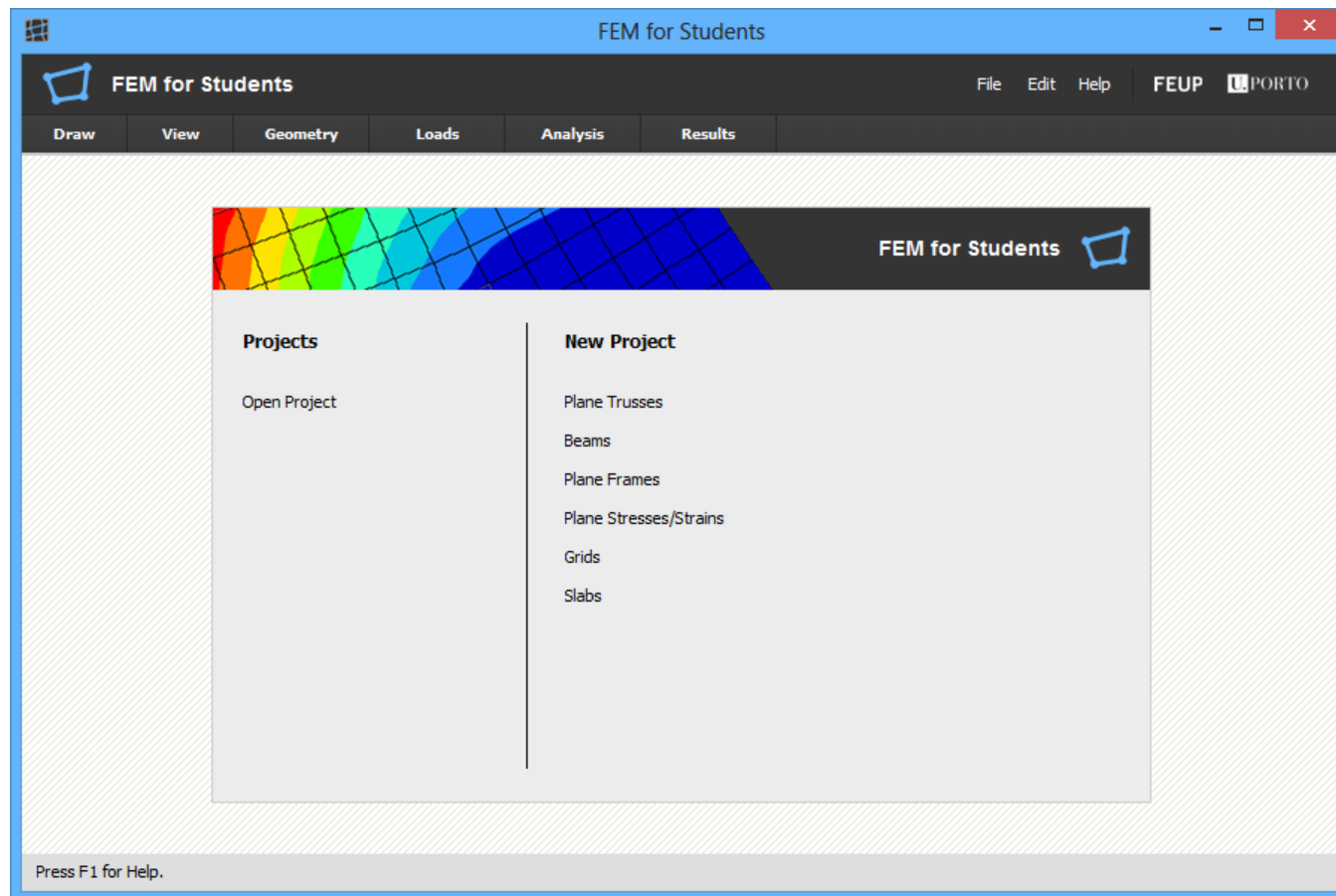# FEM for Students

MODELING AND STRUCTURAL ANALYSIS BY FINITE ELEMENT METHOD

ANDRÉ DE SOUSA
2014

PRESENTATION OF THE FEM FOR STUDENTS

The FEM for Students is a program of modeling and structural analysis by Finite Element Method.

After initialization, the program displays a panel that allows the user to choose the type of structural model that want for your project. This panel that appears in the window center also allows to open an existing project. The program allows the modeling of the generality of the structural problems found in engineering practice and who are subject of study in any university course of Civil Engineering.

PROGRAM REQUIREMENTS

The FEM for Students was written in the Java programming language.

In order to run the program on your computer you must have installed the Java SE 8. The Java Platform Standard Edition platform allows us to develop and deploy Java applications on desktops and servers.

ABOUT THE AUTHOR

The FEM for Students program was fully developed by André de Sousa in Dissertation of Structures submitted to the Faculdade de Engenharia da Universidade do Porto for partial fulfillment of the requirements the degree of Master in Civil Engineering.

INITIAL PANEL OF THE FEM FOR STUDENTS

This panel appears at the center of the program window lets you choose the type of structural model or open a project created earlier.

Of different formulations and types of finite elements available are focused on one-dimensional, two-dimensional, three-dimensional finite elements, beams and slabs. Beam elements are formulated by Euler-Bernoulli theory and Timoshenko theory. The finite elements of slab presented are formulated by Kirchhoff theory and Reissner-Mindlin theory.

Taking into account the combination of different formulations of finite elements, in this first version, the following structural models are available:

- Plane Trusses;
- Beams;
- Plane Frames;
- Plane Stresses/Strains;
- Grids;
- Slabs.

Selecting the type of model to develop the project appears the area of modeling of the program and available tools for the design of finite element and all other elements necessary for structural analysis. In order to make a simple program, such tools are arranged in a set of tabs sequentially arranged by the steps of modeling, structural analysis and display of results.

Other functionalities of the program are concentrated in the File, Edit and Help menus. So that users can store projects created and subsequently retrieve them, the program allows recording projects in the permanent memory of computer.
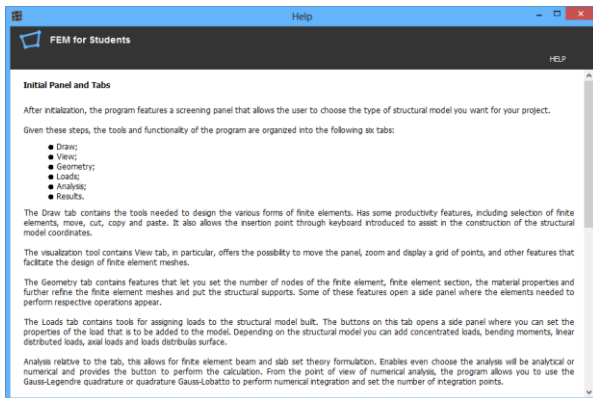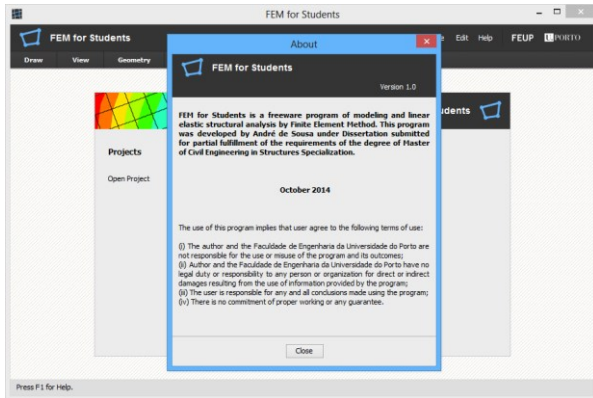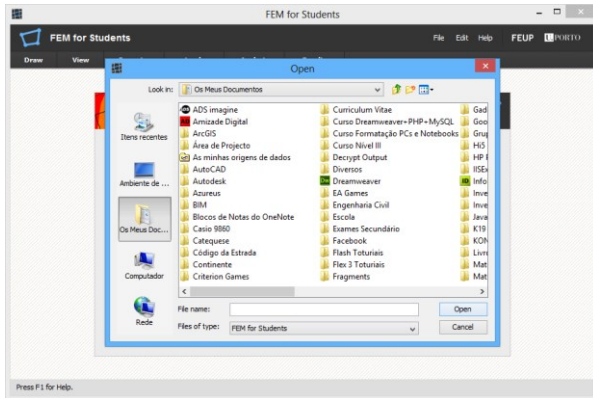
MODELLING WITH FINITE ELEMENTS

The structural model chosen for the project, the program displays the drawing panel and are available all tools for modeling arranged sequentially by a set of tabs.

Thus, the first task is to design a finite element mesh. Then, the remaining details are set as, for example, the number of nodes of the finite element, the material properties or conditions of support. Subsequently, the loads are applied to the structure. After performing the calculation the display features of the results are available.

TABS

The tools and features of the program are organized into the following six tabs:

- Draw;
- View;
- Geometry;
- Loads;
- Analysis;
- Results.

Many of the buttons shown on these tabs open side panels where are available the areas with the features that the user wants to access.
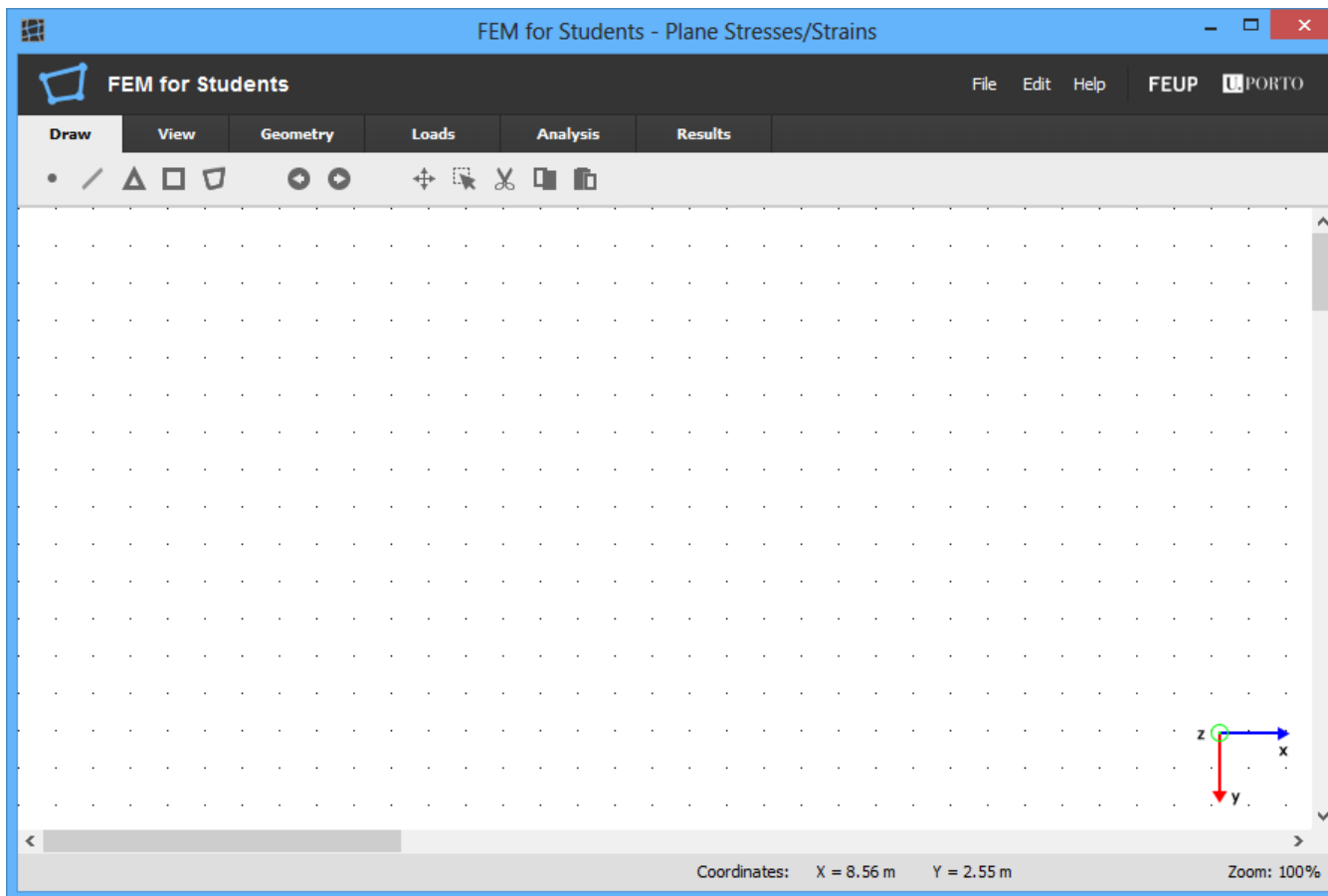
DESCRIPTION OF THE TABS

The Draw tab contains the tools needed to design the different finite elements available in the program. Has a few productivity features, including selection of the finite elements, move, cut, copy and paste.

The View tab contains visualization tools, in particular, offers the possibility to move the panel, zoom and display a mesh points and other features that facilitate the design of finite elements.

The Geometry tab contains the functionality to setting the number of nodes of the finite elements, the remaining properties of sections of finite elements, material properties, refinement of finite element meshes and placing of structural supports.

The Loads tab contains tools for assignment of loads to the structural model built. Depending on the structural model can add loads as concentrated loads, bending moments, linear distributed loads, axial loads distributed, surface loadings and/or consider the self-weight of the finite elements.

For the Analysis tab, this provides, for example, for the finite element beam and slab selection of theory formulation. Enables even choose the analysis will be analytical or numerical and the button to perform the calculation of the structure.

The Results tab, depending on the selected model, it is possible to see, for example, the global system equations, all results the level of each finite element, the deformed shape of the structure, diagrams for bars, maps of stresses and/or stresses and principal directions.
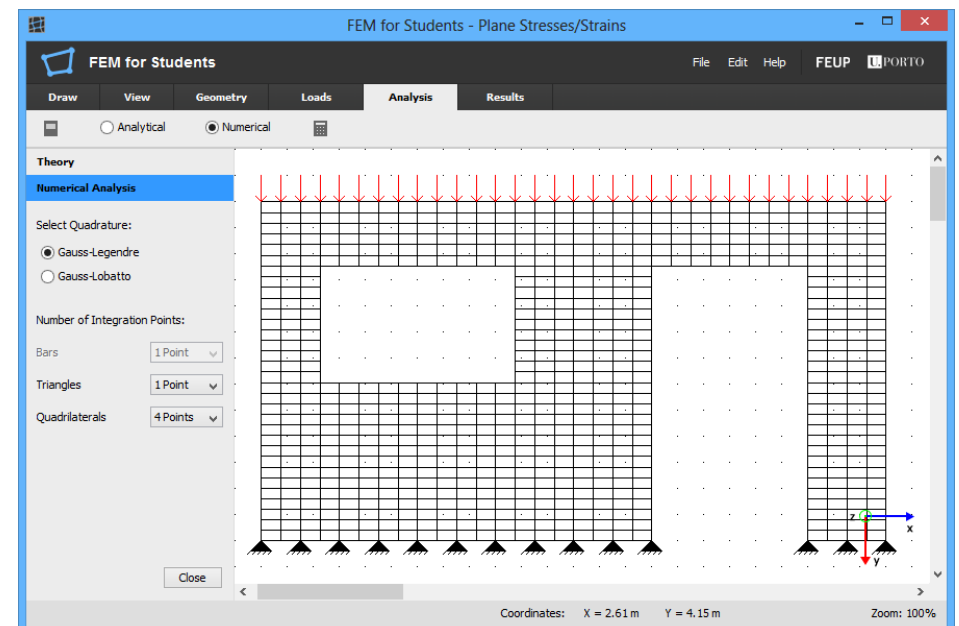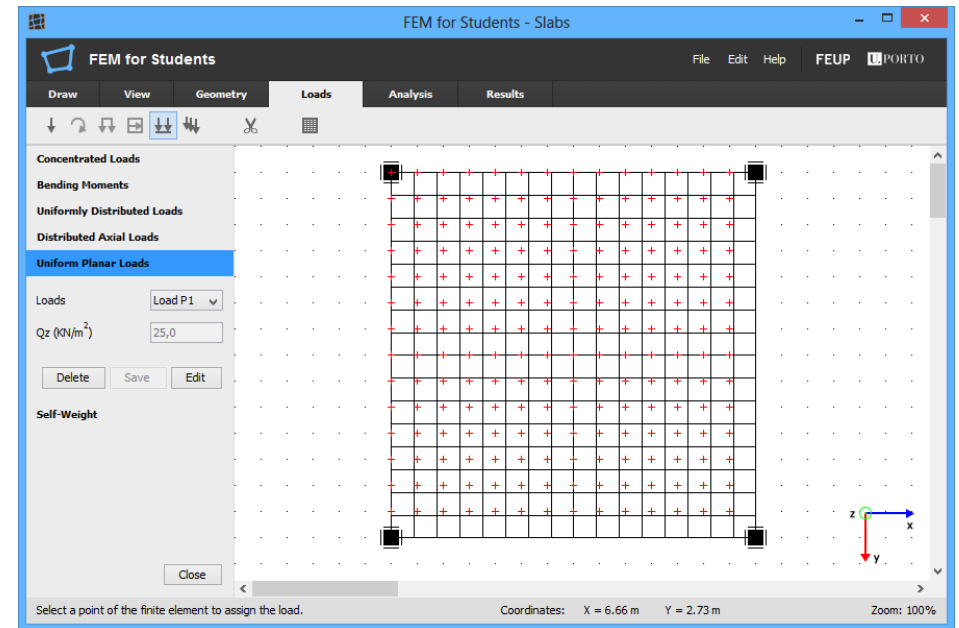
DISPLAY THE RESULTS

In the Results tab are available the options to choose the type of results obtained by finite element analysis.

In the first group of buttons, options for the visualization of the global system of equations are available. In the second group the options for viewing the results in tables at the level of finite elements are available. Thus, besides the global system of equations, you can see the balance equation for each finite element, the support reactions, nodal forces and nodal stresses. In the latter group are concentrated features for graphical representation of results.
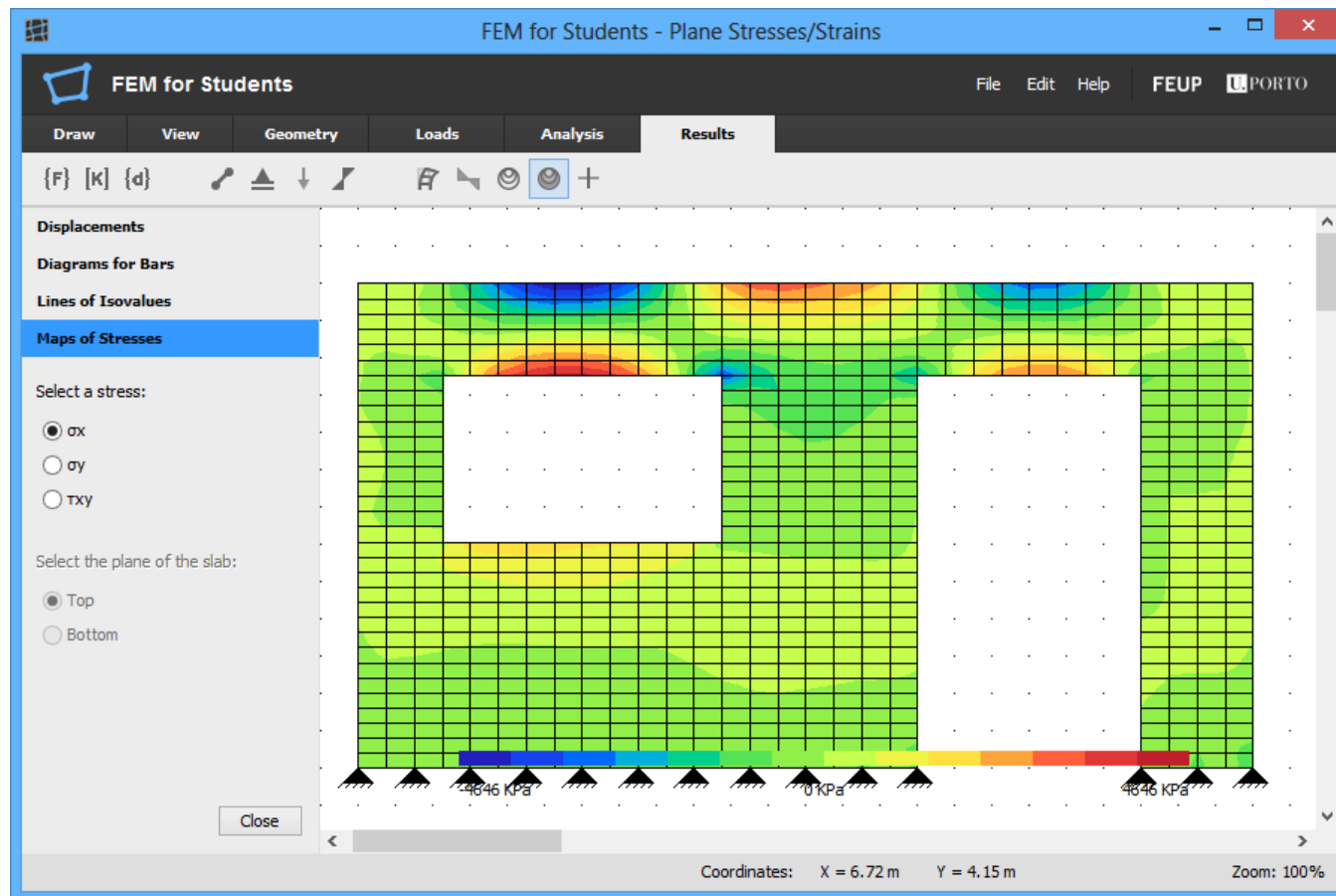
GRAPHICAL REPRESENTATION

For all models it is possible to visualize the deformed structure.

In addition, for models of bars you can view diagrams of forces and for two-dimensional models the isovalue lines, maps of stresses and stresses and principal directions.

Regarding the display of isovalue lines and maps of stresses, the user can choose that stress want to see represented.
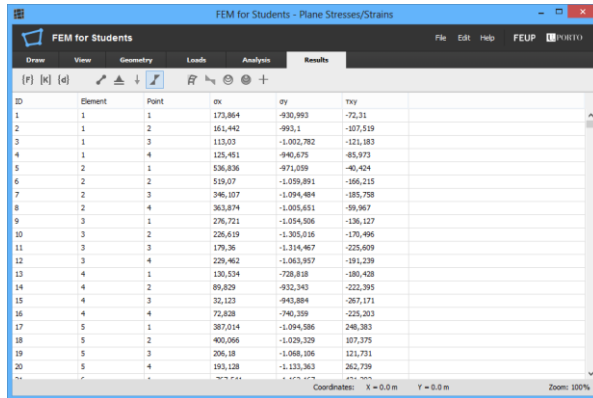
OTHER FUNCTIONS

The developed program allows modeling simple problems with various finite element formulations. Models of discrete systems available allow the study of articulated structures, frame structures, beams and grids. For solid bodies, the user can choose between a plane stress or strain and the study of slabs by the two theories available.

As already pointed out, the program enables the use of different formulations of finite elements. Furthermore, the program comprises most of the structural models used in engineering practice. From the viewpoint of numerical simulation the program allows that the changes to the model are quite intuitive and view them quickly in order to understand their impact on the behavior of the structure.
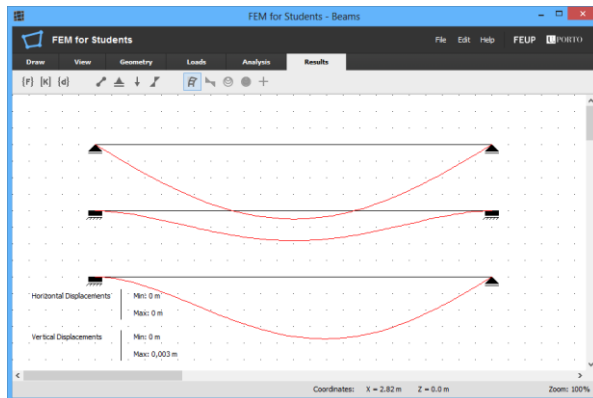
The program also presents, for example, information relating to the positioning of the mouse and tips on the operation of the selected tools. These tips are always presented to the user selects particular tool and are intended to inform you about the operating mode of this. These elements are arranged in the lower pane of the user interface.

The development of this program was part of the theme of the Dissertation in Structures where the author had to create a software tool for modeling and structural analysis by Finite Element Method oriented for university students. Thus, the program was developed in order to allow consolidate the matters addressed in a university course about the Finite Element Method.
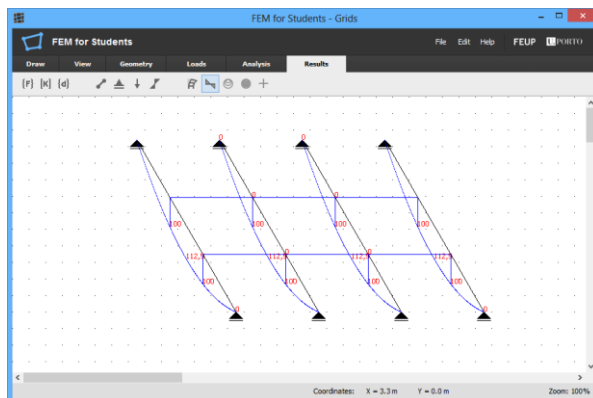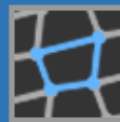
FEM for Students

The FEM for Students is a program of modeling and structural analysis by
Finite Element Method developed by André de Sousa on Structures Dissertation submitted in
fulfillment of the requirements for the degree of Master of Civil Engineering.

asousafilipe@hotmail.com