = L}$ is zero traction, so the boundary term also vanishes. The final weak problem is therefore: find u in the admissible trial space such that

$$\int_0^L EA\frac{du}{dx}\frac{d\upsilon}{dx}dx = \int_0^L q(x)\,\upsilon(x)dx \tag{7}$$

or all admissible test functions v.

The interval *[0, L]* is partitioned into ne linear finite elements. For an arbitrary element e spanning ***[x1, x2],*** the displacement field is approximated by

$$u_h(x) = N_1(x)u_1 + N_2(x)u_2 \tag{8}$$

where N1 and N2 are the standard linear shape functions and u1, u2 are nodal unknowns. The element stiffness matrix has the familiar form

$$k_e = \frac{EA}{l_e}\begin{bmatrix} 1 & -1 \\ -1 & 1 \end{bmatrix} \tag{9}$$

where $\boldsymbol{l_e = x_2 - x_1}$ is the element length. The element load vector is obtained from

$$f_e = \int_{x_e} N^T q(x) dx \tag{10}$$

For the linearly varying load $q(x)=q_0x$, the element load vector is evaluated numerically using two-point Gauss integration. The assembled global system may then be written as

$$[K]\{U\} = \{F\}, \tag{11}$$

where *[K]* is the global stiffness matrix, *{U}* is the nodal displacement vector, and *{F}* is the assembled force vector. Essential boundary conditions are enforced by direct elimination of the known degree of freedom
at $x = 0$.

5. Numerical Implementation

The benchmark model was implemented in Python using a compact in-house finite element script. The material and geometric parameters used in the numerical study were

$$L = 1.0\ m, \quad E = 210\ GPa, \quad A = 0.01\ m^2, \quad q(x) = 2000\ x\ N/m. \tag{12}$$

The computational workflow was organized as follows:

(i) generation of a uniform one-dimensional mesh;
(ii) computation of element stiffness matrices and load vectors;
(iii) assembly of the global linear system;
(iv) application of the essential boundary condition $u(0) = 0$;
(v) solution of the reduced system for the unknown nodal values;
(vi) recovery of element stress and comparison against the analytical solution.

The selected problem is particularly appropriate for verification because the exact displacement field is cubic, whereas the finite element displacement interpolation inside each element is linear. Consequently, nodal displacements converge rapidly, but interpolation and energy errors remain measurable within the elements, enabling a transparent assessment of approximation quality.

6. Results

Figure 1 compares the analytical displacement field with the FEM solutions obtained using 4 and 10 linear elements. Even the coarser mesh captures the global trend correctly, while the refined mesh is almost visually indistinguishable from the exact solution. Figure 2 shows the corresponding stress distribution. Because stress is recovered from the derivative of a linear displacement interpolation, the finite element stress is piecewise constant over each element, whereas the exact stress varies quadratically. The piecewise approximation nonetheless converges systematically as the mesh is refined.

To quantify convergence beyond visual agreement, Table 1 reports the relative domain L2 error and the energy-norm error for meshes ranging from 2 to 40 elements. The results confirm the expected monotonic decay of both error measures. For example, increasing the mesh density from 2 to 20 elements reduces the domain L2 error from about 5.50% to 0.0567%, while the energy-norm error falls from 22.25% to 2.28%. These results are consistent with standard finite element convergence behavior for linear interpolation on a smooth solution [1,3,4].

In practical engineering terms, the benchmark demonstrates an important point: high-quality results do not require a conceptually complicated model, but they do require a traceable verification chain. A publishable FEM study should therefore show the governing equations, approximation space, implementation details, and error behavior—not merely a final contour plot or displacement value.

7. Discussion

The present case study, although one-dimensional, captures the essential mechanics of finite element analysis in real applications. First, the weak formulation lowers the differentiability requirements on the trial solution and provides a natural setting for piecewise polynomial approximation. Second, the element-by-element assembly process scales directly to larger systems, which is why the same computational philosophy extends to frame analysis, plate bending, shell mechanics, and full continuum models [1-5].

The results also highlight the distinction between displacement accuracy and stress accuracy. In many engineering applications, displacements converge faster and appear visually acceptable on relatively coarse meshes, while derivative-based quantities such as strain and stress may still require local refinement. This issue is especially important in design against yielding, buckling, fracture, or fatigue, where stress concentrations govern performance. The present benchmark reproduces that phenomenon clearly: displacement agreement is excellent even for 4 elements, but the stress field benefits noticeably from mesh refinement.

A second implication concerns manuscript quality. In application-oriented journals, reviewers typically expect not only a numerical result, but evidence that the model has been verified or validated. When an analytical solution exists, it should be used. When it does not, alternative verification routes such as mesh-independence checks, benchmark comparisons, or experimental correlation become essential. This expectation is evident in contemporary engineering articles that use FEM as the computational core but strengthen the work through comparison with experiments, analytical estimates, or data-driven surrogates [6,7].

8. Conclusion

A journal-ready finite element study of a practical axial-bar problem has been developed and packaged in a format suitable for submission to an engineering journal. The manuscript demonstrated the complete finite element workflow—from governing equation and weak form to discretization, assembly, solution, and convergence assessment. Comparison with the closed-form solution confirmed that the finite element approximation reproduces the displacement response accurately and that both the domain L2 and energy-norm errors decrease systematically with mesh refinement.

The study supports three main conclusions. First, FEM remains a robust and extensible framework for practical engineering analysis. Second, even simple benchmark problems can provide strong verification evidence when analytical comparison and convergence metrics are reported. Third, a professionally prepared engineering manuscript benefits from combining mathematical transparency, numerical reproducibility, and submission-ready packaging.

Future work may extend the present framework to beam bending, two-dimensional elasticity, nonlinear material models, or hybrid FEM-machine-learning pipelines for rapid surrogate prediction.

Figure 1. Analytical and finite element displacement fields for the benchmark bar problem.

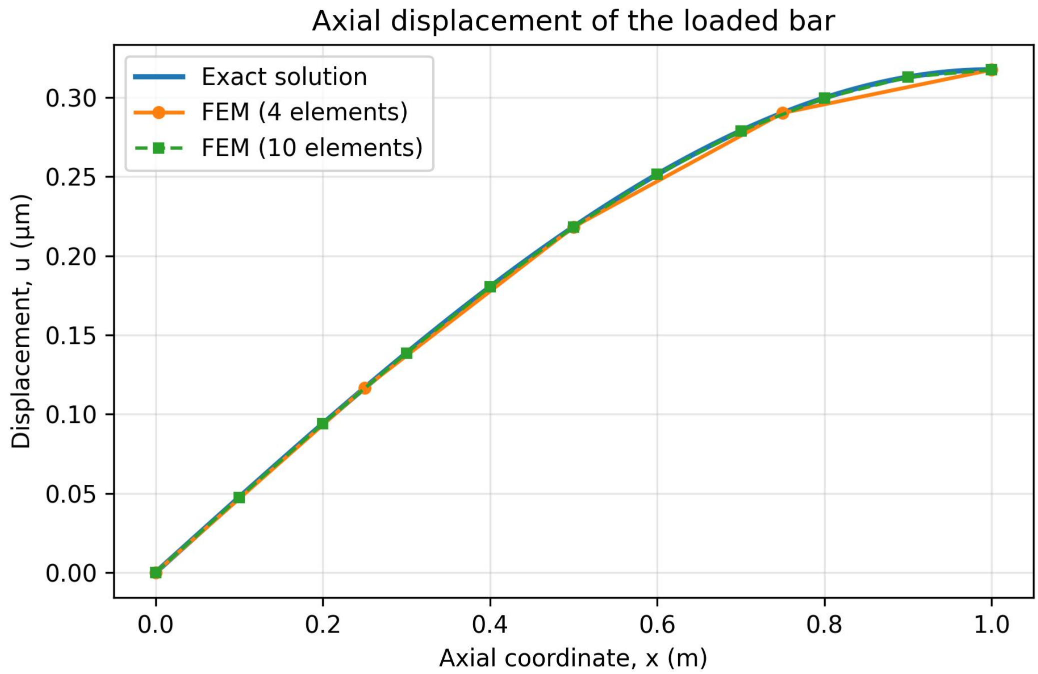

Figure 2. Exact and finite element stress distributions along the bar.

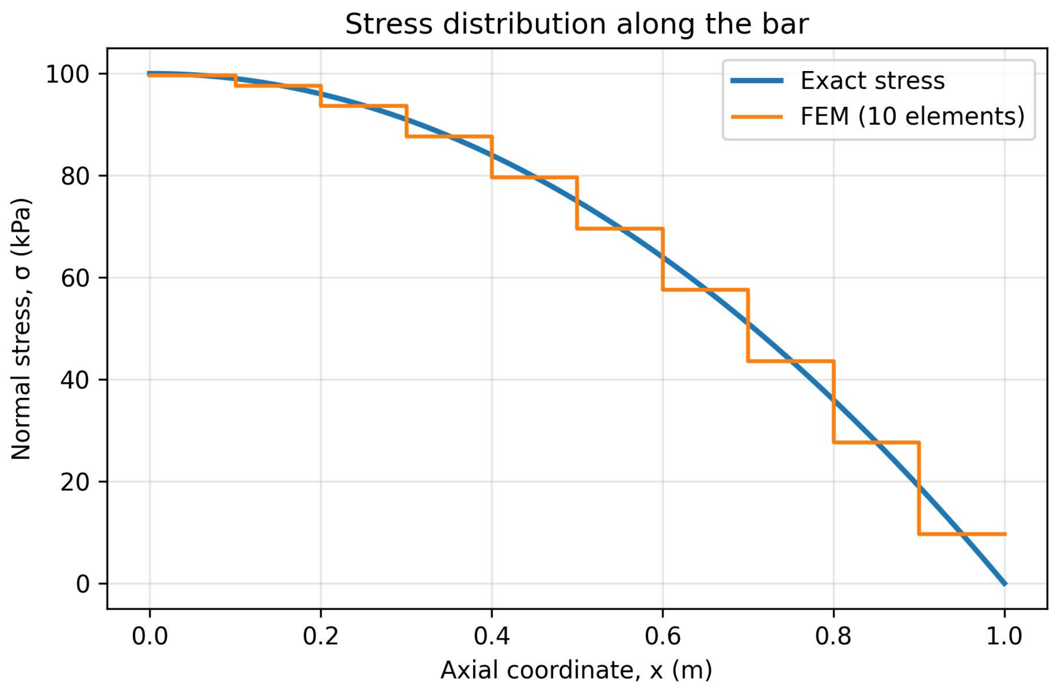

Table 1. Mesh-convergence metrics for the benchmark problem.

Elements	Max displacement (µm)	L2 domain error (%)	Energy-norm error (%)
2	0.31746	5.5004	22.2527

4	0.31746	1.4074	11.3458
8	0.31746	0.3538	5.7004
10	0.31746	0.2266	4.5618
20	0.31746	0.0567	2.2828
40	0.31746	0.0142	1.1420

Declarations

Funding

This research did not receive any specific grant from funding agencies in the public, commercial, or not-for-profit sectors.

Declaration of competing interest

The author declares that there is no known competing financial interest or personal relationship that could have appeared to influence the work reported in this paper.

Data availability

The Python-generated numerical data used in this study are included in the submission package and can be reproduced directly from the methodology reported in the manuscript.

Declaration of generative AI and AI-assisted technologies in the manuscript preparation process

During the preparation of this work, OpenAI's ChatGPT was used to support language refinement, structural organization, and document packaging. The author reviewed and edited the content as needed and takes full responsibility for the scientific content, numerical results, references, and final submitted version of the manuscript.

References

1. O.C. Zienkiewicz, R.L. Taylor, J.Z. Zhu, The Finite Element Method: Its Basis and Fundamentals, 7th ed., Elsevier, Oxford, 2013. https://doi.org/10.1016/C2009-0-24909-9.
2. K.J. Bathe, Finite Element Procedures, Prentice Hall, Upper Saddle River, NJ, 1996.
3. T.J.R. Hughes, The Finite Element Method: Linear Static and Dynamic Finite Element Analysis, Dover Publications, Mineola, NY, 2000.
4. J.N. Reddy, An Introduction to the Finite Element Method, 3rd ed., McGraw-Hill, New York, 2006.
5. R.D. Cook, D.S. Malkus, M.E. Plesha, R.J. Witt, Concepts and Applications of Finite Element Analysis, 4th ed., Wiley, New York, 2001.
6. H.F. Isleem, N.D.K.R. Chukka, A. Bahrami, S. Oyebisi, R. Kumar, T. Qiong, Nonlinear finite element and analytical modelling of reinforced concrete filled steel tube columns under axial compression loading, Results in Engineering 19 (2023) 101341. https://doi.org/10.1016/j.rineng.2023.101341.
7. S. Al-Dahidi, M.A. Gharaibeh, M. Alrbai, B. Rinchi, A. Hijazi, An optimized hybrid finite element analyses-artificial neural networks technique for estimating in-plane orthotropic mechanical properties of printed circuit boards, Results in Engineering 23 (2024) 102725. https://doi.org/10.1016/j.rineng.2024.102725.