



AN EDUCATIONAL COMPUTER PROGRAM FOR MATRIX ANALYSIS OF PLANE TRUSSES IN CIVIL ENGINEERING

Myriam Rocío Pallares M.¹, Wilson Rodríguez Calderón² and Duvan Edinson Gonzalez García¹

¹Civil Engineering Program, Faculty of Engineering, Surcolombiana University, Colombia

²Civil Engineering Program, Faculty of Engineering, Universidad Cooperativa de Colombia, Colombia

E-Mail: myriam.pallares@usco.edu.co

ABSTRACT

An educational computer program called Armadura2D was developed for matrix analysis of plane trusses, which is intended for engineering students. The program has a great advantage over commercial software, since it is designed to assist step-by-step matrix analysis of plane trusses calculation; this is essential for the lessons of the Structure Analysis course, in the civil engineering undergraduate program. This paper provides a conceptual and mathematical model, a pseudocode of the computational model developed in FORTRAN, the verification of the accuracy of the computer program with a classic problem extracted from a classical book [1] and results are compared with Ansys® simulation software. In conclusion, this application is an alternative support that enhances the traditional teaching-learning process inside classrooms, and the autonomous learning of civil engineering students of the matrix analysis, particularly for plane trusses.

Keywords: matrix structural analysis, plane trusses, Ansys®, ICT in education.

1. INTRODUCTION

The structural analysis methods are based on calculation techniques with matrices; therefore, the matrix analysis is essential for applying theoretical knowledge and numerical analysis in a clear and organized way. For that reason, the computer program developed allows students to support their own learning process, to improve the philosophy knowledge of the structural analysis and to reduce the amount of work put into the calculation of trusses in two dimensions. [2]–[4]

The goal of this work is to develop a FORTRAN code based on matrix methods. Software obtained will be used by civil engineer students who will be able to apply the theoretical knowledge and numerical analysis techniques to two-dimensional truss study.

The goal of this work is to develop a FORTRAN code based on matrix methods for civil engineer students use, applying the theoretical knowledge and numerical analysis techniques to two-dimensional truss study. [5]–[8].

It is uncommon to find computer programs that assist the step-by-step calculation of trusses using the matrix method; however, this application is useful, coherent and efficient, since it displays the matrices and operations executed. Therefore, this developing deserves disclosure in the academic community.

The Structural Mechanics training is complex due to the magnitude of the problems, the variables that define them, the mathematical theories around the subject and the characteristics of the mechanical performance; consequently, to manually solve structures is obsolete, thus it is important to support the Structural Mechanics education with a calculation computer program. [9], [10]. This educational application allows student-professor interactions, and it is suitable for in-person classes and for autonomous learning of the students.

2. METHODS

2.1 Conceptual model of the matrix analysis of two-dimensional trusses[1]

When an engineering problem is established, and it is required to start the modeling using mathematical and numerical models, it is necessary to define a conceptual model that solves fundamental questions about the analysis; hence, this step is strategic and vital to orientate the modeling process of engineering systems.

According to this, the questions solved shown from Figure-1 to Figure-4 are applied for the case of matrix analysis of plane trusses, and they describe, using the conceptual method, the problem in a general approach.

2.2 Elemental stiffness matrix for a bar element with a global coordinate system[1]

Due to the fact that trusses elements can be arranged in an arbitrary orientation (elements with an arbitrarily oriented local system), it becomes necessary to develop an elemental stiffness matrix with a global coordinate system (unique coordinate system that governs the structure, where commonly, the X axis and Y axis are considered horizontal and vertical, respectively). Thereby, when expressing this elemental matrix with a unique common global system of a truss, each one can be added to a process called assembly, in order to easily obtain the total stiffness matrix of the truss with a global coordinate system.

A general truss element is shown in Figure-5, in which the local principal axis \bar{X} has an angle ϕ with the X axis of reference system, which is called the global system of the structure or general axis system. The bar illustrates the forces and displacements related to the local axis system or elemental axis, and a macron above the letters of the axis differs them from the global system axis.

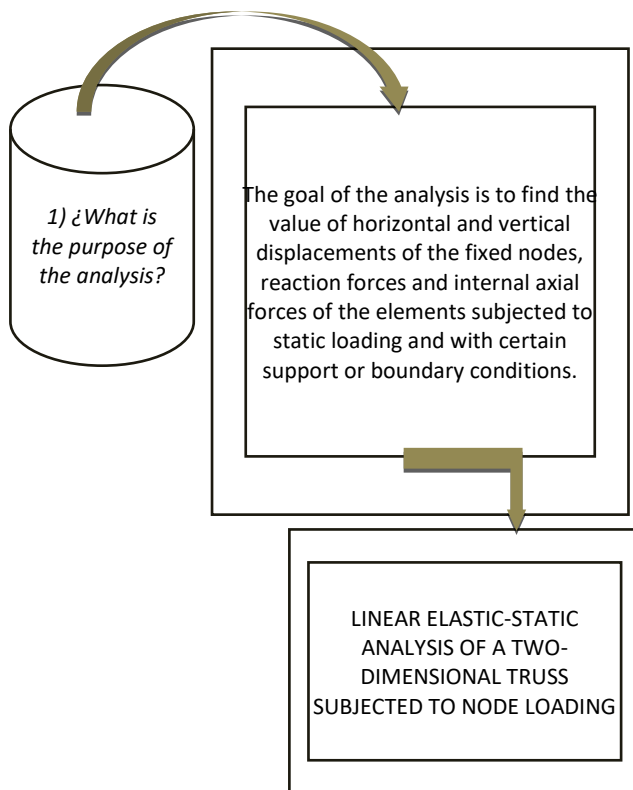


Figure-1. Question 1 for the conceptual model.

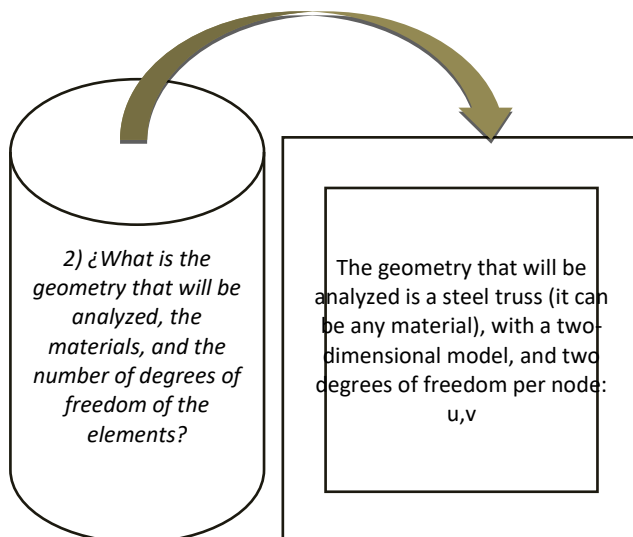


Figure-2. Question 2 for the conceptual model.

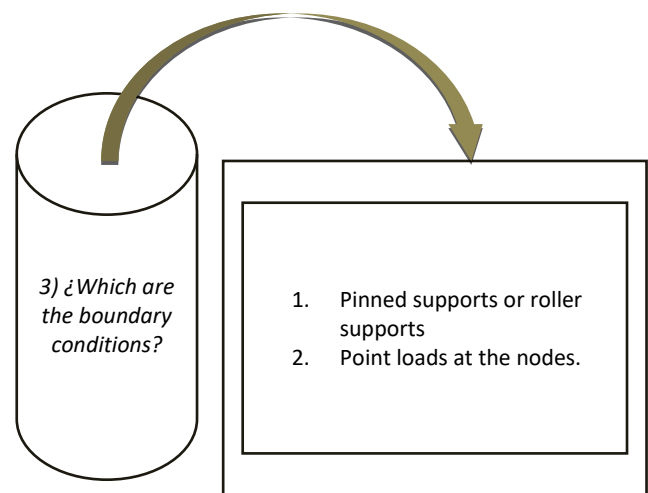


Figure-3. Question 3 for the conceptual model.

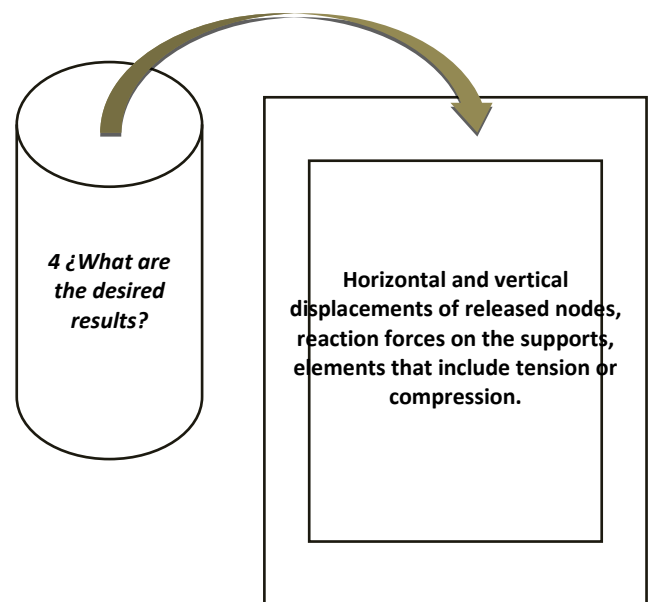


Figure-4. Question 4 for the conceptual model.

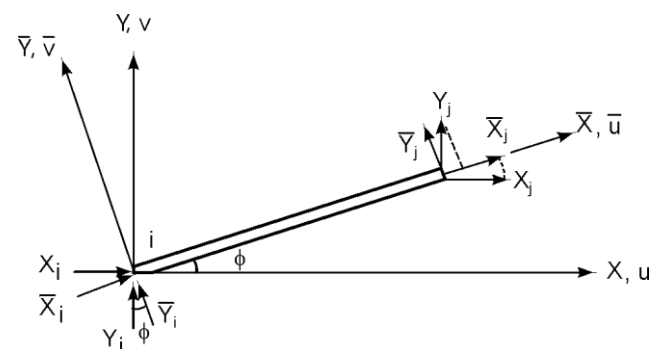


Figure-5. Bar element arbitrarily oriented with a global reference system X-Y[1].

The elemental stiffness matrix with local coordinates and expanded to size 4x4 is expressed as shown in equation (1).



$$[\bar{K}] = \frac{AE}{L} \begin{bmatrix} 1 & 0 & -1 & 0 \\ 0 & 0 & 0 & 0 \\ -1 & 0 & 1 & 0 \\ 0 & 0 & 0 & 0 \end{bmatrix} \quad (1)$$

However, it is necessary to establish the matrix of the equation (1) in global coordinates using a transformation matrix, which oversees calculating the projections of forces and displacements from the local system to the global system. The transformation matrix is expressed in terms of sine and cosine of the angle as shown in equation (2).

$$[T] = \begin{bmatrix} c & s & 0 & 0 \\ -s & c & 0 & 0 \\ 0 & 0 & c & s \\ 0 & 0 & -s & c \end{bmatrix} \quad (2)$$

Where $c = \cos(\phi)$, and $s = \sin(\phi)$.

The elemental stiffness matrix with global coordinates can be found through the matrix operations shown in equation (3),

$$[K] = [T]^T [\bar{K}] [T] \quad (3)$$

Substituting the equation (1) and (2) into (3), the elemental stiffness matrix with global coordinates is obtained as shown in equation (4).

$$[K] = \frac{AE}{L} \begin{bmatrix} c^2 & cs & -c^2 & -cs \\ cs & s^2 & -cs & -s^2 \\ -c^2 & -cs & c^2 & cs \\ -cs & -s^2 & cs & s^2 \end{bmatrix} \quad (4)$$

2.3 Internal forces of a plane truss element [1]

The transformations between the local and global systems allow obtaining the internal tension forces (positive) or the compression forces (negative) of the elements. The matrix operation is shown in equation (5)

$$S_{ij} = \frac{AE}{L} \begin{bmatrix} c & s & 0 & 0 \\ -s & c & 0 & 0 \\ 0 & 0 & c & s \\ 0 & 0 & -s & c \end{bmatrix} \begin{bmatrix} u_i \\ v_i \\ u_j \\ v_j \end{bmatrix} \quad (5)$$

The computer calculation operates the relative displacements of the nodes of the bar as shown in equation (6)

$$S_{ij} = \frac{AE}{L} [c \ s]_{ij} \begin{bmatrix} u_j & -u_i \\ v_j & -v_i \end{bmatrix} \quad (6)$$

2.4 Stiffness or displacement method general solution [1]

From equation (4), an assembly process is executed, achieving an overlap of the elemental matrices that were expanded to the size of the global matrix of the structure, in order to obtain a system of linear equations, in

which the displacement and force boundary conditions are imposed.

$$\begin{Bmatrix} F_n \\ F_a \end{Bmatrix} = \begin{bmatrix} K_{nn} & K_{na} \\ K_{an} & K_{aa} \end{bmatrix} \begin{Bmatrix} \delta_n \\ \delta_a \end{Bmatrix} \quad (7)$$

As shown in equation (7), the primary variables are the displacements, or the released degrees of freedom δ_n , and the secondary variables are the reaction forces F_a on the fixed degrees of freedom. Whenever support displacements are zero, $\delta_a = 0$, the displacements and reaction forces are obtained as shown in equation (8) and (9)

$$[\delta_n] = [K_{nn}]^{-1} [F_n] \quad (8)$$

$$[F_a] = [K_{an}] [K_{nn}]^{-1} [F_n] \quad (9)$$

2.5 Pseudocode for the matrix method of plane trusses using FORTRAN

The algorithm of the calculation developed using FORTRAN is shown in Figure-6 as a pseudocode.

do	START
	i=1, number of elements
(1)	Calculation of characteristic terms of element i and elemental matrix determination of element i using global coordinates system
enddo	
do	i=1, number of elements
(2)	Global structural matrix assembly using the elemental matrices calculated and matrix preparation for nodal displacement and reaction forces calculation
enddo	
(3)	Nodal forces placement into the global structural system and incorporation of the outline and displacement conditions
(4)	Solution of the linear system of equations for the determination of unrestricted nodal displacements
(5)	Determination of the reaction forces of the structure
do	i=1, number of elements
(6)	Calculation of the internal force of the element i.
	END

Figure-6. Pseudocode using FORTRAN.

2.6 Computational modeling using Ansys®

A step-by-step calculation of an example proposed by the professor Jairo Uribe Escamilla was executed as shown in Figure-7.

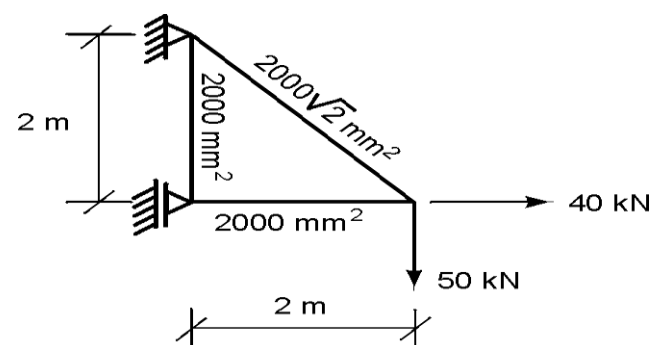


Figure-7. Reference example – Geometry [1].



The elastic modulus E of the elements is arbitrary; hence the displacements are divided by the magnitude of this parameter. The connections, node enumeration and node forces are shown in Figure-8.

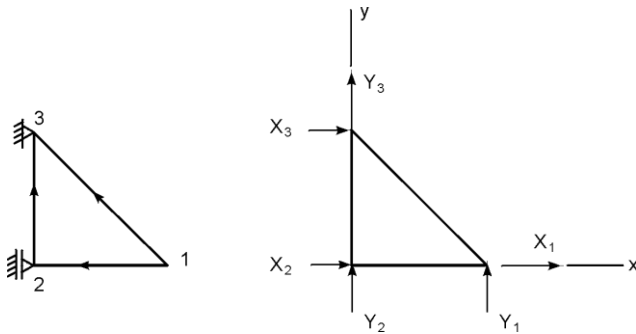


Figure-8. Reference example - Connections [1].

The Ansys® model is performed with the LINK1 element that is used to represent two-dimensional bar elements, which can support an axial compressive load or an axial tensile load. The model is a linear element with two nodes; each one has two degrees of freedom: the translations of the nodal directions X and Y . The element is defined by the two nodes, the cross-sectional area, and the material properties.

The finite element model using Ansys® is shown in Figure-9.

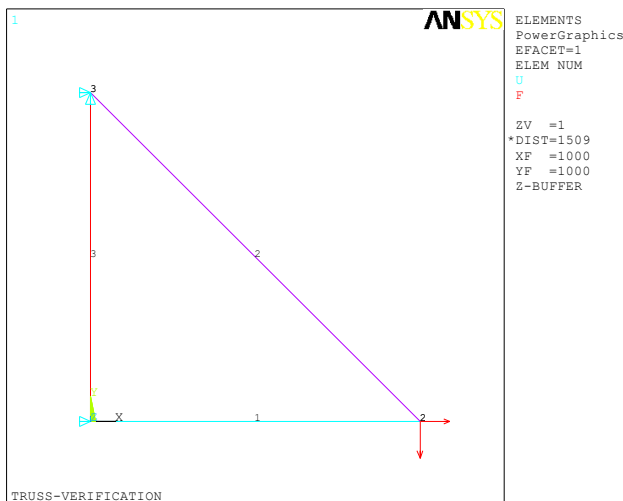


Figure-9. Geometric model using Ansys®

3. RESULTS AND DISCUSSIONS

The results of the educational computer program Armadura2D were compared with the results of the references from (2.4), taking benefit from the step-by-step calculation shown, since it displays every result obtained from the classic algorithm used in matrix analysis of trusses. This characteristic gives a didactic approach to the program computer, unlike commercial software that do not display internal calculation, resulting in a black box metaphor for the user.

The Ansys® model was developed to demonstrate that the results obtained from known finite elements software equalize a classic calculation technique with a systematic and accurate implementation of a computer program based on matrix operations equalizes, turning it into a trusted alternative for individual or classroom use.

The initial input data of the Armadura2D computer program is shown in Figure-10. The connections of the problem example [1] were used to compare the results.

```
C:\arm>armadura2d

Software educativo para armaduras 2D
Autores: Myriam R. Pallares (USCO) - Wilson Rodriguez C. (UCC)

numero de nodos=3
numero de elementos=3
Modulo Elastico de Referencia Er=1

-----****-----

x( 1 )=2000
y( 1 )=0
x( 2 )=0
y( 2 )=0
x( 3 )=0
y( 3 )=2000

Barra[ 1 ]
nodo inicial( 1 )=1
nodo final( 1 )=2
Area de la seccion( 1 )=2000
Relacion de modulos Eb/Er( 1 )=1

Barra[ 2 ]
nodo inicial( 2 )=1
nodo final( 2 )=3
Area de la seccion( 2 )=2828.4271247461900976033774484194
Relacion de modulos Eb/Er( 2 )=1

Barra[ 3 ]
nodo inicial( 3 )=2
nodo final( 3 )=3
Area de la seccion( 3 )=2000
Relacion de modulos Eb/Er( 3 )=1
```

Figure-10. Initial input data - Armadura2D.

The detailed calculation of bar 2 is shown in Figure-11 as a list, these results are the input for calculation of the elemental stiffness matrix of the bar using global coordinates.

```
Barra[ 2 ]
nodo inicial( 2 )= 1
nodo final( 2 )= 3
Area
2.828E+03
Eb
1.000E+00
L
2.828E+03
K
1.000E+00
f(grad)
1.350E+02

u[ 1 ] v[ 1 ] u[ 3 ] v[ 3 ]
5.000E-01 -5.000E-01 -5.000E-01 5.000E-01
-5.000E-01 5.000E-01 5.000E-01 -5.000E-01
-5.000E-01 5.000E-01 5.000E-01 -5.000E-01
5.000E-01 -5.000E-01 -5.000E-01 5.000E-01

cosf senf cos2f sen2f cosfsenf
-7.071E-01 7.071E-01 5.000E-01 5.000E-01 -5.000E-01
```

Figure-11. Stiffness matrix of bar 2 using global coordinates - Armadura2D.

Additionally, the results are shown in the reference example [1], trying to prove the equivalency of the results obtained step-by-step, and the fulfillment of the



goal, which is to assist the gradually calculation of plane trusses using the matrix method.

The Table shown in Figure-12 displays an input data summary of the element 1-3 (bar 2) and the elemental stiffness matrix, the results match accurately.

Element	ϕ	$\cos \phi$	$\sin \phi$	$\cos^2 \phi$	$\sin^2 \phi$	$\cos \phi \sin \phi$	A/L (mm)
1-2	180°	-1	0	1	0	0	1
1-3	135°	$-\sqrt{2}/2$	$\sqrt{2}/2$	1/2	1/2	-1/2	1
2-3	90°	0	1	0	1	0	1

$$[K]_{1-3} = E \begin{bmatrix} u_1 & v_1 & u_3 & v_3 \\ 1/2 & -1/2 & -1/2 & 1/2 \\ -1/2 & 1/2 & 1/2 & -1/2 \\ -1/2 & 1/2 & 1/2 & -1/2 \\ 1/2 & -1/2 & -1/2 & 1/2 \end{bmatrix} \begin{matrix} u_1 \\ v_1 \\ u_3 \\ v_3 \end{matrix}$$

Figure-12. Stiffness matrix of the bar 2 using global coordinates. Solution of the reference example [1].

The computer program displays the assembly shown in Figure-13. The truss matrix, or assembled matrix, is provided by a two-column list with the degrees of freedom associated with each record.

```

===== Matriz de Rigidez Global de la Armadura =====

u[ 1 ]      v[ 1 ]
1.500E+00    -5.000E-01
-5.000E-01    5.000E-01
-1.000E+00    1.225E-16
1.225E-16    -1.500E-32
-5.000E-01    5.000E-01
5.000E-01    -5.000E-01

u[ 2 ]      v[ 2 ]
-1.000E+00    1.225E-16
1.225E-16    -1.500E-32
1.000E+00    -6.123E-17
-6.123E-17    1.000E+00
-3.749E-33    -6.123E-17
-6.123E-17    -1.000E+00

u[ 3 ]      v[ 3 ]
-5.000E-01    5.000E-01
5.000E-01    -5.000E-01
-3.749E-33    -6.123E-17
-6.123E-17    1.000E+00
5.000E-01    -5.000E-01
-5.000E-01    1.500E+00

```

Figure-13. Assembled stiffness matrix - Armadura2D.

The results shown in Figure-13 are compared with the example solution shown in Figure-14, in order to verify the proper functioning of the computer program when the global stiffness matrix of the structure is assembled.

$$K = E \begin{bmatrix} u_1 & v_1 & u_2 & v_2 & u_3 & v_3 \\ 1+1/2 & -1/2 & -1 & & -1/2 & 1/2 \\ -1/2 & 1/2 & & & 1/2 & -1/2 \\ -1 & & 1 & & & \\ & & & 1 & & -1 \\ -1/2 & 1/2 & & & 1/2 & -1/2 \\ 1/2 & -1/2 & & -1 & -1/2 & 1+1/2 \end{bmatrix}$$

Figure-14. Assembled stiffness matrix using global coordinates. Solution of example[1].

The displacement and force boundary conditions are set as shown in Figure-15. The tag used to indicate the X direction is the number 1; therefore, the Y direction is tagged with the number 2.

```

===== Condiciones de Apoyo - dir X=1 , dir Y=2 =====

numero de desplazamientos fijos=3
nodo restringido( 1 )=3
direccion de restriccion( 1 )=1
    Desplazamiento x restringido para el nodo No 3
nodo restringido( 2 )=3
direccion de restriccion( 2 )=2
    Desplazamiento y restringido para el nodo No 3
nodo restringido( 3 )=2
direccion de restriccion( 3 )=1
    Desplazamiento x restringido para el nodo No 2

===== Cargas aplicadas - dir X=1 , dir Y=2 =====

numero de fuerzas aplicadas=2
nodo cargado( 1 )=1
direccion de carga( 1 )=1
Fuerza aplicada( 1 )=40
    Fuerza en x para el nodo No ( 1 ) *** [valor= 4.000E+01 ]
nodo cargado( 2 )=1
direccion de carga( 2 )=2
Fuerza aplicada( 2 )=-50
    Fuerza en y para el nodo No ( 1 ) *** [valor= -5.000E+01 ]

```

Figure-15. Boundary conditions set- Armadura2D.

Finally, the results of the displacements, reaction forces and internal forces obtained using the educational computer program are shown in Figure-16.

```

***** RESULTADOS OBTENIDOS *****

Calculo de Desplazamientos
Nodo   [ x ]      [ y ]
1      -1.000000E+01  -1.100000E+02
2       0.000000E+00   1.224647E-15
3       0.000000E+00   0.000000E+00

Resultados de Reacciones-Fuerzas Aplicadas
Nodo   [ x ]      [ y ]
1      4.000000E+01  -5.000000E+01
2      1.000000E+01  -1.577722E-30
3     -5.000000E+01  5.000000E+01

Calculo de Fuerzas Internas
Fint[ 1 ] = 1.000E+00*( -1.000E+00 1.225E-16)*
Fint[ 2 ] = 1.000E+00*( -7.071E-01 7.071E-01)*
Fint[ 3 ] = 1.000E+00*( 6.123E-17 1.000E+00)*

( 1.000E+01 1.100E+02)T= -1.00000E+01 Compresion
( 1.000E+01 1.100E+02)T= 7.07107E+01 Tension
( 0.000E+00 -1.225E-15)T= -1.22465E-15 Compresion

```

Figure-16. Displacements, reaction forces and internal forces - Armadura2D.



The results of the displacements, reaction forces and internal forces of the reference example [1] are shown in Figure-17.

$$\begin{bmatrix} u_1 \\ v_1 \\ v_2 \end{bmatrix} = \begin{bmatrix} -10/E \\ -110/E \\ 0 \end{bmatrix}$$

$$\begin{bmatrix} X_2 \\ X_3 \\ Y_3 \end{bmatrix} = \begin{bmatrix} 10\text{kN} \rightarrow \\ -50\text{kN} \leftarrow \\ 50\text{kN} \uparrow \end{bmatrix}$$

$$S_{1-2} = E \begin{bmatrix} -1 & 0 \end{bmatrix} \begin{bmatrix} 0 + 10/E \\ 0 + 110/E \end{bmatrix} = -10\text{kN} \quad (C)$$

$$S_{1-3} = E \begin{bmatrix} \frac{-\sqrt{2}}{2} & \frac{\sqrt{2}}{2} \end{bmatrix} \begin{bmatrix} 0 + 10/E \\ 0 + 110/E \end{bmatrix} = 50\sqrt{2} \text{ kN} \quad (T)$$

$$S_{2-3} = E \begin{bmatrix} 0 & 1 \end{bmatrix} \begin{bmatrix} 0 - 0 \\ 0 - 0 \end{bmatrix} = 0$$

Figure-17. Displacements, reaction forces and internal forces –Solution of example[1].

The results of the displacements and internal forces obtained using Ansys® are used to verify the accuracy of the computer program as shown from Figures 18 to 20.

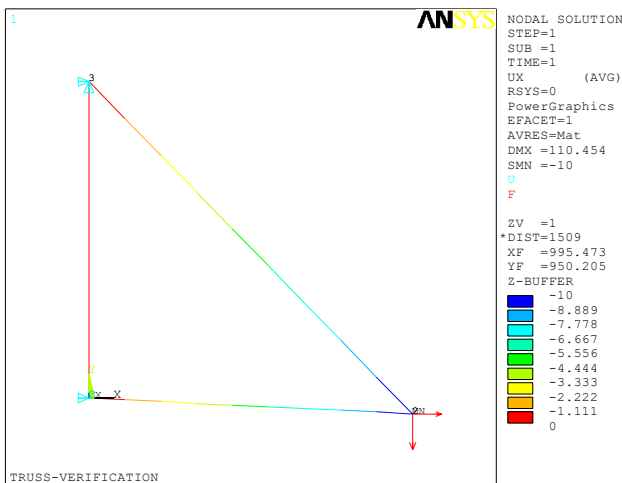


Figure-18. X axis displacements. Ansys®.

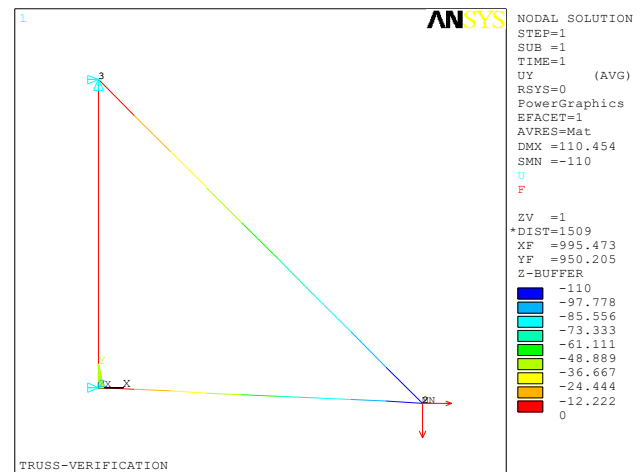


Figure-19. Y axis displacements. Ansys®.

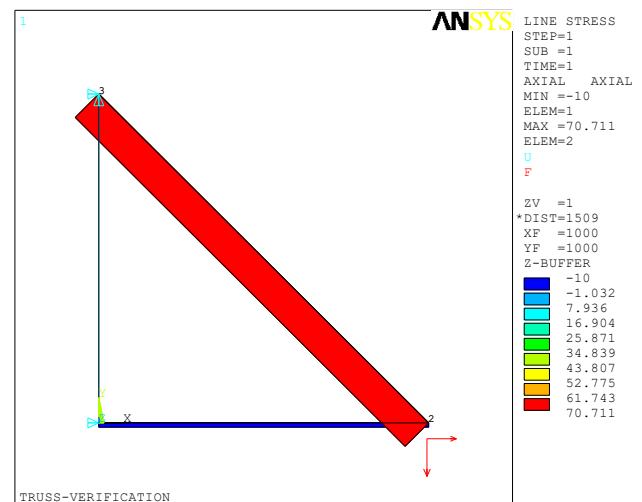


Figure-20. Internal forces. Ansys®.

The results of the horizontal and vertical displacements obtained using the three alternatives are shown in Table-1. The calculations were made using the elastic modulus; hence the results are a function of this input. Small value results are assumed as zero.

Table-1. Horizontal and vertical displacements (mm).

	Armadura2d	Reference[1]	Ansys®
U1	-10/E	-10/E	-10/E
V1	-110/E	-110/E	-110/E
V2	0	0	0

The results of the elemental internal forces obtained using the three alternatives are shown in Table-2.

**Table-2.** Internal forces (kN).

	Armadura2d	Reference[1]	Ansys®
F[1-2]	-10.000	-10	-10.000
F[1-3]	70.711	70.711	70.711
F[2-3]	0	0	0

The main results are the same for the three alternatives; therefore, the accuracy of the computer program is evidenced.

4. CONCLUSIONS

- This educational computer program used for matrix analysis of trusses is helpful, since the assisted computer calculation allows the student to comprehend the operation process and to evaluate the obtained results.
- The verified results show the versatility of the computer program to support the calculation process of plane trusses using the matrix method, and to comprehend the concepts related to the technique. The obtained results verify the accurate performance of the computer program developed.
- The computer program displays the results and generates a detailed report on a text file that allows saving the calculation memories.
- It is of great importance for professors and students to be cautious with the accuracy of the magnitude of the results; they must be compared with the values of the input data, in order to consider them as reliable.
- The great advantage of the educational computer program is its adaptation in the teaching-learning process, since it can be improved and modified to different case studies, inside the classroom or as portion of time spent on the autonomous student learning.
- Solving structural engineering problems of great importance is possible, without the expensive use of commercial software of finite elements.

ACKNOWLEDGMENT

We thank the sponsoring universities, specially to Surcolombiana University for the support given to this research (Project number 3166). This paper is the result of one of the researches developed in the "INGENIERIA Y SURDESARROLLO" group led by the first author.

REFERENCES

- [1] J. Escamilla, Análisis de estructuras / J. Uribe Escamilla. 2002.
- [2] T. H. G. Megson. 2007. Matrix Methods of Analysis. en Structural and Stress Analysis
- [3] M. Petyt. 2003. Theory of matrix structural analysis. J. Sound Vib.
- [4] A. Tena-Colunga. 2007. Análisis de Estructuras con Métodos Matriciales. México: Gedisa.
- [5] M. Paz, W. Leigh, M. Paz, y W. Leigh. 2011. Static Condensation and Substructuring. en Integrated Matrix Analysis of Structures.
- [6] W. McGuire, R. H. Gallagher, y H. Saunders. 1982. Matrix Structural Analysis. J. Mech. Des.
- [7] J. S. Przemieniecki. 2012. Finite Element Structural Analysis: New Concepts.
- [8] H. H. West. 1993. Fundamentals of Structural Analysis. Eur. J. Eng. Educ.
- [9] S. Przemieniecki J. 1969. Theory of matrix structural analysis.
- [10] R. H. Gallagher. 1975. Finite Element Analysis Fundamentals. Englewood Cliffs: Prentice-Hall.