

A NEW APPROACH TO TEACHING AND LEARNING STRUCTURAL ANALYSIS

S. Kitipornchai, H. F. Lam

Department of Building and Construction, City University of Hong Kong, Hong Kong SAR, China

T. Reichl

Mursoft OEG, Grafenbergstrasse 47c/13, A-8051 Graz, Austria

Keywords: Structural analysis, Computer software, Virtual experiment.

Abstract: After the 9/11 incident, many structural educators became aware of the importance of inculcating in their students a clear understanding of local and global structural behaviour to develop basic knowledge about the progressive collapse of structures. The authors of this paper have addressed the issue by training students through 'virtual' experiments in a self-learning environment with the help of a newly developed software application, iSA (Instant Structural Analysis). One of the outstanding features of this application is that it allows students to instantly observe the changes in structural behaviour that are induced by changes in loading conditions, structural geometry, support conditions and member properties. This paper not only discusses the need to modify traditional teaching methods in the education of the new generation of structural engineers, but also presents some of the features of iSA that serve as handy tools in the teaching and learning of structural analysis.

1 INTRODUCTION

Since the catastrophic collapse of the World Trade Centre (WTC) towers on September 11, 2001, it has become more apparent that understanding the progressive failure or collapse of structural systems is of paramount importance. Many educators and researchers have realised the importance of this, developing different types of educational packages. Al-Ansari & Senouci (1999) and Codeno-Rosete (2007) decided to employ commercial software such as Microsoft Excel, Mathcad and Scilab (<http://www.scilab.org>) in developing education packages to facilitate the teaching of structural analysis and design. With the help of rapidly developing information technology, many researchers have developed applications related to structural education that can be accessed through the Internet. Yuan and Teng (2002), Jiang et al. (2002), and Bretto et al. (2003) have applied client-side technologies such as HTML, Java and Flash in developing web-applications for students to carry out simple structural analysis or structural laboratories by using web-browsers without having

to install the application in their local computers. One of the most outstanding advantages of this type of client-side technology is that students can always gain access to the latest version of the software from the website. Elgamal et al. (2005) made use of both client-side and server-side technologies in developing the Webshaker system, which allows students to remotely control a shaking table in the laboratory through a web-browser. Common to the abovementioned packages is their ability to enable students to easily test structures under various support conditions, material and cross-sectional properties, and loading conditions.

A new approach to teaching structural analysis and design is the use of instant structural analysis software (Beer & Pilz, 1999) that is capable of stimulating rapid experimental learning as well as capable of being used as a creative design tool.

An easy-to-use instant structural analysis software application with full graphical user interface (GUI) has been developed for 2D structures – iSA (Instant Structural Analysis). iSA serves as a handy tool for teachers and engineers alike to learn about structural analysis and design. It

gives instant graphical results that accurately simulate structural responses, thus promoting self-learning and creativity. This paper introduces several features of iSA and their importance in enhancing student learning and the understanding of structural analysis.

2 ISA (INSTANT STRUCTURAL ANALYSIS) AND ITS BASIC FEATURES

iSA (<http://www.bc.cityu.edu.hk/isa>) is a user-friendly, yet powerful, 2D structural analysis package that supports modelling with a 'drawing' GUI or the use of template wizards (so-called 'structure generators') to generate structurally complicated models in minutes. Because its purpose is to provide an environment for students to simulate different structures under various loading conditions, iSA is easy-to-learn and easy-to-operate. In general, it takes only a few minutes for students to finish the computer model of a structure in iSA. According to the experience of the authors, it takes only two hours for Year 2 undergraduate structural engineering students (with knowledge of the matrix stiffness method) to evolve from beginners to experts in the use of this software.

iSA can be used for the analysis of trusses, continuous beams and frames with different types of pinned joints. Comprehensive databases of materials and standard sections are available. This feature is very important for students using iSA in learning structural design, as they can easily 'try' different sections on the structure and instantly obtain a new set of analysis results to determine whether the design is feasible. Apart from standard sections, users can define their own material and cross-sections. Analysis is automatically completed once the structural system and the loading conditions are defined, and the analysis results can be instantly presented to the user. After changes to the structural geometry, support condition, member properties, etc., the new analysis results can again be instantly presented. This ability to provide instant results makes it very suitable for carrying out 'virtual testing' (Kayvani, 2007). For example, students can remove some members, apply several hinges or change the boundary condition of the structural system to simulate the situation of a terrorist attack, and then observe the effects on the structure (e.g., what is the change in load distribution? What is the change in deformation? Is the structure still stable

under the second-order analysis?). This kind of 'virtual testing' is very efficient in helping students to develop their structural engineering sense and to understand the concept of progressive collapse.

Furthermore, iSA supports not only first-order but also second-order static analysis, dynamic modal analysis, stability analysis and moving load analysis. It is not only a versatile tool for the teaching and learning of the subject, but is also a handy tool for practising engineers in structural analysis and design.

2.1 Material and Cross-sectional Properties

iSA provides a GUI that allows users to modify the material and cross-sectional properties easily (see Figure 1).

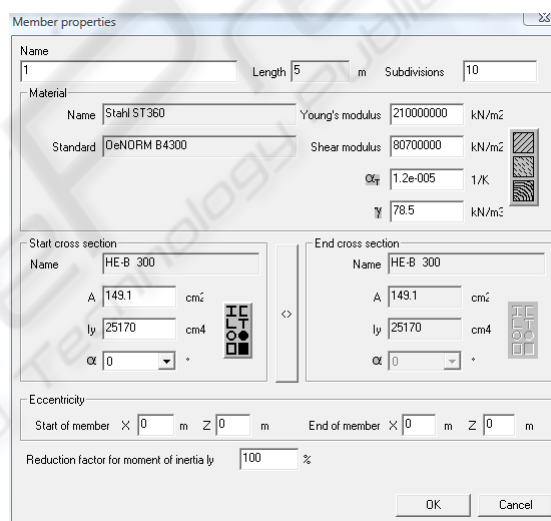


Figure 1: GUI for modifying the material and cross-sectional properties of members.

A series of commonly used materials, such as steel, concrete and aluminium are available in the system's material database (see Figure 2). A comprehensive database of cross-sections is also available for different standards, such as universal beams and columns. iSA supports both prismatic and non-prismatic members. By clicking the '$\langle \rangle$' button (see Figure 1), users can define a different section as the 'end cross section'. iSA assumes that the inverse of the second moment of area varies linearly along the member.

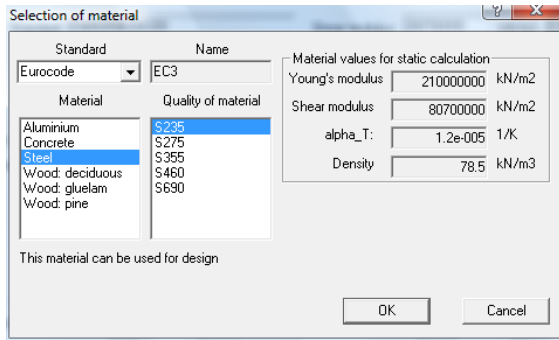


Figure 2: GUI for selecting a material from the database.

2.2 Support Conditions

iSA's GUI allows users to modify the support conditions easily. In the 'Constrained DOFs' area shown in Figure 3, users can define the support as a pin, a roller or a built-in support at any angle from the horizontal. They are also allowed to use springs to model the semi-rigid behaviour of the support.

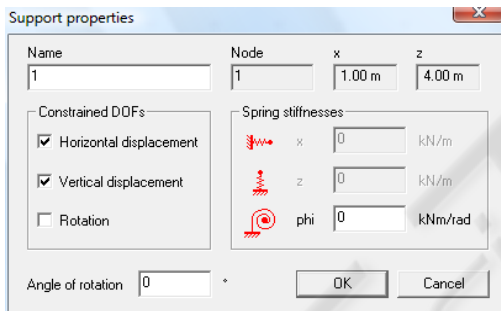


Figure 3: GUI for defining the support condition.

3 ADVANCED FEATURES

In addition to basic first-order static analysis, other types of analyses are available in iSA, such as second-order analysis, dynamic modal analysis, stability analysis and moving load analysis. Owing to the limited space, only some of these are introduced in the following sections.

3.1 Dynamic Modal Analysis

Once the structure is defined in iSA, the system stiffness and mass matrices are automatically generated. When users press the dynamic modal analysis button (at the toolbar), iSA solves the eigenvalue problem of the system stiffness and mass matrices and calculates the natural frequencies and mode shapes of the structure.

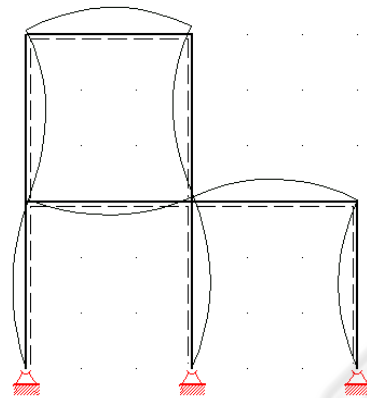


Figure 4: GUI showing the third mode of vibration of a simple frame structure.

The first mode natural frequency and mode shape is then instantly displayed in the drawing area on top of the original structure. Users can easily display the shapes of other modes through the GUI. Figure 4 shows a GUI that displays the third vibration mode of a steel frame. An animation of the mode shape is also available for students to gain a physical sense of the modes of a structural system.

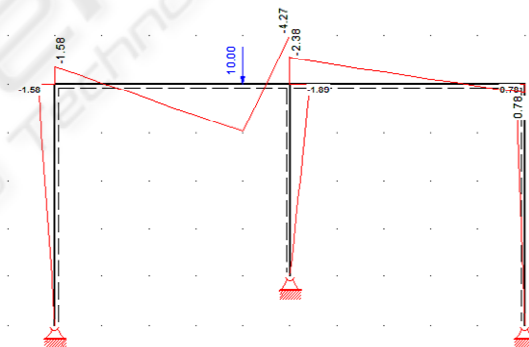


Figure 5: The bending moment diagram for one of the automatically generated moving load case.

3.2 Moving Load Analysis

Moving load analysis provides a fast and convenient feature for users to easily study the effect of a moving load on the structural system. After defining the movable load and the member on which the load is moving, a series of load cases are automatically generated under a pre-defined load case group. Figure 5 shows the bending moment diagram of the structure. In addition to the bending moment diagram, the shear force and axial force diagrams, together with their envelopes, are also available. When the structural geometry, the member

properties or the support conditions are changed, all of the analysis results are updated automatically, and the results are instantly presented through the GUI.

3.3 Teaching Progressive Collapse

The concept of progressive collapse can be introduced to students by using iSA with suitable guidance. A simple steel portal frame, as shown in Figure 6, is employed to illustrate this idea.

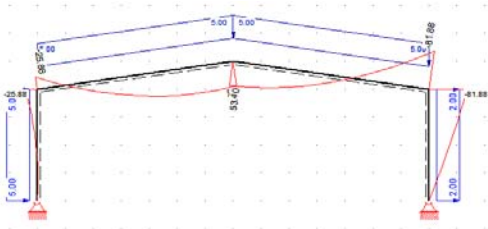


Figure 6: A portal frame to demonstrate the idea of progressive collapse.

Under the action of both the vertical and the horizontal distributed loads (as shown in Figure 6), it is clear from the results that the maximum bending moment is 81.88 kNm at the top right corner of the frame (see Figure 7).

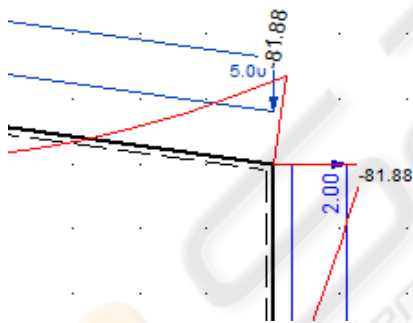


Figure 7: The maximum bending moment at the top right corner of the frame.

For demonstration purposes, 81.88 kNm is assumed to be the maximum plastic moment. Thus, a plastic hinge is formed at the top right corner of the frame. This plastic hinge can be easily modelled in iSA by first adding a full hinge at the joint and then applying appropriate moments at the ends of the members that connect the hinge. In this example, a full hinge is added to the joint at the top right corner of the frame. Then, member end moments of magnitude of 81.88 kNm and -81.88 kNm are applied at the ends of the two members that connect the full hinge, as shown in Figure 8. Note that the bending moment diagrams in Figure 7 and Figure 8

are the same, but there is a plastic hinge on the structure in Figure 8.

Users can now continue to increase the applied load on the frame. To demonstrate, only the vertical distributed load is increased. Figure 9 shows a situation in which the vertical load is increased to 6 kN/m. It is clear from the iSA result that the maximum bending moment of the structural system is on the top left member of the portal frame.

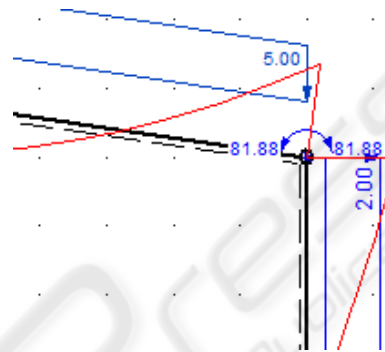


Figure 8: A plastic hinge is modelled at the top right corner of the portal frame.

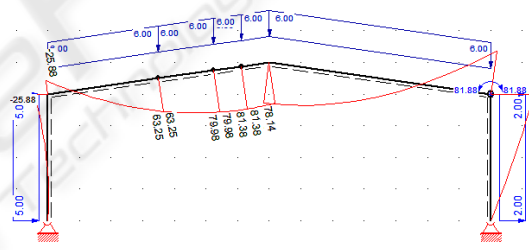


Figure 9: The applied load is further increased after the formation of the plastic hinge.

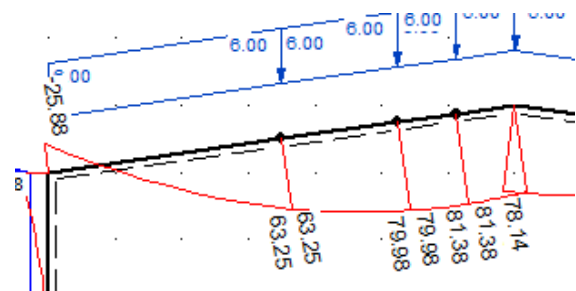


Figure 10: Additional nodes are added to locate the maximum bending moment on the top left member of the frame.

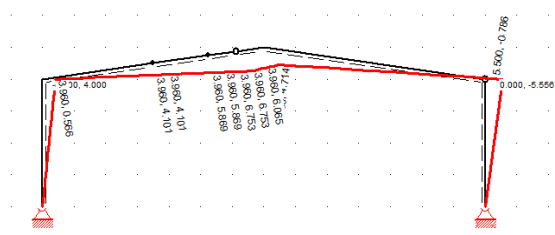


Figure 11: The additional plastic hinge makes the structure unstable, and it collapses.

To determine that location approximately, additional nodes are added on the member, as shown in Figure 10. It must be pointed out that there are many ways to identify the location and magnitude of the maximum bending moment (or axial force or shear force) on a member, and the introduction of additional nodes is only one of them. It is assumed that a plastic hinge is formed at the location of the maximum moment. When the user applies a full hinge at that location, iSA immediately shows that the structure is kinematically unstable, as shown in Figure 11. By repeating this exercise with suitable guidance from the teaching staff, students will be able to build up a sense of progressive collapse of structural systems.

4 CONCLUSIONS

The notion of having an adequate 'safety factor' in itself is not enough. Indeed, it could even give us a false sense of security. We need to emphasise to our students the importance of understanding physical, holistic structural behaviour such as the equilibrium of whole structures, the effects of compatibility, the effects of a lack of fit, etc. We need to equip our students with the fundamental and essential skills they need to be able to check and verify computer-generated results manually. We need to adopt easy-to-use structural analysis software that can provide and stimulate rapid experimental learning through the investigation of 'what if?' scenarios, that can be used to check computer-generated results to reflect on 'whether it really makes physical sense', and that can also be used as a creative design tool. We need to revise our curricula to phase out or reduce the use of old-fashioned manual calculation techniques and, instead, increase our emphasis on practical aspects and on a physical/holistic understanding of structural behaviour.

To prevent a catastrophic collapse, such as that which occurred on 9/11, we need to understand how a structure will behave and respond under all

possible loadings, including that from a terrorist attack. Therefore, we should design every structure so that:

1. It will not fail catastrophically if a part or parts of the structure are damaged or destroyed;
2. It will be able to re-distribute the load when parts of the structure have failed; and
3. It will have a high degree of redundancy and be able to provide alternative load paths to avoid sudden collapse.

iSA, a user-friendly 2D structural analysis program with an easy-to-use GUI, has been described here. This software is a handy tool for instructor teaching and student learning in structural analysis (e.g., the matrix stiffness method, second-order analysis, dynamic modal analysis, plastic analysis, stability analysis, moving load analysis, etc.). As the analysis results are presented instantly, this is an efficient and rapid self-learning tool with which students can build up their confidence, engineering sense and understanding of structural behaviour.

REFERENCES

- Al-Ansari, M.S. and Senouci, A.B., 1999. Use of Mathcad as a Teaching and Learning Tool for Reinforced Concrete Design of Footings. *Computer Applications in Engineering Education* 7, 146-154.
- Beer, G. and Pizl, G., 1999. *Civil Engineering Technology – Experiences with using the Techware Ruckzuck in a Structural Analysis Course*, Thomas Telford, UK.
- Bretto, S.F.A., Piazzalunga, R. and Ribeiro, V.G., 2003. A Web-Based 2D Structural Analysis Education Software. *Computer Applications in Engineering Education* 11, 83-92.
- Cedeno-Rosete, R., 2007. *Stiffness Matrix Structural Analysis Educational Package by Mathcad*. *Computer Applications in Engineering Education* 15, 107-113.
- Elgamal, A., Fraser, M. and Zonta, D., 2005. Webshaker: Live Internet Shake-Table Experiment for Education and Research. *Computer Applications in Engineering Education* 13, 99-110.
- Jiang, H., Kurama, Y.C. and Fanella, D.A., 2002. WWW-Based Virtual Laboratories for Reinforced Concrete Education. *Computer Applications in Engineering Education* 10, 167-181.
- Kayvani, K., 2007. Are We Winning the Battle of Computerised Structural Modelling? *Structural Engineering International* 1/2007, Editorial, 3.
- Yuan, X.F. and Teng, J.G., 2002. Interactive Web-Based Package for Computer-Aided Learning of Structural Behaviour. *Computer Applications in Engineering Education* 10, 121-136.