Leadership in Sustainable Infrastructure
Leadership en Infrastructures Durables

Vancouver, Canada
May 31 – June 3, 2017/ *Mai 31 – Juin 3, 2017*

AN INTEGRATED SIMULATION METHOD FOR PERFORMANCE-BASED ASSESSMENT OF A STRUCTURE

Huang, Xu[1,3], Sadeghian, Vahid[1], Rong, Feng[2], Kwon, Oh-Sung[1] and Vecchio, Frank[1]
[1] Department of Civil Engineering, University of Toronto, Canada
[2] S-FRAME Software Inc., Canada
[3] xu.huang@mail.utoronto.ca

Abstract: In the current engineering practice, the performance assessment of structures is carried out through a two-stage process. First, a system-level analysis is performed mostly based on equivalent linear elements. Then, the critical components identified from the system-level assessment are examined in more detail using either sophisticated nonlinear analysis tools or laboratory testing. Such two-stage process does not accurately capture the interaction of the critical elements with the rest of the structural system, particularly in highly nonlinear stages of structural response. To overcome this limitation, an integrated simulation method is proposed for an explicit consideration of nonlinear behaviour of the critical elements during the system-level analysis. In this method, the potential critical components, which are initially represented with linear elements, are modelled in a detailed FE analysis tool using more sophisticated elements. The two sub-structure models communicate through internet network to satisfy compatibility and equilibrium requirements. By using a standardized communication routine for data exchange between the substructures, any other potential elements, either numerical or physical, can be readily integrated. In this paper, the application of the proposed method is illustrated through an integration of a commercial structural analysis and design software, S-FRAME, with a state-of-the-art analysis tool, VecTor2, for the performance assessment of a reinforced concrete high-rise building. Based on the analysis results, the procedure is found to provide a more realistic behaviour of the critical elements and also considered the influence of the force redistribution due to the failure of critical elements on the performance of the structural system.

1 INTRODUCTION

Over the past few decades, the advancements in computing technology have led to the development of various types of structural analysis software enabling the analysis and design of large indeterminate structural systems. Today, these computing tools are an essential part of almost every structural design firms. The commercial structural analysis software are mainly based on the linear elastic formulations which are deemed adequate for traditional strength-based design approach. While there exist nonlinear analysis tools which are aimed at performance based design, the inelastic hysteretic behaviour of elements need to be calibrated against available experimental results or sophisticated element level model. As modern codes move toward performance-based design approach, nonlinear analysis procedures are required to provide a more realistic performance assessment of the structural behaviour. This enables achieving a set of performance objectives (e.g., a level of damage or deflection) in addition to providing life safety when the structure is subjected to an extreme loading event such as earthquakes.

To take into account the nonlinearity effects into the analysis procedure, structural analysis tools have incorporated mainly two types of elements: lumped plasticity elements and distributed plasticity elements. Lumped plasticity elements are based on linear elastic frame members with zero-length plastic hinges located at the ends (Filippou and Issa 1988; Brancaleoni et al. 1983). They are implemented in a number of commercial structural software including SAP2000 (CSI 2015) and S-FRAME (2013). Although these elements are numerically efficient and stable, due to their oversimplified formulations, they have limitations including: 1) inability to consider gradual change of nonlinear behaviour over the member length and 2) requiring the user to define spring parameters prior to the analysis. Unlike lumped plasticity methods, the distributed plasticity methods consider material nonlinearity effects at every section of an element, providing a more accurate simulation of the structural behaviour (Kaba and Mahin 1984; Zeris and Mahin 1991). However, these methods are mainly based on the assumption of “plane section remains plane” and therefore are unable to accurately capture nonlinear stress distributions at the disturbed regions (e.g., beam-column joints). The distributed plasticity methods are employed in some academic structural software such as OpenSees (Mazzoni et al. 2007) and VecTor5 (Guner and Vecchio 2011).

In addition to the above-mentioned limitations of the plasticity methods in capturing the material nonlinear effects, there are situations where a more advanced assessment of the structural behaviour is essential such as when a structure is subjected to unintended or extreme loads (e.g., fire, impact, and blast) or when a structure is deficient and requires rehabilitation. The most reliable methods to accurately capture the behaviour of complex structures are nonlinear finite element analysis (NFEA) techniques and experimental studies. However, due to the high computational and memory demands of the NFEA techniques and the expensive laboratory requirements of the experimental testing, for most instances, these methods are limited to component-level studies. One effective approach for analysis of the entire structural system while taking into account the detailed behaviour of critical components, is to integrate the global analysis software (e.g., plasticity-based tools) with local analysis programs (e.g., NFEA-based tools) and laboratory testing.

Traditionally, the integration of global and local analysis tools was performed using a two-step analysis technique known as global-local methods (Mote 1971; Noor 1986). In these methods, the data flowed only a one-way path from the global model to the local model, and therefore were unable to accurately capture the interaction between the models. Another approach was to couple different types of elements enabling mixed-dimensional analysis (Mccune et. al 2000; Garusi and Tralli 2002). The element coupling methods were implemented in some commercial structural software such as ABAQUS (2013) and ANSYS (2013) and were successfully employed for the multi-scale analysis of large structural systems (Li et al. 2009; Wang et al. 2014). In recent years, a few studies attempted to extend the mixed-dimensional modelling to multi-platform analysis by combining global and local programs in a concurrent manner. However, the existing multi-platform methods were mostly problem-dependent or limited to specific type of analysis tools. For example, Mata et al. (2008) connected two different numerical modules in a master-slave manner using a message passing library particularly for analysis of reinforced concrete (RC) frame structures. Chen and Lin (2011) developed an internet-based computing framework enhanced with two levels of parallel processing enabling efficient multi-scale simulation using ABAQUS. However, addition of a new analysis tool to the framework required extensive changes to the system.

While the number of existing concurrent multi-platform analysis methods is limited, there have been many studies on concurrent numerical-experimental simulation (i.e. hybrid simulation). Yang et al. (2002) incorporated a user-defined element in OpenSees, representing the properties of the test specimen, with a data exchange framework to conducted hybrid simulation. Takahashi and Fenves (2005) employed a similar approach and developed an object-oriented framework compatible with a wide range of testing facilities and configurations. Pan et al. (2005) adopted a sharing file technique and proposed an online testing system for geographically distributed hybrid simulation using an in-house lumped-plasticity based analysis program. Karavasilis et al. (2008) and Saouma et al. (2012) carried out real-time hybrid tests by implementing nonlinear structural elements in simulation frameworks. The main focus of the aforementioned studies was to provide a flexible testing module and little effort was made to extend the capabilities of the numerical module. However, there are situations where some of the critical members of the structure should be numerically modelled and the ability of the analysis procedure is crucial. Kwon et al. (2008) addressed this issue by proposing a hybrid simulation framework compatible with various types of analysis tools. Although the framework was successfully employed for several hybrid simulations, because the integration procedure required transferring information of all the degrees of freedom between different modules its application to large structural systems was questionable.

In this study, a new integration method is developed which attempts to address some of the limitations of existing multi-scale simulation techniques, extending their application to commercial structural software. The proposed integration method is enhanced with a standardized data exchange format greatly facilitating communication with diverse numerical analysis tools and test specimens. A recently developed interface program was employed to provide a flexible physical module compatible with a wide range of laboratory equipment and testing configurations. Static condensation technique was used to reduce size of data exchange enabling simulation of large structural systems. The feasibility of the proposed method is demonstrated by integrating a commercial structural software with an academic state-of-the-art analysis tool and laboratory testing and by performing a multi-scale analysis of a reinforced concrete high-rise building.

2 INTEGRATED SIMULATION METHOD

2.1 Concept

The incremental equilibrium equation of a structure subjected to static loads can be expressed as

[1] $K\Delta u = P_{n+1} - R(u_n)$

where P_{n+1} is the applied force at time step $n+1$; $R(u_n)$ is the structural restoring force based on the displacement at previous time step, u_n; Δu is the displacement increment; and K is the system Jacobian matrix which is equal to $\partial R(u_n)/\partial u$. For linear problems, K is equal to the initial structural stiffness and Eq. 1 can be solved directly by multiplying the unbalance force, $P_{n+1} - R(u_n)$, with the inverse of K. For nonlinear problems, nonlinear solution schemes should be used to solve Eq. 1 iteratively. For example, if the Newton Rapson method is used, K needs to be updated in each iteration.

The integrated simulation method allows a structure to be divided into several substructures, each of which can be numerically analyzed or physically tested in different modules in a concurrent manner. Then, Eq. 1 becomes

[2] $\boldsymbol{A}_{i=0}^{N} K^i \Delta u = P_{n+1} - \boldsymbol{A}_{i=0}^{N} R^i(u_n)$

where N is the number of substructures; $\boldsymbol{A}$ is the direct assembly operator.

Regardless of the number of substructures, a master program must be used to formulate and solve the equilibrium equation of the structural system, Eq. 2. Such program is referred to as an integration module in this paper. Then the substructures, either numerical (referred to as numerical substructure modules) or experimental (referred to as experimental modules), contribute to Eq. 2 in terms of Jacobian matrices of substructures, K^i, and the restoring forces, $R^i(u_n)$. For nonlinear problems, in each iteration of the simulation, the updated Jacobian matrix of each substructure module is required to be sent from the substructure module to the integration module. However, it is unfeasible to retrieve the matrix from an experimental specimen as well as from a numerical substructure module since most analysis tools cannot export the Jacobian matrix. Therefore, the predefined initial stiffness matrix of the substructure is often used resulting in a constant Jacobian matrix for nonlinear iterations.

For the proposed method, the initial stiffness and the applied external loads of the entire system should be defined in the integration module. The simulation starts by sending predicted target displacements from the integration module to the substructure modules. The substructure modules impose the received displacements to their elements and return the restoring forces resulted from their analysis back to the integration module. Then, the integration module assembles the restoring forces of all the substructures and solves the equilibrium equation of the system to determine new displacements for the next step of the simulation. This is an iterative procedure which continues until all the restoring forces and displacements are converged. This method ensures the compatibility and equilibrium requirements are satisfied in both the substructure-level and system-level.

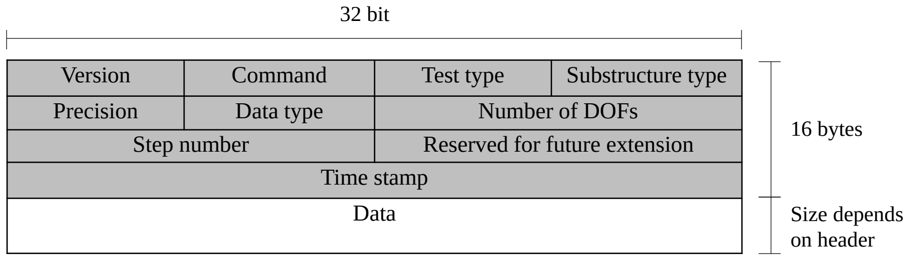
Note that the above-mentioned analysis scheme is also applicable to dynamic problems since the equation of motion can be described in the form of an equivalent static equation similar to Eq. 1.

2.2 Communication between integration and substructure modules

To allow data exchange between the integration module and substructure modules for various simulation purposes, a generalized communication library, termed as the University of Toronto Networking Protocol (UTNP), has been developed to facilitate the integration process. The library contains two components: communication protocol and standardized data exchange format.

Current development of the communication protocol in UTNP uses a sequential communication scheme. Specifically, the substructure module waits for a connection and a command from the integration module. Once the integration module is connected and a command is received, the substructure module runs analysis or experiment and sends the restoring forces back to the integration module. This communication protocol ensures the data between the integration and substructure modules are synchronized to the same loading stage.

The UTNP data exchange format is shown in Figure 1. It consists of a 16-byte header block and a data block. The header block is designed with the parameters for error-checking and most importantly covers the necessary information, such as the number of degrees of freedoms at the interface, data type (a parameter to indicate the type of displacement, velocity, acceleration, force, or any combinations of them), data precision (single or double precision), needed to determine the type and size of the data in the data block. More details about the data exchange format can be found from reference (Huang and Kwon 2015).



Version	Command	Test type	Substructure type
Precision	Data type	Number of DOFs	
Step number		Reserved for future extension	
Time stamp			
Data			

Figure 1. Standardized data exchange format in UTNP

To facilitate the implementation of the UTNP in any potential analysis tools for integrated simulations, a dynamic link library (DLL) which is compiled with a set of functions to send and receive data packets using UTNP has been developed. The dynamic library can be used with different programming languages such as MATLAB, C++, LabView, and FORTRAN. By externally linking the library, an existing analysis tool is enabled to communicate with other modules.

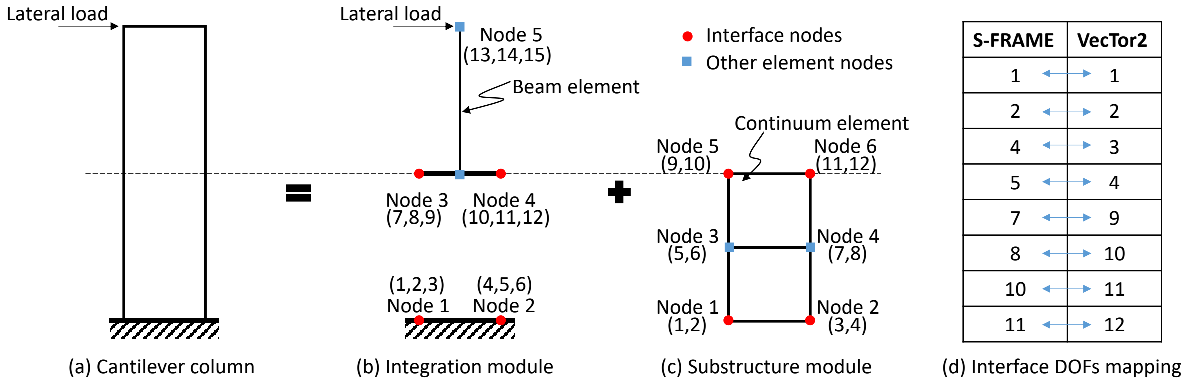
3 IMPLEMENTATION DETAILS

Many analysis-design software packages have user-friendly interface, specialized tools for design process and advanced solvers for large structures. However, most of them still use simplified methods to consider the nonlinear behaviour of structural elements. These simplified methods are computationally efficient but usually need calibrations based on experiments or more sophisticated models. The analysis programs developed in the research domain, on the other hand, employ more advanced modelling methods which are suitable for complex problems, but are computationally expensive. The integration of different analysis tools is expected to facilitate the use of these advanced modelling methods for structural performance assessment in daily practice. This section illustrates the implementation process of the proposed method by integrating the commercial analysis-design software, S-FRAME, with the state-of-the-art analysis tool, VecTor2, for analysis of reinforced concrete structures.

3.1 Integration module

In this study, the S-FRAME program is implemented as an integration module which serves as the main solver for Eq. 2. In addition to providing fast and reliable solvers, the program also features a variety of elements for different structures, such as frames, trusses, bridges, skyscrapers, shells, and cable structures. Therefore, in the integrated simulation, not all restoring forces, $R^i(u_n)$ in Eq. 2, are necessary to be obtained from the substructure modules. In other words, the majority of the structural system can be modelled in the integration module while the substructure modules only include the components that need to be presented using sophisticated finite element models or physical specimens.

The implementation of the UTNP library in the S-FRAME program, or any other potential integration module, does not need a major change to the program, but only requires adding a few lines to link the functions in the DLL library. To accurately transfer displacements and forces between the integration module and the substructure modules, the interface nodes and their degree of freedoms (DOFs) defined in both sides should be consistent. Specifically, the interface displacements and forces to be transmitted are all defined in the global coordinate system. In addition, the ordering of the interface nodes and DOFs in S-FRAME should be consistent with that defined in the substructure module. For example, Figure 2 illustrates the definition of the interface nodes for an integrated simulation of a cantilever column. The column is divided into two parts. The top half of the column is modelled with a beam element in S-FRAME while the bottom half is modelled using two continuum elements in VecTor2. The Beam element with three DOFs per node and the continuum elements with two DOFs per node are connected through two rigid link elements. In S-FRAME, the interface nodes are defined in the ordering of 1, 2, 3 and 4 and it is assumed that the DOFs follow the same numbering order to that used for the interface nodes. To meet the consistency requirement, the interface nodes in the VecTor2 model must be defined in the ordering of 1, 2, 5, and 6. Figure 2(d) shows the mapping of the interface DOFs. Since the continuum elements in VecTor2 do not have rotational DOFs, only translational DOFs are taken into account for data exchange. The rotation is transferred from the frame sub-model to the continuum sub-model based on the assumption of “plane sections remain plane”.



S-FRAME	VecTor2
1	1
2	2
4	3
5	4
7	9
8	10
10	11
11	12

(a) Cantilever column (b) Integration module (c) Substructure module (d) Interface DOFs mapping

Figure 2. Node mapping example. (The numbers in the parentheses indicate the DOF numbers in X, Y and Rz directions)

3.2 Substructure module

In this study, the VecTor2 program, a nonlinear finite element analysis program for the analysis of reinforced concrete structures, is implemented as the substructure module to work with S-FRAME. The theoretical basis of the program are the Modified Compression Field Theory (Vecchio and Collins 1986) and the Disturbed Stress Field Model (Vecchio 2000), and it has been extensively validated against experimental results and thus has been widely used for the performance assessment of RC structures. Implemented with UTNP, the program is able to export the condensed initial structural stiffness matrix at the interface DOFs, run static analysis based on the displacement command from the integration module, and send the computed restoring forces back to the integration module. Figure 3 shows the flowchart of the integrated simulation with S-FRAME and VecTor2. In fact, the use of the UTNP library has enabled the development of a generalized simulation framework at the University of Toronto (Huang and Kwon 2015). In the framework, various analysis tools have been implemented as the integration modules and/or substructure modules. To incorporate experimental specimen as a substructure module, a program referred to as Network Interface for Controllers (NICON) is also developed (Zhan and Kwon 2015). Therefore, the substructure module to work with S-FRAME is not limited to VecTor2 as presented in this paper, but any analysis tools or physical testing setups for which the UTNP library has been implemented can be connected to the integration module.

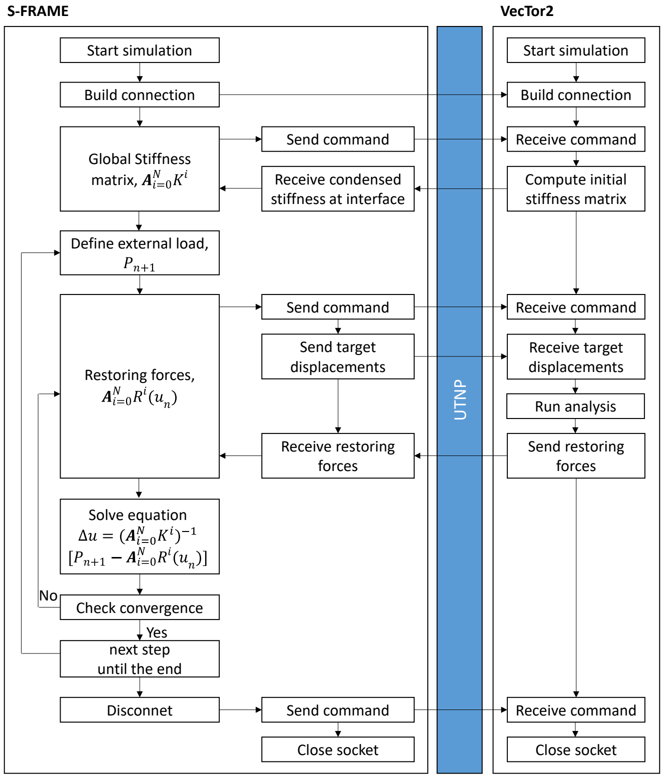
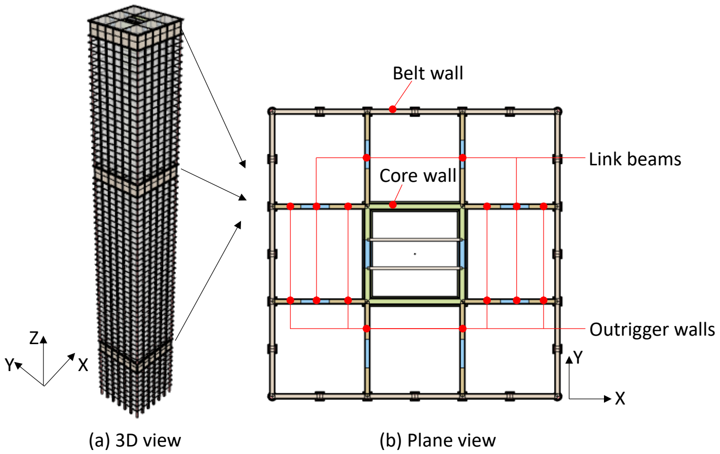


Figure 3. Flowchart of the integrated simulation with S-FRAME and VecTor2

4 APPLICATION EXAMPLE

The proposed integrated simulation method is used to assess the performance of a high-rise reinforced concrete building as shown in Figure 4(a). The main characteristics of the building are summarized in Table 1. The building has a center core wall system and an outer RC frame. These two systems are tied together through three outrigger systems at the 10~11, 44~45, and 66~67 floors. More details about the outrigger system is shown Figure 4(b). It can be seen that the system includes a perimeter belt wall which is connected to the core through outrigger walls in both the X- and Y- directions. Two adjacent outrigger walls are connected through a link beam which tends to be shear-critical under the lateral loads. The design of the building has been done in S-FRAME where all structural components are represented using linear-elastic frame and plate elements.

To investigate the influence of the link beams on the performance of the high-rise building subjected to lateral loads, a total of sixteen link beams (4 floors × 4 beams) located at the 10~11th and 44~45th floors in the X-direction, which are originally modelled with frame elements in S-FRAME, are replaced with continuum elements in VecTor2 as show in Figure 5 using the proposed integrated simulation method. In the VecTor2 model, concrete and longitudinal reinforcing bars are modelled with rectangular elements and truss elements, respectively. The out-of-plane and in-plane stirrups are smeared over the entire concrete core. To accurately capture the shear deformation, at least 10 elements are used in the vertical direction. Since the link beams modelled in VecTor2 do not have out-of-plane stiffness (in the Y-direction), only the in-plane response of the building (the X-Z plane in Figure 4(a)) is taken into account. To connect the frame elements in S-FRAME, which have two in-plane translational DOFs (in the X- and Z- directions) and an in-plane rotational DOF (along the Y-axis) per node, with the continuum elements which only have two translational DOFs per node, rigid link elements are used in S-FRAME as shown in Figure 5(b).

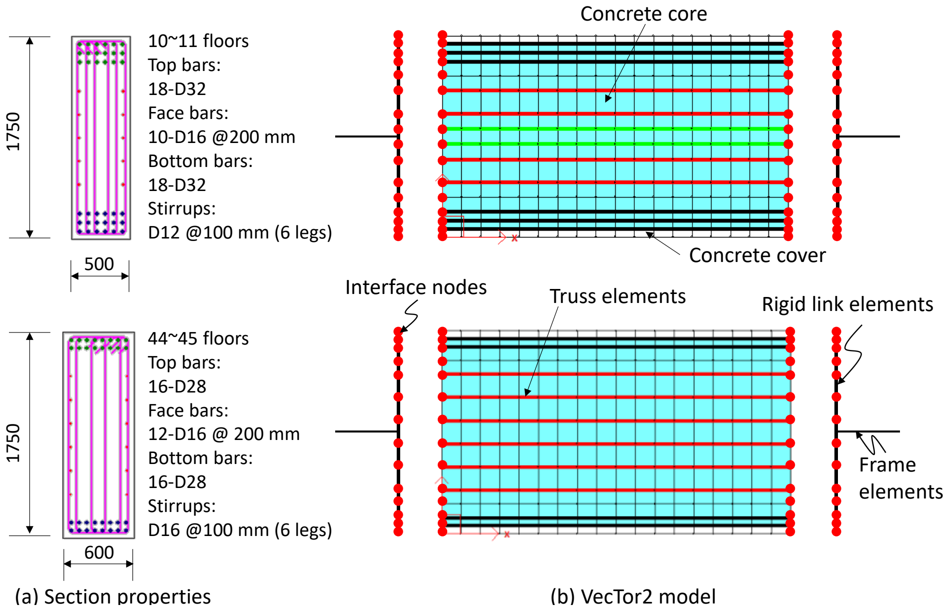


(a) 3D view (b) Plane view

Figure 4. Layout of the high-rise building

Table 1. Main parameters of the high-rise building

Parameters	Description
Stories	68
Height (m)	239.5
Normal Storey height (m)	3.5
Concrete strength (MPa)	60
Reinforcement yielding strength (MPa)	345



(a) Section properties (b) VecTor2 model

Figure 5. Section properties and numerical models of the link beams

Pushover analysis is conducted in the X-direction to evaluate the nonlinear behaviour of the link beams when lateral loads are applied. The lateral load distribution is proportional to the mass times the first mode shape of the building. Figure 6 shows the pushover curves resulted from the analysis of the integrated model and the original S-FRAME model. It can be found that both curves have the same initial behaviour up to shear weight ratio of 0.08g. After that, shear cracks are first developed in the link beams at the 44~45th floors. With the increase of the lateral loads, shear cracks are also computed in the link beams at the lower floors as shown in Figure 7. Significant shear cracks can be observed in the middle depth of the beams at the 44th floor. The nonlinear behaviour of the link beams leads to an increase of the roof drift by 18% at the base shear-weight ratio of 0.93g. In addition, as shown in Figure 8, it reduces the force demand to the adjacent beams and results in force redistribution between the elements.

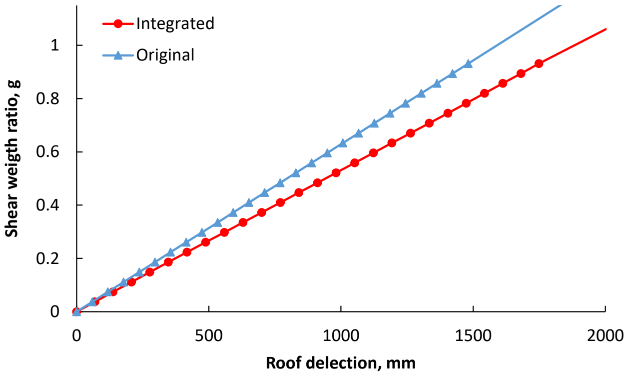
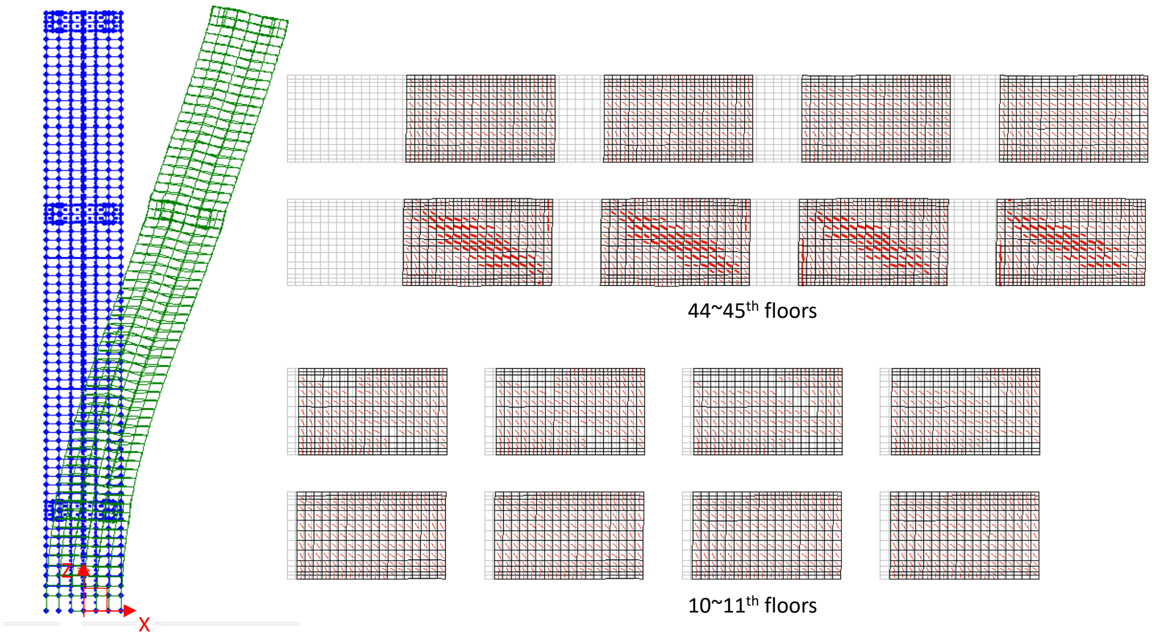


Figure 6. Predicted pushover curves from the original and the integrated model



(a) S-FRAME model

(b) VecTor2 model

Figure 7. Deformed shape and crack pattern at shear weight ratio of 0.93g

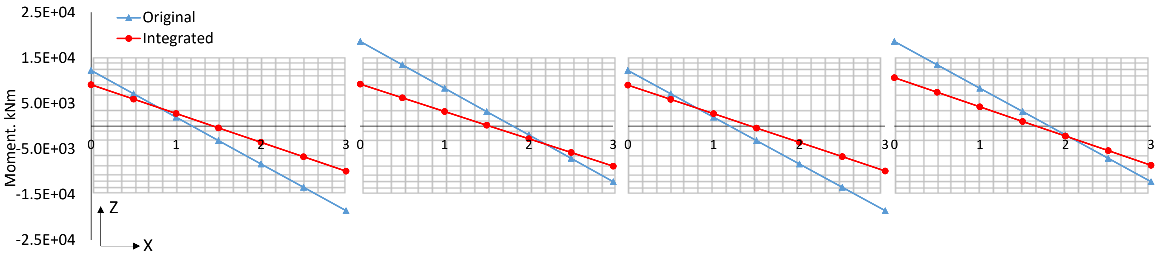


Figure 8. Moment diagrams of the link beams at the 44th floor at the base shear-weight ratio of 0.93g

To verify the proposed hybrid simulation configuration, an experimental program is planned to be carried out in which the performance of the reference reinforced concrete high rise building will be evaluated in a hybrid manner. The physical module will be a small-scale specimen representing the most critical link beam of the structure. The specimen will be tested using a 6-DOF hydraulic testing facility shown in Figure 9. Other critical link beams will be modelled in the VecTor2 finite element analysis program. The remainder of the building will be analyzed in the S-FRAME integration module using computationally fast frame-type elements.

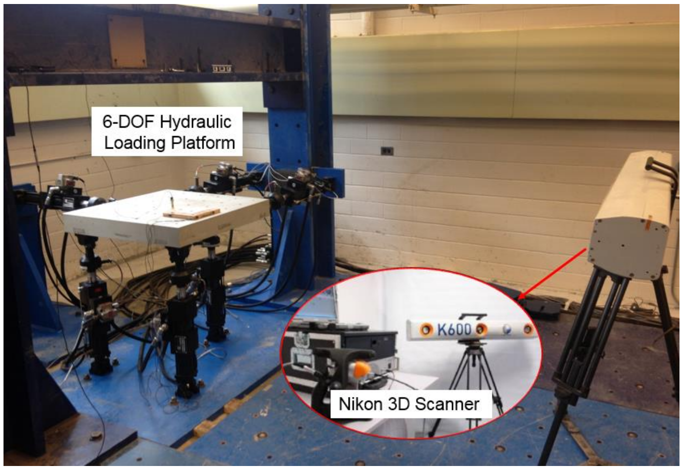


Figure 9. 6-DOF loading platform and 3D scanner

5 CONCLUSIONS

This paper presents a method to integrate different elements, either numerical or experimental, into a commercial design-analysis program, S-Frame, for structural performance assessment. The integration method is enhanced with a standardized data exchange format and communication protocol through which any potential analysis tools can be integrated to contribute to future design and analysis processes. The feasibility and potential application of the proposed method is illustrated with a pushover analysis of a real-world high-rise building. While not presented in this paper, the S-Frame model integrated with VecTor2 models is also capable of running nonlinear dynamic analysis. An experimental program is planned to validate hybrid simulation capability of the proposed integration method by combining the S-FRAME commercial structural software, the VecTor2 academic program, and a test specimen. Both the multi-scale numerical method and hybrid simulation are expected to allow practicing engineers and researchers to accurately evaluate the seismic performance of a structural system in a practical manner.

Acknowledgements

This research is carried out in collaboration with S-Frame Software Inc. through the Engage Grant funded by Natural Sciences and Engineering Research Council of Canada.

References

ABAQUS. 2013. Analysis User’s Manual. Dassault Systèmes Simulia Corp. Providence, RI, USA.
ANSYS. 2013. *User’s Guide*. ANSYS Inc. Canonsburg, PA, USA.
Brancaleoni, F., Ciampi, V. and Di Antonio, R. 1983. Rate-Type Models for Non-Linear Hysteresis Structural Behaviour. *EUROMECH Colloquium*, Palermo, Italy.
Chen, H.M. and Lin, Y.C. 2011. An Internet-Based Computing Framework for the Simulation of Multi-Scale Response of Structural Systems. *Structural Engineering and Mechanics*, **37**(1): 17–37.
CSI. 2015. Analysis Reference Manual for SAP2000, ETABS, SAFE, and CSiBridge. Berkeley, California, USA.

Filippou, F.C. and Issa, A. 1988. Nonlinear Analysis of Reinforced Concrete Frames under Cyclic Load Reversals. Earthquake Engineering Research Center, University of California, Berkeley, Report No. UCB/EERC-88/12.

Garusi, E. and Tralli, A. 2002. A Hybrid Stress-Assumed Transition Element for Solid-to-Beam and Plate-to-Beam Connections. *Computers & Structures*, **80**(2): 105–115.

Guner, S. and Vecchio, F.J. 2011. Analysis of Shear-Critical Reinforced Plane Frame Elements under Cyclic Loading. *Journal of Structural Engineering,* ASCE, **137**(8): 834–843.

Huang, X. and Kwon, O.S. 2015. Development of Integrated Framework for Distributed Multi-Platform Simulation. *6th AESE/11th ANCRiSST*, University of Illinois, Urbana-Champaign, Unites States.

Kaba, S. and Mahin, S.A. 1984. Refined Modelling of Reinforced Concrete Columns for Seismic Analysis. Earthquake Engineering Research Center, University of California, Berkeley, Report No. UCB/EERC-83/03.

Karavasilis, T.L., Seo, C.Y. and Ricles, J.M. 2008. HybridFEM User's Manual: A Program for Dynamic Time Hisotry Analysis of 2D Inelastic Frame Structures and Real-Time Hybrid Simulation. Bethlehem, PA, USA.

Kwon, O.S., Elnashai, A.S. and Spencer, B.F. 2008. A Framework for Distributed Analytical and Hybrid Simulations. *Structural Engineering and Mechanics*, **30**(3): 331–350.

Li, Z.X., Chan, T.H.T., Yu, Y. and Sun, Z.H. 2009. Concurrent Multi-Scale Modeling of Civil Infrastructures for Analyses on Structural Deterioration-Part I: Modeling Methodology and Strategy. *Finite Elements in Analysis and Design*, **45**(11): 782–794.

Mata, P., Barbat, A.H. and Oller, S. 2008. Two-Scale Approach for the Nonlinear Dyanmic Analysis of RC Structures with Local Non-Prismatic Parts. *Engineering Structures*, **30**(12): 3667–3680.

Mazzoni, S., Mckenna, F., Scott, M.H. and Fenves, G.L. 2007. The OpenSees Command Language Manual, Version 2.0. Pacific Earthquake Engineering Research Center, University of California at Berkeley, Berkeley, CA.

Mccune, R.W., Armstrong, C.G. and Robinson, D.J. 2000. Mixed-Dimensional Coupling in Finite Element Models. *International Journal for Numerical Methods in Engineering*, **49**(6): 725–750.

Mote, C.D. 1971. Global-Local Finite Element. *International Journal for Numerical Methods in Engineering*, **3**(4): 565–574.

Noor, A.K. 1986. Global-Local Methodologies and Their Application to Nonlinear Analysis. *Finite Elements in Analysis and Design*, **2**(4): 333–346.

Pan, P., Tada, M. and Nakashima, M. 2005. Online Hybrid Test by Internet Linkage of Distributed Test-Analysis Domains. *Earthquake Engineering and Engineering Dynamics*, **34**(11): 1407–1425.

Saouma, V., Kang, D.H. and Haussmann, G. 2012. A Computational Finite-Element Program for Hybrid Simulation. *Earthquake Engineering & Structural Dynamics*, **41**(3): 375–389.

S-FRAME. 2013. Structural Office Software. S-Frame Inc, Richmond, British Columbia, Canada.

Takahashi, Y. and Fenves, G.L. 2005. Software Framework for Distributed Experimental-Computational Simulation of Structural Systems. *OpenSees Developer Symposium*, UC Berkeley, USA.

Vecchio, F.J. 2000. Disturbed Stress Field Model for Reinforced Concrete: Formulation. *Journal of Structural Engineering,* ASCE, **126**(9): 1070–1077.

Vecchio, F.J. and Collins, M.P. 1986. The Modified Compression Field Theory for Reinforced Concerete Elements Subected to Shear. *Journal of American Concrete Institute*, **83**(2): 219–231.

Wang, F.Y., Xu, Y.L. and Qu, W.L. 2014. Mixed-Dimensional Finite Element Coupling for Structural Multi-Scale Simulation. *Finite Elements in Analysis and Design*, **92**(1): 12–25.

Yang, Y.S., Hsieh, S.H., Wang, K.J. and Wang, S.J. 2002. Numerical Analysis Framework for Distributed Pseudo-Dynamic Tests. *2nd International Conference on Structural Stability and Dynamics*, Singapore.

Zeris, C.A. and Mahin S.A. 1991. Behaviour of Reinforced Concrete Structures Subjected to Biaxial Excitations. *Journal of Structural Engineering,* ASCE, **117**(9): 2657–2673.

Zhan, H. and Kwon, O.S. 2015. Actuator Controller Interface Program for Pseudo-Dynamic Hybrid Simulation. *2015 World Congress on Advances in Structural Engineering and Mechanics*, Songdo, Korea.