An iStress Laboratory

Sergio Gallegos Cázares, Alejandra Balderrama López
Department of Civil Engineering,
Tec de Monterrey, Campus Monterrey, Av. Eugenio Garza Sada Sur 2501, Engineering Laboratories Building, Of. 206K, Monterrey N.L., México
sergio.gallegos@itesm.mx

Abstract

The development of a stress analysis laboratory for iOS is presented here. An app that presents a library of engineering mechanics experiments which can be manipulated using iPad or iPhone devices. Engineering students can visualize stresses and displacements with this app. Gestures are used to manipulate the types of stress and displacement contours, scale colours, and query the value at a point. A finite element model is used to interpret data and to perform reanalysis of modified models by the user.

Keywords: iPad, mobile devices, finite element, app development, engineering apps.

1. Introduction

Mobile platforms such as iPads and iPhones are continuously growing in number and in capacity. They are small wonders of technology combining high definition graphics screens, powerful central and graphics processors, a wide variety of sensors and a powerful operating system. In addition, since they have become consumer products, their prices are affordable for the general public, and many students own one or other type of tablet or mobile phone.

Applications developed for mobile devices or apps are abundant. There are apps for games, productivity, hobbies, sketching, art, and many other things; however, apps combining numerical methods, engineering procedures and graphics to produce educational tools are very scarce.

This paper is about the development of an app that simulates the behavior of different classes of structural members to help engineering students in the understanding of structural mechanics concepts. It is called StressLab2D.

The app, still under development, is implemented in Objective-C using Cocoa Touch and runs in iPad and iPhone mobile devices. It is an interactive program that allows the student to choose among different types of structural experiments that exemplify structural behavior. By playing with these experiments one can visualize different types of stresses produced inside test coupons and the corresponding displacement patterns. In addition, one can modify the original boundary conditions triggering an automatic reanalysis that shows the effects of these modifications in the behavior of the coupon. It is expected that by playing with this app the engineering student may increase his sensitivity to structural behavior and develop a better understanding of some important abstract mechanics of materials concepts.

2. The structure of iStress Lab

2.1 The library of experiments

The laboratory uses a Table View to present a repository of numerical experiments as shown in figure 1. The experiments are grouped in sections:

- Axially loaded bars.
- Beams.
- Frames.
- Plane stress.

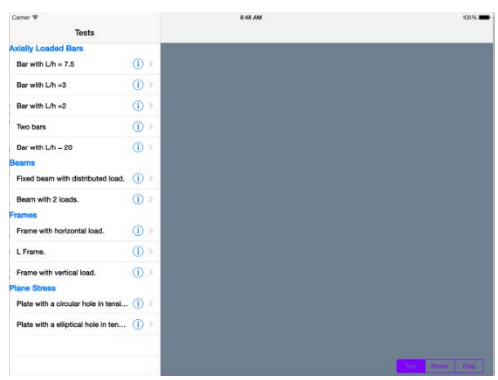

Figure 1. Groups of experiments.

Each group of experiments presents some typical laboratory tests for that kind. For example, for axially loaded bars, a sequence of axially loaded members with increasing aspect ratio is available. With this sequence it is possible to study the effect of aspect ratio of the member and justify the bar behavior for only some proportions.

For any selection the specific information for each experiment is presented by selecting the accessory button. In figure 2, the specific information for the axially loaded bar with aspect ratio L/h = 3 is presented.

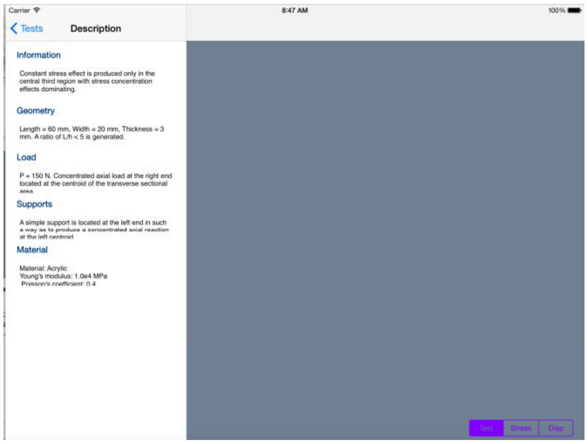

Figure 2. Specific information for the axially loaded bar with L/h = 3.

Specific information contains a brief description of the experiment, the geometry of the coupon, the type of load and support conditions, and the material characterization data.

A library of experiments is generated as plist files in an external finite element program. Then all the specific information about each experiment is organized in a directory, which is also a plist file. Only plist files can be transferred into an iOS app.

When a user presses a row of the table of experiments on the iPad screen a ViewController receives the action and calls the experiment store, which selects the corresponding plist file as shown in figure 3.

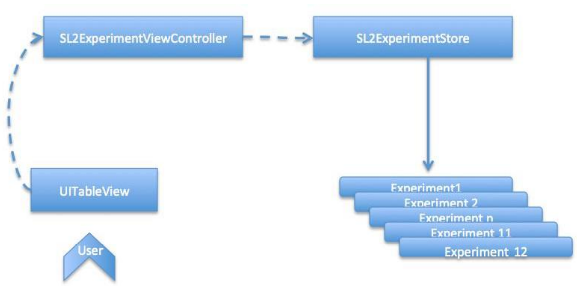

Figure 3. User interaction and selection of the experiment data.

2.2 The finite element model

In order to make sense of the plist data a finite element model is instantiated with it. The FE model knows about elements, stresses, displacements, connectivity and geometry of a model and can be queried for data. In this case, the same finite element program used externally to create the model is used inside the app to recreate the model. Xplore [1] was created to this end using Objective-C and an object oriented approach.

2.3 The graphics interface

Once a FE model has been recreated inside the app its graphical representation must be created too and loaded on the main graphics screen. An object named GLModel uses OpenGL ES to plot all graphical aspects of the model and its results on this screen. In figure 4, the result of selecting the model of a beam with two loads is presented. On this model, the supports are represented with red markers, while the loads are represented with blue markers. The boundary conditions are presented here in order to make clearer the experimental set up of the test. Also, a texture is added to make the look more appealing.

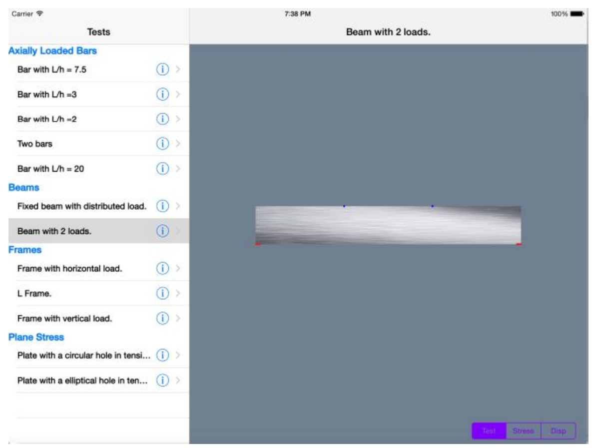

Figure 4. Model for a beam with two loads.

A Model-View-Controller strategy is used to relate the graphics view and the model data for each experiment. This is a common strategy for apps developers.

2.4 Stresses and displacements

A toggle is presented at the bottom right corner of the screen in figure 4 to select stress or displacement contours for this experiment. In figure 5 the contours of the Von Mises stress are presented on the deformed configuration of the beam. By tapping on the figure the stresses change from Von Mises, maximum shear, horizontal stress, vertical stress and shear stress.

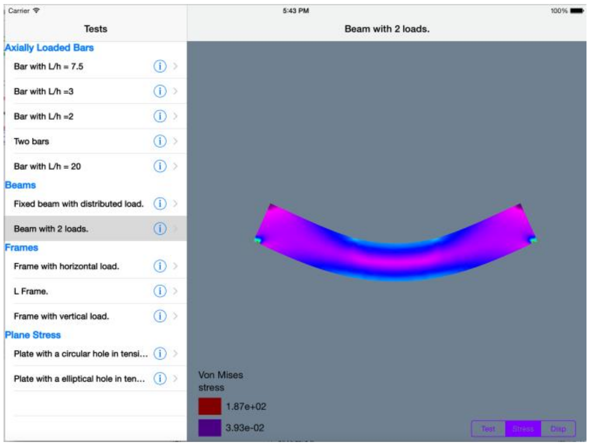

Figure 5. Von Mises stress on deformed shape.

In figure 6 the vertical displacement contours are plotted on the deformed configuration. Again, by tapping on the figure the displacement contours change from horizontal displacements, vertical displacement and displacement magnitude.

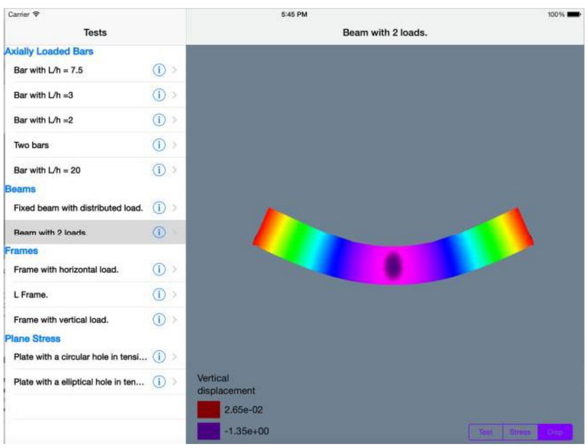

Figure 6. Vertical displacements on deformed shape.

A nice feature is that by tapping on the color scale, the colors used for plotting are changed from 2, 5, 7, 17, 25, 50 and 305 different colours. The user may select the scale that best suit his taste. An example of this is shown in figure 7.

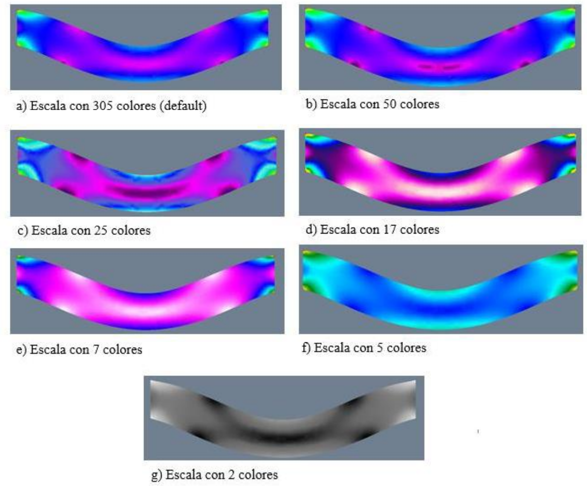

Figure 7. Different color scales obtained by tapping on the scale.

3. Analysis capabilities

An important characteristic of our app is the possibility to modify the original FE model. Since the user is assumed to be a freshman or sophomore engineering student that is not an expert using finite element programs all changes must automatically set a new FE model without user intervention. At this moment only loads are allowed to be changed during the initial view of the experiment. Loads are selected by pointing with the finger and the closest load is edited as shown in figure 8.

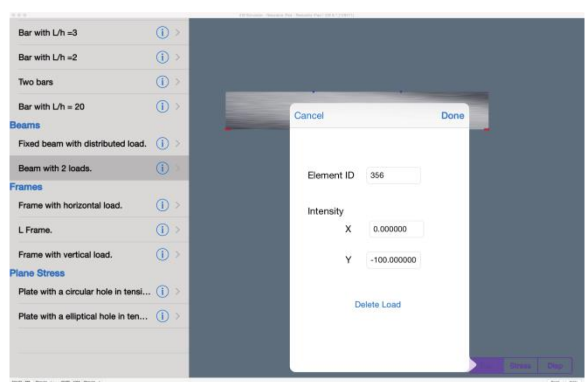

Figure 8. Load edition or deletion.

If the load is modified, the model is marked for reanalysis when the toggle is pressed to show stresses or displacements again. The reanalysis is possible due to existence of the embedded finite element program Xplore.

4. Results

A description of the current available experiments in StressLab2D is presented here.

4.1 Axially loaded bars

Figure 9 presents a sequence of bars with different aspect ratios, L/h, which can be used to discuss the importance of this parameter in the structural behavior of bars.

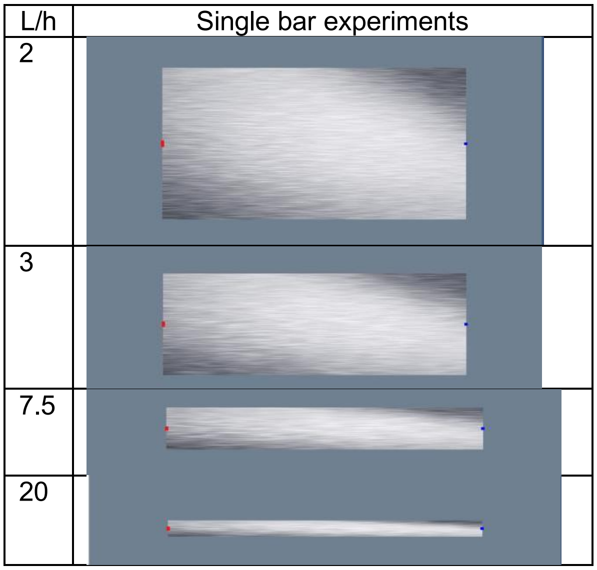

Figure 9. Bars with different aspect ratio.

In figure 10 the Von Mises stresses for each of the bars of figure 9 are presented. The concept of stress concentration can be introduced and discussed by looking at these plots. The subjective aspect ratio L/h > 5 used to categorize bars may also be justified from these plots.

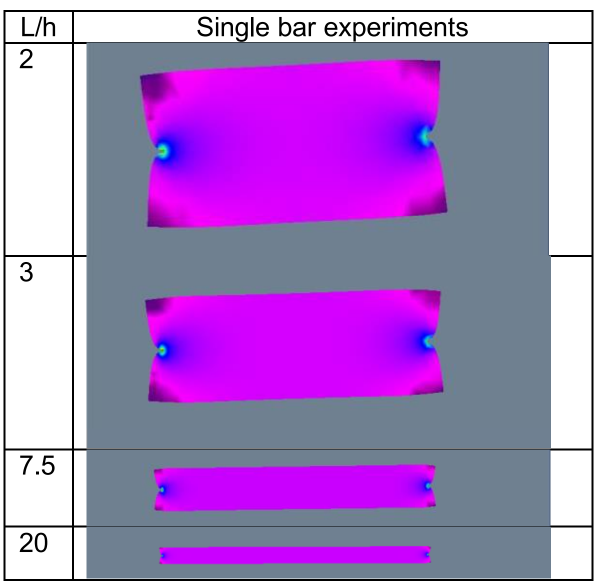

Figure 10. Von Mises stress for bars with different aspect ratios.

In figure 11, a two bar structure is presented with its Von Mises stresses. This experiment shows how a single piece of material behaves as two different mechanical bars due to the change in geometry.

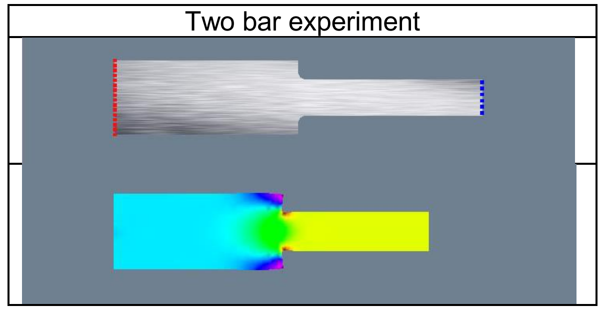

Figure 11. Two bar experiment and its Von Mises stresses.

4.2 Beams

A classical structural member studied in any mechanics course is the beam. The beam behavior is characterized by the change of longitudinal stress from top to bottom of the beam. Figure 12 shows the two beam experiments included in StressLab2D. The first one is the simply supported beam with two point symmetrical load. The second one is the fixed-fixed beam with uniformly distributed load.

Figure 13 shows the horizontal stress distributions for these two common experiments. The whole classical beam theory can be derived from the stress obtained at the central region of the two point load experiment.

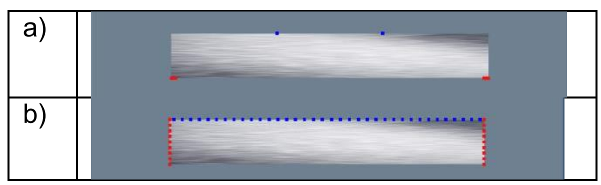

Figure 12. a) Simply supported two point loaded beam. b) Fixed-fixed beam with distributed load.

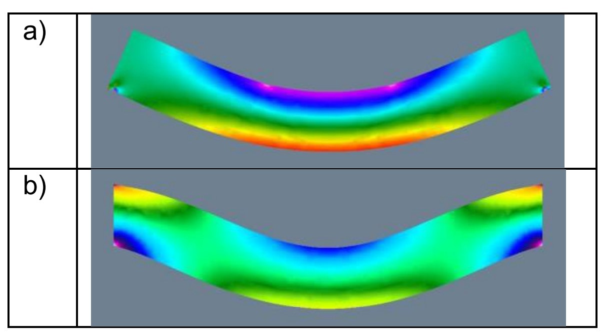

Figure 13. Horizontal stress distribution for the beams of figure 12.

4.3 Frames

Framed structures are used frequently in construction. Three different frames are included in StressLab2D: A simple frame with lateral load, a simple frame with uniform vertical load and an L frame. These frames are shown in figure 14 with the Von Mises stresses in figure 15. These are examples of a more complex structural behavior and the interaction of structural members can be explained from here.

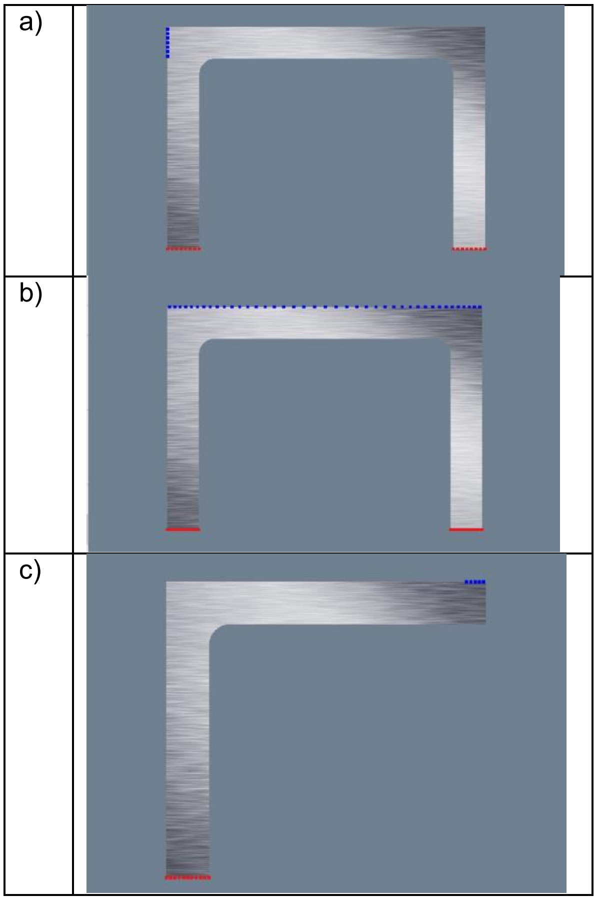

Figure 14. a) Simple frame with lateral load. b) Simple frame with uniformly distributed vertical load. c) An L frame fixed at the base.

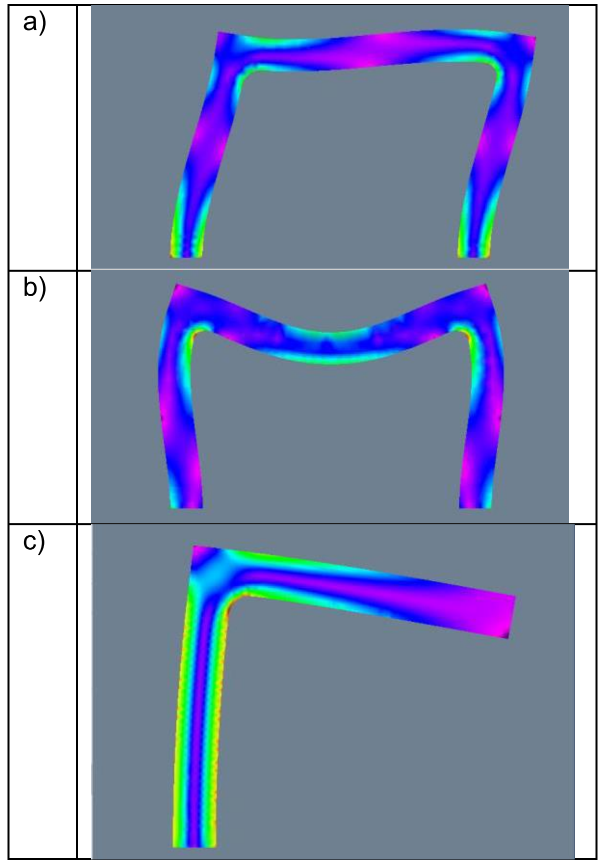

Figure 15. Von Mises stresses for the frames of figure 14.

4.4 Stress concentration

Another difficult concept for mechanics students is that of the stress concentration factor that is produced at geometric discontinuities. StressLab2D includes two examples of stress concentration produced in a plate under a tensile stress field as shown in figure 16. The first is the stress concentration around a circular hole and the second around an elliptical hole. The Von Mises stress distributions are shown in figure 17.

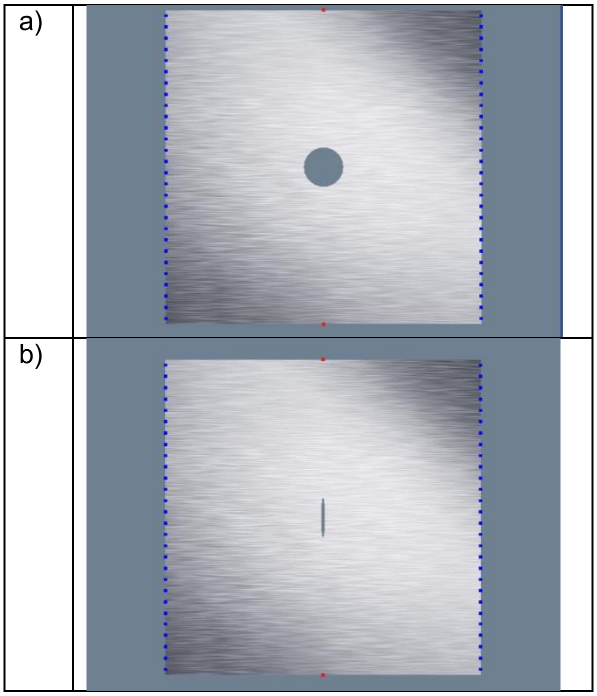

Figure 16. a) Stress concentration around a circular hole. b) Stress concentration around an elliptical hole.

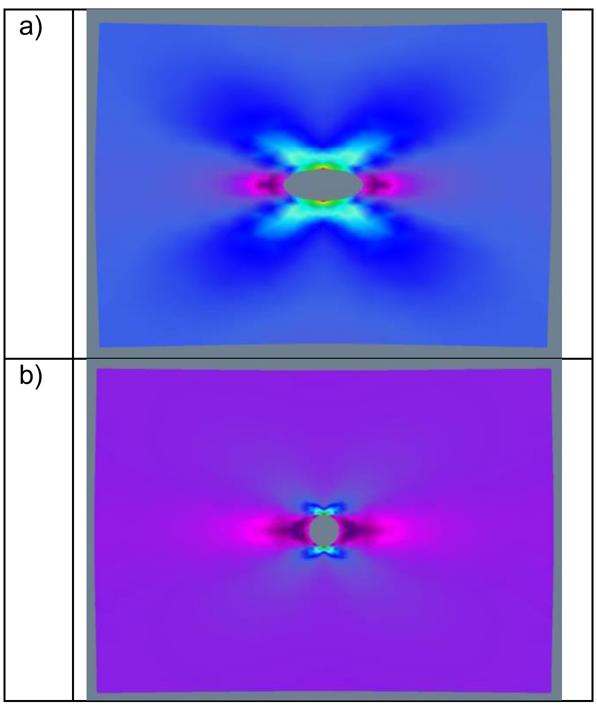

Figure 17. Von Mises stresses for the plates of figure 16.

5. Conclusions and future work

StressLab2D is our first attempt to develop apps for engineering students. The general objective is to help them in the understanding of abstract physical concepts typical of engineering mechanics. Extensive use of engineering numerical analysis is used in the development of this tool, particularly of the finite element method. An object oriented finite element code was developed specifically for this application which can run in OSX and iOS. The Mac version is used for the generation of detailed experiments which are then transferred and translated into the iOS mobile devices. The FE model allows for the manipulation of the test data which is then transformed into graphics objects with openGL ES. An automatic reanalysis capability is incorporated which regenerates an FE model when the load is modified without any user intervention. This is important since the user is not assumed to be an expert FE analyst.

Further work needs to be done measuring the speed of response in different mobile devices in order to assess their suitability for structural analysis. Additional work needs to be done trying different types of graphics in the presentation of results which may be more useful in the understanding of engineering concepts.

6. Acknowledgements

The authors would like to acknowledge the support from the NOVUS 2013 fund for this project.

7. References

[1] S. Gallegos Cázares, “Un Código de Elementos Finitos Orientado a Objetos con Objective–C y Mapas Mentales”, *Sexto Congreso de Métodos Numéricos en Ingeniería Morelia-Michoacán* (2013).