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

Ftool 5.0: Nonlinear, stability and natural vibration analyses

Rafael L. Rangel[1], Luiz Fernando Martha[2]

[1]*International Centre for Numerical Methods in Engineering (CIMNE)*
C. Gran Capità, S/N 08034 Barcelona, Spain
rrangel@cimne.upc.edu
[2]*Tecgraf Institute of Technical-Scientific Software Development of PUC-Rio (Tecgraf/PUC-Rio)*
R. Marquês de São Vicente, 225 - Gávea, Rio de Janeiro - 22451-900 - RJ, Brazil
lfm@tecgraf.puc-rio.com

Abstract. This work presents a new version of Ftool, a program for structural analysis of two-dimensional frame models. This new version has two new analysis modes: nonlinear analysis and eigenvalue analysis. The former currently considers geometric nonlinearity, allowing users to perform numerical simulations of structures with large displacements and rotations, but small deformations. The latter considers two types of general eigenvalue problem to analyze the stability and the natural vibration of structures. The stability analysis provides the critical loads and the buckling modes of the structure while the natural vibration analysis provides the natural frequencies and the vibration modes. Several options are available to perform these new types of analysis and to check the results. These options are explained and exemplified throughout this paper.

Keywords: frame models, geometric nonlinearity, stability analysis, natural vibration.

1 Introduction

The Ftool program (Martha [1]) is a graphical-interactive software for the structural analysis of two-dimensional frame models. It is widely used worldwide by students and engineers because of its user-friendly interface and the simplicity of the modeling and analyzing processes. Until then, it only had the option of linear-elastic analysis, which is sufficient for many applications in civil and mechanical engineering. However, the demand for more sophisticated types of analysis has been growing along with the needs for more economical, yet safe, engineering projects. To meet this demand, Rangel [2] developed a geometrically nonlinear analysis module for the program, which allows an extensive control of the incremental-iterative process by the user. This module was designed to include other types of nonlinearity in the future, such as the nonlinear behavior of the material and the semi-rigid connections of elements. Recent developments have also included an eigenvalue analysis module to the program, to solve the general eigenvalue problem corresponding to the stability and the natural vibration of the structure. With this new module, users can study the critical loads and buckling modes of the structural model, as well as its natural frequencies and vibration modes. These new analysis types will be available in the advanced version of the program.

In order to accommodate the new types of analysis, the graphical user interface of the program suffered some modifications. Figure 1 shows the new graphical interface of Ftool indicating some important new features and displaying a model of a semi-circle arch with a point load at the mid-node. At the top, there is a drop-down list from which users can select the desired type of analysis. When changing this option, the *Analysis Options* menu shows the options and parameters for running the selected type of analysis (when linear-elastic analysis is selected, no option or parameter is needed). An environment for plotting graphs of nonlinear analysis results is also available and can be accessed by clicking on the *Results Graph* button. In the *Edit* toolbar, on the left-hand side of the screen, there is a tool for subdividing elements. This is an important auxiliary tool for nonlinear analysis, since the accuracy of the results depend on the discretization of the elements (i.e. on the mesh refinement).

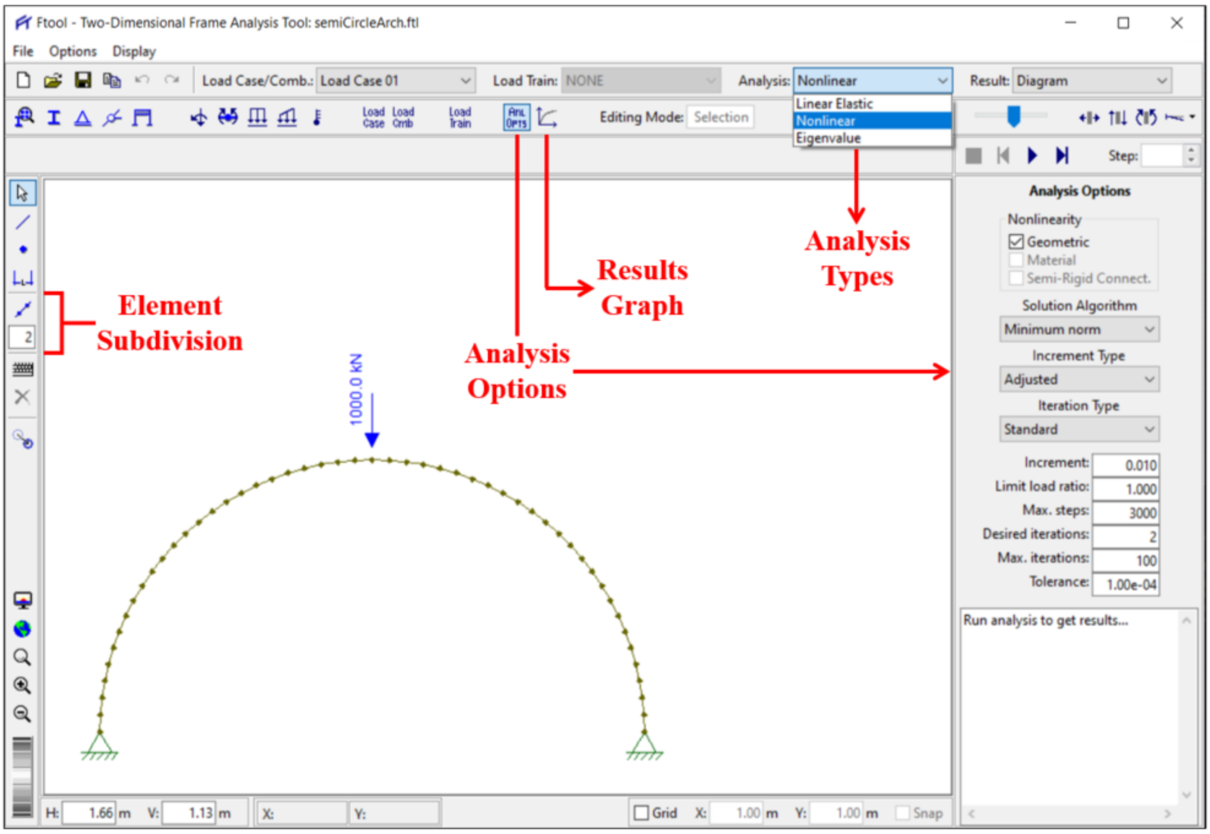

Figure 1. New graphical user interface of Ftool showing a semi-circle arch model, the available types of analysis, and the menu of nonlinear analysis options

2 Nonlinear analysis

2.1 Analysis options and control

When nonlinear analysis is selected, the *Analysis Options* menu provides several options and numerical parameters to be set for performing the analysis, as shown in Fig. 1. The first option is the source of nonlinearity, which currently allows only the geometric type. The geometric nonlinearity considers large displacements and rotations, but small deformation of elements. The formulation implemented to account for these effects is the Corotational formulation, explained by Rangel [2], Souza [3], and Crisfield [4].

The analysis options that can be set are the Solution Algorithm (Simple Step, Load Control, Work Control, Arc-Length, Minimum Norm, and Generalized Displacement); Increment Type (constant or adjusted increment sizes); Iteration Type (standard or modified Newton-Raphson iterations).

The numerical parameters are the increment size, the limit value of load ratio, the maximum number of steps, the desired number of iterations to adjust the increment size, the maximum number of iterations allowed in each step, and the tolerance for convergence.

All these options and parameters can be modified in any step during the incremental process, in what is called an interactive-adaptive analysis. For this purpose, some analysis control buttons are positioned above the *Analysis Options* menu. These buttons are the *Play*, to run full analysis, the *Forward*, to run a single step, the *Backward*, to return one step, and the *Reset*, to send the program back to model editing mode.

An analysis feedback is provided in a text field bellow the numerical parameters, where the number of iterations and the load ratio are given for each executed step.

2.2 Results visualization

The available results for nonlinear analyses are diagram results (internal forces and deformed configuration). The diagram results of any step can be checked by selecting the desired step in the text field next to the analysis control buttons. Any processed step can be selected to show the diagrams corresponding to that step.

Some interesting post-processing options are available in the *Options* menu, located on the tab at the top of the screen. There, users can find the following options for geometric nonlinear analysis results:

Deformed configuration display. The deformed configuration of the model can be drawn using straight lines between nodes or using the element shape functions in a natural coordinate system (the same used in the Corotational formulation) to better represent the deformed aspect of the structure subjected to large motions.

Internal force diagrams display. The internal force diagrams can be plotted over the undeformed or the deformed configuration of the structure.

Local P-delta effect. The local P-delta effect, associated with axial force acting on the local element deformation relative to the chord between end nodes, may be considered, or not, when computing bending moment diagrams.

2.3 Results plotting

One of the most important outputs in a nonlinear analysis is the graph of some results along the incremental process. In Ftool 5.0, it is possible create graphs in a sophisticated plotting environment. When creating a graph, the data that can be selected for each axis are nodal displacement, internal displacement, internal force, load ratio, and steps. Figure 2 shows an equilibrium path of the arch model presented in Fig. 1.

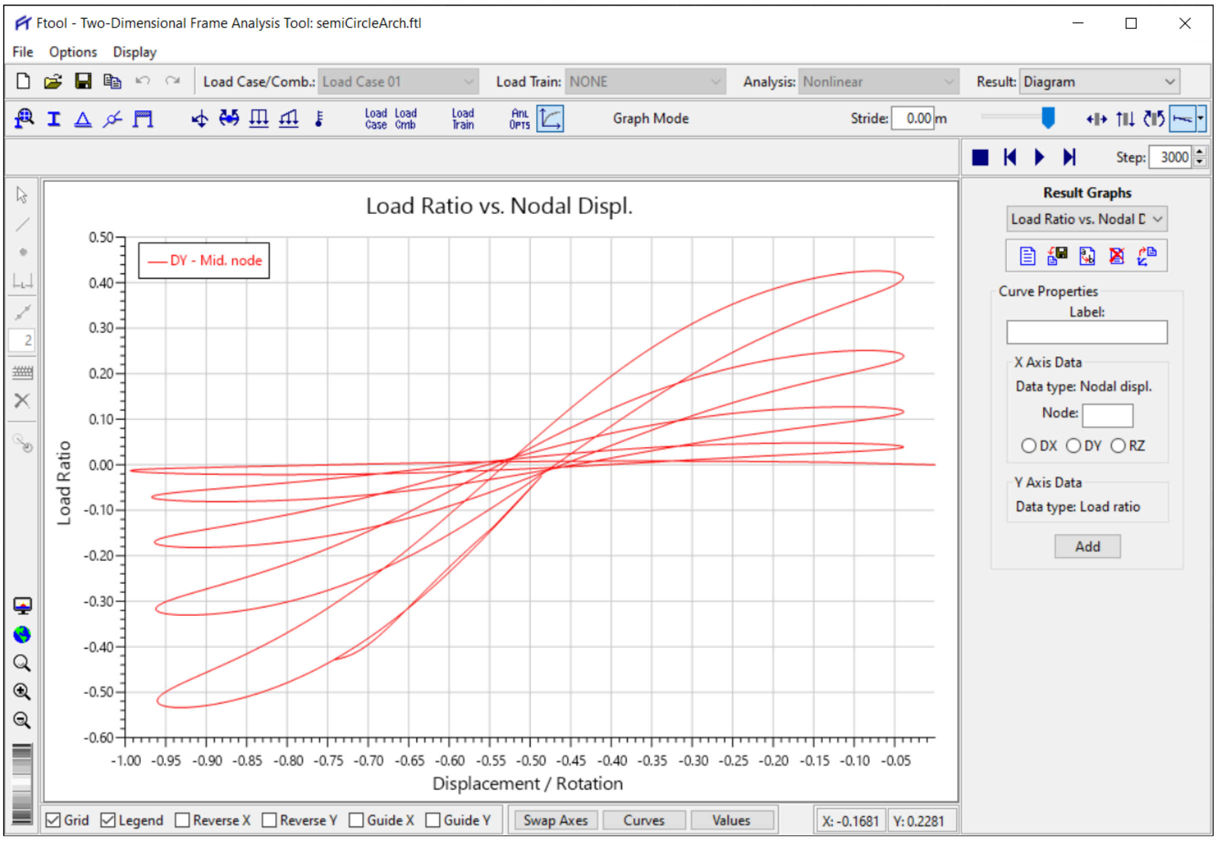

Figure 2. Equilibrium path for the vertical displacement of the middle node of the semi-circle arch model

3 Eigenvalue Analyses

Eigenvalue analyses include two types of analysis that represent different physical phenomena, but are associated with the same mathematical problem: the generalized eigenvalue problem. One of these analysis concerns the stability of the structure, and the other, the natural vibration. Each one relies on a different system matrix to formulate the eigenvalue problem. The numerical algorithm for computing the eigenvalues and eigenvectors is the Jacobi method, adapted from Press et al. [5].

When the eigenvalue analysis is selected, the *Analysis Options* menu provides the options shown in Fig. 3. The Type of Eigenanalysis option can be set to Buckling modes or Vibration modes. In the case of selecting Vibration modes, to evaluate the natural vibration of the structure, the Mass Matrix option becomes available, otherwise this option stays blocked. The desired number of modes to be computed can also be provided.

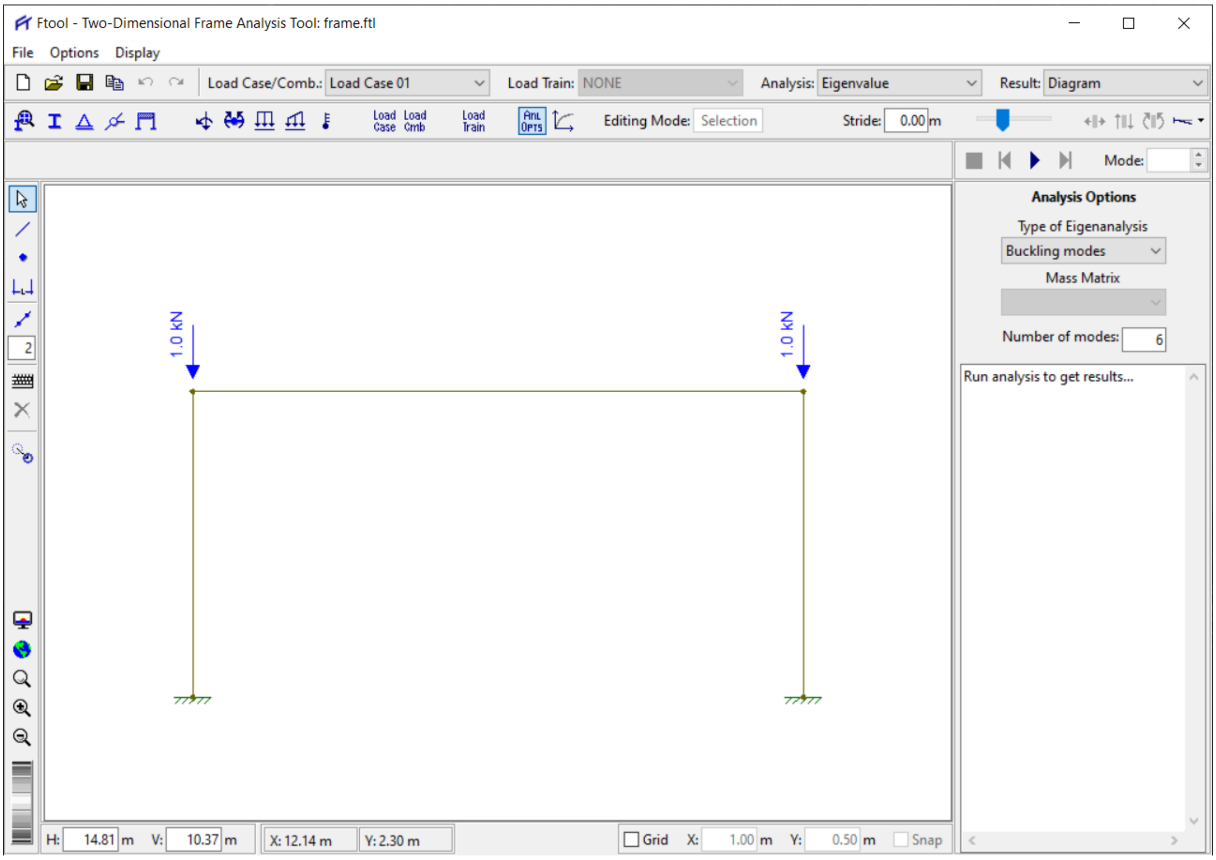

Figure 3. Simple frame model showing the analysis options for an eigenvalue analysis

3.1 Stability analysis

The goal of the stability analysis is to obtain the elastic critical loads, given by load ratios of the total applied load, and the corresponding buckling modes of the structure. The general eigenvalue problem to be solved is given in eq. (1), where $[K]$ is the global elastic stiffness matrix of the structure, $[G]$ is the global geometric stiffness matrix computed with the internal forces resulting from a linear-elastic analysis, λ are the critical load ratios (eigenvalues), and $\{v\}$ are the displacement vectors of the buckling modes (eigenvectors).

$$\left[K\right]+\lambda\left[G\right] \quad v \;=\; 0 \tag{1}$$

Figure 4 presents the results for the stability analysis of the simple frame of Fig. 3, showing the first buckling mode (the one associated with the lowest critical load), which is usually the one in which engineers are interests. Other modes can be visualized by changing the mode number in the text field next to the analysis control buttons. The critical load ratios corresponding to each buckling mode are given in the *Analysis Options* menu.

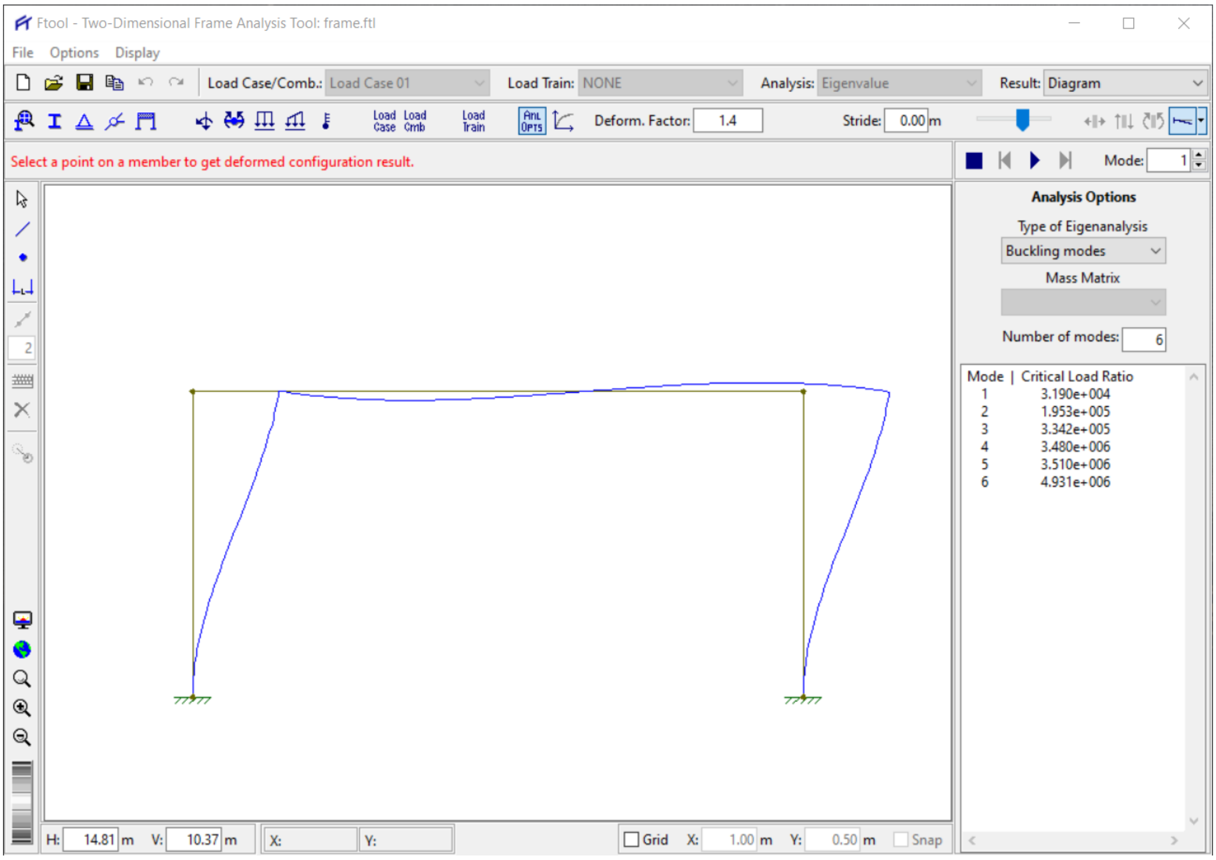

Figure 4. Results for the stability analysis of the simple frame model showing the first buckling mode

3.2 Natural vibration analysis

The goal of the natural vibration analysis is to obtain the natural frequencies and periods, and the corresponding vibration modes of the structure. The general eigenvalue problem to be solved is given in eq. (2), where $[M]$ is the global mass matrix of the structure, ω are the natural frequencies (square root of the eigenvalues), and $\{v\}$ are the displacement vectors of the vibration modes (eigenvectors). No damping is considered in this type of analysis, and the applied loads are ignored.

$$\left[K\right] + \omega^2 \left[M\right] \quad v \;=\; 0 \tag{2}$$

In the *Analysis Options* menu, users have the opportunity to choose between two types of mass matrix: consistent and lumped. The consistent mass matrix of each element is based on a variational formulation, by taking the kinetic energy as part of the governing functional and applying the FEM to interpolate the element velocity field with the same shape functions used for interpolating the displacement field. The lumped mass matrix of each element, on the other hand, is obtained by simply considering that half of its mass is concentrated in each of its end nodes and accounting only for the translational degrees-of-freedom, which results in a diagonal matrix with non-zero terms only in the translational degrees-of-freedom. The lumped mass matrix provides less accurate results than the consistent mass matrix, but it is interesting for simulating mass-spring systems, common in modeling some structures.

Figure 5 presents the results for the natural vibration analysis of the simple frame of Fig. 3, showing the second vibration mode (the one associated with the second lowest natural frequency). The natural frequencies and periods corresponding to each vibration mode are given in the *Analysis Options* menu. Users can use the scale factor slider, located above the analysis control buttons, to manually perform an animation of the vibration modes.

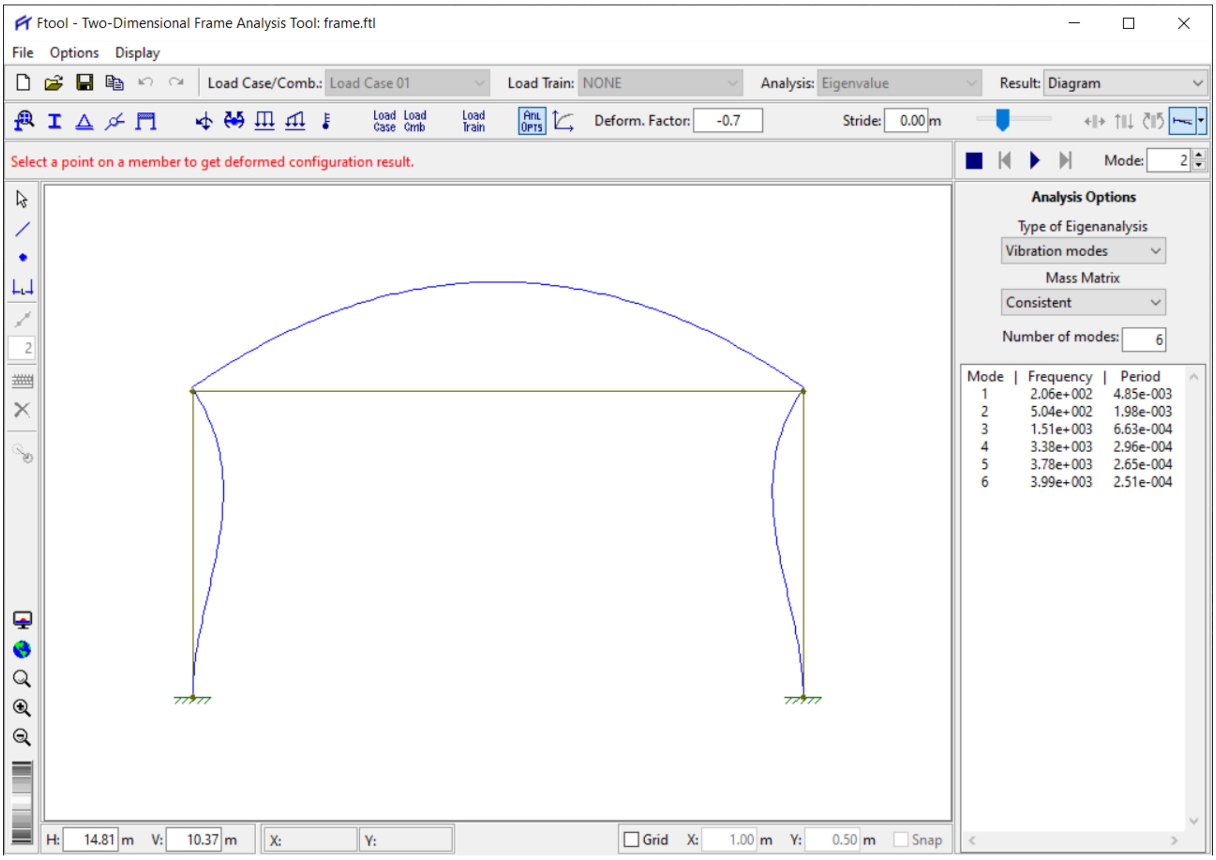

Figure 5. Results for the natural vibration analysis of the simple frame model showing the second vibration mode

4 Conclusions

With the addition of the new features to the Ftool program, it becomes a powerful tool for advanced analysis of two-dimensional frame models. One of the priorities during the development of this work was to keep Ftool philosophy of being an easy to use program with a user-friendly interface. Therefore, the authors expect that the new version of the program will be very helpful for students and engineers who need to execute advanced analysis of frames. In the following versions and sub-versions, it is intended to include the nonlinear behavior of the material and semi-rigid connections as new sources of nonlinearity, and to implement a free vibration analysis.

Acknowledgements. The authors are grateful to Professors Ricardo Silveira and Rodrigo Burgos for the shared knowledge, and to Tecgraf/PUC-Rio for providing the needed resources during the development of this work. This work was financed in part by the Coordenação de Aperfeiçoamento de Pessoal de Nível Superior – Brasil (CAPES) – Finance Code 001.

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. F. Martha, "FTOOL: A structural analysis educational interactive tool". Workshop in Multimedia Computer Techniques in Engineering Education, Institute for Structural Analysis, TU Graz, Austria, pp. 51-65, 1999.
[2] R. L. Rangel. Educational Tool for Structural Analysis of Plane Frame Models with Geometric Nonlinearity. MSc thesis, Pontifical Catholic University of Rio de Janeiro, 2019.
[3] R. M. Souza. Force-Based Finite Elements for Large Displacement Inelastic Analysis of Frames. PhD thesis, University of California, Berkeley, 2000.
[4] M. A. Crisfield. *Non-Linear Finite Element Analysis of Solids and Structures (Vol. 1)*. John Wiley & Sons, 1991.
[5] W. H. Press, S. A. Teukolsky, W. T. Vetterling, and B. P. Flannery. *Numerical Recipes: The Art of Scientific Computing*. Cambridge University Press, 2007.