ARPN Journal of Engineering and Applied Sciences

www.arpnjournals.com

EDUCATIONAL COMPUTER PROGRAM FOR THE MATRIX ANALYSIS OF PLANE FRAMES

Myriam Rocío Pallares Muñoz and Rosa Alejandra Méndez
Faculty of Engineering, Universidad Surcolombiana, Colombia
E-Mail: myriam.pallares@usco.edu.co

ABSTRACT

An educational computer program was developed for the matrix analysis of plane frames aimed at engineering students. Unlike commercial programs, the application has a plus since it is designed to assist in the step-by-step calculation of matrix analysis of plane frames subject to point loads and uniformly distributed loads perpendicular to the elements. This topic is fundamental in the teaching of the structural analysis course in engineering careers. The conceptual and mathematical model, the pseudocode of the computational model in FORTRAN, the verification of the application with SAP2000 using a classic problem from the literature are shown, concluding that the tool developed is an alternative for assisting the teaching-learning process in the classroom and for the autonomous work of engineering students in studying of the plane-frame matrix analysis method. This educational program is part of an educational-toolkit project developing to improve students' autonomous work and teacher-student communication [1].

Keywords: matrix analysis of structures, plane frames, SAP2000, educational computer program.

INTRODUCTION

Structural analysis methods are based on matrix calculation techniques, so matrix analysis of structures is fundamental for applying the theoretical bases and numerical calculation procedures in a clear and orderly way. The matrix analysis method is suited for the computer-analysis of complex structures. This method employs the members' stiffness relations for computing member forces and displacements in structures-this method is the basis for the finite element method. In the matrix analysis method, the system must be modelled as a set of elements interconnected at the nodes. The material stiffness properties are assembled into a single matrix equation that governs the entire idealized structure's behaviour. The structure's unknown displacements and forces can after be determined by solving these equations [2].

The developed computer program allows the student to support his own learning process, improve the understanding of structural analysis philosophy, and reduce the effort of calculating frames in two dimensions [1], [3]-[6]. This work aimed to develop a computer program in Fortran based on matrix methods so that the student can apply the theoretical principles and numerical calculation procedures in the analysis of two-dimensional frames [2], [7], [8].

In the scientific field, it is not common to find applications that assist the process of step-by-step calculation of frames by the matrix method. Therefore, this tool is useful, rational, and efficient because it shows the user the matrices and the operations that he executes since the results deserve to be disseminated in the academic community. Particularly, the teaching of structural mechanics is complex due to the magnitude of the problems, the variables that define them, the mathematical theories that govern the phenomenon, and the characteristics of mechanical performance, so solving a structure by hand is a task of the past, and hence the importance of supporting the teaching of structural mechanics with computer calculation tools [9], [10]. This educational program is part of an educational-toolkit project developing to improve students' autonomous work and teacher-student communication.

METHODS

Conceptual Model of the Matrix Analysis for Two-Dimensional Frames [9]

When a problem is formulated in engineering, and the first steps are taken for modeling using mathematical or numerical methods, it is necessary to define preliminarily a conceptual model that answers some fundamental analysis questions. This step leads the entire process of modeling systems and phenomena in engineering. For the matrix analysis of plane frames, the four answered questions are presented in Figures 1 to 4.

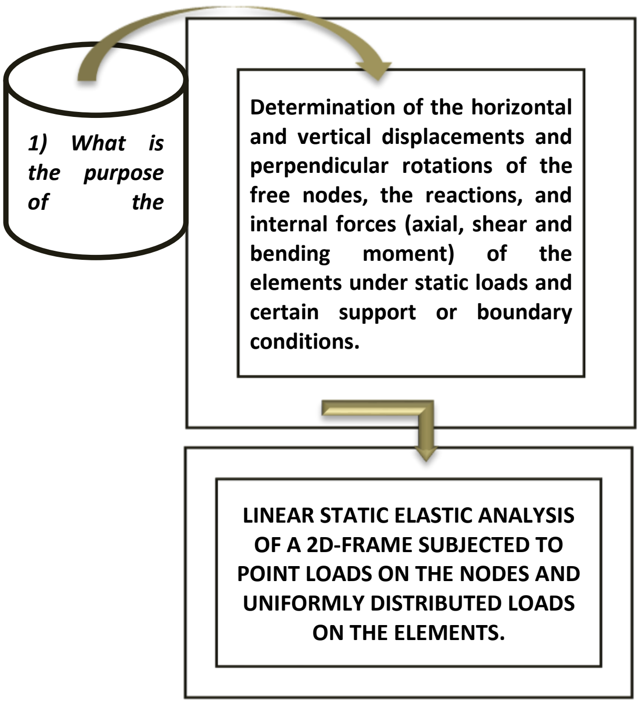

Figure-1. Question 1 of the conceptual model.

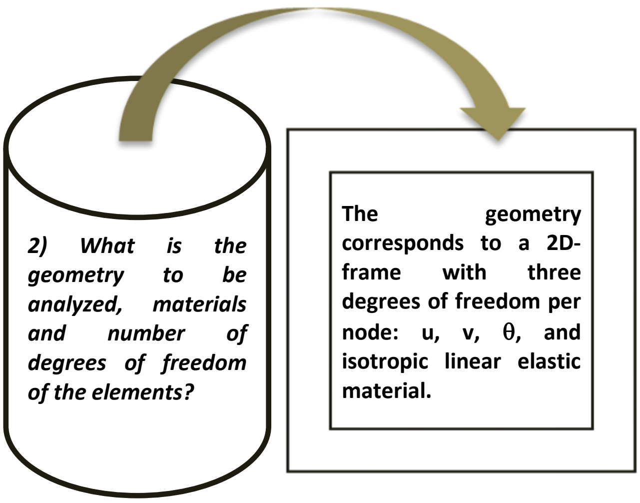

Figure-2. Question 2 of the conceptual model.

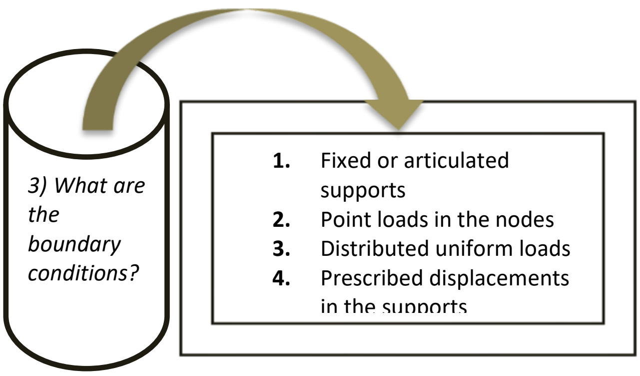

Figure-3. Question 3 of the conceptual model.

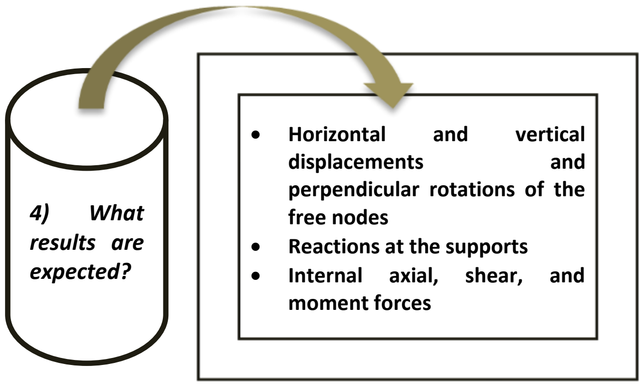

Figure-4. Question 4 of the conceptual model.

Elementary Stiffness Matrix of a Plane Frame Element in Global Coordinates [9]

Since frame elements, in general, can be arranged with arbitrary orientation (elements with an arbitrarily oriented local system), it becomes necessary to develop the elemental stiffness matrix in global coordinates (the unique coordinate system that governs the structure where commonly the X-axis is considered horizontal and the Y-axis vertical). Thus, as this elemental matrix is expressed under a single common global system, each elemental matrix can be added in an assembly process and thus obtained the frame's total stiffness matrix in global coordinates.

Figure-5 shows a general plane frame element, where the local axis $\overline{X}$ has an angle ϕ_x with the X-axis of a reference system (also called the global structure system or general axis system). The element shows the terms of force and displacement associated with the local-axes or elemental axes; these are distinguished from the global ones by having the upper mark in the notation.

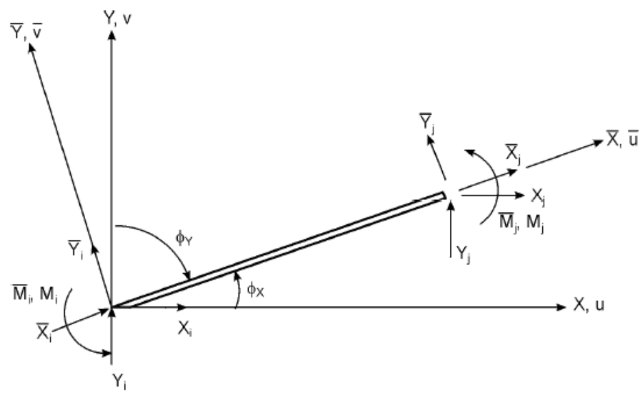

Figure-5. Plane Frame Element Arbitrarily Oriented Concerning the Global X-Y Reference System [9].

The elementary stiffness matrix in local coordinates of size 6x6 can be expressed, as shown in equation (1).

ARPN Journal of Engineering and Applied Sciences

$$[\bar{K}] = \begin{bmatrix} \frac{AE}{L} & & & & & \\ 0 & \frac{12EI}{L^3} & & & sym & \\ 0 & \frac{6EI}{L^2} & \frac{4EI}{L} & & & \\ -\frac{AE}{L} & 0 & 0 & \frac{AE}{L} & & \\ 0 & -\frac{12EI}{L^3} & -\frac{6EI}{L^2} & 0 & \frac{12EI}{L^3} & \\ 0 & \frac{6EI}{L^2} & \frac{2EI}{L} & 0 & -\frac{6EI}{L^2} & \frac{4EI}{L} \end{bmatrix} \quad (1)$$

However, it is necessary to establish the elemental stiffness matrix (1) in global coordinates using a transformation matrix (2), which is responsible for making 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 ϕ_x, as presented in Eq. (2). The elementary stiffness matrix in global coordinates can be found through the matrix operations described in equation (3).

$$[T] = \begin{bmatrix} \cos\phi_x & sen\phi_x & 0 & 0 & 0 & 0 \\ -sen\phi_x & \cos\phi_x & 0 & 0 & 0 & 0 \\ 0 & 0 & 1 & 0 & 0 & 0 \\ 0 & 0 & 0 & 0 & 0 & 0 \\ 0 & 0 & 0 & \cos\phi_x & sen\phi_x & 0 \\ 0 & 0 & 0 & -sen\phi_x & \cos\phi_x & 1 \end{bmatrix} \quad (2)$$

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

Replacing equation (1) and (2) in (3) gives the elementary stiffness matrix in global coordinates of equation (4), where $\lambda = \cos\phi_x$, $\mu = \sin\phi_x$.

$$[K] = \begin{array}{c} \begin{array}{cccccc} u_i & v_i & \theta_i & u_j & v_j & \theta_j \end{array} \\ \left[\begin{array}{ccc|ccc} \frac{AE}{L}\lambda^2 + \frac{12EI}{L^3}\mu^2 & & & & & \\ \left[\frac{AE}{L} - \frac{12EI}{L^3}\right]\lambda\mu & \frac{AE}{L}\mu^2 + \frac{12EI}{L^3}\lambda^2 & & sym & & \\ \frac{6EI}{L^2}\mu & \frac{6EI}{L^2}\lambda & \frac{4EI}{L} & & & \\ \hline -\left[\frac{AE}{L}\lambda^2 + \frac{12EI}{L^3}\mu^2\right] & -\left[\frac{AE}{L} - \frac{12EI}{L^3}\right]\lambda\mu & \frac{6EI}{L^2}\mu & \frac{AE}{L}\lambda^2 + \frac{12EI}{L^3}\mu^2 & & \\ -\left[\frac{AE}{L} - \frac{12EI}{L^3}\right]\lambda\mu & -\left[\frac{AE}{L}\mu^2 + \frac{12EI}{L^3}\lambda^2\right] & -\frac{6EI}{L^2}\lambda & \left[\frac{AE}{L} - \frac{12EI}{L^3}\right]\lambda\mu & \frac{AE}{L}\mu^2 + \frac{12EI}{L^3}\lambda^2 & \\ -\frac{6EI}{L^2}\mu & \frac{6EI}{L^2}\lambda & \frac{2EI}{L} & \frac{6EI}{L^2}\mu & -\frac{6EI}{L^2}\lambda & \frac{4EI}{L} \end{array}\right] \end{array} \quad (4)$$

Internal Forces of a Plane Frame Element [9]

The element internal forces are calculated from the transformations of the elementary displacements to the local system. Equations (5) and (6) show the corresponding matrix operations. In equation (6) $[\bar{F}^F]$ are the fixed fixed forces.

$$[\bar{\delta}] = [T][\delta] \quad (5)$$

$$[\bar{F}] = [\bar{F}^F] + [\bar{K}][\bar{\delta}] \quad (6)$$

General Solution by the Stiffness or Displacement Method [9]

From Eq. (4), the assembly or superimposition of the elementary matrices expanded to the structural global matrix size is carried out to obtain a linear system of equations on which the boundary conditions of displacement and force are imposed. The parcelled system is presented in equation (7).

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

In equation (7), the primary variables are the displacements or degrees of freedom *n*. The secondary variables are the reactions *Fa* in the restricted degrees of freedom. When the displacements are zero at the supports, then a=0, and the displacements and reactions are obtained from equations (8) and (9).

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

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

The assembly process carried out with the elemental matrices must also be carried out with the fixed forces that must be transformed into global coordinates for the assembly.

Pseudocode of the Plane-Frame Matrix Method with FORTRAN

The calculation algorithm developed in FORTRAN follows the pseudocode shown in Figure-6.

ARPN Journal of Engineering and Applied Sciences

Do	BEGIN *i*=1, number of elements
(1)	Calculation of characteristic terms of element *i* and calculation of the elemental matrix of element *i* in the global coordinate system
Enddo	
Do	*i*=1, number of elements
(2)	Global matrix assembly of the structure using the calculated elementary matrices and preparation of the matrix for the calculation of nodal displacements and reactions
Enddo	
(3)	Nodal forces in the structural global system and imposition of displacement boundary conditions
(4)	Fixed forces in the global coordinates system.
(5)	The solution of the linear system of equations for the determination of free nodal displacements
(6)	Determination of structure reactions
do	*i*=1, number of elements
(7)	The calculation of internal forces for each element *i*
Enddo	
	END

Figure-6. Pseudocode in FORTRAN.

Local Description and Signs of the Distributed Loads Applied in the Application

Two nodes define each element, an initial node *i* and an end node *j*, so that the way the element is created is decisive for identifying the sign of the applied elementary distributed loads. The local-axes or element axes are denoted with the $\bar{X}$ and $\bar{Y}$. They are defined so that the perpendicular Z-axis always be directed out of the XY plane. Following the right-hand rule and always coincide with the global Z-axis.

To describe exactly the positive local axes considered in the application of elementary distributed loads and, in general, for the interpretation of internal forces results of the elements, the four cases of the definition of the local axes according to the way the element is created (initial node *i* and final node j) are shown in Figure-7.

It is necessary to know which of the four quadrants applies to the case each time an element is created. The above is done to determine exactly the sign of the distributed load of the element. E.g., according to the quadrant where the element is located and how the element is defined: the distributed loads in Y-axis's positive direction are positive, and loads are negative in the opposite direction.

Convention on the Signs of the Application's Internal Forces

To correctly interpret the signs of the internal forces' results in the elements obtained from the computational application, the convention of signs represented in Figure-8 was considered. In this figure, the positive convention of internal forces for the initial node *i* and the final node *j* of the element analysed are shown.

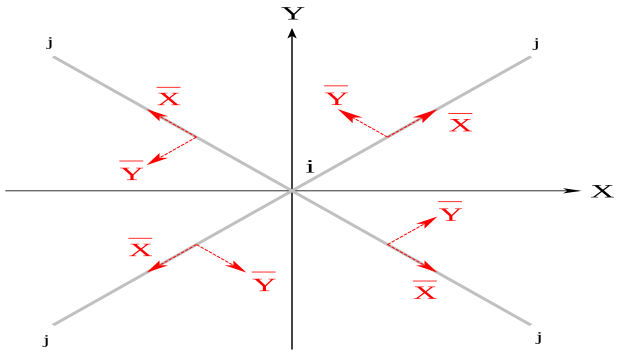

Figure-7. Local axes of the Marco2D application.

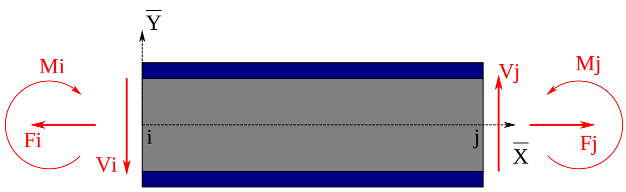

Figure-8. Convention of positive signs of internal forces - Marco2D application.

Computational Modeling in SAP2000

The application results are compared step-by-step with those of the example shown in Figure-9 [1]. Results also were verified with those of the SAP2000 software.

VOL. 16, NO. 6, MARCH 2021 ISSN 1819-6608
ARPN Journal of Engineering and Applied Sciences

www.arpnjournals.com

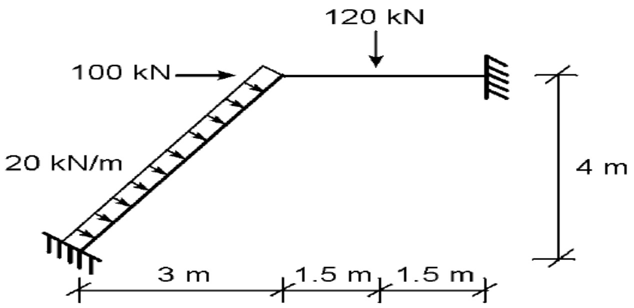

Figure-9. Reference problem - Geometry [9].

The elastic module of the elements is E=19 kN/mm^2. The connectivities, node numbering, and nodal forces are shown in Figure-10. According to these connectivities, the three elements of the frame are in the first quadrant, and as a result, the distributed load of element 3-1 is negative.

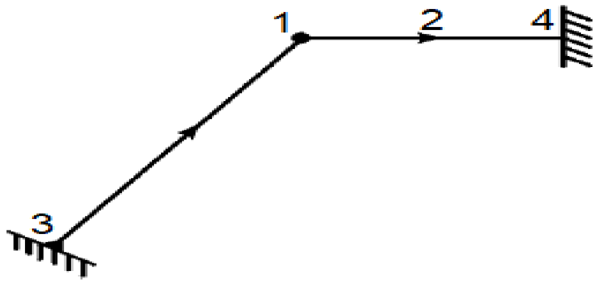

Figure-10. Reference problem - Connectivities [9].

Table-1 summarizes data for the framework's elements analyzed according to the reference [9].

Table-1. Summary of basic data.

Element	ϕ_x	λ	μ	AE/L	6EI/L^2	12EI/L^3	4EI/L	2EI/L
1-2	0°	1	0	570000	8550	5700	17100	8550
2-4	0°	1	0	570000	8550	5700	17100	8550
3-1	53.13°	0.6	0.8	456000	7300	2920	24320	12160

RESULTS AND DISCUSSIONS

The results of the Marco2D educational application were compared with the results of a model in SAP2000. Some steps are shown since the tool aims to show the classical plane-frame matrix analysis results step-by-step. This feature gives the tool a didactic meaning compared to commercial programs that are black boxes for the user. The model in SAP2000 was developed to demonstrate that a classical calculation methodology of systematic and reasonable implementation in a computer program based on matrix operations matches the results obtained from robust finite element tools, becoming a reliable option for individual and classroom use.

The finite element model in SAP2000 is shown in Figure-11, and the initial data entry of the Marco2D application is shown in Figure-12. The connectivities of the reference example [1] were used to compare the results.

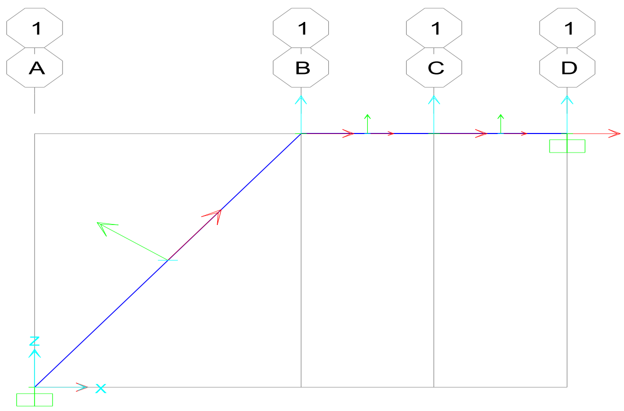

Figure-11. Geometric model SAP2000.

VOL. 16, NO. 6, MARCH 2021 ISSN 1819-6608
ARPN Journal of Engineering and Applied Sciences

www.arpnjournals.com

```
C:\>marco2drev2

 Software educativo para marcos 2D
 Autor: Myriam R. Pallares Muñoz (USCO)

 Numero de nodos=4
 Numero de elementos=3
 Modulo Elastico de Referencia Er=19e6

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

 x(  1  )=3
 y(  1  )=4
 x(  2  )=4.5
 y(  2  )=4
 x(  3  )=0
 y(  3  )=0
 x(  4  )=6
 y(  4  )=4


 Elemento[  1  ]
 Nodo inicial(  1  )=1
 Nodo final(  1  )=2
 Area de la seccion(  1  )=.09
 M. de Inercia de la seccion(  1  )=.000675
 Relacion de modulos Ee/Er(  1 )=1

 Elemento[  2  ]
 Nodo inicial(  2  )=2
 Nodo final(  2  )=4
 Area de la seccion(  2  )=.09
 M. de Inercia de la seccion(  2  )=.000675
 Relacion de modulos Ee/Er(  2 )=1

 Elemento[  3  ]
 Nodo inicial(  3  )=3
 Nodo final(  3  )=1
 Area de la seccion(  3  )=.12
 M. de Inercia de la seccion(  3  )=.0016
 Relacion de modulos Ee/Er(  3 )=1
```

Figure-12. Initial input data - Marco2D.

Figure-13 shows the elementary stiffness matrix in global coordinates of element 3-1.

```
Elemento[  3 ]
Nodo inicial(  3 )=  3
Nodo final(  3 )=  1
      Area           Ee            L            AE/L        f(grad)        cosf
   1.200E-01     1.900E+07     5.000E+00     4.560E+05     5.313E+01     6.000E-01
      I              EI          2EI/L          4EI/L        6EI/L^2      12EI/L^3
   1.600E-03     3.040E+04     1.216E+04     2.432E+04     7.296E+03     2.918E+03

   u[  3  ]      v[  3  ]      fi[  3  ]     u[  1  ]      v[  1  ]      fi[  1  ]
   1.660E+05     2.175E+05    -5.837E+03    -1.660E+05    -2.175E+05    -5.837E+03
   2.175E+05     2.929E+05     4.378E+03    -2.175E+05    -2.929E+05     4.378E+03
  -5.837E+03     4.378E+03     2.432E+04     5.837E+03    -4.378E+03     1.216E+04
  -1.660E+05    -2.175E+05     5.837E+03     1.660E+05     2.175E+05     5.837E+03
  -2.175E+05    -2.929E+05    -4.378E+03     2.175E+05     2.929E+05    -4.378E+03
  -5.837E+03     4.378E+03     1.216E+04     5.837E+03    -4.378E+03     2.432E+04
```

Figure-13. Element 3-1 stiffness matrix in global coordinates - Marco2D.

The application continues to present the assembly shown in Figure-14. The assembled matrix is reported by blocks of three columns labelled with the respectively associated degrees of freedom.

```
========= Matriz de Rigidez Global del Marco =========

      u[  1  ]        v[  1  ]        fi[  1  ]
     1.306E+06       2.175E+05       5.837E+03
     2.175E+05       3.385E+05       2.982E+04
     5.837E+03       2.982E+04       5.852E+04
    -1.140E+06       0.000E+00       0.000E+00
     0.000E+00      -4.560E+04      -3.420E+04
     0.000E+00       3.420E+04       1.710E+04
    -1.660E+05      -2.175E+05      -5.837E+03
    -2.175E+05      -2.929E+05       4.378E+03
     5.837E+03      -4.378E+03       1.216E+04
     0.000E+00       0.000E+00       0.000E+00
     0.000E+00       0.000E+00       0.000E+00
     0.000E+00       0.000E+00       0.000E+00

      u[  2  ]        v[  2  ]        fi[  2  ]
    -1.140E+06       0.000E+00       0.000E+00
     0.000E+00      -4.560E+04       3.420E+04
     0.000E+00      -3.420E+04       1.710E+04
     2.280E+06       0.000E+00       0.000E+00
     0.000E+00       9.120E+04       0.000E+00
     0.000E+00       0.000E+00       6.840E+04
     0.000E+00       0.000E+00       0.000E+00
     0.000E+00       0.000E+00       0.000E+00
     0.000E+00       0.000E+00       0.000E+00
    -1.140E+06       0.000E+00       0.000E+00
     0.000E+00      -4.560E+04      -3.420E+04
     0.000E+00       3.420E+04       1.710E+04
```

Figure-14. Assembled stiffness matrix - Marco2D.

Figures 15 and 16 show the imposition of the boundary conditions of displacements and forces. The label used to denote the X-direction is number 1; the number 2 is used for the Y-direction. For the ϕ-rotation, the number 3 is used.

```
= Condiciones de Apoyo - dir X=1 , dir Y=2, dir Fi=3 =

 Numero de desplazamientos prescritos=6
 Nodo restringido(  1  )=3
 Direccion de restriccion(  1  )=1
 Valor de la restriccion(  1  )=0
    Desplazamiento  x restringido para el nodo No.  3 *** valor=      0.000E+00
 Nodo restringido(  2  )=3
 Direccion de restriccion(  2  )=2
 Valor de la restriccion(  2  )=0
    Desplazamiento  y restringido para el nodo No.  3 *** valor=      0.000E+00
 Nodo restringido(  3  )=3
 Direccion de restriccion(  3  )=3
 Valor de la restriccion(  3  )=0
    Desplazamiento fi restringido para el nodo No.  3 *** valor=      0.000E+00
 Nodo restringido(  4  )=4
 Direccion de restriccion(  4  )=1
 Valor de la restriccion(  4  )=0
    Desplazamiento  x restringido para el nodo No.  4 *** valor=      0.000E+00
 Nodo restringido(  5  )=4
 Direccion de restriccion(  5  )=2
 Valor de la restriccion(  5  )=0
    Desplazamiento  y restringido para el nodo No.  4 *** valor=      0.000E+00
 Nodo restringido(  6  )=4
 Direccion de restriccion(  6  )=3
 Valor de la restriccion(  6  )=0
    Desplazamiento fi restringido para el nodo No.  4 *** valor=      0.000E+00
```

Figure-15. Boundary conditions - Marco2D.

```
== Cargas nod. aplic. - dir X=1 , dir Y=2, dir Fi=3 ==

 Numero de cargas nodales aplicadas=2
 Nodo cargado(  1  )=1
 Direccion de carga(  1  )=1
 Carga aplicada(  1)=100
    Carga en  x para el nodo No. (  1  ) *** valor=       1.000E+02
 Nodo cargado(  2  )=2
 Direccion de carga(  2  )=2
 Carga aplicada(  2)=-120
    Carga en  y para el nodo No. (  2  ) *** valor=      -1.200E+02

== Cargas distribuidas - dir perpendicular al elem. ==

 Numero de cargas distribuidas aplicadas=1

 No. de elementos con carga distribuida perpendicular a su eje longitudinal =  1
 Elemento cargado(  1  )=3
 Carga distribuida aplicada(  1  )=-20
     Carga distr. en el elemento No. (  3  ) *** valor=     -2.000E+01

 Calculo de Fuerzas de empotramiento en Global
 Elem    Nodo    [ F. dist ]   [FX de emp.]  [FY de emp.]  [MZ de emp]
   3       3    -2.000000E+01 -4.000000E+01  3.000000E+01  4.166667E+01
           1    -2.000000E+01 -4.000000E+01  3.000000E+01 -4.166667E+01
```

Figure-16. Loads apply - Marco2D.

VOL. 16, NO. 6, MARCH 2021 ISSN 1819-6608
ARPN Journal of Engineering and Applied Sciences

Finally, Figure-17 shows the results of displacements, reactions, and elementary internal forces obtained with the application.

```
*************** RESULTADOS OBTENIDOS ***************

 Calculo de Desplazamientos
  Nodo         [  x  ]        [  y  ]        [  fi  ]
   1       3.562156E-04  -5.598285E-04  -7.427967E-05
   2       1.781078E-04  -1.623559E-03   2.984842E-04
   3       0.000000E+00   0.000000E+00   0.000000E+00
   4       0.000000E+00   0.000000E+00   0.000000E+00

 Calculo de Reacciones-Fuerzas Aplicadas
  Nodo         [  Fx  ]       [  Fy  ]       [  Mz  ]
   1       1.000000E+02  -1.398534E-14   4.553649E-15
   2       1.226796E-14  -1.200000E+02   4.510281E-16
   3       2.304291E+01   1.161739E+02   4.529329E+01
   4      -2.030429E+02   6.382611E+01  -5.042163E+01

 Calculo de Fuerzas Internas
 Elem    Nodo    [ F. axial ]  [F. cortante]  [M. flector]
   1       1     -2.030429E+02 -5.617389E+01 -3.894328E+01
           2     -2.030429E+02 -5.617389E+01  4.531755E+01
   2       2     -2.030429E+02  6.382611E+01  4.531755E+01
           4     -2.030429E+02  6.382611E+01 -5.042163E+01
   3       3     -1.067649E+02 -5.127000E+01 -4.529329E+01
           1     -1.067649E+02  4.873000E+01 -3.894328E+01
```

Figure-17. Displacement, reactions, internal Forces - Marco2D.

The results of displacements, reactions, and internal forces obtained with the application coincide with those of SAP2000. Figures 18 to 21 show the results of displacements and internal forces obtained from SAP2000, used to verify the application.

Figure-18. Deformed shape. SAP2000.

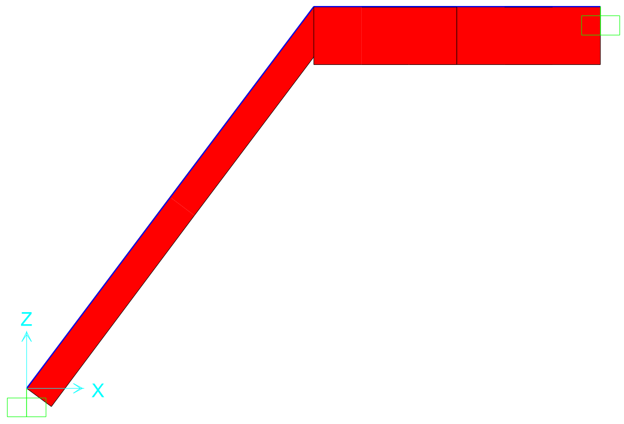

Figure-19. Axial forces. SAP2000.

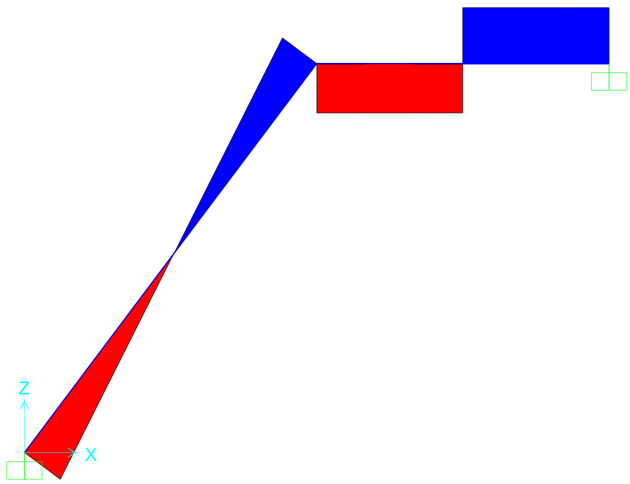

Figure-20. Shear forces. SAP2000.

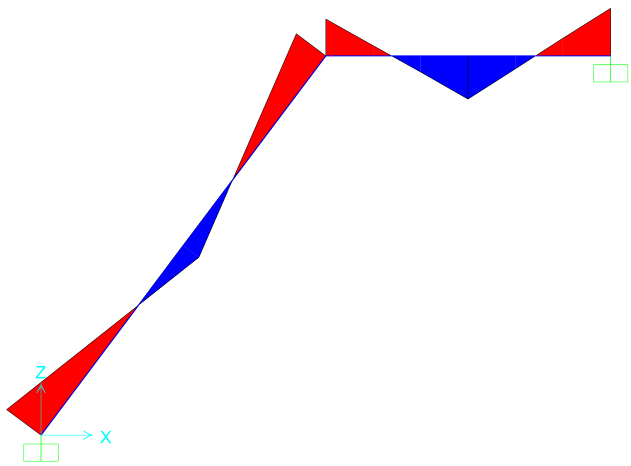

Figure-21. Flexural moments. SAP2000.

In Table-2, the results of horizontal and vertical displacements and rotations of the free nodes obtained from SAP2000 are shown, which are compared exactly with the results are given by the application developed.

Table-2. Horizontal and vertical displacement.

GDL	Marco2D/SAP2000
U1	3.56E-04 (m)
V1	-5.60E-04 (m)
ϕ1	-7.43E-05 (rad)
U2	1.78E-04 (m)
V2	-1.62E-03 (m)
ϕ2	2.98E-04 (rad)

In Table-3, the values of the elementary internal forces of axial, shear, and bending moment calculated by SAP2000 are shown. They are exactly compared with the results obtained from the developed application.

Table-3. Interval forces (kN).

Marco2D/SAP2000				
Element	**Node**	**Axial [kN]**	**Shear F. [kN]**	**Flexural M. [kN-m]**
1-2	1	-203.043	-56.174	-38.9433
	2	-203.043	-56.174	45.3175
2-4	2	-203.043	63.826	45.3175
	4	-203.043	63.826	-50.4216
3-1	3	-106.765	-51.27	-45.2933
	1	-106.765	48.73	-38.9433

The results are the same for both alternatives so that the correct functioning of the application is verified.

CONCLUSIONS

- The verified results demonstrate the application's versatility to support the matrix method's plane-frame calculation and understand its concepts. The results obtained verify the correct functioning of the application developed.
- The application registers the results on the screen. It generates a detailed report in a text file that allows the calculation memory to be secured. The results obtained demonstrated that the application is suitable for assisting the teaching-learning process in the classroom and for the autonomous work of engineering students in studying the plane-frame matrix analysis method.

ACKNOWLEDGEMENTS

Thanks to the Universidad Surcolombiana for the support given to this project (Project Number 3691). This work is one of the research products of the group "INGENIERIA Y SURDESARROLLO" led by the first author.

REFERENCES

[1] M. R. Pallares Muñoz, W. Rodriguez Calderón and D. E. González García. 2020. An educational computer program for matrix analysis of plane trusses in civil engineering. ARPN J. Eng. Appl. Sci. 15(4).

[2] J. S. Przemieniecki. 2012. Finite Element Structural Analysis: New Concepts.

[3] T. H. G. Megson. 2007. Matrix Methods of Analysis. in Structural and Stress Analysis.

[4] M. Petyt. 2003. Theory of matrix structural analysis. J. Sound Vib.

[5] A. Tena-Colunga. 2007. Análisis de Estructuras con Métodos Matriciales. México : Gedisa.

[6] M. Paz, W. Leigh, M. Paz and W. Leigh. 2011. Static Condensation and Substructuring. in Integrated Matrix Analysis of Structures.

[7] W. McGuire, R. H. Gallagher and H. Saunders. 1982. Matrix Structural Analysis. J. Mech. Des.

[8] H. H. West. 1993. Fundamentals of Structural Analysis. Eur. J. Eng. Educ.

[9] J. Uribe Escamilla. 2002. Análisis de estructuras. Bogotá D.C.