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

Development and implementation of software using the Direct Stiffness Method and Symbolic MATLAB proposing the improvement of teaching and learning processes of matrix methods in civil engineering graduation classes

Guilherme O. Berbert-Born[1], Gilberto Gomes[1]

[1] *Department of Civil and Environmental Engineering, University of Brasília*
Campus Darcy Ribeiro, 70910-900, Distrito Federal, Brazil
guiborn.eng@gmail.com, *ggomes2007@gmail.com*

Abstract. The emergence of new programming languages and philosophies allowed the possibility of more efficient calculation techniques in various engineering fields. Such techniques most often displays the result of processing without making possible for the user to visualize the process itself. This can bring difficulties to fully understand the method, impairing didactic purposes that might involve its use. The AESYM is a new structural analysis program written in MatLab language which uses the Direct Stiffness Method. This method consists in analyzing reticulated structures, statically determinate or indeterminate, based on the superposition principle, considering linearity between actions and displacements. The method assumes that the effects of rotation and translation of the bar elements in given structure due to multiple stimuli can be obtained by the combination of the effects caused by each stimulus individually. With the AESYM, the user can follow the calculation process both symbolically and numerically, allowing the understanding of the linear elastic analysis in a didactic and visual way. This makes possible for the user to observe the global stiffness matrix formation process along with its properties, displacements and forces in the elements through the resolution of the equation which governs the structure's internal behavior. The AESYM has, therefore, practical use in matrix methods teaching-learning process within graduation level engineering courses.

Keywords: Direct Stiffness, Structural Analysis, Teaching-learning processes, Symbolic *MATLAB.*

1 Introduction

The appearance of new tools and the increasingly higher processing power for computers enables more and more the use of techniques for complex problems. Adapted procedures involving modelling and graphical interface programming conditioned better use of numerical methods such as the Finite Element Method (FEM), clarified by Assan [1] and the Boundary Element Method (BEM), as exposed by Brebbia and Dominguez [2], both widely used for solving engineering problems.

It is observed that most of the computer programs for structural analysis return numerical results without the due transmission of the calculation method and procedures. As a result, it is difficult to understand the processes used to obtain efforts and displacements in structures. In the case of the Direct Stiffness Method (DSM), shown by Soriano [3], it is of great importance for the teaching-learning process to visualize the matrix assembly process, as well as the solution for the equations systems.

With this work, the use of matrix calculations and DSM applied to the MATLAB programming language [4] are used in order to develop a new visual and symbolic platform that can be used as an auxiliary tool in the teaching-learning process in the disciplines of undergraduate structural analysis courses. The use of symbolic language allows a better understanding and visualization of the elements that make up the calculation of beams, trusses and flat frames. In addition, the creation of AESYM seeks to return to the user, not only global and local results, but also the process of assembling matrices and solving the system of equations to obtain forces and displacements.

2 The Direct Stiffness Method

The Direct Stiffness Method (DSM) can be used to resolve isostatic and hyperstatic reticulated structures, where the displacements and rotations are unknown. Using the principle of superposition and assuming linear analysis, the method combines effects caused by individual stresses, obtaining global solutions for forces and displacements.

Based on Hooke's Law, clarified by Hibbeler [5], which relates strain and elasticity of a body to the stressing action, a system can be set up from which the global forces and displacements can be obtained for each node in the structure. In order to obtain the Global Stiffness Matrix (GSM), the local stiffness matrices for each element of the structure must be accumulated, these being the Degrees of Freedom (DOFs) of the truss and frame elements flat, together with the rotation matrices used indicated in Tab. 1, for each case, as shown by Halliday, Resnick and Walker [6].

Table 1 – Degrees of Freedom (DOFs), Local Stiffness Matrices and Rotation Matrices.

Plane Trusses	Plane Frames
	
$K_e = \begin{bmatrix} \frac{E\cdot A}{L} & 0 & -\frac{E\cdot A}{L} & 0 \\ 0 & 0 & 0 & 0 \\ -\frac{E\cdot A}{L} & 0 & \frac{E\cdot A}{L} & 0 \\ 0 & 0 & 0 & 0 \end{bmatrix}$	$K_e = \begin{bmatrix} \frac{E\cdot A}{L} & 0 & 0 & -\frac{E\cdot A}{L} & 0 & 0 \\ 0 & \frac{12\cdot E\cdot I}{L^3} & \frac{6\cdot E\cdot I}{L^2} & 0 & -\frac{12\cdot E\cdot I}{L^3} & \frac{6\cdot E\cdot I}{L^2} \\ 0 & \frac{6\cdot E\cdot I}{L^2} & \frac{4\cdot E\cdot I}{L} & 0 & -\frac{6\cdot E\cdot I}{L^2} & \frac{2\cdot E\cdot I}{L} \\ -\frac{E\cdot A}{L} & 0 & 0 & \frac{E\cdot A}{L} & 0 & 0 \\ 0 & -\frac{12\cdot E\cdot I}{L^3} & -\frac{6\cdot E\cdot I}{L^2} & 0 & \frac{12\cdot E\cdot I}{L^3} & -\frac{6\cdot E\cdot I}{L^2} \\ 0 & \frac{6\cdot E\cdot I}{L^2} & \frac{2\cdot E\cdot I}{L} & 0 & -\frac{6\cdot E\cdot I}{L^2} & \frac{4\cdot E\cdot I}{L} \end{bmatrix}$
$[\mathrm{R}] = \begin{bmatrix} \cos(\alpha) & \mathrm{sen}(\alpha) & 0 & 0 \\ -\mathrm{sen}(\alpha) & \cos(\alpha) & 0 & 0 \\ 0 & 0 & \cos(\alpha) & \mathrm{sen}(\alpha) \\ 0 & 0 & -\mathrm{sen}(\alpha) & \cos(\alpha) \end{bmatrix}$	$[R] = \begin{bmatrix} \cos(\alpha) & \mathrm{sen}(\alpha) & 0 & 0 & 0 & 0 \\ -\mathrm{sen}(\alpha) & \cos(\alpha) & 0 & 0 & 0 & 0 \\ 0 & 0 & 1 & 0 & 0 & 0 \\ 0 & 0 & 0 & \cos(\alpha) & \mathrm{sen}(\alpha) & 0 \\ 0 & 0 & 0 & -\mathrm{sen}(\alpha) & \cos(\alpha) & 0 \\ 0 & 0 & 0 & 0 & 0 & 1 \end{bmatrix}$

In order to accumulate the local stiffness matrices and to assemble the GSM, each local matrix must be rotated, that is, it must have its local coordinates changed to global so that the axis orientations coincide. The last row of Tab. 1 represents the rotation matrices, obtained from the direction cosines of the bars, which are used with each local matrix in eq. (1)

$$[\mathrm{K}]_G = [R]^T \cdot [\mathrm{k}]_e \cdot [R] \tag{1}$$

to obtain the global matrix. For such, the vectors of forces and global displacements have their formats represented in eq. (3) and eq. (4), where n is the number of the first node. Once imposed the support restrictions in the vector of global displacements and obtained the GSM, the system can be solved using Hooke's Law. For this, the "one and zeros" technique (setting the lines and columns corresponding to restricted DOFs to zero and their intersections to one) is applied in order to obtain the unknown displacements.

$$[F] = [F_{nx},\ F_{ny},\ M_{nz},\ F_{n+1x},\ F_{n+1y},\ M_{n+1z}\ ...] \tag{3}$$

$$[\mathrm{U}] = [u_{nx},\ u_{ny},\ \theta_{nz},\ u_{n+1x},\ \ u_{n+1y},\ \theta_{n+1z},\ ...] \tag{4}$$

At last, the Rotation Matrix is once again used to obtain the displacements and local forces given the global displacement and forces vectors. This can be done by using eq. (5) and eq. (6), respectively

$$[u] = R \cdot [\mathrm{U}]' \tag{5}$$

$$[f] = K_e \cdot [u]\ . \tag{6}$$

3 Computational implementation

3.1 Pre-processing and modelling

The analysis of cross-linked structures using the DSM requires that not only the layout of the elements and nodes be known, but also certain factors that influence the overall behavior of the structure. These factors include boundary conditions (supports), concentrated and distributed loads, temperature variations (uniform or not), imposed displacements (settlements), cross-section geometry (area and moment of inertia) and the material's Elasticity Module.

AESYM allows the user to model the structure and insert its properties. The modeling interface of the program, shown in fig. (1), collects the model information and fills eight matrices obtained from the table fields.

- “L1” Matrix - General structure information (number of nodes, number of elements, number of restricted nodes, number of loaded nodes and number of loaded elements);
- "L2" Matrix – Structure node coordinates;
- “L3” Matrix –Each element's starting and ending nodes, as well as its properties (area, moment of inertia and modulus of elasticity).
- "L4" Matrix - Restrictions on x, y and θ in nodes;
- “L5” Matrix - Concentrated loads at nodes;
- "L6" Matrix - Distributed loads in the elements;
- "L7" Matrix - Imposed displacements on x, y and θ at nodes;
- "L8" Matrix - Coefficient of thermal expansion, temperature variation and section height

These matrices will be used in DSM processing and can be stored and loaded into the computer's memory, through the “Save / Open” menu. This menu gives access to a structure manager, in which the user can save, open, view or delete models.

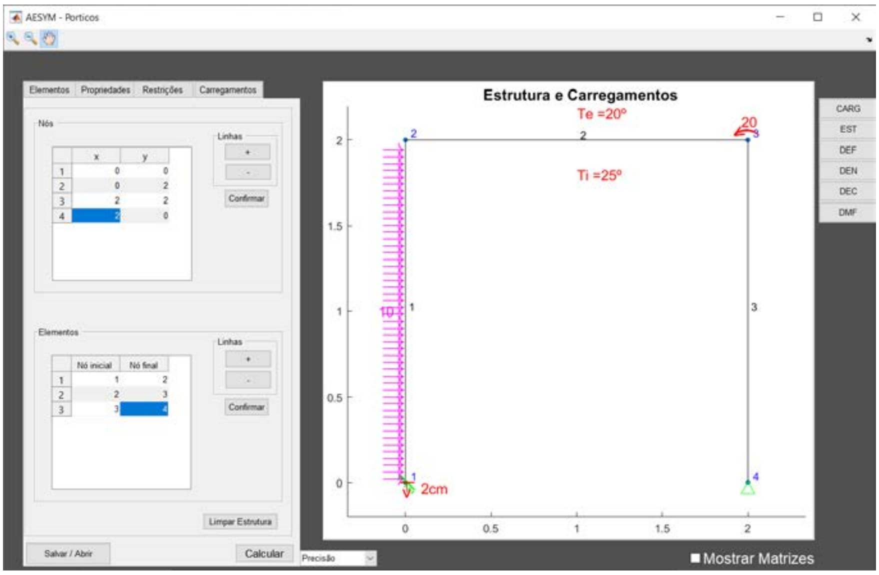

Figure 1. AESYM's modelling interface

3.2 DSM processing

Once the property matrices are obtained, the calculation process is initiated at the “Calculate” button. The first step of the calculation is the analysis of the “L1” matrix, necessary to limit the recursive functions (loops) and repeat the same procedures for all members and nodes.

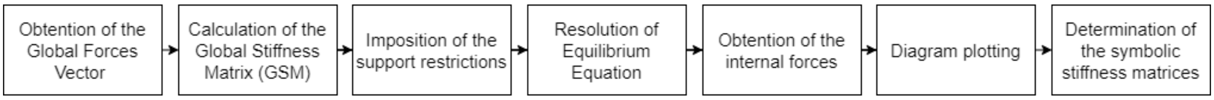

Figure 2. DSM Processing Flowchart

The first stage of the processing is the Global Forces Vector obtention. The concept of equivalent loads, exposed by Martha [7] is used to obtain the reactions in each element of the bar separately, with the efforts being concentrated on the nodes and later added to other bars with common nodes. In the event of temperature variation or settlement, this vector is also changed. This vector's size is three times the number of nodes, representing the forces at x, y and the moment at each node.

The second stage consists of the calculation of the Global Stiffness Matrix (GSM). For each element of the structure, the director cosine is calculated, which represents the slope of the bar. Then, the physical and geometric properties of the element are inserted in the Local Stiffness Matrix (LSM) and the Rotation Matrix is assembled. With this, it is possible to rotate the local matrix using the eq. (2) for the assembly of the GSM, consisted of the sum of all local rotated matrices.

The third step of the processing is the imposition of the support restrictions. Starting with the "L4" Matrix, the GSM and the Global Forces Vector are manipulated to suit the boundary conditions. For each restricted DOF, the corresponding row and column in the GSM is set to zero, being the value of the intersection between them set to 1. In addition, the Global Forces Vector is set to zero at the corresponding restricted DOF row. In the case of imposed displacements, the value of the corresponding force is equal to the displacement, in meters. The support settlement is then subtracted from all other forces of the vector, multiplied by the corresponding value in the GSM, as shown by Martha, [7].

Once the GSM and the Global Forces Vector are ready, the next step is the resolution of Equilibrium Equation and the calculation of the displacements. From the resulting Global Displacement Vector, the Deflection Diagram is plotted based on the starting and ending position of each node.

In order to obtain the values of the internal forces, the Rotation Matrix and the LSM are once again used, as well as the physical and geometric properties of each element. This is done by calculating the local displacements using eq. (5) and eq. (6). After that, the internal forces and the support reactions are determined according to local coordinates. A "Sl" matrix is created, in which the lines represent each element of the structure and the columns number is six, containing the initial and final values of axial forces, shear forces and bending moments in the element.

After the internal forces of the member are determined, the diagrams are plotted using the nodes and members coordinates along with the initial and final positions of each node consisting the Global Displacement Vector. The AESYM plot navigation buttons are used to navigate through 6 different resulting plots: Structure with loads (CARG button), Structure without loads (EST button), Structure Deflection (DEF button), Axial Force and Support Reactions Diagram (DEN button), Shear Force Diagram (DEC button) and Bending Moment Diagram (DEM button)

The last step is the determination of the symbolic stiffness matrices. All the mentioned procedures are repeated without numerical values being inserted in the local matrices. The MATLAB Symbolic ToolBox is used to make symbolic calculations. The Local, Rotation and Global matrices obtained are stored for post-processing stage user visualization.

3.3 Post-processing and results

At the end of the calculation process, the user may access the results on the interface. Diagrams, numerical results, symbolic and numerical matrices can be viewed globally or locally, as shown in fig. (2). The diagrams obtained during processing are available immediately after their completion, accessed with the plot navigation buttons.

The numerical results are the values for support reactions, internal forces and global displacement. The matrices (local, rotation, rotated and GSM) can be visualized by checking the "Mostrar Matrizes" checkbox. Within the menu, the user may also navigate between the global results (button "Estrutura") or local (member) results (button "Elementos"), both located in the lower left corner.

The numerical and symbolic GSMs can also be accessed using the side navigation buttons. The symbolic matrix is plotted with LaTex, and the user can interactively browse by dragging the cursor across the matrix. The numerical matrix is presented in the form of a table where columns and rows have customizable size. The font size in each matrix can be changed in order to improve the visualization.

For the visualization of local results, the "Elementos" button may be pressed, which will open a window with a list of the structure elements. When selecting an element, the user is able to see its properties, internal forces and displacements. In addition, the buttons "Matriz de Rigidez Local", "Matriz de Rotação" and "Matriz Rotacionada" can be used to show the corresponding local stiffness, rotation or rotated matrix.

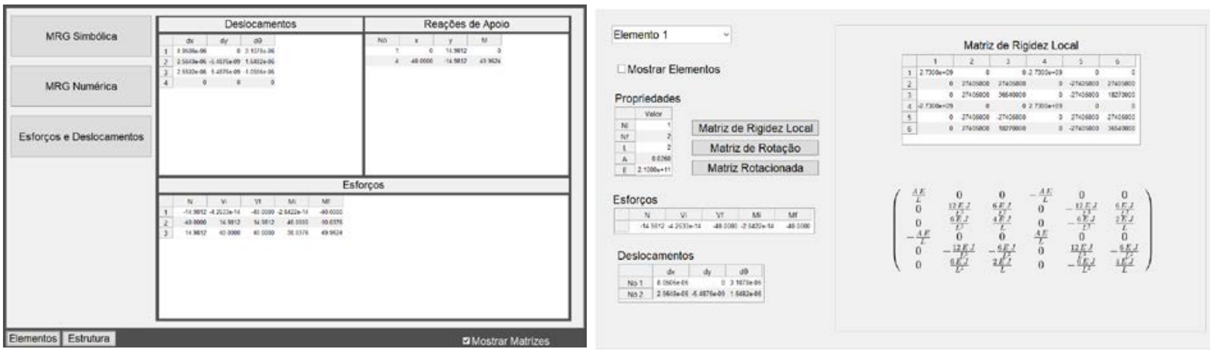

Figure 3 – Global Forces and Displacements Window / Local Results and Local Stiffness Matrix

4 Conclusions

The visual interface allows the user greater control over modeling, as well as more speed and practicality. Still through the interface, results can be obtained in a simpler and more controlled way, in addition to the possibility of greater understanding of the DSM and the structure's behavior. Thus, AESYM is a program with great educational potential for academic purposes.

Thus, the convenience of using DSM in computational implementation is notable, since in this environment there are powerful tools for matrix calculations. Another advantage of this implementation is its adaptability and for the most diverse purposes. Compared to the results of the educational structural analysis software FTool, developed by Martha [8], the calculations made by the AESYM have shown good precision and liability.

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] A.E. Assan, Método dos Elementos Finitos – Primeiros Passos, Editora da Unicamp, Campinas, São Paulo, 1999.
[2] BREBBIA, C. A. and DOMINGUEZ, J. Boundary Elements, An Introductory Course, 2nd Edition, Computational Mechanics Publications, McGraw-Hill Book Company
[3] SORIANO, H. L. Análise de Estruturas - Formulação Matricial e Implementação Computacional. 1ª. ed. Rio de Janeiro: Editora Ciência Moderna Ltda, v. único, 2005.
[4] MATLAB M. The language of technical computing. The MathWorks, Inc, 2012. Natick, Massachusetts, United States. Available at: http://www mathworks.com.
[5] HALLIDAY David, RESNICK Robert and WALKER Jearl Fundamentos da Física, Volume I. Rio de Janeiro: LTC, 2012.
[6] Martha, L.F., O Método da Rigidez Direta sob um Enfoque Matricial, Apostila editada pela Coordenação de Extensão e Treinamento Profissional – Escritório Técnico da Escola de Engenharia, UERJ, 1984.
[7] Martha, L.F., Análise de Estruturas. Conceitos e Métodos Básicos, Pontifícia Universidade Católica do Rio de Janeiro – PUC-Rio, 2010.
[8] MARTHA, L.F. FTOOL - Um programa gráfico-interativo para ensino de comportamento de estruturas. Versão educacional 2.11, Rio de Janeiro, August 2002, 33p.