σ Journal of Engineering and Natural Sciences
Mühendislik ve Fen Bilimleri Dergisi

Sigma 33, 119-128, 2015

Research Article / Araştırma Makalesi

AN EDUCATIONAL TOOL FOR OBSERVING SEISMIC BEHAVIOR OF 3D FRAME STRUCTURES

Rafet ŞİŞMAN*, Abdurrahman ŞAHİN

Yildiz Technical University, Department of Civil Engineering, Davutpasa-ISTANBUL

Received/Geliş: 28.09.2014   Accepted/Kabul: 21.12.2014

ABSTRACT

This paper presents new educational software named Earthquake Simulator 3D which analyzes three dimensional frame structures under ground motion effect by using OpenSees and visualizes deformed shape of structure during earthquake by using MATLAB. The plan of structure can be created with the interactive drawing screen. Earthquake record and direction can be selected by user in the program. Parameters such as elastic modulus, shear modulus and surcharge loading can also be defined by user. This software may be used to give examples about structural behavior during an earthquake and can be useful for both civil engineering and earthquake engineering students. In addition to transient analysis, modal analysis can be carried out, mode shapes and modal frequencies can be calculated and observed.
Keywords: Structural behavior, earthquake simulation, educational software, ground motion effect, modal analysis, seismic behavior, 3D frame structures.

3B ÇERÇEVE YAPILARIN SİSMİK DAVRANIŞINI GÖZLEMLEMEK İÇİN EĞİTİCİ BİR ARAÇ

ÖZET

Bu çalışma, yer hareketi etkisine maruz üç boyutlu çerçeve sistemleri OpenSees yazılımını kullanarak analiz eden ve MATLAB yazılımını kullanarak yapının deprem süresince yaptığı şekil değiştirmeleri görselleştiren, eğitici bir yazılımı sunmaktadır. Yapının planı interaktif bir çizim ekranıyla oluşturulabilir. Deprem kaydı ve deprem yönü program içerisinde kullanıcı tarafından belirlenebilir. Elastisite modülü, kayma modülü ve ilave yükler de kullanıcı tarafından tanımlanabilir. Bu yazılım yapının deprem anındaki davranışını örneklemek için kullanılabilir ve hem inşaat mühendisliği öğrencileri için hem de deprem mühendisliği öğrencileri için yararlı olabilir. Zaman tanım alanında analize ilave olarak, modal analiz de yapılabilmekte, mod şekilleri ve modal frekanslar hesaplanıp görüntülenebilmektedir.
Anahtar Sözcükler: Yapısal davranış, deprem simülasyonu, eğitim yazılımı, yer hareketi etkisi, modal analiz, sismik davranış, 3B çerçeve yapılar.

1. INTRODUCTION

Despite human knowledge about earthquakes has increased so far, the behavior of structures during earthquakes is still an undigested issue for many of civil and earthquake engineering students. This situation can result in indifference to earthquake related courses. After computer-aided learning has begun to occupy an important part of engineering education, the indifference problem can be overcome.

* Corresponding Author/Sorumlu Yazar: e-mail/e-ileti: rafet@yildiz.edu.tr, tel: (212) 383 52 11

This paper presents new developed educational software named Earthquake Simulator 3D which performs time history analysis and visualizes seismic behavior of three dimensional frame structures. The dynamic analysis is performed by using OpenSees (Open System for Earthquake Engineering Simulation) [1] which is a software framework for simulating the seismic response of structural and geotechnical systems. OpenSees is advanced Nonlinear Finite Element Analysis [2-5] software and has been developed for performance-based earthquake engineering at the Pacific Earthquake Engineering Research Center (PEER) and is widely used for nonlinear seismic analysis of structures. Earthquake Simulator 3D uses advanced dynamic analysis capabilities of OpenSees and advanced graphical properties of MATLAB [6] which is a mathematical tool developed by MathWorks. The graphical User Interface (GUI) in MATLAB is used and a user friendly interface has been created. The difficulties about creating structural system for OpenSees have been overcome owing to the user friendly interface created by using MATLAB.

There are some important advantages of this software. First, users may design their creative frame structures interactively. They can apply any earthquake record in each direction and observe the seismic behavior of structures under different earthquake effects. Above all, the design and analyze can be done easily. The modal analysis can also be performed and mode shapes and natural frequencies may be observed interactively.

2. DESIGN PROCESS IN THE PROGRAM

The structural system is interactively defined with a graphical user interface (GUI). The floor plans are created interactively; number and heights of the stories are defined by using GUI. After saving a current plan, user can go on with creating another plan for upper stories by erasing the unwanted beams and columns from existing plan which is just saved.

The main window of the Earthquake Simulator 3D is presented in Fig. 1. As it can be seen from this figure, user needs to click the new project tool to start designing.

By clicking the new file tool, a new window is opened as shown in Fig. 2. The floor plan may be created on the screen. This window is equipped with design tools to help user to create the plans easily.

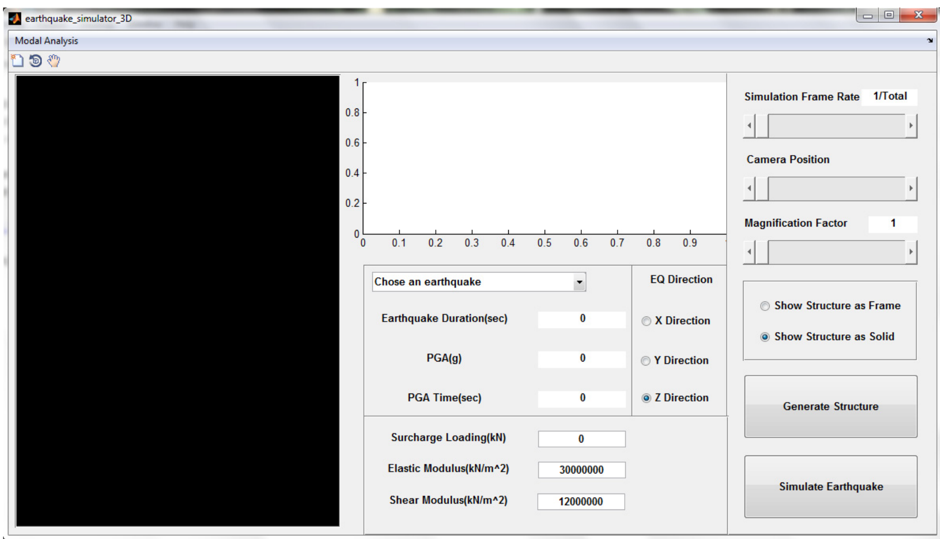

Figure 1. The main window of Earthquake Simulator 3D

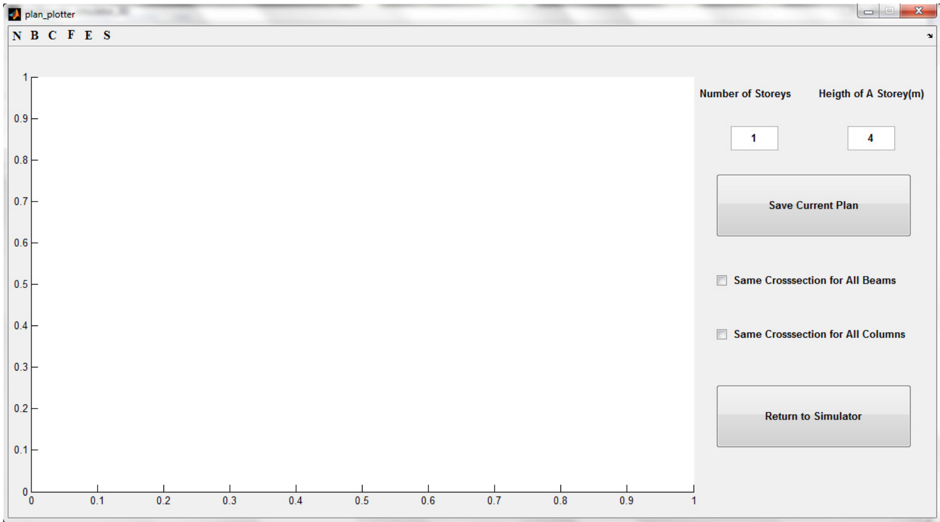

Figure 2. Floor plan creating window

The functional explanations of the abbreviations given in the floor plan creating window are as follows:

- N: creates plan template and adds nodes
- B: adds beams between selected nodes
- C: adds columns to selected nodes
- F: fixes selected column ends as footings
- E: erases selected beams(selected by clicking end nodes) and columns(selected by clicking nodes)
- S: defines edges of structure to be shown in solid demonstration mode

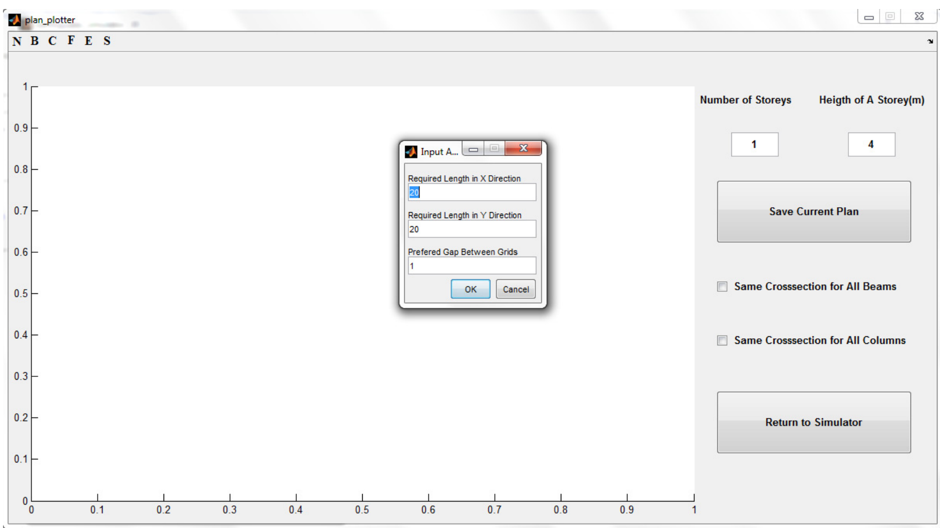

Figure 3. Template size and grid gap input screen

2.1. Determining Grid System

First, template size and grid gaps are defined in model creating process as shown in Fig. 3. The boundaries of the structure are determined and grid system is automatically created by this way.

After determining the boundaries and grid system, the last form of screen is given in Fig. 4.

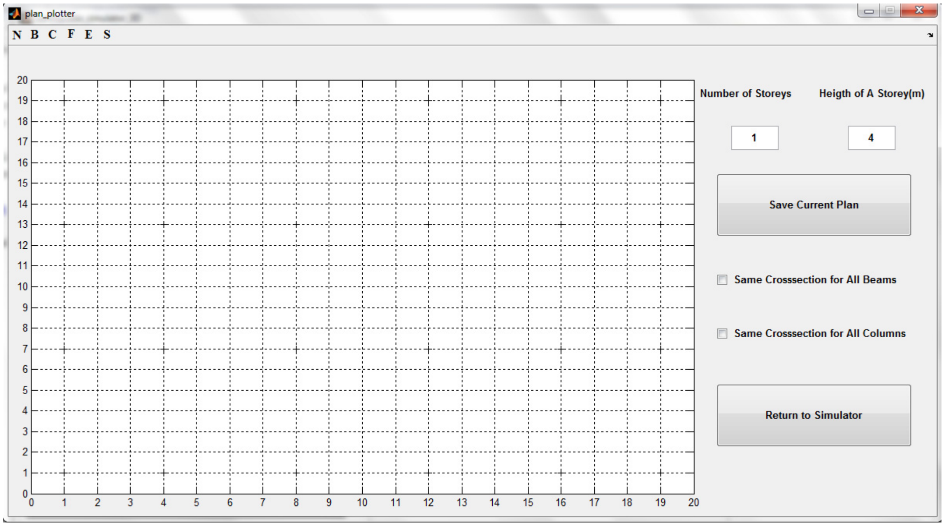

Figure 4. The form of plan after grid system is determined

2.2. Determining Frame Elements and Floor Plan

After the grid system is created, the columns and beams are determined. First, the cross sections for columns and beams are defined. At this stage, the cross sections of all beams and columns may be same as an option by ticking the checkboxes given in Fig.5. In this situation, user will need to define element cross-sections once only. This is required for both beam element and column element. The screen given in Fig. 5 shows the input stage for column elements.

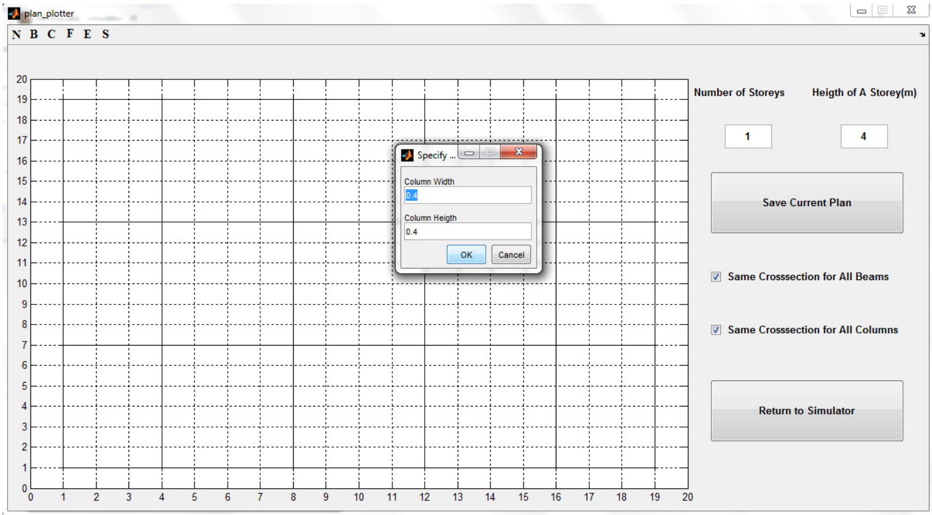

Figure 5. Cross section define screen for column element

After the floor plan is drawn, user needs to select nodes to be fixed. The screen after fixing is completed is shown in Fig. 6. As it can be seen in Fig. 6, the form of plan includes all beam and column elements.

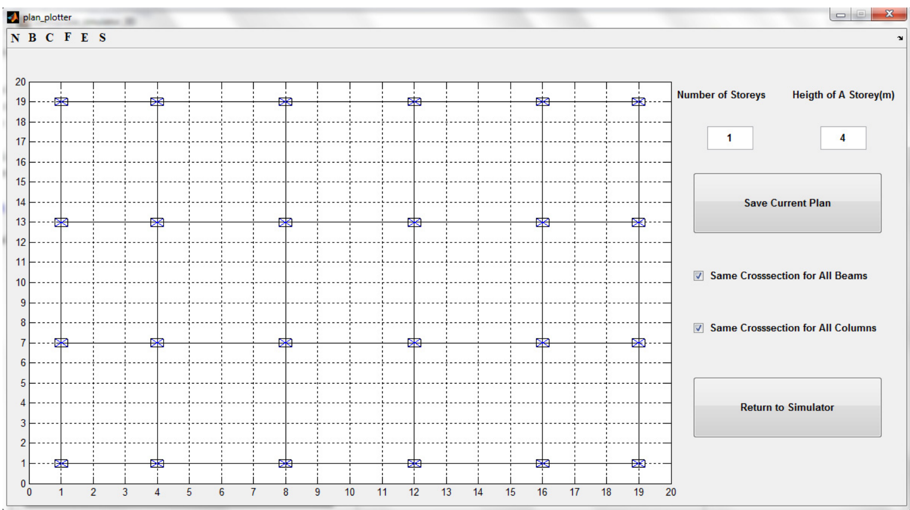

Figure 6. Determined floor plan for sample building

2.3. Modeling Whole Building

After the general floor plan is created, user can replicate it and decide the height of each storey. The replicated stories may be edited by using the erase tool. When the erase tool is used, the color of erased beam and column elements become blue as shown in Fig. 7. This means that these elements will not be created at upper stories. This feature is added to provide easy design for users.

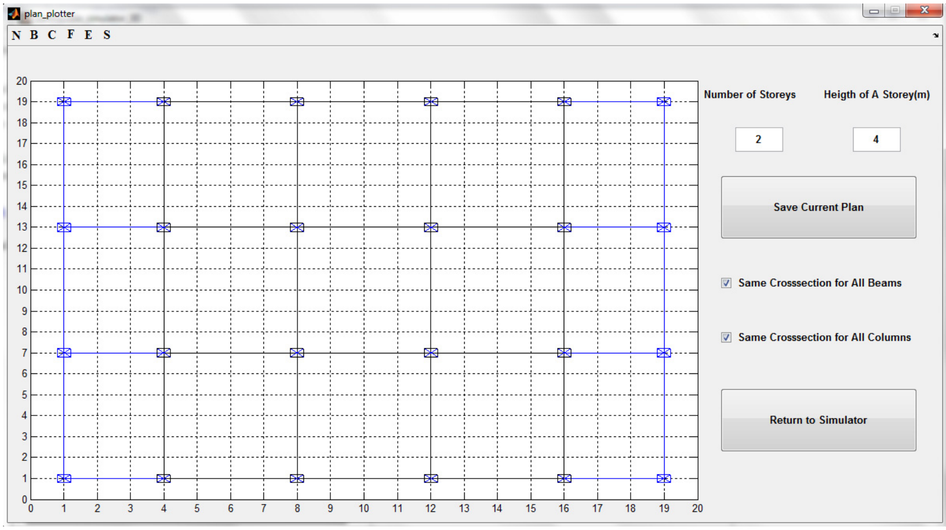

Figure 7. Edited plan of the sample structure

Erase tool can be used every time by saving each floor plan. After this editing process user needs to use solid edge tool and define the outer edge of each floor plan. The edges of different plans will be shown in different colors as it can be seen in Fig. 8. In this Figure, the edges of first story are drawn in green color and the second is drawn in red color. After the completion of plan designation, user can return to Earthquake Simulator 3D main screen by clicking the "Return to Simulator" button.

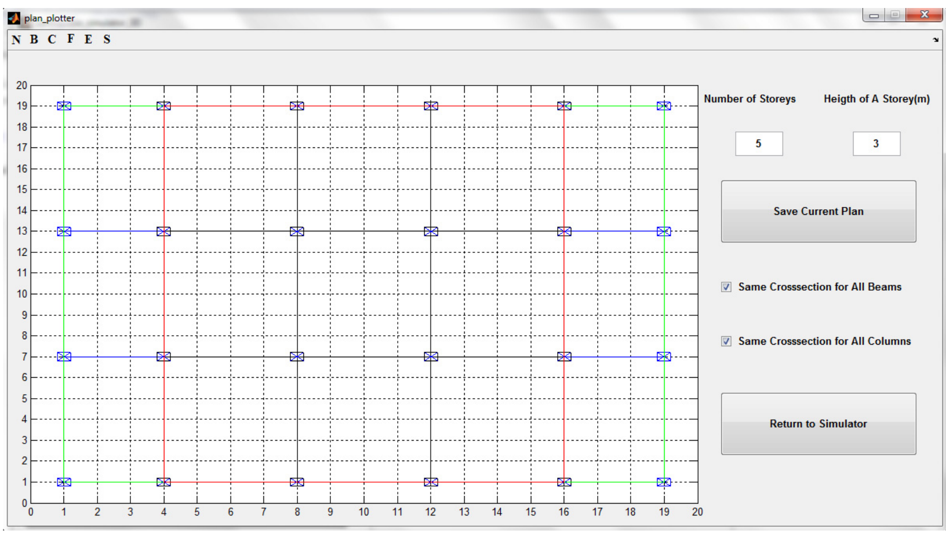

Figure 8. Solid edges defined for each floor plan

3. ANALYSIS PROCESS IN THE PROGRAM

3.1. Time History Analysis

The whole analysis of structure is performed by using OpenSees which is an open source structural analysis program. OpenSees determines nodal displacements for each step of earthquake motion. At the end of the analysis, the seismic behavior of the structure during the whole earthquake may be observed by using graphical tools.

After selecting earthquake from the popup menu and defining ground motion direction, user can visualize 3d structure by clicking "Generate Structure" button. Now it is ready to simulate structural behavior as given in Fig. 9. The user can define camera position, magnification factor for displacements and frame rate which controls frequency of drawing deformed shape (for example 50 refers to draw deformed shape for every 50x0.02 seconds). It is recommended to set higher frame rates for complicated structures with regard to user's computer performance.

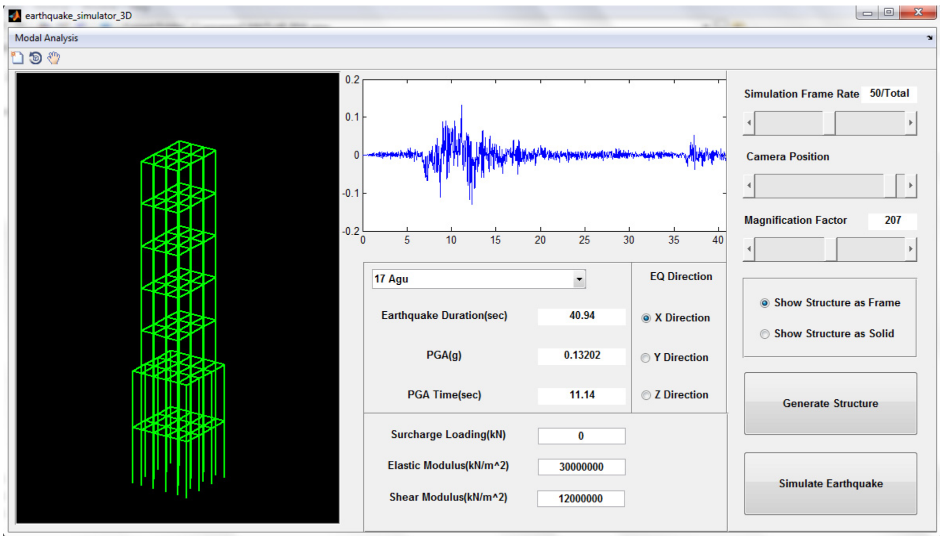

Figure 9. Main simulation window for observing structural behavior during earthquake

The seismic simulation starts after clicking the "Simulate Earthquake" button. User will see different deformed shapes at different times of earthquake motion as shown in Figs. 10-11. The structural behaviors at peak ground acceleration (PGA) times are presented in these figures. The seismic behavior of the structure during the whole earthquake may be animated and observed.

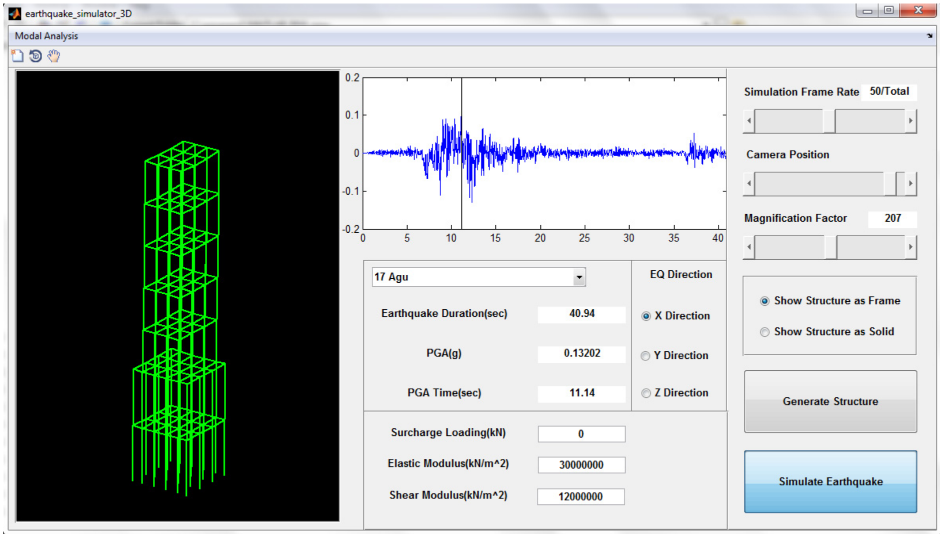

Figure 10. Deformed shape of the sample structure at peak ground acceleration (PGA) time

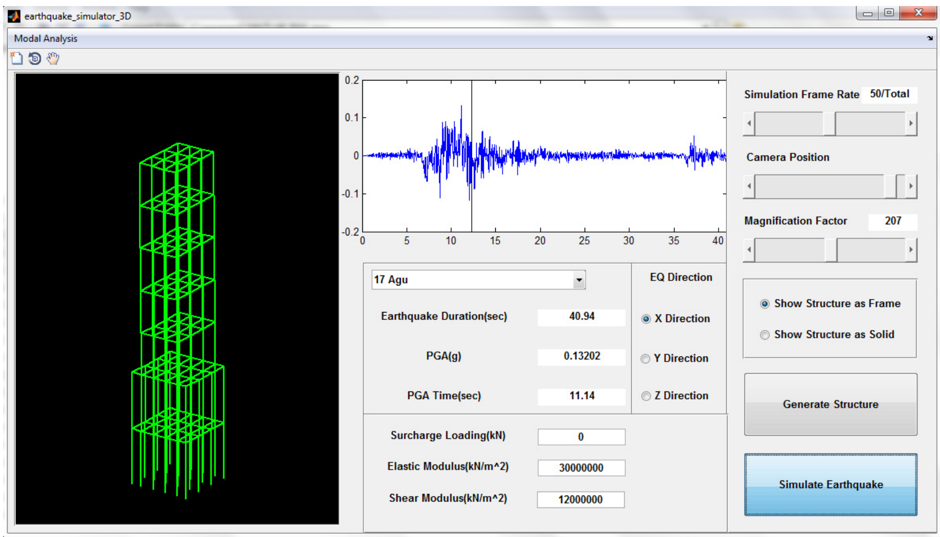

Figure 11. Deformed shape of the sample structure at negative peak ground acceleration (PGA) time

3.2. Modal Analysis

In structural engineering, modal analysis is performed to find the various periods and mode shapes at which the structure will naturally resonate. In design process, structural engineers does not want to see that natural frequency of the structure coincides with the frequency of expected earthquakes in the region in which the structure is to be constructed. It is a well-known physical behavior that if a structure's natural frequency coincides with an earthquake's frequency, the structure may continue to resonate and seismic damage may be observed. Therefore, modal analysis is very important for structural design process [7-10].

Modal analysis is performed by using OpenSees and results are visualized by using MATLAB. OpenSees analyzer is called by main program and the results are presented by using advanced graphical capabilities of MATLAB. As an example, the existing model which was designed for transient analysis can be used for modal analysis. By selecting the "Run Modal Analysis" option from "Modal Analysis" menu, the user will be wanted to define number of modes and then analysis will be completed. The obtained mode shapes may be investigated by selecting "Show Mode Shapes" option. A new screen will appear and demonstrate mode shapes and periods as shown in Fig. 12.

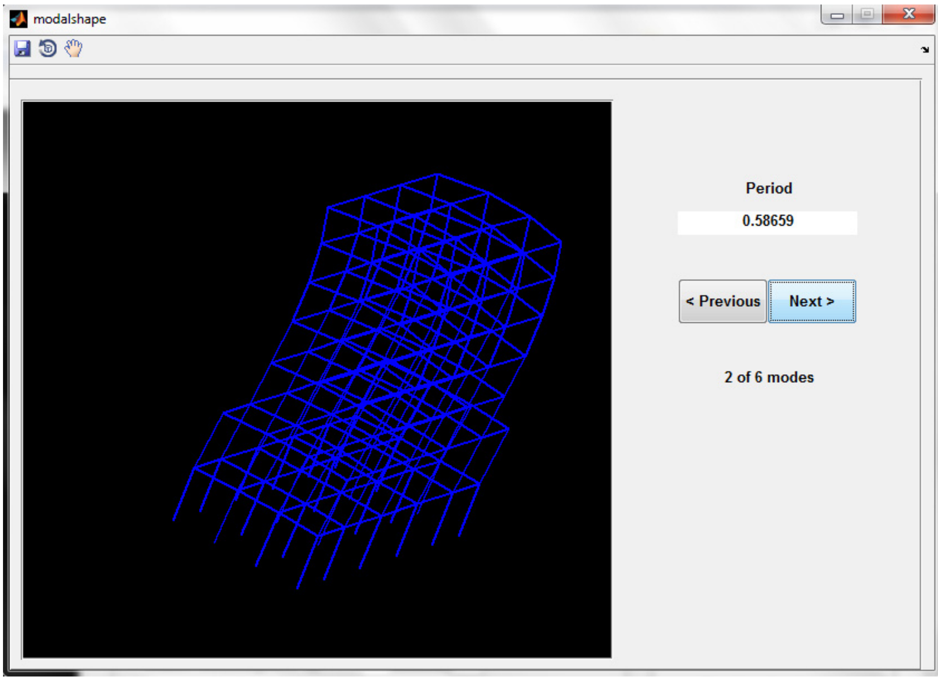

Figure 12. Modal analysis animation window

4. CONCLUSIONS

This paper presents new developed educational software named Earthquake Simulator 3D which analyzes three dimensional structures and visualizes their behavior under the earthquake motions. This program can be used by both civil engineering and earthquake engineering students. This program provides a user friendly interface for data input and display.

The Earthquake Simulator 3D is developed by utilizing MATLAB software which is a mathematical tool developed by Mathworks. OpenSees software which is developed by Pacific Earthquake Engineering Research(PEER) Center is used as analyzer. The input data preparation for OpenSees is very difficult and time consuming. The Earthquake Simulator 3D has user friendly interface and the structural system and seismic loads may be easily defined to the system. The program automatically creates input files for OpenSees and run the analyzer. The analysis results data is read by the program and the results are presented to the users graphically by using the advanced graphical capabilities of MATLAB.

REFERENCES / KAYNAKLAR

[1] OPENSEES Open System for Earthquake Engineering Simulation, Pacific Earthquake Engineering Research Center

[2] Zienkiewicz, O.C., The Finite Element Method in Engineering Science, McGraw-Hill, London, 1977.

[3] Zienkiewicz, O.C.,Taylor, R.L., Zhu, J.Z. (2005). The Finite Element Method: Its Basis and Fundamentals (Sixth ed.). Butterworth-Heinemann. ISBN 0750663200.

[4] Bathe, K.J. (2006). Finite Element Procedures. Cambridge, MA: Klaus-Jürgen Bathe. ISBN 097900490X.

[5] Azer A. Kasımzade, Finite Element Method: Fundamentals and Application on Structural Mechanics, Birsen Yayınevi, 2004. (Language: Turkish)

[6] MATLAB The Language of Technical Computing (R2010a), The MathWorks, Inc.

[7] D. J. Ewins: (1995). Modal Testing: Theory, Practice and Application

[8] Jimin He, Zhi-Fang Fu (2001). Modal Analysis, Butterworth-Heinemann. ISBN 0-7506-5079-6.

[9] Heylen,W., Lammens, S., and Sas, P., Modal Analysis Theory and Testing, Department of Mechanical Engineering, Katholieke Universiteit Leuven, Faculty of Engineering, Division of Production Engineering, Machine Design & Automation, Leuven, Belgium, 2007.

[10] Allemang, R.J., "Experimental Modal Analysis," Report No. UC-SDRL-CN 20-263-663/664, Structural Dynamics Research Laboratory, Dept. of Mechanical, Industrial and Nuclear Engineering, Univ. of Cincinnati, Cincinnati, OH, 1999.