Integrated graphical environment for support nonlinear dynamic software for the analysis of plane frames

H. Rodrigues
Department of Civil Engineering, University of Aveiro, 3810-193 Aveiro, Portugal

H. Varum
Department of Civil Engineering, University of Aveiro, 3810-193 Aveiro, Portugal

A. Costa
Department of Civil Engineering, University of Aveiro, 3810-193 Aveiro, Portugal

ABSTRACT: Nonlinear structural analyses allow reproducing in a more realistic sense the behaviour of structures subjected to several types of complex loading conditions, e.g. earthquakes. However, it is largely recognized that these analyses normally generate a considerable amount of results, being difficult its interpretation. Over the last years considerable progresses have been made in structural nonlinear behaviour modelling, associated to the fast growing development of numerical algorithms for structural analysis and computer capacities. However, a similar growth in the development of graphical results visualization tools has not been witnessed. To face this, a graphical processor called VISUALANL was developed for an existing nonlinear dynamic analysis program for plane frame structures, PORANL.

1 INTRODUCTION

Numerous programs developed over the past at the academic level for nonlinear structural analysis are not being used by engineers because, in most cases, these programs lack graphical interfaces, a downside creating strong barriers to their use. Traditionally, data input for these programs was carried out through extensive text files, with a pre-defined rigid sequence. Normally, the results are also presented in long text files, without any or poor graphical visualization.

To face this, a graphic tool called VISUALANL was developed to provide a user frontend for an existing nonlinear dynamic analysis program for plane frames, PORANL (Varum, 1996). By using VISUANL, introduction and modification of data are easier thanks to interactivity. Graphical representation at run-time is possible using dialogue windows. It should be noted that the philosophy followed in this graphical tool does not enforce a rigid sequence in the data input. For results interpretation, the program provides an easier visualization by supplying a series of graphical representations suited for different needs.

The development of VISUANL, which incorporates the necessary aspects for preparation, manipulation and visualization of a structural analysis problem, that may consider material nonlinearity and dynamic behaviour, was carried out in the programming language Visual Basic, version 6.0 (see for example Thayer, 1999) that enables an easy implementation of the necessary graphical concepts and tools.

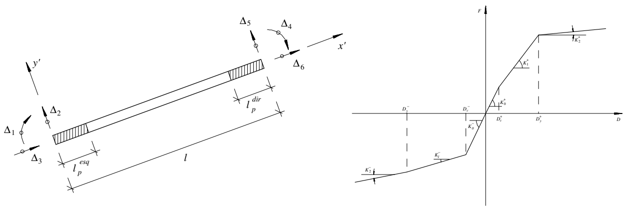
Besides highlighting the potentialities of the graphical processor VISUANL at the levels of data generation and results visualization, this article briefly describes the structural models available in the program PORANL.

2 AVAILABLE BEHAVIOUR MODELS

2.1 *Structural behaviour models*

Varum (1996) developed the initial version of the nonlinear analysis program PORANL in which a hysteretic behaviour model suited for seismic analysis. The non-linear model implemented is the Costa-Costa model (1987, 1996), a modified Takeda hysteretic model suited for seismic analysis of reinforced concrete (RC) frame elements predominantly subjected to bending.

The structural member modelling strategy associated to this hysteretic model considers that inelastic deformations are concentrated in the vicinity of the member's extremities, in which nonlinear behaviour is expected under earthquakes. Each structural element is a macro-element subdivided into three sub-elements (as represented in Figure 1-a), namely a central sub-element with linear elastic behaviour, connected at each end to a sub-element with nonlinear behaviour (plastic hinge). The behaviour of each plastic hinge under cyclic loading is represented by the referred hysteretic model. This model enables the numerical modelling of several known behaviour characteristics of members subjected to earthquake-type loading, namely: i) stiffness degradation with load reversals; ii) strength degradation due to cycling loading; iii) pinching effects accounting for the importance of shear forces; iv) slipping effects accounting for reinforcement slippage; v) possibility to define different monotonic envelopes for positive and negative bending. The reader is referred to CEB (1996) for a more detailed description of these structural effects.



a) b)

Figure 1. a) Macro-element of a structural member; b) Monotonic envelope for RC cross sections under bending

The behaviour of each plastic hinge under monotonic loading is represented by a trilinear moment-curvature envelope, Figure 1-b, for each bending direction, thus allowing the analysis of asymmetric RC cross sections. Trilinear moment-curvature envelopes of RC cross sections can be obtained using a fibre model approach as the one implemented in the program BIAX developed by Vaz (1996). The possibility to use the BIAX program was also included in the graphical framework of VISUALANL (see Figure 2).

In the fibre model approach, cross sections are divided into slices (in the case of uniaxial bending) or filaments (in the case of biaxial bending). In the fibre approach, the cross section constitutive behaviour is obtained from the uniaxial behaviour of the fibres but shear strains across the section are not modelled. The basic assumptions that cross sections remain plane after deformation and that only small deformations take place are also considered. Fibre modelling allows for the calculation of the field of axial extensions in the section by equilibrium as a function of the axial extension and curvatures. Based on the deformation of each fibre, stress at each fibre can be obtained from the stress-strain relation of each material. Then, integrating the stress field across the cross section yields the internal forces.

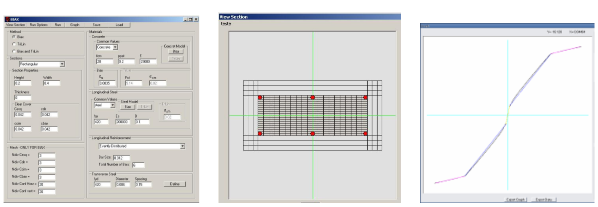


Figure 2. Graphical interface for the fibre model: data generation, section visualization, nonlinear monotonic envelope

2.2 *Infill masonry model*

It is a common misconception that masonry infills in structural RC frames can only increase the overall lateral load capacity, and therefore, must always be beneficial to seismic performance. Even when relatively weak, masonry infills can drastically modify the global stiffness of the structure and consequently the structural response, attracting forces to parts of the structure that have not been designed to resist them (Paulay and Priestley, 1992).

A hysteretic model for the simulation of the cyclic behaviour of infill masonry panels based on the equivalent truss model, as presented in the Figure 3, was implemented in the program PORANL by Rodrigues (2005). In this model, each infill panel is represented by diagonal trusses with linear behaviour and a central element that simulates the nonlinear behaviour of the infill panel. The nonlinear behaviour of each panel defined by the central element is represented by an hysteretic model, contemplating important effects, such as: i) stiffness degradation with load reversals; ii) strength degradation due to cycling loading; iii) pinching and slipping effects accounting for the influence of shear forces.

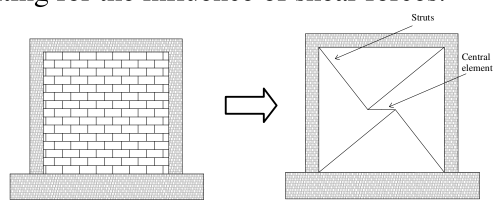


Figure 3. Proposed equivalent truss model

Monotonic behaviour of the panels is characterised by a piecewise-linear envelope with five branches for each loading direction, as represented in Figure 4, thus allowing for the analysis of infill panels with non-symmetrical behaviour e.g., a panel with a non-centred opening.

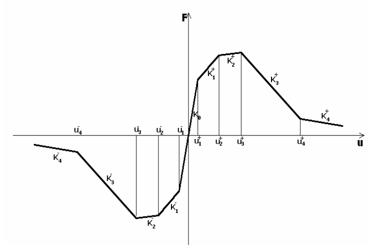


Figure 4. Monotonic envelope for the infill panel element

3 PRE-PROCESSING DATA

The graphical framework VISUALANL integrates both pre and post-processing operations, necessary to carry out a structural analysis run. The basic steps of data input using this new interface, namely, definition of geometry, constraints, loads , material characteristics (for members having linear or nonlinear behaviour) are described in the following. Description of the pre-processor steps will be carried out in a logical sequence for the definition of a structural problem. However, one of the major advantages of VISUALANL is the possibility of data definition without following to a rigid sequence.

3.1 *Geometry of the structure*

The geometry of the structure can be defined in two different ways (see Figure 5): i) defining an initial support mesh for the structure's geometry, establishing the number of storeys and bays, and corresponding dimensions; ii) importing a DXF file with the complete structural geometry. After either of the previously referred forms for geometry definition, the user is able to change the length of a bay or the height of a storey, add or delete frames or change the coordinates of the nodes.

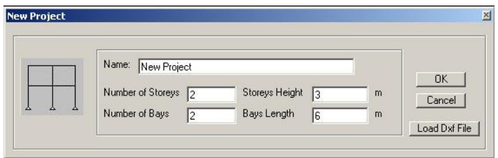
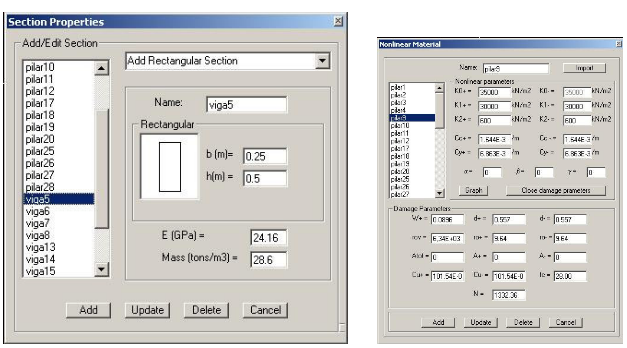


Figure 5. Input of the geometry of the structure

3.2 *Linear and nonlinear sections*

As stated previously, each structural element having nonlinear behaviour is modelled with three sub-elements: a central element with linear behaviour and two sub-elements in the vicinity of the joints with nonlinear behaviour (see Figure 1-a). To define the characteristics of the sub-elements with linear behaviour (see Figure 6-a), the following parameters are necessary: width and depth of the member cross section, or its corresponding moment of inertia and area; volumetric mass of the material and its Young's modulus. For each sub-element with nonlinear behaviour are defined: the plastic hinges (Varum, 1996), parameters defining their monotonic behaviour (trilinear moment curvature envelope), the hysteretic behaviour rules and the damage index parameters. In addition, properties of the linear and nonlinear sub-elements can be imported from a text file to VISUALANL, and can be modified or deleted at any time.



a) b)

Figure 6. a) Definition of elements with linear behaviour; b) Definition of elements with nonlinear behaviour

3.3 *Frame properties*

The following properties are assigned to each frame or group of frames (Figure 7): i) the material defining the elastic properties of the member; ii) the length of the plastic hinges at the member ends; iii) the material defining the nonlinear behaviour of each plastic hinge.

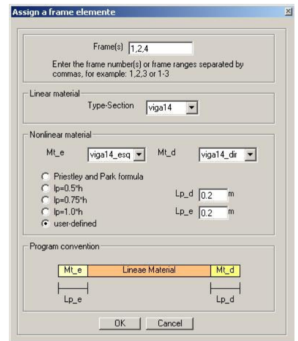


Figure 7. Frame element properties

3.4 *Nodal constraints*

Following the basic formulation of a 2D structural model, the user can restrict the degrees of freedom of a node, or group of nodes, namely, displacements in the x and y directions and the rotation (Figure 8). To facilitate the definition of constraints shortcut keys were defined for the different constraint types.

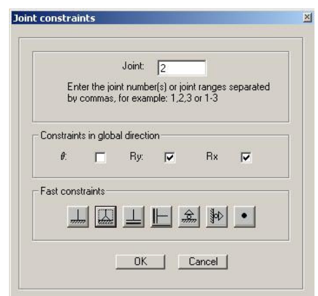
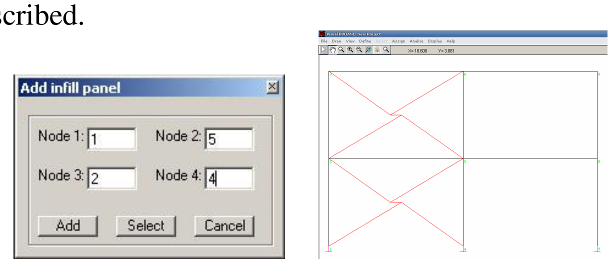


Figure 8. Definition of nodal constraints

3.5 *Infill panels*

To add an infill masonry panel to the frame structure, the user has to select the 4 nodes surrounding the panel. This operation can be performed in two different ways, namely, the user can write the node labels in a dialog box or directly selecting in the structure, with the mouse, the area where the infill panel must be considered (Figure 9-a). After adding the infill panel, VISUALANL represents it in the global structure (Figure 9-b), and the user is now able to define the masonry properties previously described.

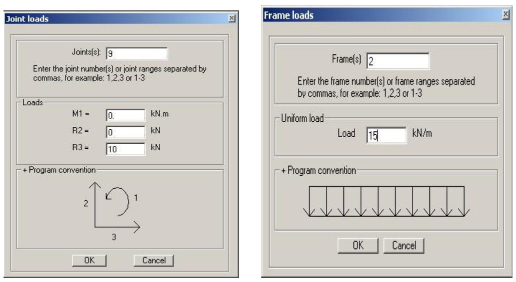


a) b)

Figure 9. a) Infill panel definition; b) Infill panels representation

3.6 *Static loads*

The static loads considered in the PORANL are concentrated loads in nodes and uniformly distributed loads in members. The concentrated loads applied directly to the nodes can be forces in x and y directions and also concentrated bending moments (see Figure 10-a). With respect to the distributed loads, only uniform loads applied on the total member length and normal to its axis are considered (as represented in the Figure 10-b). Any other different load case in members can be considered as a set of equivalent concentrated nodal forces. For a correct definition of the loading, the load sign conventions adopted in the PORANL program are displayed in the dialogue window.



a) concentrated in nodes b) distributed in frames

Figure 10. Static loads

4 PROBLEM CALCULATION

After completing the input of the structural problem using VISUALANL, the program prepares a series of text files that constitutes the text based input data for the analysis engine PORANL and runs it. Next, VISUALANL reads the results from the PORANL output text files, and, using a series of post-processing options, enables the fast graphical visualisation of the structural response results. As can be interpreted from the above, two different programs are used (VISUALANL and PORANL), although the user only works in the graphic environment VISUALANL without interacting directly with the analysis engine PORANL.

The program PORANL contemplates several analysis types, namely: i) linear elastic static analysis; ii) nonlinear static analysis, enabling to perform the push-over analyses; iii) nonlinear dynamic analysis; iv) nonlinear displacement controlled; and, v) calculation of the natural frequencies and vibration modes of the structure. Each one of these options was implemented in VISUALANL.

4.1 *Static analysis*

After the definition of the geometry, material properties and loading conditions, no additional data is necessary to perform a static analysis. Therefore, the user only has to choose whether the analysis is linear or nonlinear, though for the latter the user must supply the nonlinear material characteristics referred.

4.2 *Dynamic analysis*

To perform a nonlinear dynamic analysis the user has to define: i) the accelerogram (imported from a text file); ii) the dynamic equilibrium equation integration method to be used (Wilson-θ, Newmark or Central Differences Method); and, iii) the parameters for the definition of the Rayleigh-type damping matrix. These parameters can be calculated in VISUALANL as a function of an assumed damping value for two different natural frequencies.

After importing the selected accelerogram, it can be visualized (Figure 11) and the user can scale it to increase or reduce its intensity, alter the integration time-step or eliminate part of the accelerogram. In the dialogue window of the accelerogram definition, Figure 11, the user can also define the time interval for which the program PORANL will save results of the analysis in the output files.

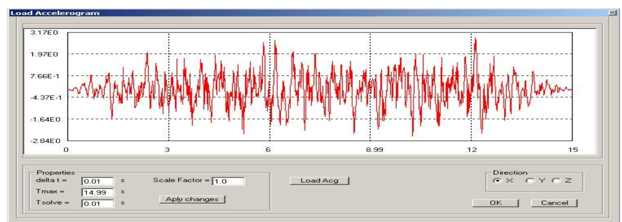


Figure 11. Accelerogram option window

4.3 *Displacement controlled nonlinear analysis*

In this type of analysis, a set of nodal displacement evolution laws are imposed to one or more nodes in a specific direction. As for the accelerograms, the displacement evolution laws can be imported from text files, and scaling factors can also be applied to increase or reduce the intensity of each individual law (see Figure 12).

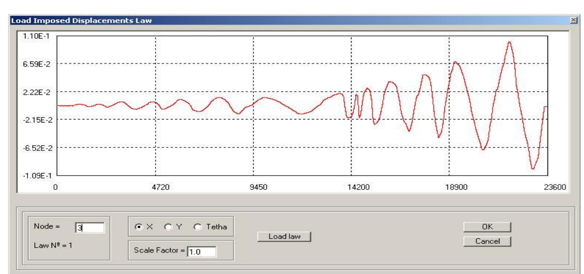


Figure 12. History of imposed displacements

4.4 *Natural frequencies and vibration modes*

In this option natural frequencies and their corresponding vibration mode configurations are calculated for the structure. The frequencies are presented in a table and the vibration modes are represented graphically. After obtaining the natural frequencies, the Rayleigh-type damping matrix coefficients can be calculated.

5 POST-PROCESSING RESULTS

The post-processing operations consist in the visualization of the structural analysis results and imply that all pre-processing instructions have been carried out. The availability of the visualization options will depend on the type of analysis that was performed. For instance, for dynamic analyses it is possible to visualize the time-history evolutions of internal member forces, an option not available for results of a static analysis. Any of the graphics generated in the post-processing options can be exported to an image format or to a compatible MS Excel text file. Also, graphics can be visualized in detail within VISUALANL using common graphical tools like the zoom or pan operations.

5.1 *Deformed shape plot*

This option, Figure 13, represents both the original undeformed and the deformed shapes of the structure, for an automatically calculated scale factor.

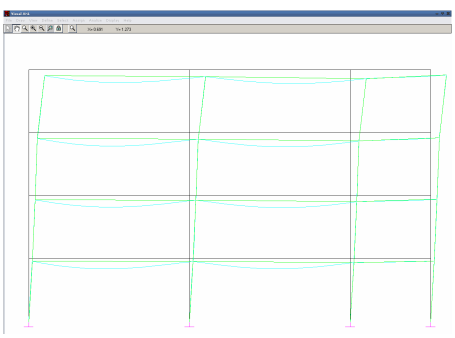


Figure 13. Deformed shape of the structure

However, the user can change it, in order to visualize the deformed structure in a more convenient scale. When the deformed shape is active, the user can select a member with the mouse to additionally visualize the deformed shape of that member in the local reference axes, the maximum positive and negative values of the displacements and the value of the displacement at any point within the member (Figure 14).

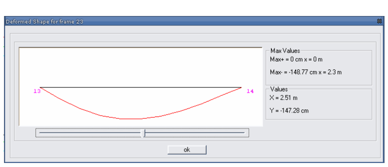


Figure 14. Deformed shape of an element

5.2 *Internal forces diagrams plot*

Diagram visualization follows a procedure similar to that of the deformed shape. The user selects the time-instant, for dynamic analysis results, or the step number, for displacement controlled analysis results, for which the representation of the global internal forces diagrams is required (axial force N, shear force V or bending moment M). Figure 15 shows an example of a bending moment diagram.

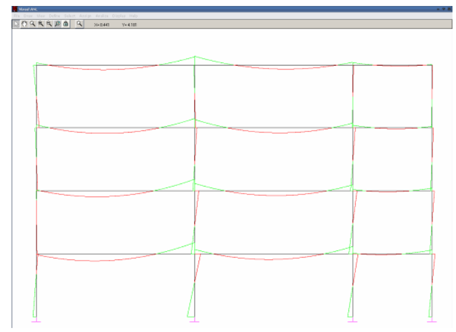
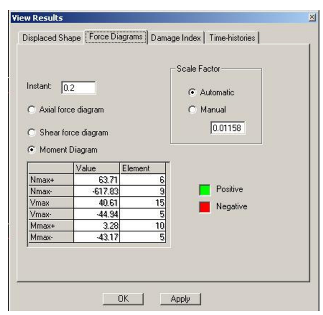
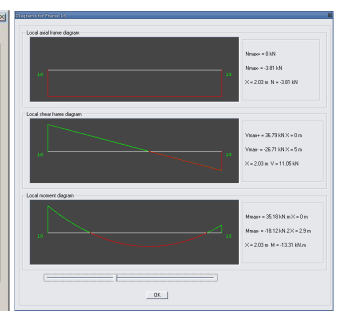


Figure 15. Distribution of bending moments across the structure

When the diagram plot is active, an options window enables the display of the maximum positive and negative internal force of the whole structure as well as the identification of the member where they occur (Figure 16-a). In addition, the user can select a member and visualize the three diagrams at once with indication of the maximum and minimum values (Figure 16-b). Internal force values at any point in the member can also be displayed.



a)



b)

Figure 16. a) Options for diagrams of internal forces; b) Axial force, shear force and bending moment diagrams of a single frame element

5.3 *Damage index plot*

Following the previously referred damage index, two different types of results can be visualized: the evolution of the global damage index for the structure and the local damage index for each member for a certain time-instant, in the case of a dynamic analysis, or step number, in the case of a displacement controlled analysis (Figure 17). In addition, another measure of damage is also available for visualization. This measure of damage is the maximum inter-storey drift profile that represents the maximum relative displacement between consecutive floors normalized by the height of the storeys.

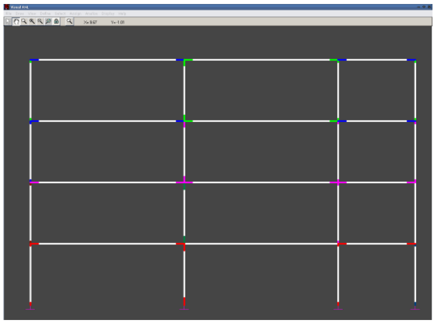


Figure 17. Local damage index distribution across the inelastic zones of the structure

For the graphical representation of the different damage intensity values, VISUANL considers a pre-defined colour scale where each colour is assigned to a damage level. However, the damage limits for each damage level, as well as the colour scheme representing each damage level can be altered by the user.

5.4 *Time-wise and step-wise evolutions*

In this option, the user can visualize the time-wise evolution, in the case of a dynamic analysis, or the step-wise evolution, in the case of a displacement controlled analysis, of internal forces (N, V and M), of curvatures in a plastic hinge and of nodal displacements for a specific direction. In addition the user can also visualize the moment-curvature evolution plot at a plastic hinge (Figure 18).

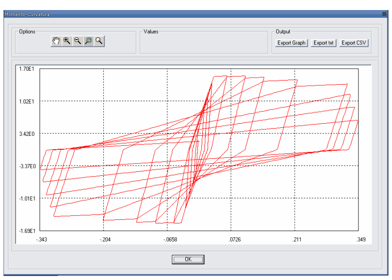


Figure 18. Moment-curvature diagram of a frame element under cyclic loading

6 FINAL COMMENTS

Structural analysis programs that include nonlinear models are valuable tools in the analysis and verification of structural safety, giving the engineer capacity to represent more precisely the real behaviour of the structures. For design of new structures or capacity assessment of existing ones, nonlinear analyses allow for a better representation of the structural response under any loading condition, and under earthquake loading in particular. Furthermore, nonlinear models can also be used in the calibration of more simplified numerical models.

Recent standards suggest nonlinear analyses as the reference analysis type to be used in capacity evaluation and verification of structural safety. For instance, Eurocode 8 (CEN, 2003) recommends the use of nonlinear time-history analysis for assessment of existent RC structures.

Nonlinear time-history analyses generate a considerable amount of information (nodal displacements, internal forces in frame elements, deformations, etc.), at each step of the analysis, thus making the treatment and interpretation of the results a rather difficult task. Most of the currently available tools for nonlinear analysis are research programs lacking convenient user-interfaces, thus presenting several shortcomings from a practical point of view. Although a considerable number of structural programs include state-of-the-art nonlinear material models, few have adequate graphical tools for results visualization. Therefore, the development of a graphical processor like VISUALANL is considered to be a helpful tool for nonlinear analysis as a support for the design of new structures and, especially, in the capacity assessment of existing ones, under severe static and/or dynamic loads.

7 REFERENCES

- *H. Varum (1996), Modelo numérico para a análise sísmica de pórticos planos de betão armado, MSc Thesis, FEUP, Porto.*
- *Thayer, R. (1999), Visual BASIC 6 Unleashed, Professional Reference Edition. Indianapolis, Indiana: SAMS Publishing.*
- *A.G. Costa (1989), Análise sísmica de estruturas irregulares, PhD Thesis, FEUP.*
- *C.T. Vaz (1996), Análise não-linear de pilares de betão armado sob cargas cíclicas, LNEC, Lisboa.*
- *H. Varum and A.G. Costa (1997), Modelo não-linear para a análise estática e/ou dinâmica de pórticos de betão armado, 3º Encontro de Sismologia e Engenharia Sísmica, IST, Lisboa.*
- *CEN (2003), Eurocode 8: Design of structures for earthquake resistance - Part 1-3: Strengthening and repair of buildings - European prEN 1998-1-3, European Committee for Standardization, Brussels, Belgium.*
- *A.G. Costa and A.C. Costa (1987), Modelo histerético das forças-deslocamentos adequado à análise sísmica de estruturas, Relatório Técnico, Núcleo de Dinâmica Aplicada, L.N.E.C., Lisboa.*
- *H. Rodrigues, H. Varum and A.G. Costa (2005), Modelo Numérico não-linear para painéis de Alvenaria de Enchimento em Pórticos de Betão Armado, Congreso Métodos Numéricos en Ingeniería, E.T.S. de Ingenieros de Caminos de la Universidad de Granada.*
- CEB (1996), RC frames under earthquake loading, Comité Euro-International du Béton, Bulletin nº231.