

# ANALYSIS OF INTERNAL FORCES AND DISPLACEMENTS OF PLANE FRAMES USING MATLAB: COMPARISON WITH SAP2000 AND THE FINITE ELEMENT METHOD

TỪ HỒNG NHUNG<sup>1</sup> - NGUYỄN THÀNH CÔNG<sup>1</sup>

<sup>1</sup>College of Engineering and Technology, Tra Vinh University

Trường Kỹ thuật và Công nghệ, Đại học Trà Vinh

Received date: July 21, 2026

Revised date: August 10, 2026

Accepted date for publication: August 25, 2026

---

## Abstract:

*This study examines the internal forces and displacements of plane frames by comparing analytical calculations with finite element analyses performed in SAP2000 and MATLAB. Two representative structures are considered: a one-story, one-bay frame and a two-story, two-bay frame. The results of the three approaches are compared in terms of accuracy, computational practicality, strengths, and limitations. The findings demonstrate the applicability of the finite element method to plane-frame analysis and provide a basis for selecting a suitable approach to calculating and verifying internal forces and displacements in structural engineering.*

**Keywords:** *finite element method, plane frame, internal forces, displacement, SAP2000, MATLAB*

---

## Phân tích nội lực và chuyển vị của khung phẳng bằng MATLAB: So sánh với SAP2000 và phương pháp phần tử hữu hạn

### Tóm tắt:

*Nghiên cứu này khảo sát nội lực và chuyển vị của kết cấu khung phẳng thông qua việc so sánh kết quả tính toán giải tích với kết quả phân tích bằng phương pháp phần tử hữu hạn trên SAP2000 và MATLAB. Hai mô hình kết cấu điển hình được xem xét gồm khung một tầng, một nhịp và khung hai tầng, hai nhịp. Kết quả từ ba phương pháp được so sánh về độ chính xác, tính thuận tiện trong tính toán, ưu điểm và hạn chế. Những phát hiện của nghiên cứu cho thấy khả năng ứng dụng phương pháp phần tử hữu hạn trong phân tích khung phẳng, đồng thời cung cấp cơ sở để lựa chọn phương pháp phù hợp khi tính toán và kiểm tra nội lực, chuyển vị trong kỹ thuật kết cấu.*

**Từ khóa:** *phương pháp phần tử hữu hạn, khung phẳng, nội lực, chuyển vị, SAP2000, MATLAB*

## 1. Introduction

Load-bearing frame structures are among the most widely used structural systems in construction, particularly in civil and industrial buildings. Accurate determination of internal forces and displacements is essential for structural design, verification, and assessment of load-bearing capacity. For plane frames subjected to horizontal and vertical loads as well as concentrated moments, the distribution of internal forces and structural deformations should be comprehensively analyzed to ensure adequate strength, stiffness, and overall structural stability.

With recent advances in information technology, computer-based approaches have become increasingly practical and accessible for solving engineering problems. To effectively perform computational structural analysis, the behavior of real structural systems must be appropriately modeled, computational algorithms must be formulated, and the resulting mathematical models must be transformed into systems of equations that can be solved efficiently and reliably using computers. Various analytical approaches are available for frame structures, including the force method, displacement method, and FEM (Nguyen & Nguyen, 2006).

The FEM is one of the most widely used numerical approaches in structural analysis. In FEM, a structural system is discretized into a finite number of simple elements, from which element stiffness matrices, load vectors, and global equilibrium equations are formulated. The resulting system of equations is then solved to determine the nodal displacements, which are subsequently used to evaluate the internal forces in each element. FEM has been extensively applied to a wide range of engineering problems. Nevertheless, the development and investigation of alternative simulation approaches that are reliable, straightforward, and well suited to specific classes of problems remain necessary. Owing to its versatility, applicability to various structural systems, and high accuracy when appropriately modeled, FEM has become an essential computational tool for the research, design, and analysis of engineering structures.

In addition to analytical methods, specialized structural analysis software such as SAP2000 and computational programming platforms such as MATLAB are widely employed for the analysis of frame structures. SAP2000 provides several advantages, including intuitive graphical modeling, efficient computation, and practical applicability to structural engineering design. In contrast, MATLAB is a widely used and versatile computational platform for implementing finite element formulations and developing customized numerical algorithms for structural analysis (Daryl, 2007; Kattan, 2010). The MATLAB enables researchers to independently develop computational algorithms and systematically control the procedures for assembling stiffness matrices, imposing boundary conditions, constructing load vectors, and solving the resulting equilibrium equations. Therefore, a comparative assessment of the results obtained from analytical solutions, MATLAB-based FEM computations, and SAP2000 analyses is essential for evaluating the reliability, advantages, limitations, and applicability of each approach.

In this study, the internal forces and displacements of plane frame structures are investigated using the FEM through two representative models: a one-story, one-bay frame and a two-story, two-bay frame. The results obtained from different computational approaches are compared and evaluated in terms of discrepancies, accuracy, and applicability. Based on the comparative results, an appropriate computational approach is proposed to support the design, verification, and structural analysis of plane frames in civil engineering applications.

## 2. Finite element methods for plane frame analysis

### 2.1. Analytical method

In the FEM, the formulation and solution of a plane frame problem generally involve nine steps (Chung, 2026; Thinh & Cuong, 2011):

Step 1: Reading the input data

Step 2: Discretizing the structure

Step 3: Establishing the differential equations in the strong form

Step 4: Establishing the weak form equations, also known as the energy equations

Step 5: Constructing the shape functions

Step 6: Calculating the stiffness matrix and load vector

The equilibrium equation of the entire structural system is then established as follows:  $[K] \times \{U\} = \{F\}$  where:  $\{U\}$  is the global nodal displacement vector of the entire structural system in the global coordinate system;  $[K]$  is the global stiffness matrix of the entire structural system in the global coordinate system;  $\{F\}$  is the global nodal load vector of the entire structural system in the global coordinate system.

Step 7: Assembling the element equations and applying boundary conditions

After the boundary conditions and compatibility conditions are imposed, the modified equilibrium equation can be expressed as follows:  $[k] \times \{u\} = \{f\}$

where:  $\{u\}$  is obtained from  $\{U\} \setminus \{U\} \setminus \{U\}$  by eliminating the displacement components equal to zero;  $[k]$  is obtained from  $[K]$  by removing the rows and columns corresponding to the displacement components equal to zero;  $\{f\}$  is obtained from  $\{F\}$  by removing the rows corresponding to the displacement components equal to zero.

Step 8: Solving the global system of equations  $\{u\} = [k]^{-1} \times \{f\}$

Step 9: Result analysis

Determination of internal forces: Based on the obtained results, together with the prescribed boundary conditions, the nodal displacement vector of each element in the local coordinate system can be determined. Consequently, the internal forces, as well as the strains and stresses at any point within the element, can be evaluated using the established relationships in Structural Mechanics and the Theory of Elasticity.

## 2.2. Application of SAP2000 software

SAP2000 is a useful software package for structural design and strength analysis. At present, SAP2000 is a comprehensive software suite widely used to simulate the behavior of engineering structures. In other words, SAP2000 is a commercial software package that employs the FEM to calculate internal forces, stresses, strains, displacements, vibrations, and other structural responses under the effects of external and internal actions, including both static and dynamic loads.

## 2.3. Using MATLAB software

MATLAB is an integrated computational platform incorporating graphical interfaces and specialized toolboxes. Through numerical programming, it enables the modeling and analysis of structural behavior, including stress, strain, displacement, and structural responses under static and dynamic loading. Owing to its powerful computational capabilities and flexibility, MATLAB has become a widely used tool in scientific research, engineering analysis, and structural engineering applications.

The procedure for determining the internal forces and displacements of frame structures using FEM implemented in MATLAB consists of the following steps (Trung & Hung, 2015):

Step 1: Discretize the structural domain into finite elements. Define the element and node numbering, and assign the corresponding degrees of freedom associated with the nodal displacements.

Step 2: Establish the global equilibrium equations of the structural system.

$$[K] \times \{u\} = \{F\}$$

Step 3: Determine and impose the boundary and compatibility conditions. The modified equilibrium equations can then be expressed as follows:

$$[k] \times \{u\} = \{f\}$$

Step 4: Solve the resulting system of equilibrium equations to determine the nodal displacements.

$$\{u\} = [k]^{-1} \times \{f\}$$

Step 5: Determine the internal forces in each structural element.

### 3. Results and discussion

#### 3.1. Finite element analysis of a one-story, one-bay plane frame

##### a. Analytical Method

Problem 1: The structural system considered in this study is a one-story, one-bay plane frame, as illustrated in Figure. 1. The objective is to:

Determine the internal forces and displacements of the plane frame shown in Fig.1. The elastic modulus is  $E = 200 \times 10^6 \text{ kN/m}^2$ , and the cross-sectional area is  $A = 10 \text{ cm}^2$  for all elements. The moment of inertia is  $I = 80 \text{ cm}^4$  for Elements (1) and (3), and  $0.5I = 40 \text{ cm}^4$  for Element (2).

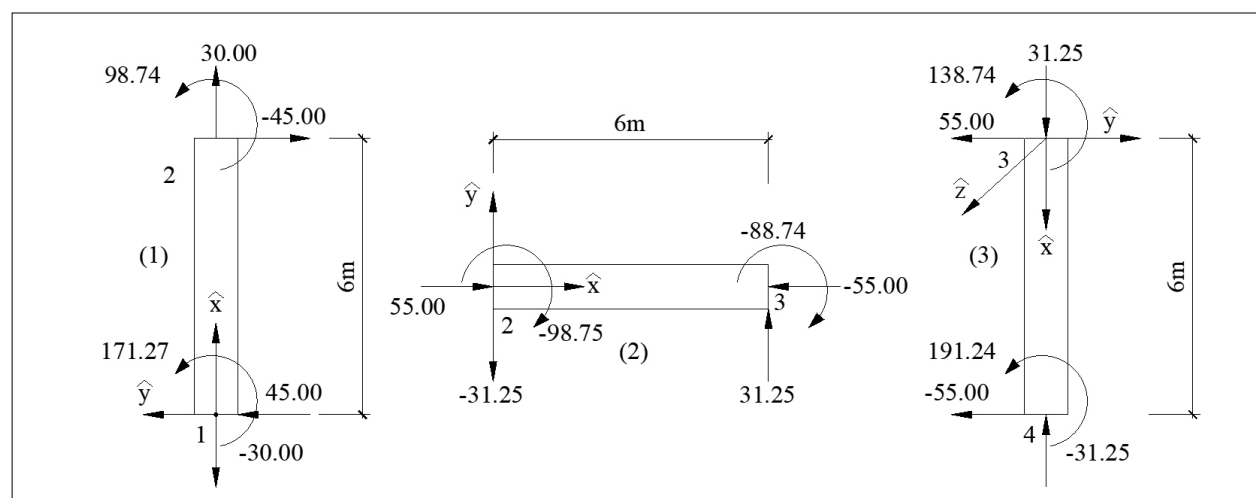
Solution: Applying the FEM yields the following results:

Table 1. FEM analysis results of the one-story, one-bay plane frame

| Element | Node | Displacement (m) và Rotation (rad) |            |              | Internal Force (kN, m) |        |        |
|---------|------|------------------------------------|------------|--------------|------------------------|--------|--------|
|         |      | $d_{i,jx}$                         | $d_{i,jy}$ | $\phi_{i,j}$ | $N_x$                  | $Q_y$  | $M_z$  |
| (1)     | 1    | 0                                  | 0          | 0            | -30                    | 45     | 171.27 |
|         | 2    | 9.14                               | 0.00       | -1.36        | 30                     | -45    | 98.74  |
| (2)     | 2    | 9.14                               | 0.00       | -1.36        | 55                     | -31.25 | -98.75 |
|         | 3    | 9.14                               | -0.00      | -0.98        | -55                    | 31.25  | -88.74 |
| (3)     | 3    | 9.14                               | -0.00      | -0.98        | 31.25                  | 55     | 138.74 |
|         | 4    | 0                                  | 0          | 0            | -31.25                 | -55    | 191.24 |

Source: Author's calculation

Figure.2. Internal Force Diagrams Obtained Using the FEM



*b. MATLAB-Based Method*

The MATLAB-based analysis yields the following results:

**Table 2. The MATLAB analysis results of the one-story, one-bay plane frame**

| Element | Node | Displacement (m) và Rotation (rad) |            |              | Internal Force (kN, m) |        |        |
|---------|------|------------------------------------|------------|--------------|------------------------|--------|--------|
|         |      | $d_{i,jx}$                         | $d_{i,jy}$ | $\phi_{i,j}$ | $N_x$                  | $Q_y$  | $M_z$  |
| (1)     | 1    | 0                                  | 0          | 0            | -31.25                 | 45.00  | 171.27 |
|         | 2    | 9.14                               | 9.37e-04   | -1.36        | 31.25                  | -45.00 | 98.75  |
| (2)     | 2    | 9.14                               | 9.37e-04   | -1.36        | 55.00                  | -31.25 | -98.75 |
|         | 3    | 9.14                               | -9.37e-04  | -0.98        | -55.00                 | 31.25  | -88.74 |
| (3)     | 3    | 9.14                               | -9.37e-04  | -0.98        | 31.25                  | 55.00  | 138.74 |
|         | 4    | 0                                  | 0          | 0            | -31.25                 | -55.00 | 191.24 |

Source: Author's calculation

**Figure.3. Internal force diagrams of the one-story, one-bay plane frame obtained using MATLAB**

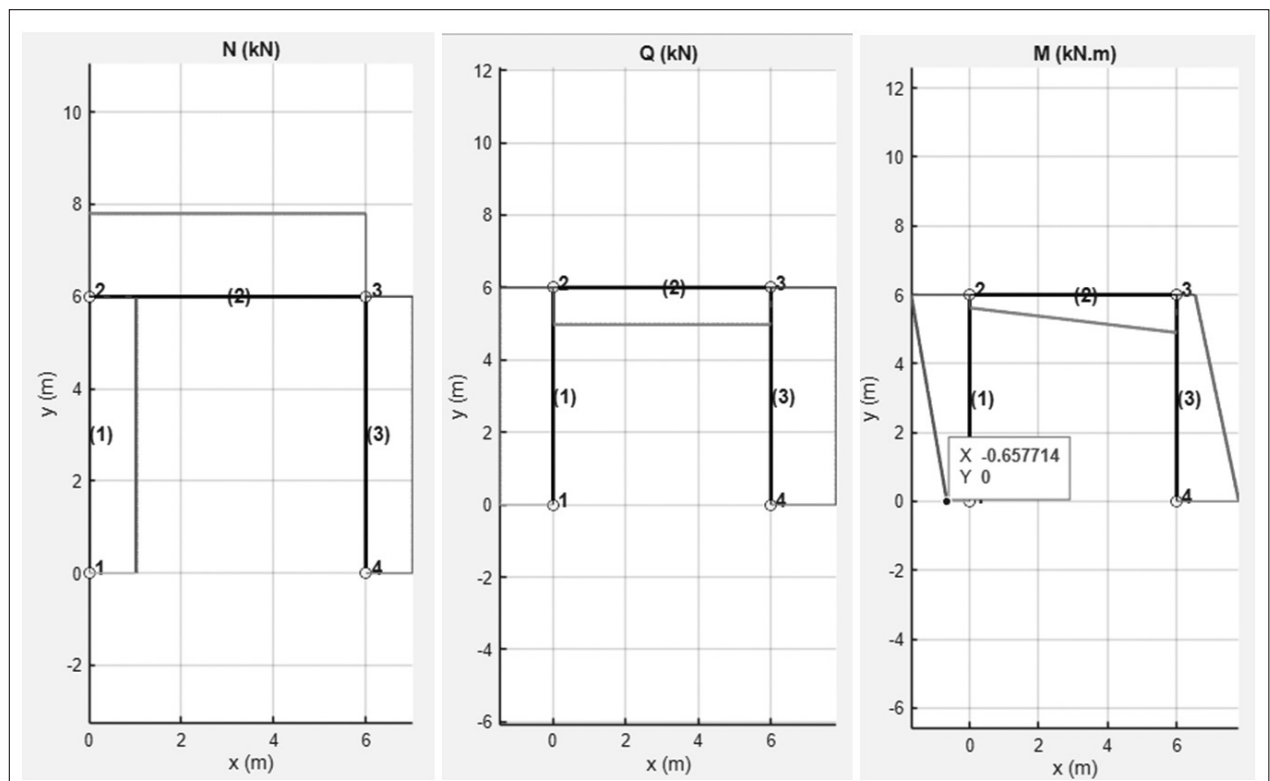
*c. SAP2000-Based Method*

**Table 3. The SAP2000 Analysis Results of the One-Story, One-Bay Plane Frame**

| Element | Nodes | Displacement (m) và Rotation (rad) |            |              | Internal Force (kN, m) |        |        |
|---------|-------|------------------------------------|------------|--------------|------------------------|--------|--------|
|         |       | $d_{i,jx}$                         | $d_{i,jy}$ | $\phi_{i,j}$ | $N_x$                  | $Q_y$  | $M_z$  |
| (1)     | 1     | 0                                  | 0          | 0            | -31.25                 | 45.00  | 171.29 |
|         | 2     | 9.14                               | 9.37e-04   | -1.36        | 31.24                  | -45.00 | 98.74  |

|     |   |      |           |       |        |        |        |
|-----|---|------|-----------|-------|--------|--------|--------|
| (2) | 2 | 9.14 | 9.37e-04  | -1.36 | 55.00  | -31.24 | -98.74 |
|     | 3 | 9.14 | -9.37e-04 | -0.98 | -55.00 | 31.24  | -88.73 |
| (3) | 3 | 9.14 | -9.37e-04 | -0.98 | 31.24  | 55.00  | 138.73 |
|     | 4 | 0    | 0         | 0     | -31.24 | -55.00 | 191.25 |

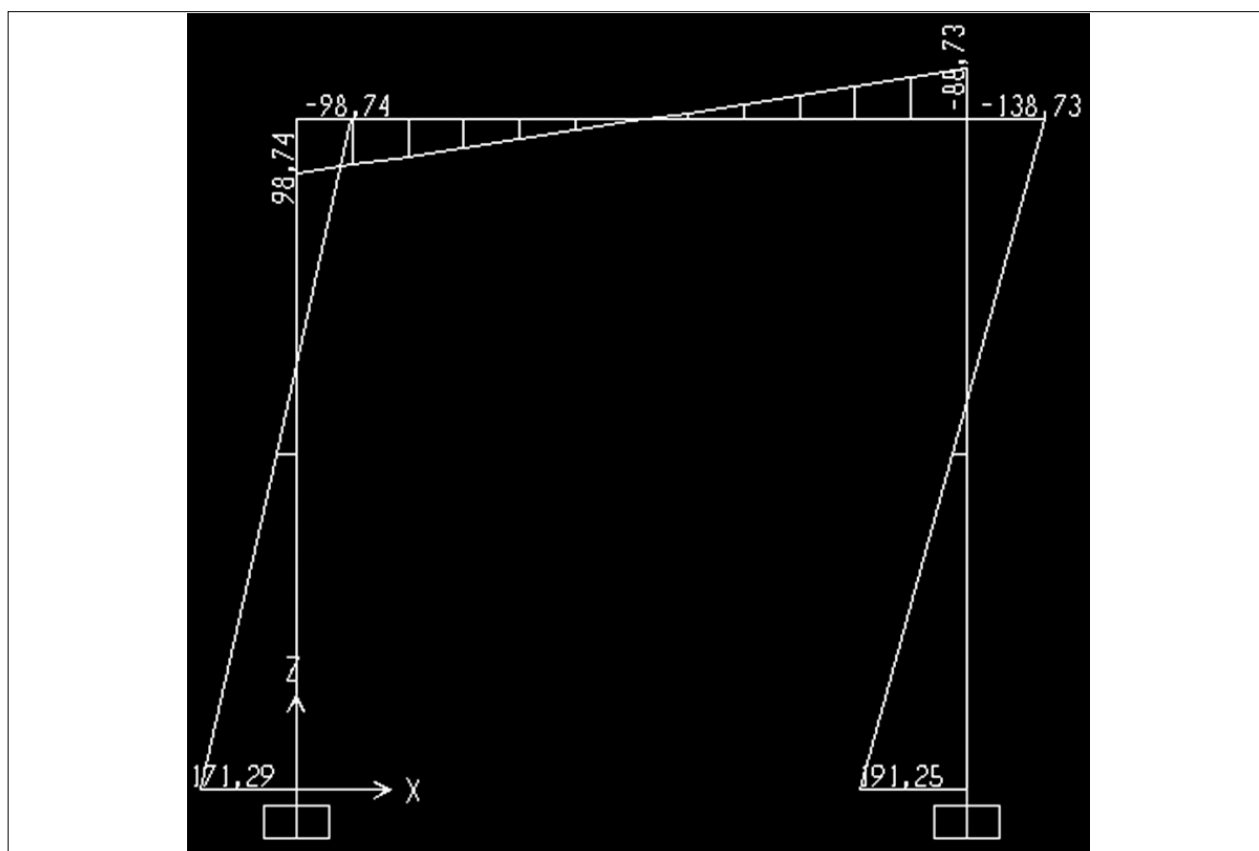
### 3.2. Discussion of Results

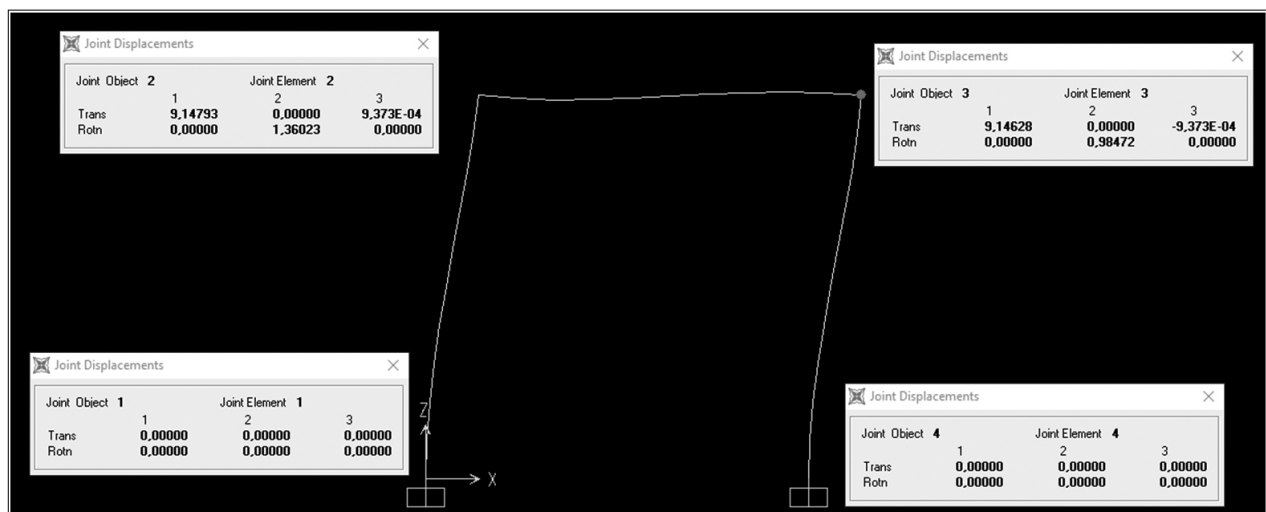
Table 4. Comparison of internal forces and displacements at Nodes 1 and 2

| Internal<br>Forces and<br>Displacements | Node 1-Unit: kN, m        |      |         | Internal<br>Forces and<br>Displacements | Node 2-Unit: kN, m        |      |         |
|-----------------------------------------|---------------------------|------|---------|-----------------------------------------|---------------------------|------|---------|
|                                         | Percentage Difference (%) |      |         |                                         | Percentage Difference (%) |      |         |
|                                         | Matlab                    | FEM  | SAP2000 |                                         | Matlab                    | FEM  | SAP2000 |
| M                                       | 0                         | 0    | 0.01    | M                                       | 0                         | 0    | 0.01    |
| Q                                       | 0                         | 0.01 | 0.01    | Q                                       | 0                         | 0.01 | 0.01    |
| N                                       | 0                         | 4.16 | 0       | N                                       | 0                         | 4.16 | 0       |
| d <sub>x</sub>                          | 0                         | 0    | 0       | d <sub>x</sub>                          | 0                         | 0    | 0.01    |
| d <sub>y</sub>                          | 0                         | 0    | 0       | d <sub>y</sub>                          | 0                         | 0    | 0       |
| ϕ                                       | 0                         | 0    | 0       | ϕ                                       | 0                         | 0    | 0       |

Source: Author's calculation

Figure.4. SAP2000 analysis results for the one-story, one-bay plane frame





Due to the extensive computational workload, Tables 1–4 of this study present only the analysis of discrepancies at Nodes 1 and 2. The results for the remaining nodes will be investigated and reported in a subsequent study.

The results presented above indicate that, in terms of internal forces, the FEM results exhibit relatively small discrepancies, ranging from 0% to 4.16%. The most significant differences are observed in the axial forces at Nodes 1 and 2. For shear forces and bending moments, the SAP2000 results are in close agreement with those obtained using MATLAB, with discrepancies ranging from 0% to 0.01%. Regarding displacements, the differences among the computational approaches are generally negligible. The FEM results are in complete agreement with the MATLAB results, while SAP2000 shows a discrepancy of only 0.01% in the dx displacement component at Node 2.

The maximum discrepancies of 4.16% in axial forces and 0.01% in horizontal displacement are attributed to differences in material, section, and boundary-condition definitions in SAP2000. In contrast, the FEM and MATLAB approaches primarily use elastic modulus and moment of inertia. These modeling differences affect structural stiffness, consequently resulting in discrepancies in the calculated internal forces and displacements among methods.

Overall, the MATLAB-based approach and FEM yield highly consistent results for plane frame problems, demonstrating the reliability of the developed MATLAB program. However, for more complex structures, such as multi-story and spatial frames, discrepancies may increase due to differences in modeling assumptions, material and cross-sectional properties, and numerical rounding during computation.

Based on the results of this study, the application of MATLAB-based computational methods to frame structural analysis is considered feasible and reliable. The findings also highlight the importance of carefully controlling structural modeling, numerical simulation, parameter definitions, and element conventions to minimize discrepancies among different computational approaches.

#### 4. Conclusions

Based on the theoretical investigation of the FEM for plane frame analysis, together with analytical calculations, MATLAB-based numerical computations, and SAP2000 simulations, the following conclusions can be drawn:

The implementation of the FEM in MATLAB for the analysis of internal forces and displacements provides a flexible computational approach applicable to a wide range of load-bearing frame structures, from simple to complex systems, without introducing significant mathematical complexity.

The internal forces and displacements obtained using MATLAB exhibit slight discrepancies compared with those obtained from SAP2000 and FEM. The maximum discrepancy in axial force is 4.16%, while the maximum displacement discrepancy is only 0.01%. Overall, these discrepancies are relatively small, thereby confirming the reliability of the developed MATLAB-based computational program.

The complexity of frame structures, variations in modeling procedures, definitions of material and cross-sectional properties, element stiffness, boundary conditions, numerical rounding, and software-related constraints are among the key factors contributing to the discrepancies observed in the structural analysis results.

Further research should validate the MATLAB-based approach using broader benchmark problems, commercial structural analysis software, and field monitoring data. The framework will be extended to more general frame structures involving various loading conditions, complex boundary conditions, variable cross-sections, and multi-bay and multi-story systems, thereby establishing a systematic research pathway toward increasingly complex structural configurations and enhancing overall computational reliability ■

## REFERENCES:

- Daryl, L. L. (2007). A first course in finite element methods. Cengage Learning.
- Kattan, P. I. (2010). MATLAB guide to finite elements: An interactive approach. Springer Science & Business Media.
- Nguyen, C. C., & Nguyen, T. H. L. (2006). Plane frame analysis by modified discrete element method using displacement model. *Science & Technology Development*, 9(8), 53 - 63.
- Nguyen, T. T., & Nguyen, X. H. (2015). The finite element method using MATLAB. Construction Publishing House.
- Nguyen, V. C. (2026). Advanced finite element method: Lecture notes [Course material for Ph.D. students]. Ho Chi Minh City University of Technology and Education.
- Tran, I. T., & Nguyen, M. C. (2011). The finite element method: Theory and exercises. Vietnam Education Publishing House.