LEARNING THE FINITE ELEMENT METHOD AIDED BY EXPERIMENTS

Marcilio Alves, maralves@usp.br
Caio Cesar Fukumori Brasilino, caio.fukumori@gmsie.usp.br
Miguel Angel Gonzales Calle, miguel.calle@gmsie.usp.br
Renato Toshio Yamassaki, renato.yamassaki@gmsie.usp.br
Larissa Driemeier, driemeie@usp.br
Carlos Alberto Nunes Dias, candias@usp.br
Group of Solid Mechanics and Structural Impact, University of Sao Paulo, Sao Paulo, Brazil

Abstract. *The main purpose of this article is to present a learning/teaching approach of the finite element method (FEM) in the context of structural mechanics, by comparing numerical results with experimental ones. A set of four experiments involving the dynamic and static response of simple mechanical structures was developed. These experiments were made in class and the measured results compared with the ones provided by the students FE programs. Various manifestations of the students were very positive in the way FEM, programing and experiments were merged.*
Keywords*: finite element method, experiments, learning methodologies.*

1. INTRODUCTION

Curricula of engineering schools are organized around theoretical and practical courses, which are usually taught independently from one another. This lack of interplay can be somewhat deleterious for young students seeking ways to see their vocational choices be fulfilled. Although important courses, such as the ones related to mathematics, should be more theoretically orientated, there are courses that, by blending theory and experimentation, would be more attractive to students. Such courses could, at a later stage, be revisited by the student and their theoretical aspects emphasized.

Books on finite element (FE) analysis either emphasize the mathematical aspects of the method, or programming techniques and applications. Whatever the approach, it is not in the scope of finite element text books to motivate the student by the use of experiments. Being aware of such a scenario, the senior authors decided to create a finite element course applied to structures in which experiments would be used to motivate the students, showing them, at the same time, the limitations of the modeling procedure.

A set of somewhat simple experiments that can be used in a finite element course are presented herein. The experiments use simple structures, loaded statically and dynamically. A salient feature to be also described is that the experiments are compared with a commercial finite element software and with the programs developed by the students. This tripartite study warrants a much more in-depth learning of the finite element technique, at the same time raising the necessary motivation for one to pursue further related studies.

The course taught by the senior authors is divided into 13 3-hour lectures, according to Table 1. The students are asked to write their programs as the course progresses and they end up with some 5 different programs. Some students, later in the course, assemble all these programs under a single main code, in which choices of the various finite elements and cases are implemented. It is observed that, as the course unfolds, the students are more motivated and they further sophisticate their program by implementing both data pre and post processing, the latter being more common. Indeed, the FE results shown here and referred to as *in-house program*, present the results as yielded by the junior author program. As for the adopted language, most of the students opt for Matlab, but codes in Fortran, C++, Maple and Excel were also developed by the students.

Students are evaluated through exercises given at each lecture. Every two to three lectures, students should hand in the various programs, with a report in which comparisons with the experiments are performed. An exam is also applied at the end of the course and the final lecture gives a final opportunity for the programs to be handed in.

Table 1 – Finite element course outline.

Lecture	Subject
1	Introduction to solid mechanics Matrix analysis of 1D structures
2	Introduction to the finite element method Applications: structures, fluids
3	Fiinite elements: exercises
4	Finite element for beams Programing the finite element method
5	Experiment 1: frame Modal analysis of beams
6	Experiment 2: Modal analysis of beams Dynamic response of beams: implicit/explicit methods
7	Experiment 3: beam loaded dynamically Plane finite element Isoparametric finite element
8	Numerical integration
9	Experiment 4: Plate with a hole
10	Exam
11	Hand in of the FE programs

2. FRAME STRUCTURE

After three lectures, the students are introduced to the beam finite element and to programming techniques. They are asked to implement a program with beams finite elements. The beams can have any orientation in a plane, thus axial forces and coordinate transformation are also considered. This program is then used to analyze the frame shown in Fig. 1. The frame is built with riveted 20 × 1.6 mm aluminium strips and the load is applied by dead weights at points C, via a cable and a pulley, and A, via dead weights hooked at this point. The horizontal displacement of point B is measured by a laser vibrometer.

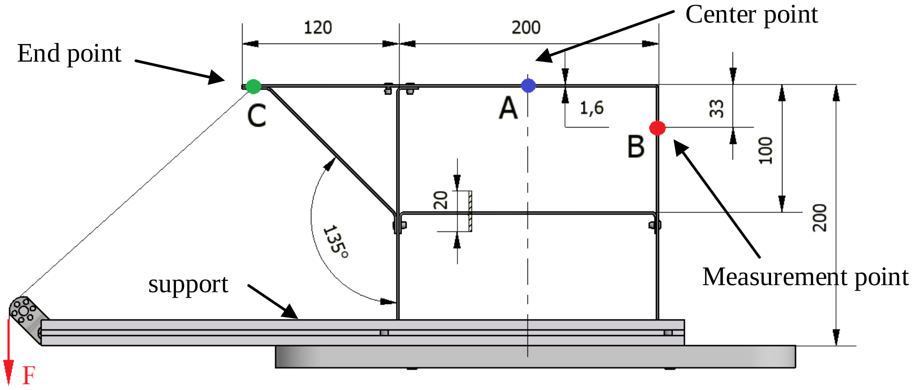
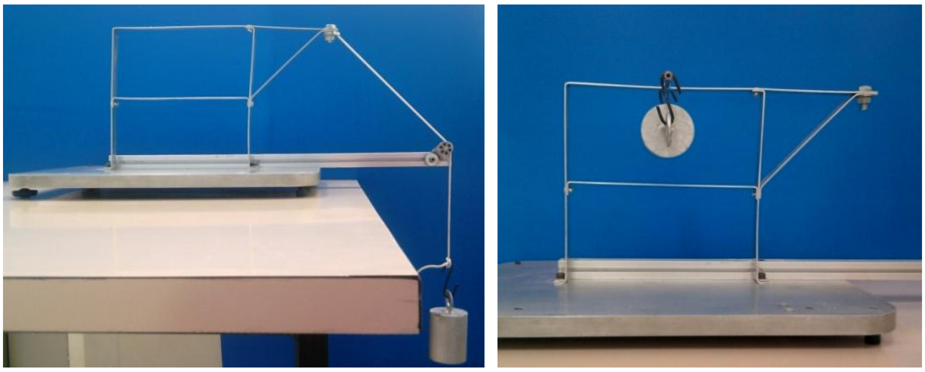


Figure 1 – Geometry of the portico structure, in mm.

The elastic properties of the aluminium are obtained via a tensile test in a universal test machine, with repetitive load and unload steps in the elastic range. The elastic modulus was measured to 70GPa. The material density, to be used in the dynamic analysis, was assumed to be 2768 kg/m^3.

Two concentrated loads are applied separately to the structure, as shown in Fig. 2. In the first test, a 1.13 kg load was applied at 45° at point C in Fig. 1. In a second test, a 1.22 kg load was applied at the central point A in Fig. 1.



(a) (b)

Figure 2 – Layout of experimental tests of the structure, for load at (a) point C and (b) point A.

The measurement system illustrated in Fig. 3 is used to measure the horizontal displacement of the structure at point B in Fig. 1. The system comprises a laser sensor, a data acquisition board and a computer, together with a routine developed in Matlab language to acquire the laser signal.

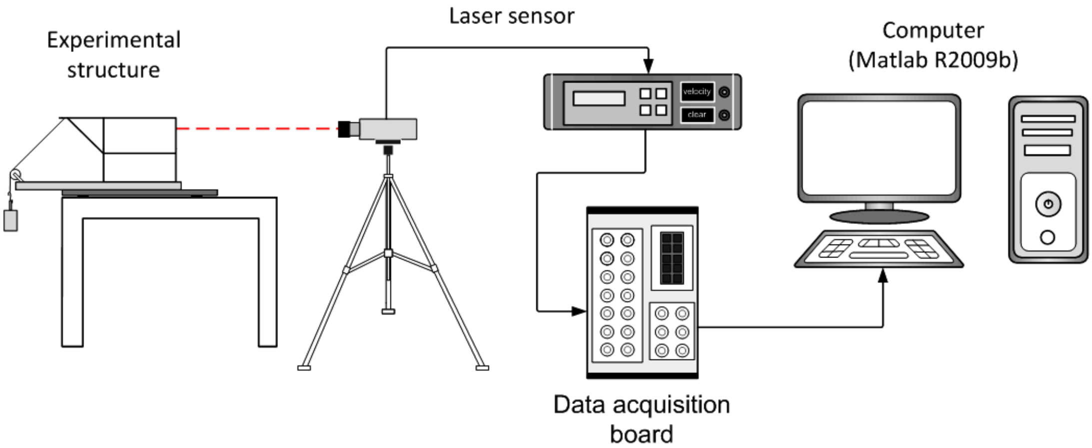
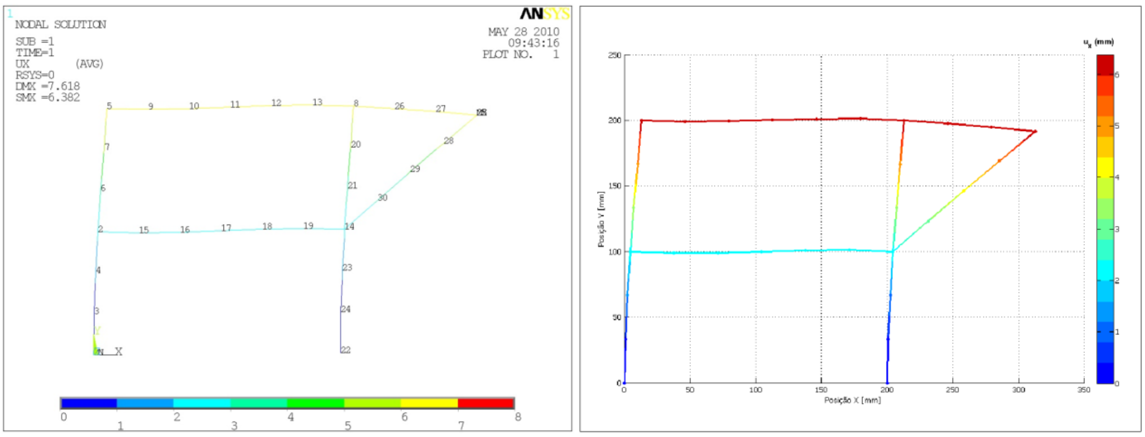
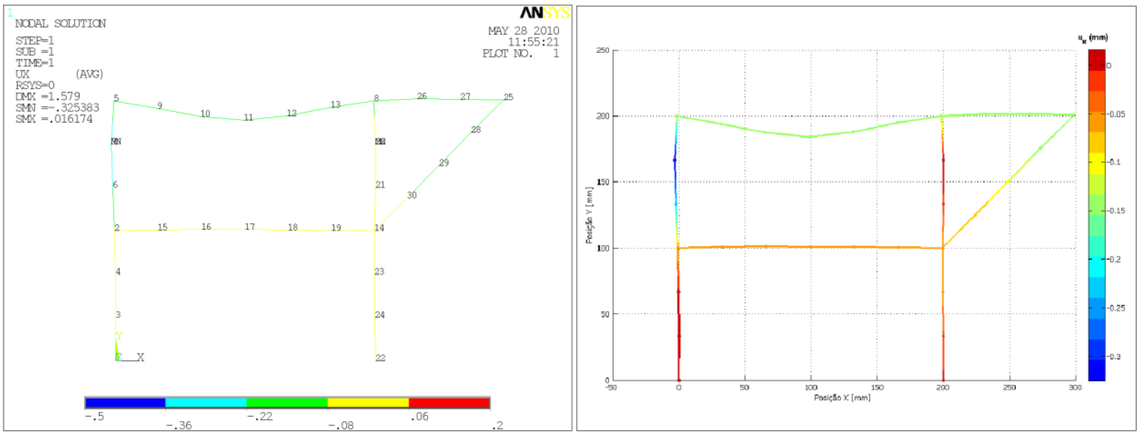


Figure 3 - Measurement system for experiment 1.

A finite element simulation of this structure, loaded in the two configurations shown before, was carried out in Ansys v.10.0, and the results were taken as a reference. FE analysis is also performed using an in-house software developed by one of the students during the course, who this paper co-authors. A quasi-static structural analysis is considered and displacements of the whole structure after loading is analyzed. The model for both programs is created using 31 beam elements. Fig. 4 depicts the displacements of the frame structure, aided by a color scale. Table 2 lists the displacement of point B, including the experimental result.



(a) (b)



(c) (d)

Figure 4 – Displacements, in mm, of the frame for load applied at (a),(b) point C and (c), (d) point A. Figs. (a), (c) and (b), (d) show Ansys and in-house code results, respectively.

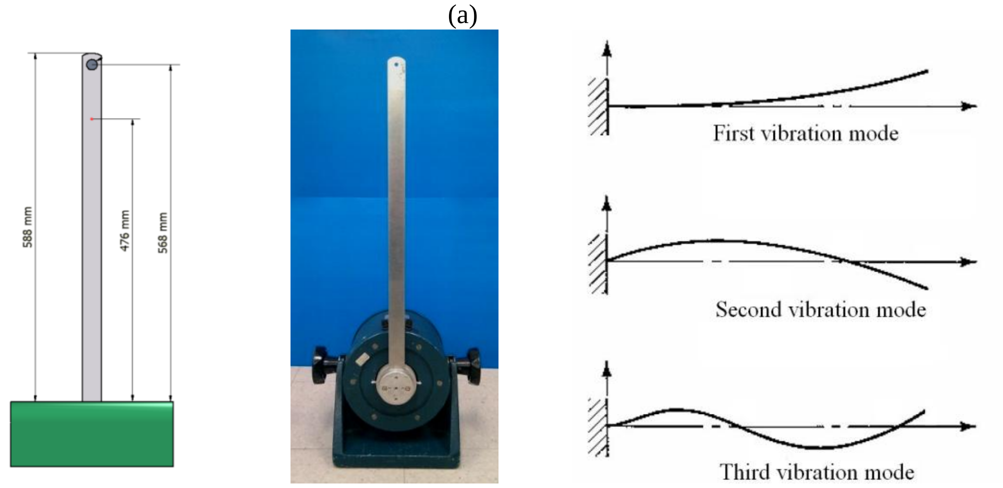
Table 2– Displacement of the portico structure.

Point	Load (kg)	Displacement at point B (mm)				
		FE (Ansys)	FE (in-house)	Experimental (Laser)	Error (Ansys/Exp)	Error (in-house/Exp)
A	1.22	0.33	0.32	0.45	26.66 %	28.88%
C	1.13	5.13	5.13	5.63	8.88 %	8.88%

3. MODAL ANALYSIS OF A CANTILEVER BEAM

During the FE course, a review on vibration of discrete and continuous simple systems is presented together with an experiment, in which different excitations are applied to a cantilever, a simple 588 × 30.82 × 1.2 mm stainless steel rule, fixed in a shaker as depicted in Fig. 5. The first three resonant frequencies of the beam are searched by visually observing maximum displacements and the expected modes of vibration shown in the figure. Here, the students react with a sense of wonder, specially if the experiment is improved by using a strobe light in the darkened classroom.

(a)



(a) (b) (c)

Figure 5 – Cantilever beam: (a) geometry; (b) setting of the beam on the shaker and (c) first three theoretical vibration modes.

The measurement system adopted is depicted in Fig. 6. It includes a wave generator (to control the oscillation frequency of the shaker), an amplifier (to amplify the wave generator output) and a scope (to aid in the visualization of the signals). In a somewhat more sophisticated system, an accelerometer is used (and its respective charge amplifier and capture data board) fixed at the top end of the beam. The accelerometer output can then be compared with the input to the shaker signal and the exact natural frequency of the beam measured when the phase attains a 90-degree value.

In terms of finite elements, the students are introduced to the concept of mass matrix at this stage. At some position of the global mass matrix, it is possible to add a concentrated mass to mimic the accelerometer, if necessary. Basic modal analysis and modal parameters extraction concepts are now introduced and the students are asked to implement features to their basic beam finite element program so as to make it capable of performing the required modal analysis.

The material parameters are not measured but taken from the open literature as E=195 GPa, Poisson coefficient ν=0.3 and density ρ=7850 kg/m^3. Damping was thoroughly disregarded.

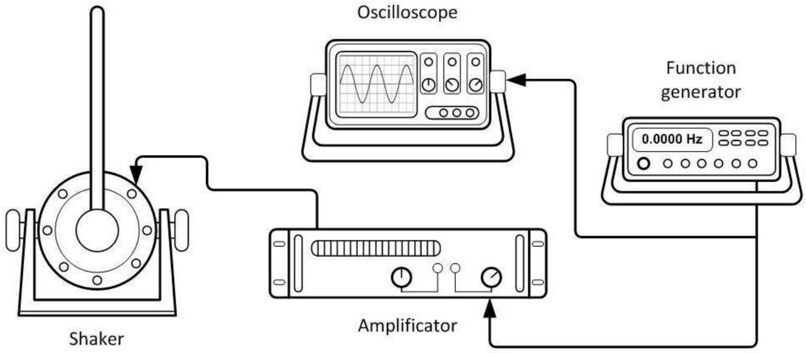
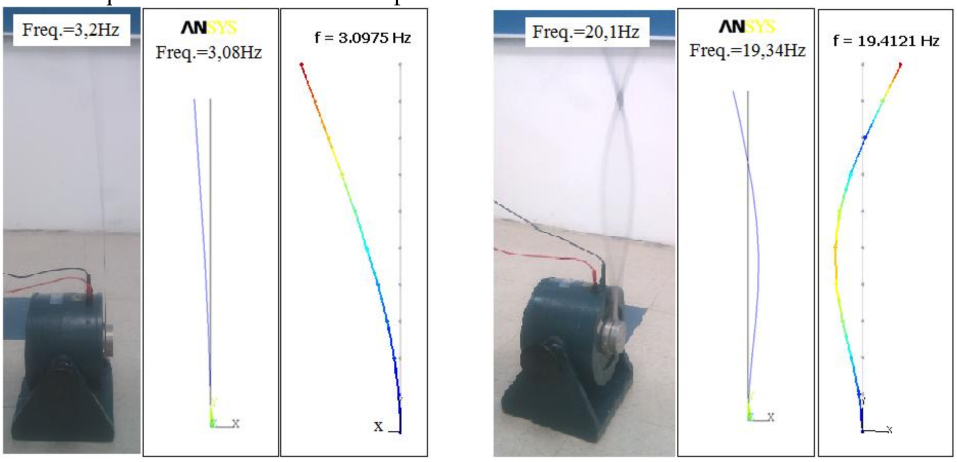


Figure 6. Measurement system for the modal analysis experiment.

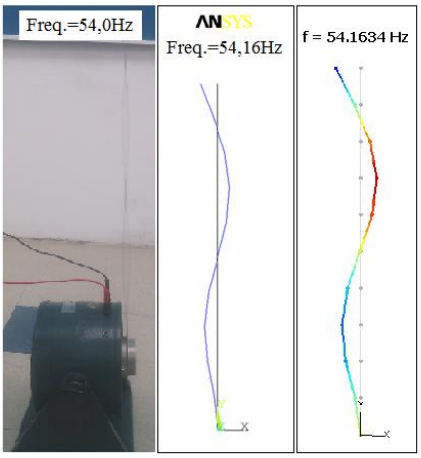
The results of the experimental tests, together with the ANSYS and in-house FE code, are depicted in Figs. 7 (a), (b) and (c) for the first, second and third natural frequencies of the cantilever beam, respectively. The natural frequencies of a cantilever beam are also calculated theoretically as,

$$\omega_{\mathrm{i}} = (nL)_i^2 \sqrt{\frac{EI}{wL^4}} \tag{1}$$

with $(nL)_i$ being the coefficient related to the natural frequency of the *i*th mode. For the first four numerical solutions, one has: $(nL)_1$ = 1.875, $(nL)_2$ = 4.694, $(nL)_3$ = 7.855 and $(nL)_4$ = 10.996. The theoretical, numerical and experimental results for the natural frequencies are listed and compared in Table 3.



(a) (b)



(c)

Figure 7 – Experimental and numerical results for the (a) first, (b) second and (c) third natural frequencies of the stainless steel cantilever beam.

Table 3. Natural frequencies for the cantilever beam.

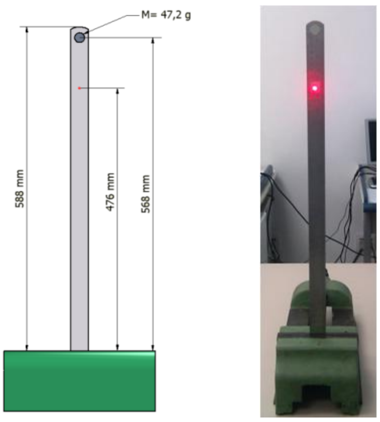
	Natural frequencies (Hz)				Error	
	FE (Ansys)	FE (in-house)	Theoretical	Experimental (Shaker)	FE/Exp (Ansys)	FE/Exp (in-house)
Mode 1	3.087	3.097	2.754	3.2	3.53%	3.22%
Mode 2	19.350	19.412	17.262	20.1	3.73%	3.42%
Mode 3	54.193	54.163	48.339	54.0	0.36%	0.30%

4. FREE VIBRATION OF A CANTILEVER BEAM

This experiment was devised as an application of the time integration techniques learned around the middle of the course. Explicit and implicit time integration techniques are taught and the students are asked to implement both techniques in their program. By now, they have a FE program capable of handling linear static and dynamic analysis of frame structures of any size, with the common boundary conditions.

In terms of experiments, the same stainless steel cantilever beam used in the modal analysis experiment was adopted. A concentrated mass of 47.2g, in fact an accelerometer, was installed at the end of the beam, adding a small difficulty to the analysis. The layout of this experiment is shown in Fig. 8.

The experiment is run by applying an initial displacement of around 40 mm in the free edge. The amplitude of displacements during free vibration was measured using the same system employed in the first experiment. The natural damped frequency of the experimental test was obtained from an optimization algorithm programmed in Excel.



(a) (b)

Figure 8 – Layout for the free vibration experiment.

The displacement of the analyzed point in the beam, both experimental and from the in-house program with 59 beam elements, is depicted in Fig. 9 for the first 5 seconds. The results for ANSYS are not shown for the sake of clarity since they overlap with the in-house FE program results.

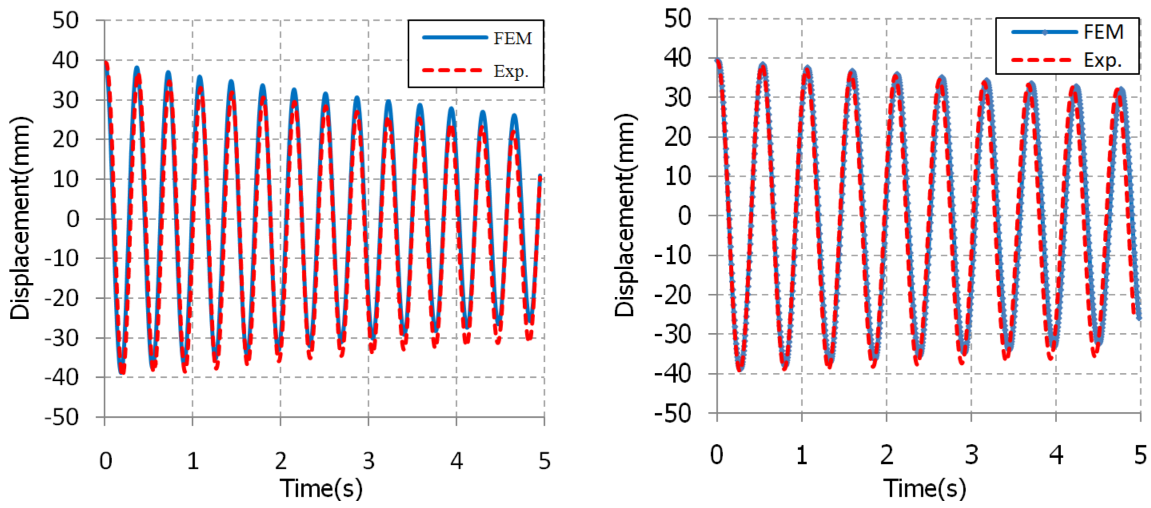


Figure 10 – Experimental and in-house FE tip displacement for the beam during free vibration, (a) without and (b) with added mass.

5. PLANE STRESS ANALYSIS OF A PLATE WITH A HOLE

Towards the end of the FE course, students are introduced to plane stress-strain and axisymmetric finite elements. Isoparametric FE and numerical integration are by now all known and students are asked to write a FE program to handle 2D linear problems.

Following the strategy adopted in the course, another experiment is performed, this time a plate with a hole pulled by a standard tensile machine. The aluminium plate in Fig. 11 is pulled only in the elastic regime and the hole region is monitored by taking a series of high definition pictures. These images are then processed by the software 7D, which adopts digital image correlation (DIC) techniques. This allows measuring the strains around the hole, which can be used for comparison with the FE programs.

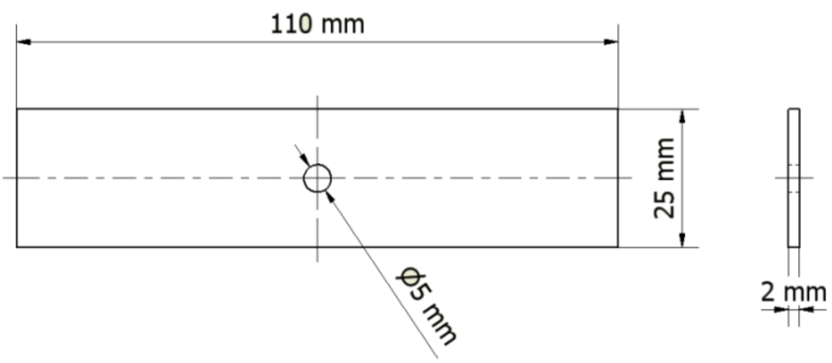


Figure 11 – Plate geometry.

The instrumentation shown in Fig. 12 includes: universal machine Instron 3369, tripod, camera Nikon D60 DSLR (Digital Single Lens Reflex), lens Vivitar series 1L 105 mm f 2.5 macro and LED lighting system. The resolution of the camera is 10.2 MP and its settings are ISO 400, aperture f22 and exposure time of 1/30 seconds. These parameters guarantee low noise, good light sensitivity and best image focus, all important characteristics to obtain a good contrast quality in the images for the post-processing in the 7D software. The LED lighting system guarantees a uniform illumination along the entire test.

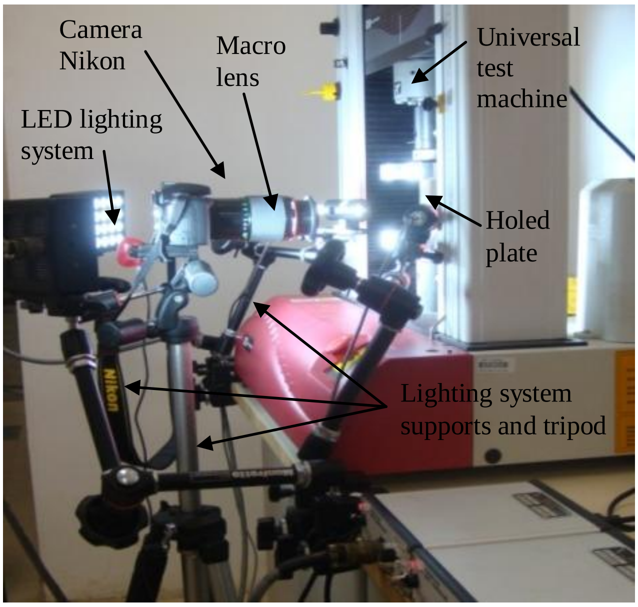


Figure 12 – Photo instrumentation and lighting system for the test.

The mesh for both FE and DIC are shown in Fig. 14. For a load of 4.8 kN and using standard elastic properties for the aluminium plate, the ANSYS, in-house FE programs and the DIC measurement of strains in the load direction at the middle section of the plate adjacent to the hole are listed in Table 3.

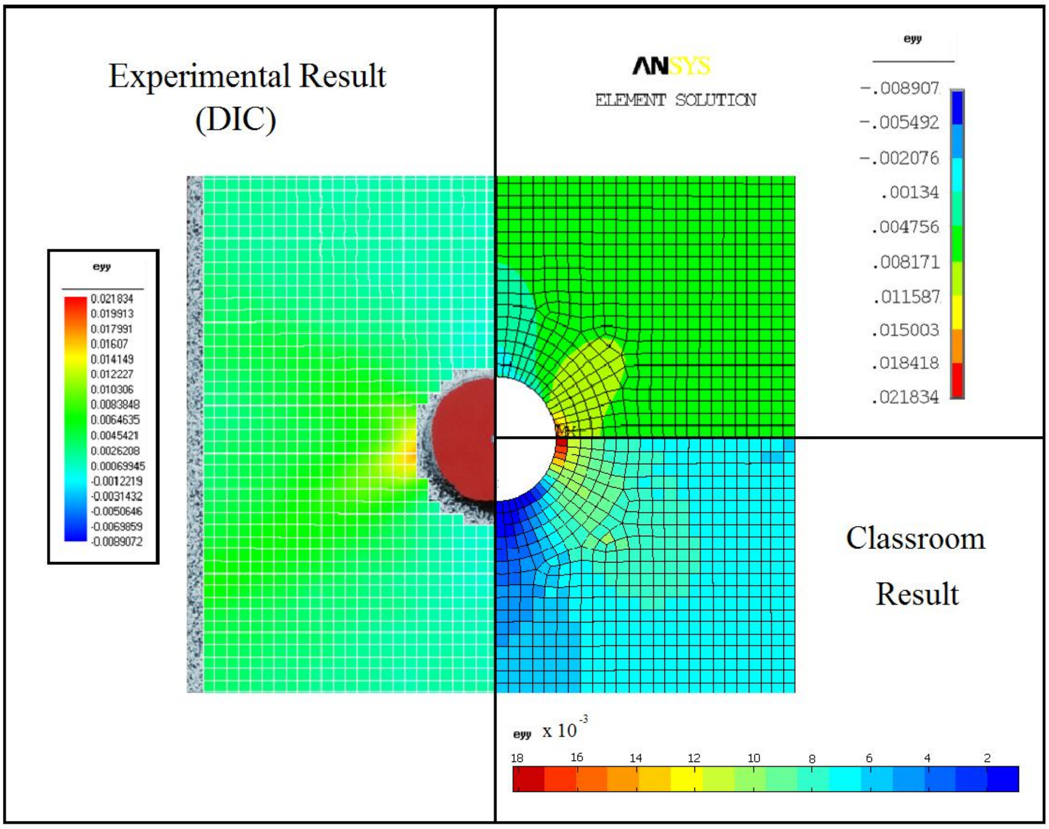


Figure 14 – Results for the strain in the y direction.

Table 4 – Maximum strains in *y* direction obtained in the region adjacent to the hole.

	Theoretical	FE (Ansys)	FE (in-house)	Experimental DIC	Error (Ansys/Exp)	Error (inho/Exp)
Maximum strain	0.0259	0.0204	0.0182	0.0175	11.69%	4.0%

6. CLOSURE

The experiments described here were chosen along the years the FE course was taught. At the beginning, the course was more theoretically oriented. Later on, the programming side was emphasized and this was seen as an already very important step in motivating students. Indeed, it was felt that programming the FEM is a fundamental aspect in the learning process, paramount for understanding the method.

Given the experimental facilities in the authors' research group, it was natural to introduce some experiments in the course as a matter of students' motivation. Only recently were these experiments used as a tool to compare the FE results yielded by the students' program with the ones measured.

It should be stressed that the experiments are carried out during the class. The experimental results are listed in Tables in the blackboard or in a computer with data show. Depending on the class, on the day of the experiments, some students had their programs ready and a comparison could be made at once. To the wonder of the class, this comparison usually gives good results, as shown in the various tables.

Clearly, the use of experiments in teaching FEM is also helpful for the definition of the boundary conditions and the way loads should be applied. Such a discussion is important in class, contributing to the maturity of the students in the structural analysis field.

Positive comments by the students at the end of the course only reinforce the overall impression of the senior authors that the use of experiments in teaching bridges the theoretical and real world in a very appealing way, with advantages for both realms.

7. ACKNOWLEDGEMENTS

The authors would like to thank Engs. Leonardo M. Mazzariol and Rafael T. Moura for their help with image processing.

8. RESPONSIBILITY NOTICE

The authors are the only responsible for the printed material included in this paper.