A Web-Based Architecture for Interactive Finite Element Program

F. Hejazi[1], M. Shahpasand[2], M. Mirnezhad[2], M. S. Jaafar[1], A. A. Abang Ali[1]

[1]Department of Civil Engineering, Faculty of Engineering, University Putra Malaysia, Malaysia, farzad@upm.edu.my
[2]Departments of Computer Science, Faculty of Computer Science and Information Technology, University Putra Malaysia, Malaysia, m.shahpasand@gmail.com

ABSTRACT

In the recent decade, using of Personal Digital Assistants (PDAs) and smart phones had incredible growth and they can be used as a small computer in business, communication and education. Most of current civil engineering software are created for desktop computers in specific operating system and platforms without any portability and flexibility feature. This paper proposed a web-based architecture for an interactive Finite Element (FE) program. The architecture supports linear and nonlinear analysis of reinforce concrete buildings subjected to static and dynamic loading. Structure modeling is performed on the client-side via any internet browser. Data is send to the server and analyzed by FE program which has been installed on the server. Then, the result of analysis which is response of structure under imposed load is send back to the client-side to show and plot output of analysis. The developed software which is named Analysis of Reinforced Concrete Structure (ARCS) implemented in secure and user friendly interface. Running software in offline mode is possible for modeling but internet connection is needed to transfer input data to the server and transfer output of analysis from server to the client-side. The most important advantage of proposed model is operating system and platform independency. Backup capability of all data and authority access control are other features of ARCS. Therefore the developed software is able to use extensively by engineers to carry out modeling and analysis of structures by any PDAs or smart phones through any operating systems and platforms.

INTRODUCTION

In the recent decade, using of PDAs and smart phones had incredible growth and they can be used as a small computer in business, communication and education. Most of current civil engineering software are created for desktop computers in specific operating system and platforms without any portability and flexibility features. Recently many powerful software for structural analysis have been developed such as MASTAN2 (Ronald D. Ziemian 2000), DIANA (BV 2003), SAP2000 (SAP 2008), STAAD Pro (Bentley 2011), NARCBEEEDS (Hejazi 2009 2011 2013), ETABS (ETABS 2013) but all of them are needed to install in desktop and are not able to use in mobile operating system like Android or ios. Civil companies have started moving their software products to the new platforms in cyber space and there are a few web-based software to cover part of modern structure requirements.

Almeida Barretto et al. (2003) created a web-based program to analyze plane-framed structures, under static or dynamic loading. Yang et al. (2004) developed a web-based simulation for nonlinear seismic ground response using Cyclic1D and a framework was introduced for building a FEA program as a distributed web service (Peng and Law 2004). Marante et al. (2005) implemented Portal of Damage as a program for numerical simulation of reinforced concrete framed structures under earthquakes. An attempt has been made by Network for Earthquake Engineering Simulation Consortium (NEES) to reduce the impact of earthquake and tsunami disasters in the human infrastructures by linking research laboratories around the US (Alonso et al. 2007). Bittencourt et al. (2008) presented a web application service provider for finite element analysis with services for mesh generation. Also, Weng (2011) implemented WebDFEA as a post-processor for finite element analysis results. Huynh et al. (2011) developed a real-time solution for parameterized partial differential equations by Android mobile and a high performance computing (HPC) grid by offloading the heavy numerical computation to the grid and using the mobile for only displaying the results. An integrated deterministic solver was presented for uncertainty management and HPC by Patelli et al. (2012). Ari et al. (2012) discussed the design and implementation of cloud computing service for finite element analysis.

The greatest weakness of the current civil engineering software is user accessibility. Analysis software for the new mobile phone platforms is another shortage. However, a study of alternative competing software available in the market needs to be performed. Currently, there is no any available proper software in the market to simulate and inelastic analysis of three dimensions Reinforced Concrete Structures subjected to static and dynamic loads by mobile phone platforms. In this paper, a web-based architecture plan is proposed and implemented for Analysis of Reinforced Concrete Structure (ARCS) software for an interactive Finite Element program. The software was designed in three components, the Finite Element program as processor part, web-based interfaces as pre-processor and post-processor. Therefore, input data for modeling of structure and output data of structural analysis are provided and deliver easier in an acceptable and accessible way for a vast majority of end-users using the latest available computing platforms. Most important ARCS features are standalone, easy backup, user-friendly interface, access control by server-side ARCS and prevent unauthorized usage.

PROPOSED ARCHITECTURE

In this research, a web-based architecture was proposed for an interactive Finite Element program. The FEM program supports linear and nonlinear analysis of reinforce concrete buildings subjected to static and dynamic loading. The proposed architecture was implemented in 3D web-based ARCS software and shown in Figure 1. Three dimension features has been implied through Three.js WebGL libraries. It allows the creation of GPU accelerated 3D animations using the JavaScript language as part of a website without relying on proprietary browser plugins. The application architecture is used as a blueprint to ensure that the underlying modules will support future growth. In the proposed architecture, structure modeling is performed in the client-side by pre-processor and data is sent to the server via any internet browser.

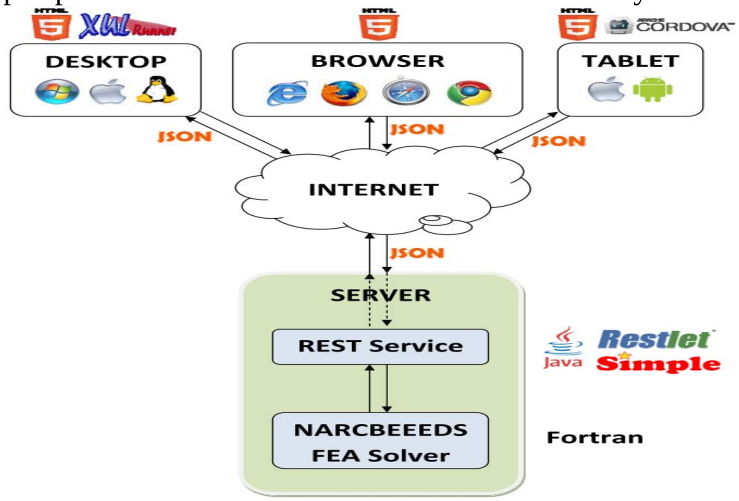


Figure 1. ARCS architecture.

Sent data is analyzed by using of Finite Element program which has been installed on the server as a processor part and its Java Wrapper will lie in the domain logic or the Model layer. This will ensure that the same code can be reused for cloud computing and HPC platforms. The result of analysis which is response of structure under imposed load is sent back to the client-side and backup has been made on selected path on server. Showing and plotting output of analysis is done on client-side by post-processor program. It can be used for both desktop based and server based applications. Running software in offline mode is possible for modeling but internet connection is needed to transfer input and output data between server and client as shown in Figure 1. Data is transferred in JSON format between the server and the client which is a lightweight data-interchange format and completely language independent. Pop-up windows in web browser will show the results and user can save or print it.

Client side contains all the Java script, HTML, CSS codes and performs modeling process. Server Side is the complete Eclipse Project for the Java-based Server side application and performs analysis on data. The NARCBEEDS utilizes the client-side (web browser) technologies used in Web 2.0 development which include the Ajax and JavaScript frameworks such as THREE.js, Dojo Toolkit and Prototype JavaScript Framework. The data fetched by an Ajax request is formatted in JSON (JavaScript Object Notation) format and transfer to server for processing.

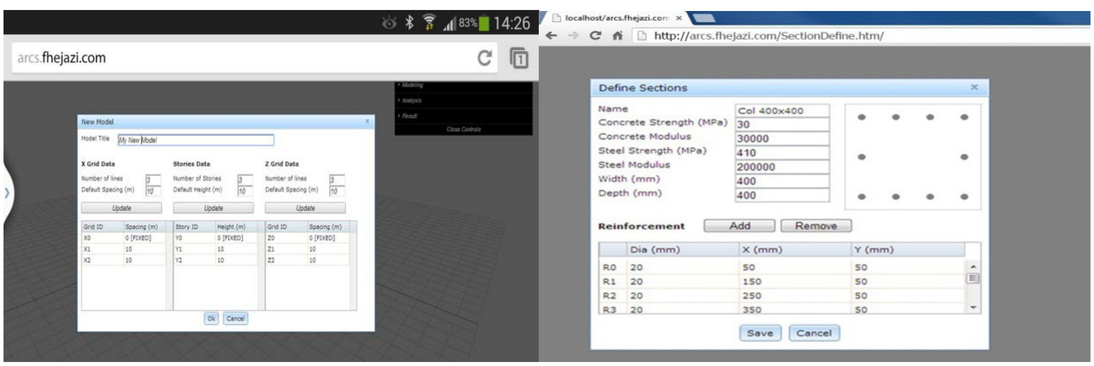
The HTML5 application will either execute as a native application on Windows, Mac OSX, Linux, Android and iOS. Therefore the developed software is able to use extensively by engineers to carry out modeling and analysis of structures by any Desktop PCs, PDAs or smart phones through any browsers, operating systems and platforms. The application will communicate with the server via HTTPS and Access to the program can be controlled directly from the server by an access control system, thus enabled the administrator to allow or deny end-users access to the program. For every single PDA the users only need to access one of the web browsers like Safari, Chrome, Firefox or Opera. It is a standalone application and there is no need to install any specific GUI.

PRE-PROCESSOR

The preprocessor has been facilitated by a graphical user interface to the end-user for modeling the structure and analyzing in details of material, sections and loadings. It also has to integrate with the processor by providing input and displaying feedback to the user based on the response of the processor. The pre-processor GUI clearly makes it very easy for a user to define a structural model to be analyzed especially when compared to the previous method of defining using text input files. The user interface mimics a normal application and will be easy for users to learn. The developed software has all common options such as creating new model, save, open, edit, export and Import, and help menu. The main menu will explain following paragraphs.

Defining the Model Grids. the user is define the grid for the frame structure by defining the horizontal distances between columns (X Grid Data) and then the vertical heights between floor levels (Stories Data) as shown in Figure 2-a.

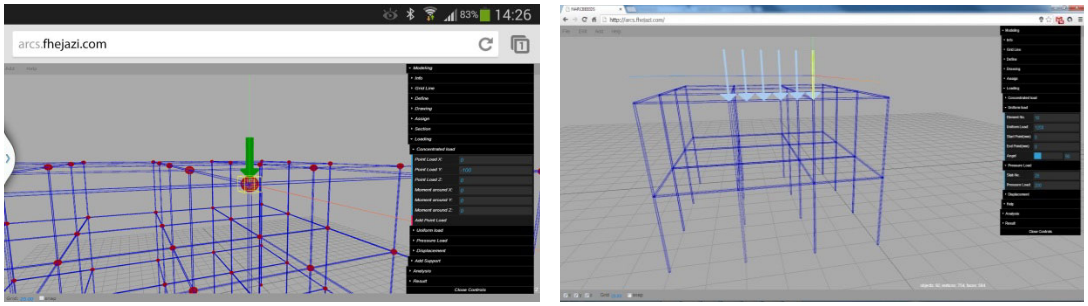
Define Beam, Column Sections. Detail of beam, column sections as compulsory component is added to the model as shown in Figure 2-b.



(a)- Defining the grids-mobile platform (b)- Define beam, column sections

Figure 2. Define new model grids and new section.

Adding Point Load and Uniformly Distributed Load. Any Nodes, Elements and Supports can be added by provided software options. Users can assign point loads to any node. Negative values can be used to specify loads in opposite direction. Also, Uniformly Distributed Load (UDL) can be assign to any node as shown in **Error! Reference source not found.**. Time history accelerations of earthquake and blast loading are able to define in loading part.



(a) Mobile platform (b) Desktop platform

Figure 3. Assign point load and udl to the mode.

PROCESSOR

Finite element is a method of computation for solving engineering problems and formulated as partial differential equations, integral equations or functional minimization. It can be used to solve both static and dynamic problems. In the developed system, NARCBEEEDS finite element program for linear and non-linear analysis of reinforced concrete structures under static and dynamic loadings (Hejazi 2012) is implemented as a processor on server side. Figure 4 shows the FEM program structural model.

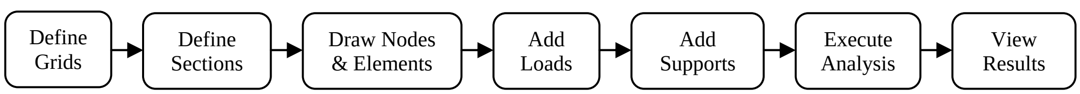


Figure 3. Analysis flowchart

Input data represents the structure, section geometry, number of segments, material properties, nonlinear constants, damper properties and applied static and dynamic loads. Once all the required data of structure has been defined, the program performs inelastic analyses and output file which is structural response will be generated. If there is any missing data in the model, the program will display an error message to the user.

POST-PROCESSOR

The post-processor GUI is integrated into the pre-processor part of the software on client side and it is used for showing different option of result. User can identify which part of output is needed to be transfer from server side for show and download. The output of the processor for analysis includes the displacement of nodes, forces in elements, reactions at the supports, the locations of the plastic hinges in structural members. For dynamic analysis, the processor outputs the time history displacements of nodes due to dynamic loadings such as earthquake excitation. All analysis outputs of FEM program are shown in Figure 6.

Analysis Report. The user can generate a detailed report in two parts as shown in Figure 7. It displays a diagram of the model along with the node IDs and the table below the diagram shows the resulting displacements and rotations of each node.

Element analysis shows the axial force, shears, torsion and moments within each element.

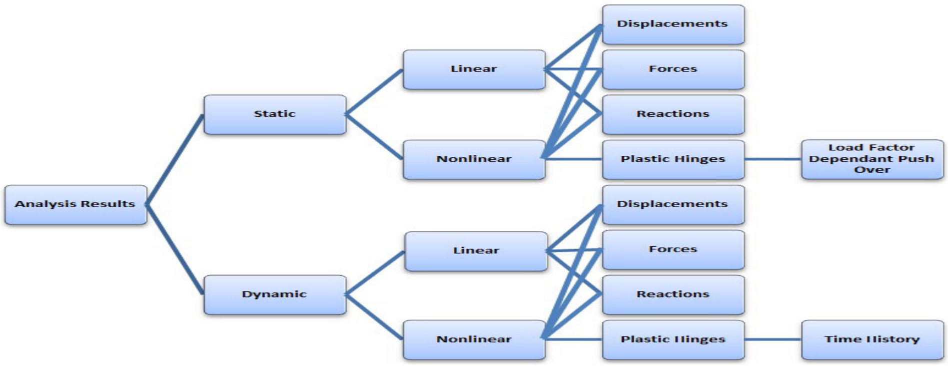


Figure 4. Analysis output

APPLICATION IN 3 STOREY REINFORCE CONCRETE BUILDING

A three dimensional, 3-storey reinforced concrete structure is considered and modeled as shown in Figure 5 and nonlinear analysis was performed for evaluation inelastic response of 3D building under multi support excitation.

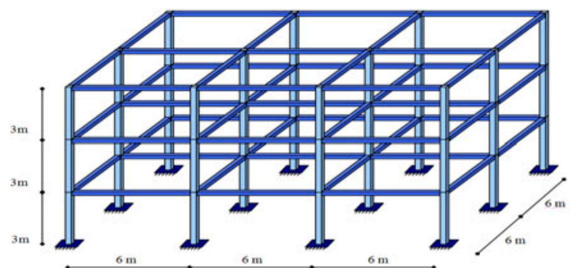


Figure 5. Geometry of 3D, 3-storey reinforced concrete frame structure.

The model is consisted of 48 Nodes, 51 Beam Elements 36 Column Elements. 12 Boundary Conditions are defined in structure nodes for ground support as rigid nodes. Beam and columns cross sections, reinforcement and material properties are shown in Figure 6. The all material properties for concrete and steel material and beam, column sections are defined in the ARCS application and considered structure configuration has been modeled as shown in Figure 7-a. Distribution load on structure beams are considered as 25 kN/m and concentrated load on columns are 40 kN. The structure is subjected to Elcentro earthquake record (1940-USA) and after assign the loading, the modeling part is completed.

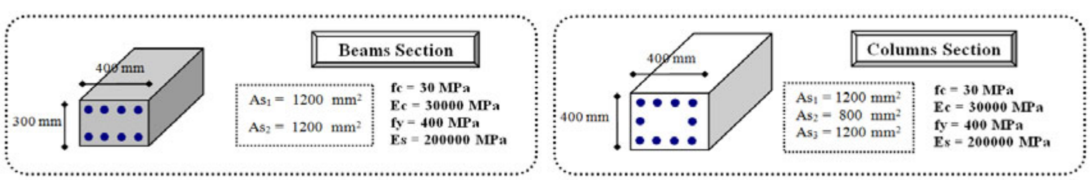
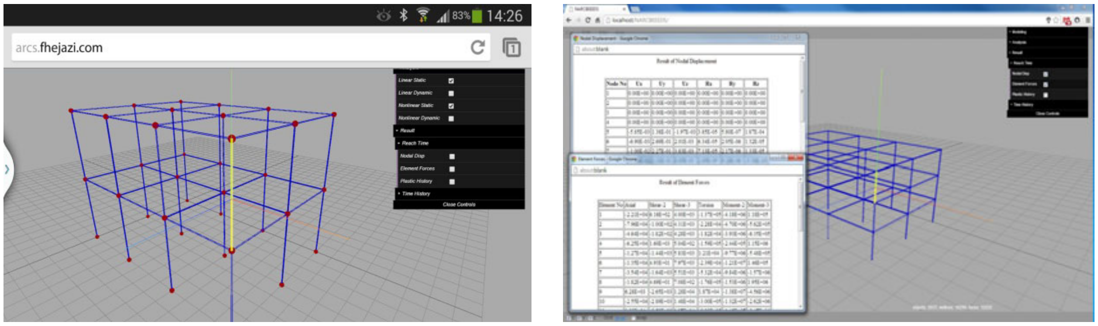


Figure 6. Beam, column cross sections and material properties.

So, the all information about structure is send from client-side to the server via internet and inelastic analysis has been done through processor by Finite Element program. The result results of inelastic analysis in terms of displacements of structure in 6 degrees of freedom (space state) and force in beam and column sections are send back to the client-side through internet and plotted as showed in Figure 7-b.



(a) Structure model-Mobile platform (b) Inelastic analysis results

Figure 7. 3 story RC structure model in ARCS and inelastic analysis results.

CONCLUSION

Increasing usage of Internet, PDAs, and smart phones highlights the need of portability, operating system and platform independency in all type of software. In the present paper, a web-based architecture was proposed for an interactive Finite Element program which implemented and imply in ARCS (Analysis of Reinforced Concrete Structure) software. The architecture supports linear and nonlinear analysis of reinforce concrete buildings subjected to static and dynamic loading. Structure modeling data provides in the client-side and sends to the server to analyze by FE program. The result of analysis sends back to the client-side to show and plot the output. The ARCS facilitate Analyzing of Reinforced Concrete Structure through secure and user friendly interface, operating system and platform independency. Moreover, backup capability of all data and authority access control are provided for ARCS software. Access to the program can be controlled directly from the server, thus enabling an administrator to allow or deny end-users access to the program.

The same code can be reused for cloud computing and high performance computing platforms in future. Also for further enhancement, the display of results by the post-processor can be improved by showing the deflected shape of the structure and force distribution diagram.

ACKNOWLEDGMENTS

This work received financial support from Housing Research Center of UPM and NAEIM Company and was further supported by the Ministry of Higher Education of Malaysia through FRGS Research Projects No. 5524254 and No. 5524256 and the supports are gratefully acknowledged.

REFERENCES

Almeida Barretto, S. F., Piazzalunga, R., & Ribeiro, V. G. (2003). "A web-based 2D structural analysis educational software." *Computer Applications in Engineering Education*, 11(2), 83-92.

Alonso, J. M., de Alfonso, C., García, G., & Hernández, V. (2007). "GRID technology for structural analysis." *Advances in Engineering Software*, 38(11), 738-749.

Ari, I. and N. Muhtaroglu (2012). "Design and implementation of a cloud computing service for finite element analysis." *Advances in Engineering Software*.

Bentley. (2011). "STAAD.Pro." *Structural Analysis & Design Engineering Software*.

Bittencourt, M. L., Borges, E. G., Carvalho, E. F., & Augusto, R. A. (2008). "An application service provider for finite element analysis." *Advances in Engineering Software*, 39(11), 899-910.

BV, T. D. (2003). "DIANA." *Displacement Analyzer Software*.

SAP, CSI. (2013). "Sap2000" *Structural analysis Program, Computer and Software Inc*.

ETABS, CSI. (2013). "ETABS2013." *Integrated Building Design, Computer and Software Inc*.

Hejazi, F., Jaafar. M. S. (2012). "NARCBEEEDS, Program Code for Nonlinear Analy-sis of Reinforced Concrete Buildings Equipped with Earth-quake Energy Dissipation System", *Copyright by University Putra*.

Hejazi, F., Kojouri, S. J., Noorzaei, J., Jaafar, M. S, Thanoon W. A., and Abdullah, A. A. A. (2011). "Inelastic Seismic Response of RC Building with Control System". Key Engineering Materials Journal, 462-463 241-246

Hejazi, F., Noorzaei, J., Jaafar, M. S., & Abdullah, A. A. (2009). "Earthquake Analysis of Reinforce Concrete Framed Structures with Added Viscous Damper." *World Academy of Science, Engineering and Technology*, 3(57), 130.

Hejazi, F., Toloue, I., Jaafar, M. S., & Noorzaei, J. (2013). "Optimization of earthquake energy dissipation system by genetic algorithm." *Journal of Computer-Aided Civil and Infrastructure Engineering* 28(10), 796-810.

Huynh, D. B. P., Knezevic, D. J., Peterson, J. W., & Patera, A. T (2011). "High-fidelity real-time simulation on deployed platforms." *Computers & Fluids*, 43(1): 74-81.

Marante, M. E., Suárez, L., Quero, A., Redondo, J., Vera, B., Uzcategui, M. and Flórez-López, J. (2005). "Portal of damage: a web-based finite element program for the analysis of framed structures subjected to overloads." *Advances in Engineering Software*, 36(5): 346-358.

Patelli, E., Murat Panayirci, H., Broggi, M., Goller, B., Beaurepaire, P., Pradlwarter, H. J., and Schuëller, G. I. (2012). "General purpose software for efficient uncertainty management of large finite element models." *Finite elements in analysis and design*, 51: 31-48.

Peng, J., & Law, K. H. (2004). "Building finite element analysis programs in distributed services environment." *Computers & structures*, 82(22): 1813-1833.

Ronald D. Ziemian, W. M. (2000). "MASTAN2", *Matlab Based Interactive Structural Analysis Program*.

Weng, W.-C. (2011). "Web-based post-processing visualization system for finite element analysis." *Advances in Engineering Software*, 42(6): 398-407.

Yang, Z., Lu, J., & Elgamal, A. (2004). "A web-based platform for computer simulation of seismic ground response." *Advances in Engineering Software*, 35(5): 249-259.