2D STRUCTURAL ANALYSIS EDUCATIONAL SOFTWARE: A FORCE PROCESS SIMULATOR USING FREECAD

Mateus Felipe Marques de Oliveira[1], Christian Carneiro de Oliveira[1], João Carlos Cordeiro Barbirato[1]

[1]*Tecnology Center, Federal University of Alagoas*
Lourival Melo Mota Ave, n/n, Tabuleiro do Martins, Maceió, 57072-970, Alagoas, Brazil
Mateus.oliveira@ctec.ufal.br, Christian.oliveira@ctec.ufal.br, Jccb@ctec.ufal.br

Abstract. The technological advances and the consequent modernization in various sectors of society evidence and foster the emerging tendencies to the usage of computational tools in behalf of the analysis of structures, more specifically with regard to the field of Civil Engineering. Along with it, the training of future engineers with the use of new methodologies that assist in the teaching-learning process, is frequently debated and encouraged. Among these educational tools, stand out those that have an appeal in computational mechanics based in the use of numerical methods with a very well characterized and absorbed theoretical basis. In this context, knowledge of the behavior of structural systems is essential, especially when using numerical analysis methods and their computational implementations. In undergraduate courses that deal with hyperstatic structures, two solution methods are studied: the Displacement Method and the Force Method. The Force Method, despite having a very consistent physical appeal and being the one that better represents the structural behavior, does not have an interesting approach for an integral computational implementation. In this circumstance, the present work intends to present an expert software developed for computer-assisted learning of the Force Process. The idea is to have a Simulator of the Flexibility Process with the explanation of each of its steps, so that the user, a structural mechanics student, can interact not only by entering initial data and obtaining the final analysis data. On the contrary, he will have the definition/explanation of his choice of the fundamental isostatic model and the subsequent steps of its systems, leading to the obtention of the flexibility matrix of the simulated problem. The computational code was written in Python, using FreeCAD, an open parametric modeling software, as the user interaction platform, taking advantage of its graphical directives and its interoperability with the Python language. The simulation is done by using personalized buttons for data entry, inherent to the pre-processing stage of the application, that is, the stage of modeling the structure, which is presented visually, in order to facilitate the user´s understanding. Subsequently, the program calls for the computational processing of the plane structure and presents each step of the Force Method. For the due validation process of the code developed, applications taken from other scientific works are presented.

Keywords: Force Method, Educational Software, Computational Mechanics.

1 Introduction

With the advent of the new DCN (National Curriculum Guidelines) of engineering sciences introduced in Brazil in the year of 2019, the massive insertion of technology in the labor market and the need to modernize engineering education as a whole, there has been a greater focus on issues related to active learning by students. As a consequence of this mobilization, in engineering courses there is already an increasing adoption of information technology as an essential component for development, as explained by Massukado and Schalch [1]. In this sense, the use of educational software aimed at engineering learning has been increasingly present in university environments, becoming more and more relevant in tasks related to calculations and modeling, as an additional resource for a better understanding of the problem, as mentioned by Andreatta-da-costa, Bernardes and Ramiro [2].

When it comes to the structural analysis of hyperstatic systems, the related disciplines of several engineering courses around the world mainly addresses two classical methods: the Displacement Process and the Force Process.

The first one basically consists on isolating each displaceability of the original structure in a different structural system that satisfies its compatibility conditions, but does not supply its original equilibrium conditions (since new forces are generated to nullify the displacements that are not related to each system). The original equilibrium is then restored through superposition of each system, finally providing the hyperstatic values. This one is widely explored computationally, since it is more suited to computational implementation, once it has a matricial formulation that is convenient for a computational implementation, as explained by Martha [3]. For that reason, it has been implemented in several works on this field, such as the software FTOOL©, developed by researchers at PUC-RJ and widely used by undergraduate and graduate courses in Brazil and abroad. Furthermore, since the main purpose of most software developed for didactic purposes was to obtain the final result, the best option available was to use the Displacement Process.

On the other hand, also according to Martha [4], the Force Method is based on the sum of basic solutions that satisfy the equilibrium conditions, but do not supply the original compatibility conditions of the structural system, which are then re-established once the superposition of the basic cases is done. The structure used as basis to this superposition is an isostatic auxiliar frame originated by the flexibilization of the original hyperstatic structure, which happens through elimination of displacement constraints. This method, however, despite being more tangible from the physical point of view, allowing a more intuitive structural analysis, is not as amenable to computational implementation in its entirety, since the flexibilization process involved leads to infinite possibilities of resolution.

In this context, it can be said that there is a gap in the supply of software that constitutes a didactic tool focused on structural analysis, for the proper understanding of the behavior of reticulated structures using the concepts of the Force Process. Therefore, the present work aims to develop a software, for academic use, which simulates each step of the implementation of the classic Force Process (Flexibility Process), with graphical routines that presents the main and the auxiliary systems and its respective internal forces diagrams, as well as the flexibility and hyperstatic parameters inherent to the process.

2 Methodology

The methodology used in this work was based on two main steps: literature review and computational implementation of the Force Process simulator. Both will be detailed below.

In the literature review step, the objective was to deepen the knowledge regarding the two methods previously presented for the solution of hyperstatic structures. Furthermore, since the proposed simulator has a strong educational appeal, case studies related to the implementation of software in the context of engineering education were researched.

As for the simulator's computational implementation, it was initially chosen to use the open-source language Python, due to the fact that this is a high-level language easily adaptable for the purposes in question. Furthermore, the development of the simulator in question was carried out through three sub-stages, namely: pre-processing, data processing and post-processing. The pre-processing phase consists in providing input fields that receive data related to the modeling of the structure and the acting forces, such as the choosing of the user’s desired way of flexibilization of the hyperstatic structure. Once the data fields are filled, the hyperstatic structure and the fundamental isostatic structure, along with their respective basic systems, are visually presented – in order to facilitate the user's comprehension of the problem to be solved, the pre-processing phase was designed to have a strong visual appeal. To this end, some python libraries were studied, such as *kivy* and *tkinter*, related to the creation of graphical user interfaces. However, in order to speed up this stage, it was decided to use the open-source parametric modeling software, FreeCAD, since it already had its own graphic directives, which can be manipulated through great interoperability with the python language.

In terms of data processing, these were performed once the previous stage was concluded, using the Direct Stiffness Method (or Displacement Process), where, in short, all the information inherent to the hyperstatic structure along with its binding conditions, as well as the fundamental isostatic and their respective complementary systems were processed, leading the software to the determination of nodal parameters such as displacement and internal forces and reactions. Then, is open the possibility to determine the flexibility coefficients (δij) of the structural system, which by definition are the represented by the displacement in the node of the common system related to the unitary hyperstatic i in the direction of the related coordinate, in the common system j.

Then, having information about flexibility coefficients, reactions and internal forces for each node of each system, the post-process stage is triggered, presenting the obtained data, along with the internal forces diagrams for each bar of each system.

3 Results

In order to verify the results obtained, the hyperstatic system shown in Fig. 01 was calculated. Then, a comparison was made between the values presented in the FTOOL software and the simulator presented in this paper.

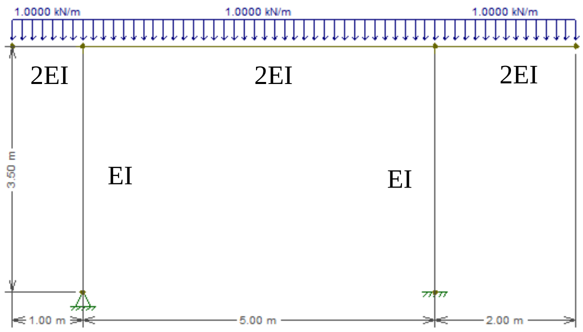

Figure 01. Hyperstatic structure used to verify the results

The insertion of this model into the FreeCAD software is done by filling the data inherent to the modeling of the structure through custom buttons, checkboxes and input boxes, following a logical procedural sequence, as presented in Fig. 02.

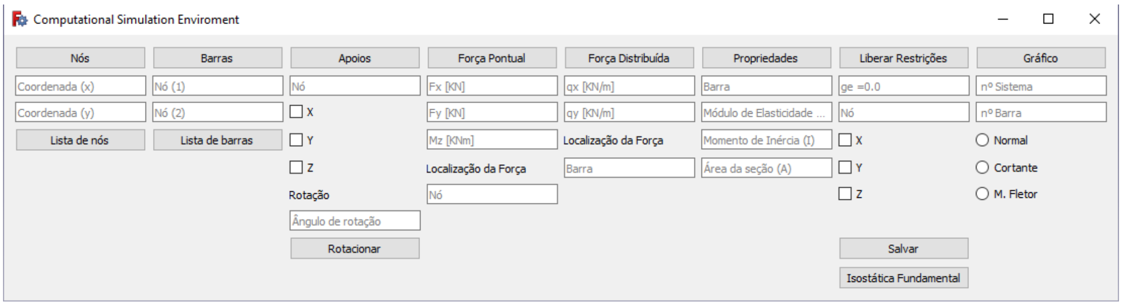

Figure 02. Graphical interface for modeling the structure

Once the modeling of the hyperstatic structure is complete, the user is presented to it, along with the fundamental isostatic (which had its flexibilization by liberating the horizontal displacement in the support to the left, while also freeing rotation on the support on the right pillar) with its corresponding basic systems, as illustrated in Figures 03 and 04. After the data is processed, the values related to the support reactions relative to the hyperstatic system, as well as the values corresponding to the flexibility coefficients, are displayed on the screen.

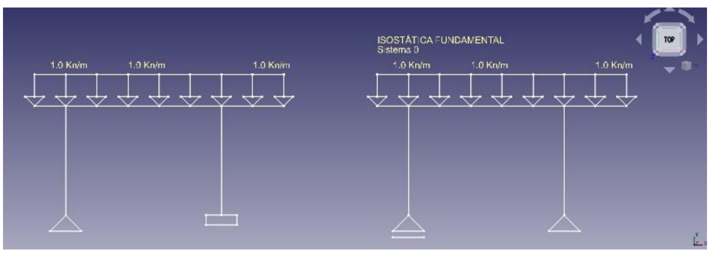

Figure 03. Simulator's hyperstatic and fundamental isostatic models

Fig 04. Simulator's presented basic systems

The comparative analysis between the simulator and the FTOOL software showed that the support reactions obtained using the simulator are in accordance with those obtained in the mentioned software. This fact is represented in Fig. 05 – where the box on the left presents the simulator's output of the system's reactions and flexibility coefficients, and "nó 0" represents the node on the bottom of the left column, while "nó 4" stands for the node on the bottom of the right one. The FTOOL results output for hyperstatic reactions are presented to the left of the previous mentioned box by the red arrows.

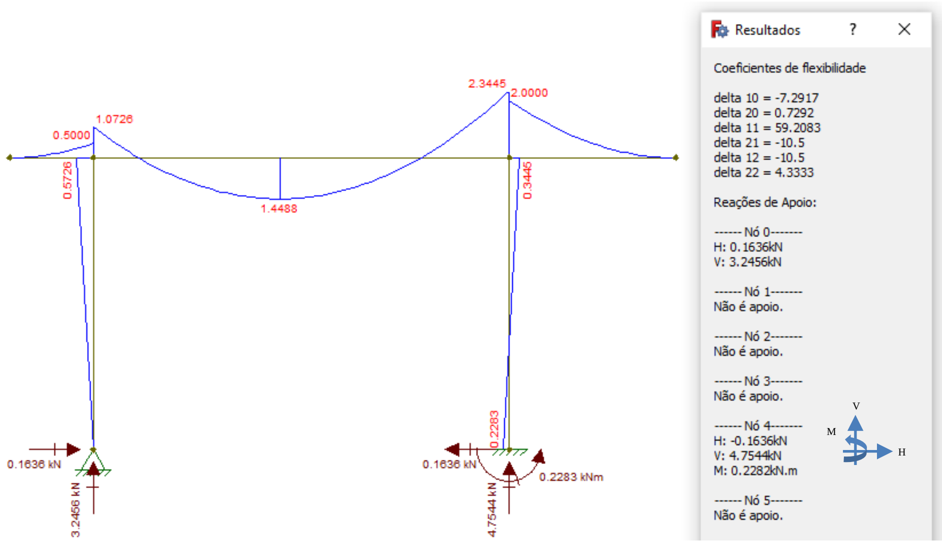

Figure 05. Comparative analysis of the reaction values obtained for the support reactions

Furthermore, Tab. 1 shows that the values obtained for the flexibility coefficients using the simulator are in accordance with the displacements, relative to the respective systems at the flexed points, provided by the FTOOL software. Positive horizontal displacement has a left-right direction, as well as positive rotation is counter-clockwise.

δij	FTOOL	Simulator
$\delta 10$	-7.2917	-7.2917
$\delta 20$	0.7292	0.7292
$\delta 11$	5.9208	5.9208
$\delta 21$	-10.5	-10.5
$\delta 12$	-10.5	-10.5
$\delta 22$	4.333	4.333

Table 1. Comparative analysis between the flexibility coefficients

Finally, a comparison was made between the internal forces' diagrams obtained in both softwares. For this, the horizontal bar with the largest span was chosen, and the bending moment diagram was generated by indicating the system and the corresponding bar. The results obtained demonstrate the simulator's consistency in determining the internal forces acting on the structure, as its possible to notice comparing the obtained diagrams in Fig. 05 and Fig. 06.

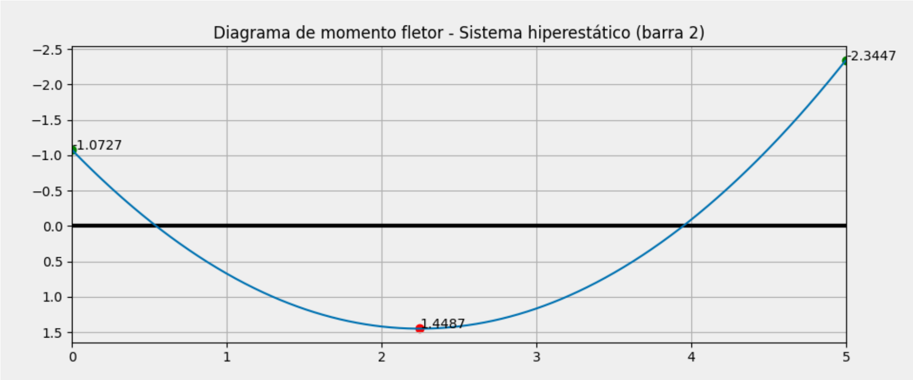

Figure 06. Bending moment diagram for the selected bar

4 Conclusions

In view of the above, it can be concluded that the simulator developed meets the proposed objectives, since it presents all the steps inherent to the classic Force Process, making use of graphic routines for presenting the auxiliary and final diagrams as well as the corresponding flexibility and hyperstatic parameters.

The next steps will be based on the application of the simulator in the discipline of Theory of Structures 2, taught at the Technology Center of the Federal University of Alagoas, to verify the influence of the tool in the students' understanding. Moreover, the product of this work can be passed on to a startup or institutionalized project, through the support foundation of the Federal University of Alagoas, able to support and explore its distribution.

Acknowledgements. This research was supported by the Fundo Nacional de Desenvolvimento da Educação (National Education Development Fund) and the Ministério da Educação (Ministry of Education), through the Programa de Educação Tutorial (Tutorial Education Program), to which we are immensely grateful.

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] L.M. Massukado and V. Schalch, "Simulation in engineering education - Evaluating the application of Simgere software under the 'Learn To Learn' paradigm". *Engineering Education Journal*, vol. 26, n. 2, pp. 40-46, 2007.
[2] L. Andreatta-da-costa and J.A. Bernardes and F.S. Ramiro, "Educational software – its use and importance in the teaching-learning of civil engineering students". *Brazilian Congress on Engineering Education*, 2014.
[3] L.F. Martha, "Direct stiffness method for linear and elastic structural models". Rio de Janeiro-RJ.
[4] Basic Methods of Structural Analysis. Available at:
<http://coral.ufsm.br/decc/ECC1002/Downloads/_Cap_5_Metodo_das_forcas.pdf>, pp. 129-160.