XLIII CILAMCE
21-25, November, 2022
FOZ DO IGUAÇU - PR - BRAZIL

Development of a computational routine in Fortran language for analysis of plane and space trusses using the finite element method

Lucas A. de Aguiar[1], Marcos B. Guimarães[1], Daniele K. Monteiro[1], Patrick de O. B. da Costa[1]

[1]*Department of Civil Engineering, Federal University of Rio Grande do Sul*
Av. Osvaldo Aranha 99, 99, 90035-190, Rio Grande do Sul, Brazil
lucas.a.aguiar@hotmail.com, bressan.marcos@hotmail.com,
danielekauctz@hotmail.com, ecivilpatrick@yahoo.com

Abstract. Computational tools applied to the teaching and learning of structural analysis have been increasingly developed and improved to optimize computational time and cost. The method to solve the analysis, the language to be chosen, the organization of the algorithm, as well as the programming paradigm adopted, are factors that must be taken into account in numerical analysis. Thus, this paper aims to study the behavior of 2D and 3D trusses in the linear-elastic range using a code developed with Fortran programming language, showing the importance of computational application and numerical methods in civil engineering. To solve this problem, the bar finite element formulation was used. Three different benchmark examples were analyzed and verified with a commercial software, one plane truss and one space truss. The results of these structures in stresses and displacements were compared with MASTAN2, proving the efficiency of the built algorithms to solve the studied problems.

Keywords: Trusses, Fortran, FEM.

1 Introduction

The advancement of technology and the development of numerical methods have been gaining more and more space in the field of structures, both in engineering offices and in pedagogical education. Among the main reasons that can be highlighted are the use of open source, the possibility of greater cooperation and integration between projects and disciplines, the versatility of the code, and the greater security for the proposed solutions. In this context, the computationally implemented finite element method (FEM) becomes a powerful tool with wide application in Structural Engineering. This method proposes to simulate the behavior of a real structure through a geometrical discratization of the proposed problem, where the results can be approximated numerically [1].

Trusses are an example of a structure widely used in civil construction and easy to implement numerically. This type of system is a viable solution for large spans due to its: high stiffness with reduced weight; high degree of hyperasticity with excellent effort redistribution; and easiness of transport and assembly. According to Martha [2], both plane and space trusses are considered reticulated structures, i.e., the bars are designed only to work under axial tensile and/or compression forces.

Thus, this work aims to study the behavior of 2D and 3D trusses in the linear-elastic range using a code developed with the FORTRAN programming language. Consequently, the importance of computational application and numerical methods in educational practice is shown, using finite element method.

2 Finite element formulation for trusses

The proposed problems are solved through the adoption of bar elements, using a computational routine in FORTRAN based on the Displacement Method. To analyze these systems, it is necessary to: define the element associated with the problem and the global reference system, identify the members by name and their respective connectivity, and specify the material properties and their cross-section.

With these predefined data and knowing the restrictions of the degrees of freedom (DOF), it is possible to calculate the stiffness matrix and the load vector of each element. Therefore, by applying the boundary conditions and solving the equilibrium equation, the generalized displacements of the nodes are obtained and the stresses on the members are calculated. In the following subsections, the formulation for plane (2D) and space (3D) trusses will be detailed.

2.1 Plane trusses

To formulate the equilibrium equation, the simple example of the spring can be used, mainly because this element only transmits axial forces as well as trusses. Consider a spring subjected to an external pulling force (F). Note that the internal force of the spring can be expressed by the product of the stiffness coefficient (k) and the displacement (u), thus, defining the generic equilibrium equation $ku = F$ (see Filho [3]).

According to Filho [3], analogous to the spring problem, the generic equilibrium equation can be extended to plane truss bars, according to the scheme in Fig. 1, where the coefficient k is the axial stiffness of the bar with $k = EA/L$.

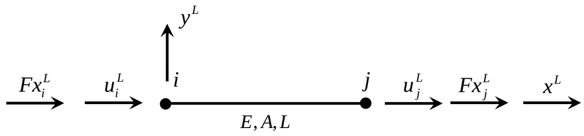

Figure 1. Plane truss bar equilibrium diagram

Filho [3] and Soriano [4] detail the procedures and equations described below related to the Displacement Method. In summary, a unitary and positive displacement is applied at node "i", i.e., $u_i^L = 1$ and the node "j" is blocked. Thus, the load that causes the displacement (Fx_i^L) and the respective reaction in "j" (Fx_j^L) are given by the set of expressions in eq. (1).

$$Fx_i^L = \tfrac{EA}{L} u_i^L = \tfrac{EA}{L} = k_{ii}; \quad Fx_j^L = -\tfrac{EA}{L} = k_{ji} \tag{1}$$

Applying a positive unit displacement at node "j" and blocking node "i", in the same way as shown in eq. (1), the terms k_{jj} and k_{ij} are obtained. Therefore, a generic term k_{ij} indicates the action of node "i" due to a unit displacement at node "j". As stated by the theorems of Betti and Maxwell $k_{ij} = k_{ji}$, where k_{ii} and k_{jj} are always positive. Hence, the set of expressions in eq. (2) express a general case of forces Fx_i^L and Fx_j^L, and displacements u_i^L and u_j^L.

$$Fx_i^L = -\tfrac{EA}{L} \Delta L_i = -\tfrac{EA}{L} \left(u_j^L - u_i^L\right); \quad Fx_j^L = -\tfrac{EA}{L} \Delta L_j = \tfrac{EA}{L} \left(u_j^L - u_i^L\right) \tag{2}$$

This can also be expressed in matrix form, according to eq. (3).

$$\frac{EA}{L} \begin{bmatrix} 1 & -1 \\ -1 & 1 \end{bmatrix} \cdot \begin{pmatrix} u_i^L \\ u_j^L \end{pmatrix} = \begin{bmatrix} k_{ii} & k_{ij} \\ k_{ji} & k_{jj} \end{bmatrix} \cdot \begin{pmatrix} u_i^L \\ u_j^L \end{pmatrix} = \begin{pmatrix} Fx_i^L \\ Fx_j^L \end{pmatrix} \tag{3}$$

Since truss elements only transmit axial forces, and no forces or displacements acting in the y^L direction, the matrix expression that represents the equilibrium of bars can be described by eq. (4) when referring to local axes. Where u_i^L and v_i^L are displacements on the x-axis and the y-axis.

$$\frac{EA}{L} \begin{bmatrix} 1 & 0 & -1 & 0 \\ 0 & 0 & 0 & 0 \\ -1 & 0 & 1 & 0 \\ 0 & 0 & 0 & 0 \end{bmatrix} \cdot \begin{pmatrix} u_i^L \\ v_i^L \\ u_j^L \\ v_j^L \end{pmatrix} = \begin{pmatrix} Fx_i^L \\ 0 \\ Fx_j^L \\ 0 \end{pmatrix} \tag{4}$$

Similar to the generic expression $ku = F$, one can write the matrix equilibrium equation for plane truss elements (eq. (4)) in a compact form, as presented in eq. (5), where m indicates the number of the bar, L represents the local system, and $\underset{\sim}{K}$, $\underset{\sim}{U}$ and $\underset{\sim}{P}$ are the stiffness matrix, displacement vectors, and forces respectively. (see McGuire et al. [5]).

$$\underset{\sim}{K}^{m,L}\underset{\sim}{U}^{m,L} = \underset{\sim}{P}^{m,L} \tag{5}$$

The stiffness matrix of an element is normally obtained in the local coordinate system, as well as the nodal forces. However, while results from the equilibrium equation are given in the global reference system. Therefore, it is necessary the rotation of the axes as shown in Fig. 2, whose formulation is established by Soriano [4].

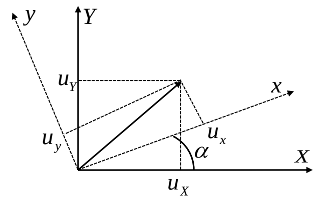

Figure 2. Rotation of axes in the plane, adapted from Soriano [4]

As shown in Fig. 2, the vector $\vec{u}$ is considered in the XY (global) and xy (local) references, according to eq. (6).

$$\begin{pmatrix} u_x \\ u_y \end{pmatrix} = \begin{bmatrix} \cos\alpha & \sin\alpha \\ -\sin\alpha & \cos\alpha \end{bmatrix} \cdot \begin{pmatrix} u_X \\ u_Y \end{pmatrix} \tag{6}$$

The rotation matrix ($\underset{\sim}{r}$) and its respective expansion ($\underset{\sim}{R}$) for plane trusses are indicated in eq. (7), which is described in nodal partitions [4, 5].

$$\underset{\sim}{r} = \begin{bmatrix} \cos\alpha & \sin\alpha \\ -\sin\alpha & \cos\alpha \end{bmatrix} \quad \rightarrow \quad \underset{\sim}{R} = \begin{bmatrix} \underset{\sim}{r} & \underset{\sim}{0} \\ \underset{\sim}{0} & \underset{\sim}{r} \end{bmatrix} \tag{7}$$

Finally, using the rotation $\underset{\sim}{K}^{m,G} = \underset{\sim}{R}^{T}\underset{\sim}{K}^{m,L}\underset{\sim}{R}$, according to the procedure indicated by Soriano [4] and McGuire et al. [5], the global stiffness matrix of plane trusses (eq. (8)) is obtained.

$$\begin{pmatrix} F_{x1} \\ F_{y1} \\ F_{x2} \\ F_{y2} \end{pmatrix} = \frac{EA}{L} \begin{bmatrix} \cos^2\alpha & \sin\alpha\cos\alpha & -\cos^2\alpha & -\sin\alpha\cos\alpha \\ \sin\alpha\cos\alpha & \sin^2\alpha & -\sin\alpha\cos\alpha & -\sin^2\alpha \\ -\cos^2\alpha & -\sin\alpha\cos\alpha & \cos^2\alpha & \sin\alpha\cos\alpha \\ -\sin\alpha\cos\alpha & -\sin^2\alpha & \sin\alpha\cos\alpha & \sin^2\alpha \end{bmatrix} \cdot \begin{pmatrix} u_1 \\ v_1 \\ u_2 \\ v_2 \end{pmatrix} \tag{8}$$

2.2 Space trusses

Space trusses (3D) are defined by the Cartesian coordinate system (x,y,z) and their bars and loads can be in any direction in space. Taking eq. (4) as a reference, and knowing that space truss elements also only transmit axial forces (there are no forces or displacements acting on the y^L and z^L axes), the matrix expression of eq. (9) is defined. It represents the equilibrium equation for space truss bars, referring to local axes, or in a compact form as already demonstrated in eq. (5). Where w_i^L is displacements on the z-axis.

$$\frac{EA}{L}\begin{bmatrix} 1 & 0 & 0 & -1 & 0 & 0 \\ 0 & 0 & 0 & 0 & 0 & 0 \\ 0 & 0 & 0 & 0 & 0 & 0 \\ -1 & 0 & 0 & 1 & 0 & 0 \\ 0 & 0 & 0 & 0 & 0 & 0 \\ 0 & 0 & 0 & 0 & 0 & 0 \end{bmatrix} \cdot \begin{pmatrix} u_i^L \\ v_i^L \\ w_i^L \\ u_j^L \\ v_j^L \\ w_j^L \end{pmatrix} = \begin{pmatrix} Fx_i^L \\ 0 \\ 0 \\ Fx_j^L \\ 0 \\ 0 \end{pmatrix} \tag{9}$$

The rotation matrix $\underset{\sim}{r}$ is defined in three-dimensional space with its lines referring to the directional cosines of the x, y, and z axes, as indicated in eq. (10) [4]. The y-axis directional cosines are illustrated in Fig. 3.

$$\underset{\sim}{r} = \begin{bmatrix} \lambda_{xX} & \lambda_{xY} & \lambda_{xZ} \\ \lambda_{yX} & \lambda_{yY} & \lambda_{yZ} \\ \lambda_{zX} & \lambda_{zY} & \lambda_{zZ} \end{bmatrix}. \tag{10}$$

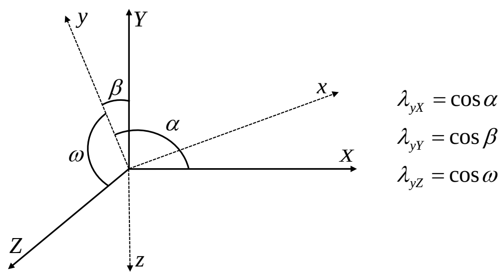

Figure 3. Rotation of axes in three-dimensional space, adapted from Soriano [4]

The x-axis passes through the initial (i) and final (j) nodal points of the bar of coordinates (X_i, Y_i, Z_i) and (X_j, Y_j, Z_j). Thus, the directional cosines of each axis are given by eq. (11), as proposed by Soriano [4].

$$\lambda_{xX} = \frac{X_j - X_i}{\text{L}} \quad , \quad \lambda_{xY} = \frac{Y_j - Y_i}{\text{L}} \quad , \quad \lambda_{xZ} = \frac{Z_j - Z_i}{\text{L}} \tag{11}$$

where L defines the length of the bar, expressed by eq. (12).

$$L = \sqrt{(X_j - X_i)^2 + (Y_j - Y_i)^2 + (Z_j - Z_i)^2} \tag{12}$$

Therefore, the rotation matrix $\underset{\sim}{R}$ for space trusses (dimensions $6x6$) is defined in a compact form in eq. (7), using the rotation matrix $\underset{\sim}{r}$ of three-dimensional space (eq. (10)). Again, applying $\underset{\sim}{K}^{m,G} = \underset{\sim}{R}^T \underset{\sim}{K}^{m,L} \underset{\sim}{R}$, the global stiffness matrix for space trusses (eq. (13)) is obtained.

$$\underset{\sim}{K}^{m,G} = \frac{EA}{L} \begin{bmatrix} \lambda_{xX}^2 & \lambda_{xX}\lambda_{xY} & \lambda_{xX}\lambda_{xZ} & -\lambda_{xX}^2 & -\lambda_{xX}\lambda_{xY} & -\lambda_{xX}\lambda_{xZ} \\ \cdot & \lambda_{xY}^2 & \lambda_{xY}\lambda_{xZ} & -\lambda_{xY}\lambda_{xX} & -\lambda_{xY}^2 & -\lambda_{xY}\lambda_{xZ} \\ \cdot & \cdot & \lambda_{xZ}^2 & -\lambda_{xZ}\lambda_{xX} & -\lambda_{xZ}\lambda_{xY} & -\lambda_{xZ}^2 \\ \cdot & \cdot & \cdot & \lambda_{xX}^2 & \lambda_{xX}\lambda_{xY} & \lambda_{xX}\lambda_{xZ} \\ \cdot & \cdot & \cdot & \cdot & \lambda_{xY}^2 & \lambda_{xY}\lambda_{xZ} \\ \text{Sim}\cdot & \cdot & \cdot & \cdot & \cdot & \lambda_{xZ}^2 \end{bmatrix} \tag{13}$$

3 Numerical examples

In this chapter, two problems were proposed: a plane truss and a space truss. The solution was developed computationally in FORTRAN language, following the formulation detailed in the previous chapter.

3.1 Plane Truss

The routine for plane trusses was tested with the Warren truss shown in Fig. 4. The structure has width $L = 10.5m$, height $H = 3m$ and elastic modulus $E = 200GPa$. Table 1 indicates the cross-sectional area of the bars.

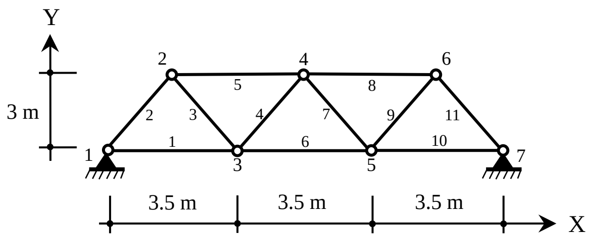

Figure 4. Warren truss

Table 1. Cross sectional areas of the members of the Warren truss

Group	Member number	Area (m^2)
Inferior chord	1, 6 e 10	0.0060
Diagonals	2, 3, 4, 7, 9, 11	0.0040
Superior chord	5 e 8	0.0080

Initially, to use the FORTRAN routine, it is necessary to build a data input file in *.txt* format, containing the material properties, including its geometry, the connectivity vector, the relationship that informs the number of bars connected, and the coordinate vector.

Next, the program must be informed of the boundary conditions. In this example, it was applied to nodes 1 and 7 movement restrictions in the X and Y directions, imposing zero value. As for the loading, two $150kN$ loads were applied vertically downward at nodes 3 and 5. In the computational routine developed, the vectors and matrices must have the appropriate dimensions for the plane truss solution, considering two degrees of freedom per node, i.e., $K \in \mathbb{R}^{14x14}$ e $F, P \in \mathbb{R}^{14x1}$. The allocation of the individual stiffness matrix of each member in the global stiffness matrix is done by the "*do*" command. This creates a loop that traverses the rows and columns of the matrix, adding the corresponding degree of freedom portion.

After the processing of eq. (8), the support reactions (*VX* and *VY*) were equal at nodes 1 and 7, since the plane truss structure and the load's application are symmetrical. The results obtained by the FORTRAN code resulted in $VX = 116.67kN$ and $VY = 150.00kN$ at the support points. As for the axial forces in the bars, the results obtained were compared with the software MASTAN2, which has its formulation based on McGuire et al. [5]. Table 2 shows the results of both analyses for the Warren truss. Note that the results obtained were identical in both analyses, except for the diagonal bars 2, 3, 9, and 11. There was a divergence of only 0.02%.

Table 2. Axial forces of the plane truss bars

Member number	FORTRAN (kN)	MASTAN2 (kN)	Member number	FORTRAN (kN)	MASTAN2 (kN)
1	-29.17	-29.17	7	0.00	0.00
2	-173.66	-173.70	8	-175.00	-175.00
3	173.66	173.70	9	173.66	173.70
4	0.00	0.00	10	-29.17	-29.17
5	-175.00	-175.00	11	-173.66	-173.70
6	58.33	58.33	-	-	-

3.2 Space Truss

The proposed space truss was a 25-bar electricity transmission tower, presented by Ribas [6] (Fig. 5). The structure has elastic modulus $E = 210GPa$, cross-sectional area of the bars $A = 0.0016m^2$, height $H = 5m$, and length $L = 1.9m$. Motion constraints were applied in the three main directions ($X, Y,$ *and* Z). As for the loading, those present in Tab. 3 were considered.

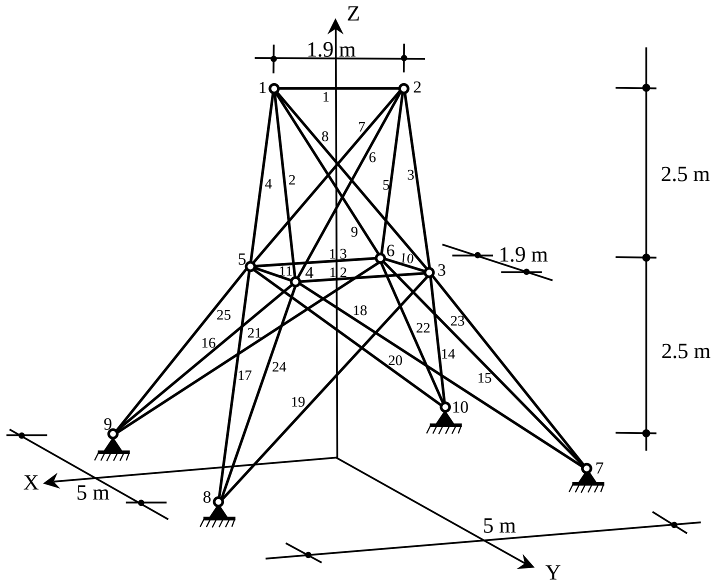

Figure 5. Electricity transmission tower

From the plane truss formulation, the vectors and matrices must be expanded in the computational routine, as shown in the 3D truss formulation in Chapter 2. Furthermore, it should be noted that for allocation in the global stiffness matrix, the "*do*" loop must be modified from the value 4 to 6, because each space truss bar has 6 degrees of freedom. The results obtained in the numerical method were also compared with the software MASTAN2, as shown in Tab. 4. It can be seen that the values obtained in the support nodes are identical.

Finally, Tab. 5 presents the axial forces in each bar of the electricity transmission tower. The largest variation found between the FORTRAN and MASTAN2 values was 0.03%, thus verifying the results obtained.

Table 3. Total loads of the electricity transmission tower (*kN*)

Nós	X	Y	Z
1	4.48	135.57	-44.84
2	0.00	135.57	-44.84
3	2.24	0.00	0.00
6	2.24	0.00	0.00

Table 4. Support reactions of the space truss

	FORTRAN (kN)			MASTAN2 (kN)		
Nós	X	Y	Z	X	Y	Z
7	131.82	-82.47	154.63	131.82	-82.47	154.63
8	-136.31	-87.86	161.35	-136.31	-87.86	161.35
9	96.41	-47.71	-109.79	96.41	-47.71	-109.79
10	-100.89	-53.10	-116.51	-100.89	-53.10	-116.51

Table 5. Axial forces of the space truss bars

Member number	FORTRAN (kN)	MASTAN2 (kN)	Member number	FORTRAN (kN)	MASTAN2 (kN)
1	8.60	8.60	14	-45.66	-45.66
2	-91.29	-91.29	15	31.98	31.98
3	-87.42	-87.42	16	-48.59	-48.59
4	68.71	68.71	17	29.05	29.05
5	72.57	72.57	18	-85.79	-85.79
6	-143.54	-143.54	19	-86.54	-86.54
7	107.68	107.70	20	68.38	68.38
8	-140.39	-140.40	21	67.63	67.63
9	110.83	110.80	22	138.29	138.30
10	2.87	2.87	23	-166.19	-166.20
11	4.69	4.69	24	-172.43	-172.40
12	21.16	21.16	25	132.06	132.10
13	-20.33	-20.33	-	-	-

4 Conclusions

The diffusion of knowledge involving linear static analysis, through the finite element method of truss structures, is extremely relevant for academic students and engineers. However, in order to solve problems of large variables or various complexities, it is recommended to implement FEM numerically to optimize time and process.

Thus, this work developed a code in FORTRAN language using the FEM formulation for truss bars. The computational routines were applied to two problems: a plane truss and a space truss. In summary, the results obtained were satisfactory in both problems, with a maximum variation of 0.03% in relation to MASTAN2.

Acknowledgements. The authors are grateful for the financial support provided by CAPES and CNPq.

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] H. L. Soriano. *Método de Elementos Finitos em Análise de Estruturas Vol. 48*. Edusp, 2003.
[2] L. F. Martha. *Análise matricial de estruturas com orientação a objetos*. Elsevier Brasil, 2018.
[3] A. A. Filho. *Elementos Finitos–A base da tecnologia CAE*. Saraiva Educação SA, 2018.
[4] H. L. Soriano. *Análise de Estruturas: Formulação e Implementação Computacional*. Ciência Moderna, 2005.
[5] W. McGuire, R. H. Gallagher, and R. D. Ziemian. *Matrix Structural Analysis*, 2014.
[6] H. S. Ribas. *Projeto Ótimo da Geometria de Treliças Espaciais*. Master's thesis, Federal University of Rio Grande do Sul, 1980.