XLI CILAMCE
XLI Ibero-Latin American Congress on Computational Methods in Engineering
16-19 November, 2020 FOZ DO IGUAÇU PARANÁ - BRAZIL

Finite Elements Method use for displacement calculus in trusses

Chiquesi P. L. Amós[1], Pegoretti dos S. Thaís[1]

[1]*Dept. of Civil Engineering, University of Araraquara*
Rua Carlos Gomes, 1338, Centro, 14801-340, São Paulo/Araraquara,Brazil
aplchiquesi@uniara.edu.br, tdspegoretti@uniara.edu.br

Abstract. The Finite Element Method (FEM) is a numerical procedure for analyzing structures and continuous media. It is a technique used to obtain approximate solutions of differential equations. Trusses are rigid elements whose definition is based on loads applied only to moorings (knots). This project proposes the use of the FEM in trusses in order to discover displacement values and the intensity of the tensile and compression forces in each element of the structure. Initially, the bar was studied and implemented, then the beam. In this work, one example is presented to illustrate the application of the FEM, for the case of a static analysis of a truss, using one-dimensional bar elements including bibliographic results.The stiffness matrixes of each element are presented and the global matrix assembly is developed. To compare the results of the example with a simulation, oneFtool® model is constructed and both results are compared. This project is able to expand the student's understanding of FEM, modelling and results interpretation.

Keywords:FEM, displacement, trusses, global matrix.

1. Introduction

The Finite Element Method (FEM) was a system developed by Walter Ritz in 1909. His intention was to determine approximate results of problems in deformable solids. In 1943 Richard Courant applied new methods to the Ritz system, using linear functions over triangular shapes to solve torsion problems [1].

A priori, the commonly used MEF is based on the Ritz method that foresees the division of the domain of integration in continuous means in a finite number of small known regions and called finite elements.

In this method, the model is divided into simple shapes, called finite elements, such as flat models, usually triangles or quadrilaterals. Fig. 1 illustrates a two-dimensional model of a continuous hollow structure with a hole in its upper area and its division into finite area elements.

Figure 1

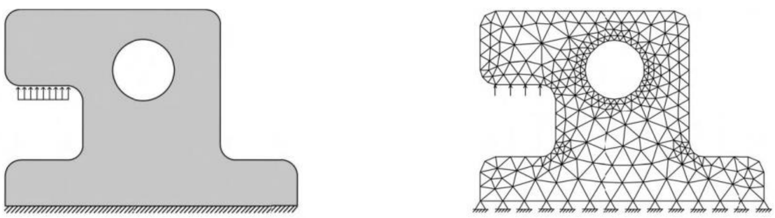

Figure 1: Continuous structure exemplifying the Finite Element Method [2]

This division according to the figure above is called a finite element network, which can be increased or decreased varying in size. Each point of intersection in this network is called a node.

Through this methodology, it is possible to study structures from different areas: automobile, aeronautics, civil, among others. It is possible to determine the distribution of internal forces in the structures and their deformation, when subjected to external static or dynamic loading.

There are structural elements connected continuously. The continuous body is artificially subdivided into a certain finite number of elements, connected only in nodes, thus making the approximate representation of a continuous body.

Trusses are frames formed by the connection of rigid elements, such as wood, steel and others. Its definition is based on loads applied only to the nodes (moorings), leaving the bars receiving only tensile or compression efforts.

The origin of the trusses is believed to have been around the 18th century, but used and applied by Alexandre GrahamBell in 1907. And later being better developed for any applications [3].

The main elements that make up the trusses are, [4];

- Rope or flange: set of bars that limit the upper or lower truss;
- Amount: vertical bar of the trusses;
- Diagonal: bar with the axis coinciding with the diagonal of a panel;
- Node: meeting point and joining the ends of the bars.

Figure 2 represents a truss with the elements described above showing their respective locations.

Figure 2

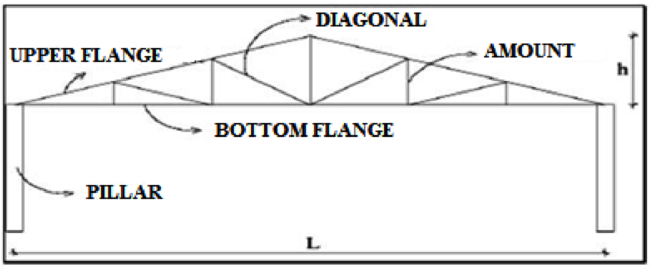

Figure 2: Image exemplified with the types of bars that make up a trellis

Figure 3 illustrates the image containing the magnitude of the modal shift of a truss generated by the commercial software RFEM ®.

Figure 3

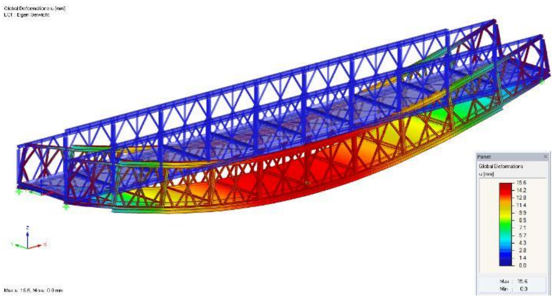

Figure 3: Image by: RFEM, Representation of deformation of a bridge in RFEM,[5]

The Fig 1 and Fig. 3 showed, there are commercial software(s) that calculate the internal forces and the displacements of the structures. The present work aims at a complete understanding of the MEF through its computational implementation using a relatively simple example: a trellis.

In addition, after such understanding, the researcher will be able to make some analyzes with more property on the subject, such as: varying the type of finite element chosen, evaluating the influence of the quantity of elements used and even evaluating the possibility of including dynamic loads.

2. Purpose

The general objective of the work is to use the finite element method to calculate displacements in structures: bars, beams and trusses.

The specific objectives of this work are:

- Improve research and scientific writing skills;
- Increase knowledge about numerical methods for calculating internal forces and displacements in structures;
- Develop programming skills.

3. Methodology

A bibliographic search on the finite element and structure method (bars, beams and trusses) is be carried out. One truss example is detailed studied and its results are be compared with examples already published in the references and with simulations in Ftool®.

4. Development

The main idea of finite elements is to divide the domain of an infinite problem into finite subdomains to obtain results about the element in question.

4.1 Stiffness matrix of a bar

It is a system of differentialand symmetric equations whose set with the Finite Element Method matrix equations are obtained. Its equation is obtained through:

$$[K^e]\{u^e\} = \{F^e\} \quad (1)$$

Where $[K^e]$ is the Rigidity Matrix of an element, $\{u^e\}$ is the vector of nodal displacements, and $\{F^e\}$nodal forces of the element.

This matrix represents the relationship between nodal efforts and nodal displacements.

4.2 Assembly techniques of the global system - Displacement method

The displacement method consists of writing the global equilibrium equations from the local equations. It will be presented through the case study of this work.

The purpose of this example is to illustrate the application of the Finite Element Method, for the case of a static analysis of a truss, using one-dimensional bar elements.

Considering the problem of the flat truss, shown in Fig. 4, subject to two forces; one at node 3 and another at node 4; a roller at 2 that restricts vertical travel and a bezel at node 5 that restricts the two degrees of freedom (x and y) at that node.

Figure 4

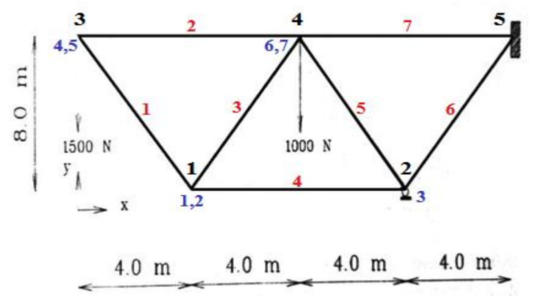

Figure 4: Structure of a truss using one-dimensional elements of a bar

The free degrees of freedom of the truss were numbered according to the Tab. 1.

Table 1: Numbering of the structure's degrees of freedom

Node	X	Y
1	1	2
2	3	-
3	4	5
4	6	7
5	-	-

Thus, for each of the 5 elements, a vector called LOC (location) was assembled, which organizes the numbers of degrees of freedom that relate to the nodes of each element. Then, the local stiffness matrix of each element is assembled and, from the LOC vector, the elements of the local matrix are allocated in the global matrix. In this way, the assembly of the global matrix is clearly presented, as follows.

The location vector of element 1 is:

$$LOC^1 = [1 \quad 2 \quad 4 \quad 5] \tag{2}$$

The local matrix of element 1 is:

$$[K^1] = \begin{bmatrix} K^1_{(1,1)} & K^1_{(1,2)} & K^1_{(1,3)} & K^1_{(1,4)} \\ K^1_{(2,1)} & K^1_{(2,2)} & K^1_{(2,3)} & K^1_{(2,4)} \\ K^1_{(3,1)} & K^1_{(3,2)} & K^1_{(3,3)} & K^1_{(3,4)} \\ K^1_{(4,1)} & K^1_{(4,2)} & K^1_{(4,3)} & K^1_{(4,4)} \end{bmatrix} \tag{3}$$

The assembly of the elements of element 1 in the global matrix is:

$$[K^1] = \begin{vmatrix} K^1_{(1,1)} & K^1_{(1,2)} & 0 & K^1_{(1,3)} & K^1_{(1,4)} & 0 & 0 \\ K^1_{(2,1)} & K^1_{(2,2)} & 0 & K^1_{(2,3)} & K^1_{(2,4)} & 0 & 0 \\ 0 & 0 & 0 & 0 & 0 & 0 & 0 \\ K^1_{(3,1)} & K^1_{(3,2)} & 0 & K^1_{(3,3)} & K^1_{(3,4)} & 0 & 0 \\ K^1_{(4,1)} & K^1_{(4,2)} & 0 & K^1_{(4,3)} & K^1_{(4,4)} & 0 & 0 \\ 0 & 0 & 0 & 0 & 0 & 0 & 0 \\ 0 & 0 & 0 & 0 & 0 & 0 & 0 \end{vmatrix} \tag{4}$$

The location vector of element 2 is:

$$LOC^2 = [4 \quad 5 \quad 6 \quad 7] \tag{5}$$

The local matrix of elements 2, 3, 4, 5, 6 and 7 were built similarly to the local matrix of element 1, so they are not presented in the following development.

The assembly of the elements of element 2 in the global matrix is:

$$[K^2] = \begin{vmatrix} 0 & 0 & 0 & 0 & 0 & 0 & 0 \\ 0 & 0 & 0 & 0 & 0 & 0 & 0 \\ 0 & 0 & 0 & 0 & 0 & 0 & 0 \\ 0 & 0 & 0 & K^2_{(1,1)} & K^2_{(1,2)} & K^2_{(1,3)} & K^2_{(1,4)} \\ 0 & 0 & 0 & K^2_{(2,1)} & K^2_{(2,2)} & K^2_{(2,3)} & K^2_{(2,4)} \\ 0 & 0 & 0 & K^2_{(3,1)} & K^2_{(3,2)} & K^2_{(3,3)} & K^2_{(3,4)} \\ 0 & 0 & 0 & K^2_{(4,1)} & K^2_{(4,2)} & K^2_{(4,3)} & K^2_{(4,4)} \end{vmatrix} \tag{6}$$

The location vector of element 3 is:

$$LOC^3 = [1 \quad 2 \quad 6 \quad 7] \tag{7}$$

The assembly of the elements of element 3 in the global matrix is:

$$[K^3] = \begin{vmatrix} K^3_{(1,1)} & K^3_{(1,2)} & 0 & 0 & 0 & K^3_{(1,3)} & K^3_{(1,4)} \\ K^3_{(2,1)} & K^3_{(2,2)} & 0 & 0 & 0 & K^3_{(2,3)} & K^3_{(2,4)} \\ 0 & 0 & 0 & 0 & 0 & 0 & 0 \\ 0 & 0 & 0 & 0 & 0 & 0 & 0 \\ 0 & 0 & 0 & 0 & 0 & 0 & 0 \\ K^3_{(3,1)} & K^3_{(3,2)} & 0 & 0 & 0 & K^3_{(3,3)} & K^3_{(3,4)} \\ K^3_{(4,1)} & K^3_{(4,2)} & 0 & 0 & 0 & K^3_{(4,3)} & K^3_{(4,4)} \end{vmatrix} \tag{8}$$

The location vector of element 4 is:

$$LOC^4 = [1 \quad 2 \quad 3 \quad 0] \tag{9}$$

The assembly of the elements of element 4 in the global matrix is:

$$[K^4] = \begin{vmatrix} K^4_{(1,1)} & K^4_{(1,2)} & K^4_{(1,3)} & 0 & 0 & 0 & 0 \\ K^4_{(2,1)} & K^4_{(2,2)} & K^4_{(2,3)} & 0 & 0 & 0 & 0 \\ K^4_{(3,1)} & K^4_{(3,2)} & K^4_{(3,3)} & 0 & 0 & 0 & 0 \\ 0 & 0 & 0 & 0 & 0 & 0 & 0 \\ 0 & 0 & 0 & 0 & 0 & 0 & 0 \\ 0 & 0 & 0 & 0 & 0 & 0 & 0 \\ 0 & 0 & 0 & 0 & 0 & 0 & 0 \end{vmatrix} \tag{10}$$

The location vector of element 5 is:

$$LOC^5 = [3 \quad 0 \quad 6 \quad 7] \tag{11}$$

The assembly of the elements of element 5 in the global matrix is:

$$[K^5]= \begin{vmatrix} 0 & 0 & 0 & 0 & 0 & 0 & 0 \\ 0 & 0 & 0 & 0 & 0 & 0 & 0 \\ 0 & 0 & K^5_{(1,1)} & 0 & 0 & K^5_{(1,3)} & K^5_{(1,4)} \\ 0 & 0 & 0 & 0 & 0 & 0 & 0 \\ 0 & 0 & 0 & 0 & 0 & 0 & 0 \\ 0 & 0 & K^5_{(3,1)} & 0 & 0 & K^5_{(3,3)} & K^5_{(3,4)} \\ 0 & 0 & K^5_{(4,1)} & 0 & 0 & K^5_{(4,3)} & K^5_{(4,4)} \end{vmatrix} \tag{12}$$

The location vector of element 6 is:

$$LOC^6 = [3 \quad 0 \quad 0 \quad 0] \tag{13}$$

The assembly of the elements of element 6 in the global matrix is:

$$[K^6]= \begin{vmatrix} 0 & 0 & 0 & 0 & 0 & 0 & 0 \\ 0 & 0 & 0 & 0 & 0 & 0 & 0 \\ 0 & 0 & K^6_{(1,1)} & 0 & 0 & 0 & 0 \\ 0 & 0 & 0 & 0 & 0 & 0 & 0 \\ 0 & 0 & 0 & 0 & 0 & 0 & 0 \\ 0 & 0 & 0 & 0 & 0 & 0 & 0 \\ 0 & 0 & 0 & 0 & 0 & 0 & 0 \end{vmatrix} \tag{14}$$

The location vector of element 7 is:

$$LOC^7 = [6 \quad 7 \quad 0 \quad 0] \tag{15}$$

The assembly of the elements of element 7 in the global matrix is:

$$[K^7] = \begin{vmatrix} 0 & 0 & 0 & 0 & 0 & 0 & 0 \\ 0 & 0 & 0 & 0 & 0 & 0 & 0 \\ 0 & 0 & 0 & 0 & 0 & 0 & 0 \\ 0 & 0 & 0 & 0 & 0 & 0 & 0 \\ 0 & 0 & 0 & 0 & 0 & 0 & 0 \\ 0 & 0 & 0 & 0 & 0 & K^7_{(1,1)} & K^7_{(1,2)} \\ 0 & 0 & 0 & 0 & 0 & K^7_{(2,1)} & K^7_{(2,2)} \end{vmatrix} \tag{16}$$

Thus, the final global matrix becomes:

$$
\begin{vmatrix}
K^{1}_{(1,1)}+K^{3}_{(1,1)}+K^{4}_{(1,1)} & K^{1}_{(1,2)}+K^{3}_{(1,2)}+K^{4}_{(1,2} & K^{4}_{(1,3)} & K^{1}_{(1,3)} & K^{1}_{(1,4)} \\
K^{1}_{(2,1)}+K^{3}_{(2,1)}+K^{4}_{(2,1)} & K^{1}_{(2,2)}+K^{3}_{(2,2)}+K^{4}_{(3,2} & K^{4}_{(2,3)} & K^{1}_{(2,3)} & K^{2}_{(2,4)} \\
K^{4}_{(3,1)} & K^{4}_{(3,2)} & K^{4}_{(3,3)}+K^{5}_{(1,1)}+K^{6}_{(1,1)} & - & - \\
K^{1}_{(3,1)} & K^{1}_{(3,2)} & - & K^{1}_{(3,3)}+K^{2}_{(1,1)} & K^{1}_{(3,4)}+K^{2}_{(1,2)} & \dots \\
K^{1}_{(4,1)} & K^{1}_{(4,2)} & - & K^{1}_{(4,3)}+K^{2}_{(2,1)} & K^{1}_{(4,4)}+K^{2}_{(2,2)} \\
K^{3}_{(3,1)} & K^{3}_{(3,2)} & K^{5}_{(3,1)} & K^{2}_{(3,1)} & K^{2}_{(3,2)} \\
K^{3}_{(4,1)} & K^{3}_{(4,2)} & K^{5}_{(4,1)} & K^{2}_{(4,1)} & K^{2}_{(4,2)}
\end{vmatrix}
$$

$$
\begin{vmatrix}
K^{3}_{(1,3)} & K^{3}_{(1,4)} \\
K^{3}_{(2,3)} & K^{3}_{(2,4)} \\
K^{5}_{(1,3)} & K^{5}_{(1,4)} \\
K^{2}_{(1,3)} & K^{2}_{(1,4)} \\
K^{2}_{(2,3)} & K^{2}_{(2,4)} \\
K^{2}_{(3,3)}+K^{3}_{(3,3)}+K^{5}_{(3,3)}+K^{7}_{(1,1)} & K^{2}_{(3,4)}+K^{3}_{(3,4)}+K^{5}_{(3,4)}+K^{7}_{(1,2)} \\
K^{2}_{(4,3)}+K^{3}_{(4,3)}+K^{5}_{(4,3)}+K^{7}_{(2,1)} & K^{2}_{(4,4)}+K^{3}_{(4,4)}+K^{5}_{(4,4)}+K^{7}_{(2,2)}
\end{vmatrix}
$$

5. Results

The truss of Fig. 4 simulated data are: Young modulus $1{,}99.10^{9}$ N/m^2, section area 0,009 m^2 and Poisson coefficient 0,3 [6].The comparison results obtained in the reference were displacements and reaction forces that are listed in Tab. 2, with the simulation results obtained in Ftool®.It is possible to observe that the maximum error is 1,93%, in the Displacement 2, so the results were considered equivalent. Figure 5 illustrates the deformed configuration of the truss includingits reactions.

Table 2: Comparison between reference and simulation.

	Reference	Simulation Results	Error %
Displacement 1	$0{,}749.10^{-4}$ m	$0{,}740.10^{-4}$ m	1,20
Displacement 2	$-1{,}087.10^{-4}$ m	$-1{,}066.10^{-4}$ m	1,93
Displacement 4	$-0{,}155.10^{-4}$ m	$-0{,}154.10^{-4}$ m	0,65
Displacement 5	$-1{,}633.10^{-4}$ m	$-1{,}613.10^{-4}$ m	1,22
Reaction in node 2	8000 N	8000 N	0
Reaction in node 5	-5500 N	-5500 N	0

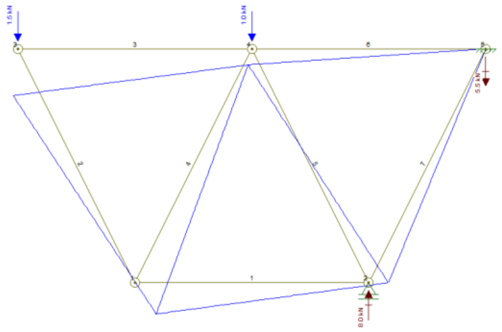

Figure 5: Deformed configuration of the truss including its reactions

The reference and simulation displacement numbers in Tab. 2 are really close, while the reaction values are exactly the same. So the model was considered correct.

6. Conclusion

In this work, the FEM method was applied in a truss example. First, the stiffness matrix of each element was presented and a methodology to build the global matrix was shown. Each free degree of freedom was numbered and the truss elements were related to them in order to assembly the stiffness matrix of each element um the global stiffness matrix.

Then, the truss was modeled in Ftool®. The joints that permit rotation among the elements were included in all nodes, so the elements do not suffer shear stress or bending moment. Theoretical results for displacements and reaction forces were compared to the simulated ones and the greatest error valued 1,93% and the reaction forces errors were 0%, so the model was considered correct.

Future work can be projected with this knowledge base:

1. Implement the methodology to build the global matrix in another software such as Matlab®;
2. Analyze different structures: porticoes and grids;
3. Consider dynamic efforts;
4. Study a real truss using the same methodology.

7. References

[1] CAMPOS, Marcos Donisete de, O Método de Elementos Finitos aplicado à Simulação Numérica de Escoamentos de Fluidos, 2006, acesso em 05/04/2018,http://www.mat.ufg.br/bienal/2006/poster/marcodonisete.pdf.

[2]MARTHA, Luiz Fernando, 1955 - Análise de estruturas: conceitos e métodosbásicos / Luiz Fernando Campos Ramos Martha. – Rio de Janeiro: Elsevier, 2010.

[3] SOUZA, Alex Sander Clemente, GONÇALVES, Roberto Martins, – Análiseteórica e experimental de treliçasespaciais, 2006. Acessoem (27/04/2019). Disponível em: http://www.set.eesc.usp.br/cadernos/nova_versao/pdf/cee31_31.pdf

[4] SOUZA, Marta Francisca Suassuna Mendes de, RODRIGUES, Rafael Bezerra – Sistemasestruturais de edificações e exemplos, 2008. Acesso em (13/03/2019). Disponível em: http://www.fec.unicamp.br/~nilson/apostilas/sistemas_estruturais_grad.pdf

[5] HANK, JansonBridging – Representação de deformação de umaponte no RFEM, 2013. Acesso em (29/04/2019), disponível em: https://www.dlubal.com/pt/download-e-informacao/referencias/projetos-de-clientes/000714

[6] PAVANELO, Renato, Notas de aula Introdução ao Método dos Elementos Finitos, Campinas, 1997.

Acknowledgements: **ToPIBIQ and University of Araraquara.**