XLVI CILAMCE
VITÓRIA - ES - BRAZIL - 2025
XLVI IBERO-LATIN AMERICAN CONGRESS ON COMPUTATIONAL METHODS IN ENGENEERING

Development of a Python Program for Analysis of Frame Structures Using the Finite Element Method

Diogo M. Cardoso[1,2], Maria S. M. Sampaio[1,3]

[1]*Dept. of Civil Engineering, University of Amazonas State*
Darcy Vargas Avenue, 69050-020, Amazonas/Manaus, Brazil
[2]*dmc.eng22@uea.edu.br,* [3]*msampaio@uea.edu.br*

Abstract. Structural analysis of frames is essential in Civil Engineering. The use of computational tools allows for a more efficient and accurate approach to calculating internal forces, optimizing the structural design and sizing process. However, it is extremely important to have a basic understanding of structural analysis methods, as using computer programs without adequate knowledge of structural analysis can be very dangerous, as incorrect analysis results can cause serious accidents in civil engineering projects. This paper aims to present a Python program based on Structural Matrix Analysis and the Finite Element Method to determine the displacements, internal forces, and support reactions of frames. The implementation will allow for the input of structural data, the construction of the global stiffness matrix, and the determination of the variables of interest - namely, nodal displacements - and from these, the determination of the forces in the elements and the reactions at the structure's supports. The program was validated through known examples and comparisons with theoretical solutions. Therefore, the project aims to provide an accessible and educational resource for students in the field, facilitating structural analysis without the need for expensive commercial software.

Keywords: Frame Structures, Finite Element Method, Structural Analysis, Python Programming.

1 Introduction

A frame is a rigid structure composed of interconnected beams and columns, modeled by one-dimensional bar elements, forming a structure resistant to normal, shear, and bending forces. This type of structure is widely used in industrial warehouses, bridges, and buildings. The structural analysis of frames is fundamental in Civil Engineering, and the use of computational tools allows for a more efficient and accurate approach to calculating internal forces, optimizing the structural design and sizing process.

The growing demand for fast and accurate solutions motivates the development of accessible and effective computational tools. However, knowledge of the fundamentals of basic structural analysis methods is extremely important, as using computer programs without adequate knowledge of structural analysis can be very dangerous, as erroneous analysis results are the cause of serious accidents in civil construction.

Engineering programs related to structural analysis assist the user in finding solutions to a given problem. Many engineering problems can be expressed in terms of partial differential derivatives and have analytical solutions. However, in real-world problems, the analytical solution is complex, requiring the use of a numerical tool to obtain approximate solutions.

A widely used method for calculating approximate solutions is the Finite Element Method (FEM). Its main feature, as confirmed by Hughes [1], Zienkiewicz and Taylor [2], Oden and Reddy [3], and Reddy [4], is to divide a body into finite elements, connected by nodes, and obtain an approximate solution to the problem under analysis. Structural analysis can now be viewed as a computational simulation of structural behavior. According to Martha [5], "it is important to keep in mind that it is inconceivable, in the 21st century, to perform structural analysis tasks, even in the case of bar structures, without the use of computers and computer graphics." With the advancement of computer technologies, the need for specialized tools for structural analysis becomes evident. Therefore, the development of this program aims to provide an accessible and educational resource for engineers and students in

the field to solve frame structures, such as steel structures, facilitating structural analysis without the need for expensive commercial software.

2 General formulation of the FEM for analysis in frame structures

Consider a plane frame finite element with length (L), cross-sectional area (A), moment of inertia (I), and modulus of elasticity (E). The element is subjected to a distributed load (q) along its length, resulting in axial forces and bending moments, as well as associated shear reactions. When a structure is loaded, specific points, called nodes, undergo unknown displacements. These displacements are called degrees of freedom (dofs), as shown in Figure 1.

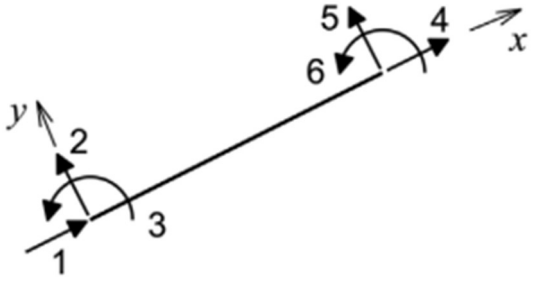


Figure 1. Degrees of freedom (Martha [5]).

The nodal degrees of freedom associated with the initial and final nodes of the element include axial displacement (in the direction of the element), transverse displacement (perpendicular to the element), and rotation θ associated with bending. Thus, each node of a plane frame has 3 degrees of freedom, totaling 6 degrees of freedom per element, as shown in Figure 1.

2.1 Fundamentals of the stiffness method

Both matrix analysis and FEM yield the same stiffness matrix for the structural element. While matrix analysis uses the displacement method, as described by Martha [5], FEM can use variational or energetic methods to obtain the element's stiffness matrix, as described by Vaz [6]. However, FEM is more general and can be used to formulate more general 2D and 3D elements.

The stiffness method consists of subdividing the structure into finite elements, whose endpoints are nodes representing the connections between the members. The force–displacement properties of each element are determined and related through the equilibrium equations at the nodes, forming the global stiffness matrix K. From this, the nodal displacements for an applied load are determined, and, using these displacements, the internal and external forces of the structure are calculated. To do this, we must first establish an element stiffness matrix [K'] shown in Eq. (1) for each frame element, Hibbeler [7].

$$[k'] = \begin{bmatrix} +EA/l & 0 & 0 & -EA/l & 0 & 0 \\ 0 & +12EI/l^3 & +6EI/l^2 & 0 & -12EI/l^3 & +6EI/l^2 \\ 0 & +6EI/l^2 & +4EI/l & 0 & -6EI/l^2 & +2EI/l \\ -EA/l & 0 & 0 & +EA/l & 0 & 0 \\ 0 & -12EI/l^3 & -6EI/l^2 & 0 & +12EI/l^3 & -6EI/l^2 \\ 0 & +6EI/l^2 & +2EI/l & 0 & -6EI/l^2 & +4EI/l \end{bmatrix} \tag{1}$$

The equilibrium equation of the bar that relates displacements, forces and the stiffness matrix in the local coordinate system is given by:

$$[K'].\{d_e\} = \{f_e\} \tag{2}$$

From the stiffness matrix and the nodal forces of each element, the stiffness matrix K and the global nodal forces F of the structure are then assembled as a function of the connection between the elements (incidence), obtaining the global equilibrium equation of the structure:

$$[K].\{D\} = \{F\} \tag{3}$$

In Eq. (3), $\{D\}$ are the global displacements and $\{F\}$ are the vector of applied external forces. Finally, with the known displacements, the internal forces (axial force, bending and shear moments) in each element are obtained, as well as the reactions at the supports.

3 General flowchart for the analysis of frame structures

A general flowchart was created to demonstrate the processing flow for solving frame structure analyses. It describes each step to arrive at the solution for the structure's displacements, forces, and nodal moments as shown in Figure 2.

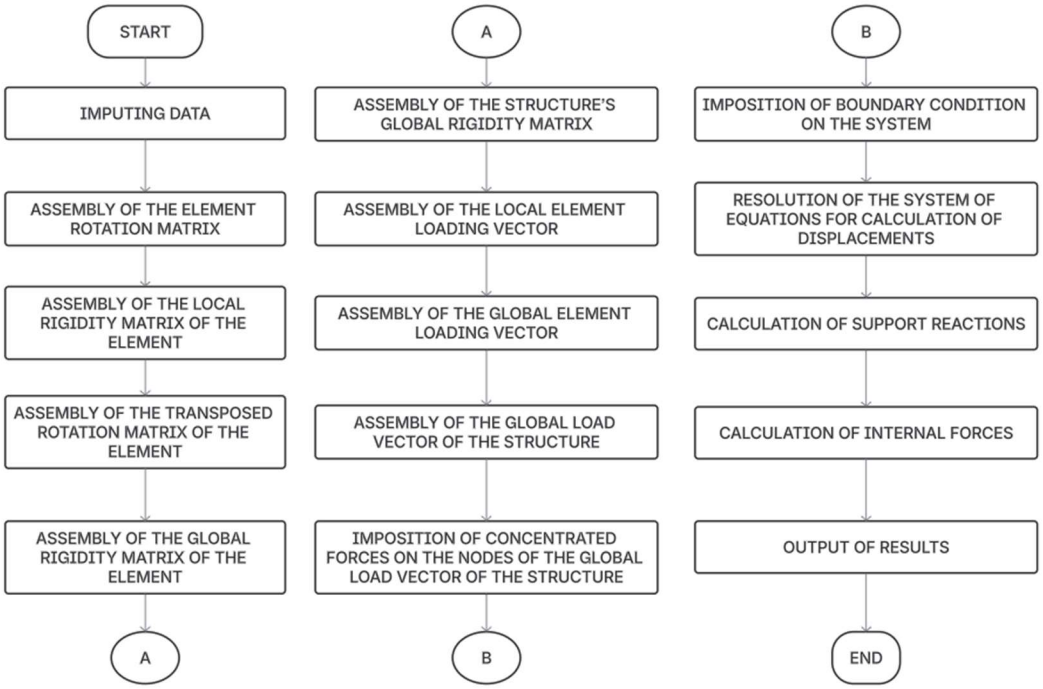


Figure 2. Flowchart of the developed python program.

The flowchart presented in Figure 2 illustrates in a clear and organized manner the computational process adopted for the analysis of frames. It allows us to visualize the logical sequence of operations, from defining the initial parameters to calculating the final displacements and forces. This representation facilitates understanding of the method, highlighting critical steps such as assembling the global stiffness matrix, applying boundary conditions, and solving the system of equations. Furthermore, the flowchart serves as a basis for algorithmic implementation and verification of the analysis procedure.

4 Numerical Results and Validation

Numerical examples available in the literature were tested, and the results demonstrated the good performance of the implemented tool for the tested examples. This section will cover some examples.

4.1 Two-member structure subjected to a horizontal force presented by Hibbeler [7]

The problem consists of a two-member structure with lengths L = 6 m, as shown in Figure 3. Consider I = $180x10^{-6}$ m², A = $6x10^{-3}$ m² and E = 200GPa for both members. A horizontal force of 20kN is applied at one of the nodes. Tables 1 and 2 compare the displacement solutions, and Tables 3 and 4 compare the reaction solutions, at each node, given in Hibbeler [7]. As can be seen, the numerical solution converges with Hibbeler's solution.

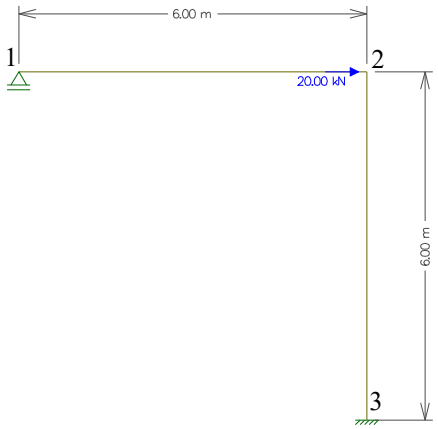


Figure 3. Two – member structure (Hibbeler [7]).

Table 1. Reference displacements

Node	Dx(mm)	Dy(mm)	Dz(rad)
1	17,51	0,00	0,001243
2	17,51	-0,03747	-0,002505
3	0,00	0,00	0,00

Table 2. Reference displacements

Node	Dx(mm)	Dy(mm)	Dz(rad)
1	17,514054	0,00	0,001243
2	17,514054	-0,037477	-0,002505
3	0,00	0,00	0,00

Table 3. Reference reactions

Node	Fx(kN)	Fy(kN)	Mz(kNm)
1	0,00	-7,50	0,00
2	0,00	0,00	0,00
3	-20,00	7,50	75,00

Table 4. Present solution of reactions

Node	Fx(kN)	Fy(kN)	Mz(kNm)
1	0,00	-7,495	0,00
2	0,00	0,00	0,00
3	-20,00	7,495	75,028

4.2 Frame structure with forces applied at the node presented by Hibbeler [7]

The second problem consists of a frame structure, as shown in Figure 4. Considering I = $250x10^{-6}$ m², A = $12x10^{-3}$ m² and E = 200GPa, for both members. A horizontal force of 20 and a vertical force of 30kN are applied at node 1. Tables 5 and 6 show the results of the reference and calculated displacements, respectively. As well as tables 7 and 8 present the solutions of reference and calculated reactions, for the problem presented by Hibbeler [7]. It is observed that there is a great approximation of the calculated values with the reference ones, showing great accuracy of the program.

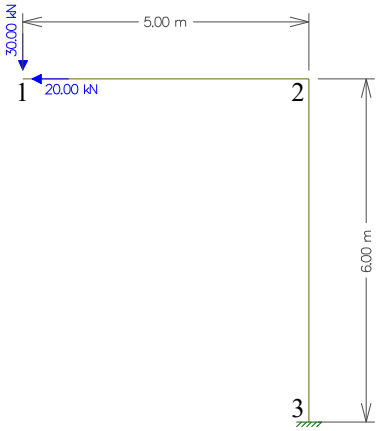


Figure 4. Frame structure (Hibbeler [7]).

Table 5. Reference displacements

Node	Dx(mm)	Dy(mm)	Dz(rad)
1	-82,84	-151,1	0,03270
2	-82,80	-0,07500	0,02520
3	0,00	0,00	0,00

Table 6. Present displacements solution

Node	Dx(mm)	Dy(mm)	Dz(rad)
1	-82.841667	-151.075	0.0327
2	-82.8	-0.075	0.0252
3	0,00	0,00	0,00

Table 7. Reference reactions

Node	Fx(kN)	Fy(kN)	Mz(kNm)
1	0,00	0,00	0,00
2	0,00	0,00	0,00
3	20,00	30,00	-270,00

Table 8. Present solution of reactions

Node	Fx(kN)	Fy(kN)	Mz(kNm)
1	0,00	0,00	0,00
2	0,00	0,00	0,00
3	20,00	30,00	-270,00

5 Conclusions

The development of a Python program for analyzing frame structures using the Finite Element Method (FEM) proved to be an efficient and flexible approach to structural analysis. By implementing the stiffness method in a modular and automated manner, the program enables the calculation of nodal displacements, internal forces, and support reactions with high reliability. It is important to emphasize that the program is still under development, seeking improvements in both the graphics and processing aspects. Therefore, the use of Python, combined with its numerical and visualization libraries, has proven to be a suitable choice for developing open-source tools for both teaching and research. This approach allows for customization and code expansion, encouraging innovation and the dissemination of knowledge in computational methods applied to engineering.

Acknowledgements. The authors would like to thank the University of Amazonas State for the financial support granted through the “ICT Ações Afirmativas” Program: 2025-2026 Edition/SISPROJ Project Number 57952.

Authorship statement. The authors hereby confirm that they are the sole liable people 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] T.J.R Hughes. *The Finite Element Method – linear static and dynamic finite element analysis*. Prentice-Hall International, Inc. Englewood Cliffs, New Jersey, 1987.

[2] O. C. Zienkiewicz and R. L. Taylor, *The Finite Element Method: Basic Formulation and Linear Problems,* 4^{th} Edition, vol. 1. McGraw-Hill, 1989.

[3] J.T. Oden and J.N. Reddy, *An introduction to the mathematical theory of finite elements*. John Wiley & Sons, New York, 1976.

[4] J.N Reddy. *Energy and variational methods in applied mechanics*. John Wiley & Sons, New York, 1984.

[5] L. F Martha. *Análise de Estruturas: Conceitos e Métodos Básicos*. Rio de Janeiro: Elsevier, 2010.

[6] L. E. Vaz. *Método dos elementos finitos em análise de estruturas.*Rio de Janeiro: Elsevier, 2011.

[7] R. C. Hibbeler. *Análise das Estruturas.* 8º ed.Pearson Education do Brasil, São Paulo, 2013.