Engineering Structures 181 (2019) 260–270

Contents lists available at ScienceDirect

Engineering Structures

journal homepage: www.elsevier.com/locate/engstruct

# A framework for multi-platform simulation of reinforced concrete structures

Vahid Sadeghian[a,*], Oh-Sung Kwon[b], Frank Vecchio[b]

[a] Department of Civil and Environmental Engineering at the Carleton University, Ottawa, ON, Canada
[b] Department of Civil and Mineral Engineering at the University of Toronto, Toronto, ON, Canada

ARTICLE INFO

*Keywords:*
Reinforced concrete
Shear behaviour
Finite element modelling
Multi-scale analysis
Beam-column joint behaviour
Soil-structure interaction

ABSTRACT

This study presents a framework for multi-platform analysis and hybrid simulation of reinforced concrete (RC) structures. In this approach, each subpart of the structure, based on its mechanical characteristics, is modelled using the most suitable finite element analysis tool or represented with a test specimen. The proposed framework combines all the substructure modules and takes into account the interactions between them by satisfying compatibility and equilibrium requirements. The main contribution of the study lies in demonstrating the effectiveness of multi-platform modelling in accurate and practical analysis of complex RC structures or multi-disciplinary RC systems with a particular focus on shear behaviour. Three application examples including a wide-flange shear wall, a three-storey frame with critical joints, and a soil-structure interaction simulation are discussed in detail. It is concluded that the multi-platform analysis can compute the behaviour of such structures with a level of accuracy that was previously difficult to achieve with most single-platform analysis software.

## 1. Introduction

Over the past few decades, significant progress has been made with respect to nonlinear analysis of reinforced concrete (RC) structures. With advancements in computing technology, nonlinear analysis procedures have been implemented into various types of structural software, expanding the size and complexity of the problems that can be analyzed. Despite these great improvements, because of the complex behaviour of cracked reinforced concrete, there is no analysis software that can perform well for all types of structures and loading conditions. Typically, each structural software has its own advantages and disadvantages and is only suitable for certain types of problems. In addition, there are situations where the behaviour of a structure can be influenced by the surrounding soils and therefore a soil-structure interaction analysis is required. However, most structural programs do not have advanced soil modelling capabilities and geotechnical software cannot accurately model structural behaviour. Therefore, for accurate and practical analysis of complex RC structures in a multi-scale manner or modelling multi-disciplinary RC systems, more advanced simulation methods are required.

The most common approach that has been used for multi-scale modelling of structures is a two-step technique known as the global-local method. In this approach, first a global analysis of the entire structure with a relatively coarse numerical model is performed to determine the internal forces and displacements. Then, the critical components of the structure are analyzed using sophisticated local models with the boundary values being the displacements obtained from the global analysis. In general, there are two types of global-local methods: non-iterative methods [1–4] and iterative methods [5,6]. The iterative methods satisfy both local and global equilibrium conditions and therefore are more accurate than the non-iterative methods; however, they are limited to linear and simple nonlinear problems. Also, since the analysis is not performed in a concurrent manner, the force redistribution due to the stiffness changes in the system is not fully considered.

With advancements in computer science, parallel simulation methods have been developed to improve the computational performance of analysis tools. These methods have been implemented into structural analysis programs either as parallel equation solvers [7,8] or parallel processing techniques [9,10], also known as domain decomposition methods. Although these methods can significantly improve the computational performance of the analysis, for large or complex problems, they require advanced computing facilities which are expensive and may not be available in a typical engineering design office.

Another approach for analyzing complex structures in both component- and system-level is mixed-dimensional modelling where two or more types of elements with different numbers of degrees of freedom (DOFs) are coupled in a single finite element (FE) model. Several researchers successfully employed mixed-dimensional modelling methods for multi-scale analysis of RC structures. Li et al. [11] developed a

* Corresponding author.
*E-mail addresses:* vahidsadeghian@cunet.carleton.ca (V. Sadeghian), os.kwon@utoronto.ca (O.-S. Kwon), fjv@ecf.utoronto.ca (F. Vecchio).

https://doi.org/10.1016/j.engstruct.2018.12.023
Received 9 November 2017; Received in revised form 2 November 2018; Accepted 7 December 2018

mixed-dimensional model of a three-span RC bridge structure using the LS-DYNA software [12]. Yue et al. [13] analyzed a flexure-critical composite steel-concrete frame structure in a multi-scale manner using the ABAQUS program [14]. In both studies, the critical regions of the structure were modelled using solid elements, while the remaining regions were modelled with beam elements. The mixed-dimensional analysis results agreed well with those obtained from the full-solid model. Sadeghian et al. [15] proposed a new beam-membrane interface element specifically formulated for RC members and applied it to analysis of a series of RC beams with different failure modes and an RC frame structure. There was a good correlation between the analysis results and experimentally observed behaviour.

In recent years, some studies attempted to extend mixed-dimensional modelling to multi-platform simulation. In this approach, each potentially critical or complex member is modelled in detailed using the most suitable FE analysis tool or is represented with a physical specimen, while the rest of the structure is modelled with a computationally fast global analysis software. An integrated simulation procedure is required to take into account the interaction between the substructure modules. Most of the published multi-platform studies have focused on hybrid (numerical-experimental) simulation where the numerical modules were analyzed using frame-type programs [16–18]. There have been only a few studies which focused on integrating different analysis tools to extend the modelling capabilities of single-platform programs. Mata et al. [19] combined a global frame-type analysis tool with a local FE analysis program for integrated simulation of RC structures. However, the integrated procedure was only verified against the full-frame model and not the detailed FE analysis or experimental results. In addition, the method is limited to in-house programs. Chen and Lin [20] developed an internet-based simulation framework enhanced with two levels of parallel processing. In this method, the data flows only in a one-way path from the global model to the local models, compromising the ability of the method to accurately capture the interaction between the models. Kwon et al. [21] proposed a simulation framework which enables integration of different analysis tools and experimental equipment. With this method, the integrated procedure requires a substantial amount of data exchange, and therefore its practical application to analysis of large RC structures is disputable.

In addition to the above-mentioned limitations, none of the existing studies specifically investigated the application of multi-platform modelling to analysis of shear-critical RC structures. The majority of the studies used frame-type methods which are based on the assumption of 'plane sections remain plane' and therefore cannot accurately capture shear behaviour. Others used detailed FE programs with concrete damage models calibrated for flexure-critical structures. Because of the more complicated nature of shear behaviour compare to flexure behaviour, accurate prediction of structural response requires more comprehensive analysis procedures. Structural software with such analysis capabilities are generally highly time consuming and limited to the component-level modelling. Multi-platform modelling can improve both accuracy and practicality of the analysis method by taking into account both the component- and system-level behaviour.

This study, presents a generalized framework for multi-platform analysis and hybrid simulation of complex or multi-disciplinary reinforced concrete systems. The framework addresses the common deficiencies of previous methods; namely, (1) their inability to fully consider the interactions between substructure modules, (2) their need for substantial amount of data exchange, (3) their limited application which is mostly restricted to in-house analysis tools and specific testing configurations, and (4) their inability in capturing shear-critical behaviour. For application of the framework to hybrid testing and multi-platform analysis of repaired RC structures, refer to Sadeghian et al. [22] and Sadeghian et al. [23], respectively.

## 2. Proposed simulation framework

### 2.1. Overview

The recently developed multi-platform simulation framework, Cyrus, is written in the C++ programming language using the Microsoft Foundation Classes. The architecture of the framework is based on an object-oriented methodology where each class is written in a standard form making it suitable for adopting by other potential analysis tools. The framework enables integration of both numerical and experimental substructure modules. To date, eight different nonlinear analysis tools have been integrated into the simulation framework: Zeus-NL [24], OpenSees [25], ABAQUS [14], and the VecTor suite of software which includes VecTor2 [26], VecTor3 [27], VecTor4 [28], VecTor5 [29], and VecTor6 [30]. The VecTor programs are based on the Modified Compression Field Theory [31] a well-recognized theoretical model for RC structures which was developed about 40 years ago and has been extended to advanced research areas in recent years [32]. For the programs with accessible source codes, the communication and integration functions are implemented in the source code. For other programs and experimental modules, two interface program, NICA [21,35] and NICON [33], are used to provide network communication and integration capabilities. In the following sections, a comprehensive discussion of the integrated simulation procedure, communication methods, and interface programs is provided.

### 2.2. Combination of different nonlinear solution schemes

To consider the geometry and material nonlinearities of RC structures, analysis programs use different types of iterative solution schemes which can be mainly categorized into two groups: tangent stiffness-based methods and secant stiffness-based methods. Both groups can be represented either with incremental formulations or total formulations. In this section, the integration of different solution schemes for multi-platform simulation is investigated.

#### 2.2.1. Incremental formulations

With the incremental tangent stiffness-based methods, the force-displacement relationship at iteration *i* is defined as follows:

$$\{F\} - \{f\}^{i-1} = [K]_t^{i-1}(\{U\}^i - \{U\}^{i-1}) \tag{1}$$

where {F} is the external load vector, {f} is the internal force vector computed based on the element stresses, {U} is the displacement vector, and $[K]_t^{i-1}$ is the incremental tangent stiffness matrix representing the slope of the load-deflection response at the previous iteration and calculated as follows:

$$K^i{}_t = \frac{\partial f^i}{\partial U^i} \tag{2}$$

As shown in Fig. 1(a), by updating the incremental tangent stiffness matrix at every iteration of the solution scheme, new values for the displacement vector are computed. This procedure is repeated until the displacement values converge within a predefined error limit.

The incremental secant stiffness-based methods are based on a similar load-deflection relationship except the stiffness matrix ($K_s$) is defined as the slope of the line which connects the previous iteration ($u_{i-1}$, $f_{i-1}$) and current iteration ($u_i$, $f_i$) points (see Fig. 1(b)); Thus:

$$K_s^i = \frac{\Delta f^i}{\Delta U^i} = \frac{\{f\}^i - \{f\}^{i-1}}{\{U\}^i - \{U\}^{i-1}} \tag{3}$$

For sufficiently small load steps, as shown in Fig. 1(c), the incremental tangent stiffness can be approximated as being equal to the incremental secant stiffness. Since structural software with shear analysis capabilities generally require small load steps to reach convergence, the integration of the incremental forms of tangent and secant solution schemes for multi-platform simulation is feasible.

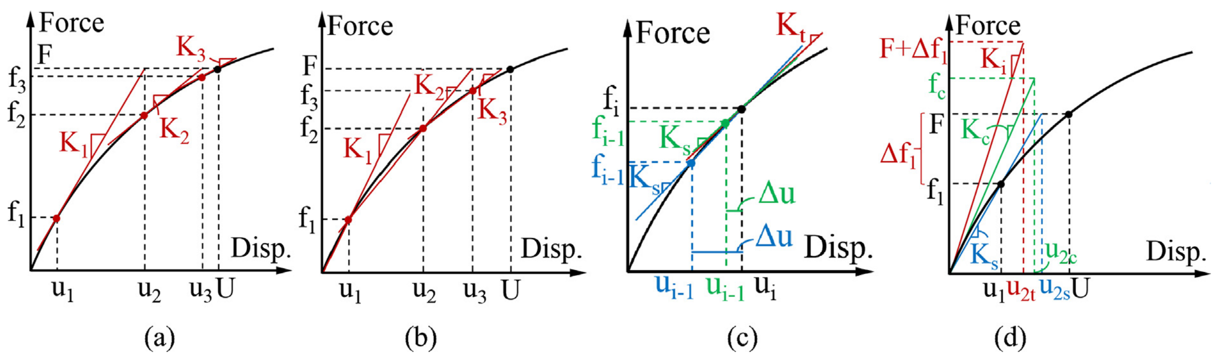


**Fig. 1.** (a) incremental tangent-based method; (b) incremental secant-based method; (c) approximation of secant method with tangent method; (d) combination of total solution methods.

#### 2.2.2. Total formulations

The integration of the tangent and secant stiffness-based methods can also be shown for the total forms of the methods. For the total form of the tangent stiffness-based method that is presented here, the stiffness matrix is constant throughout the iterations and load steps, and is equal to the initial stiffness matrix of the structure. The nonlinear behaviour of the structure is taken into account by updating the unbalanced force values in each iteration. A similar discussion can be presented for the total tangent stiffness-based method in which the stiffness matrix is updated at every load stage. With the total secant stiffness-based method, the stiffness is defined as the slope of the line which connects the current iteration point ($f_1$, $u_1$) to the origin. To take into account the nonlinearity of the response, the stiffness matrix is updated at every iteration. Fig. 1(d) presents the total forms of the tangent and secant stiffness-based methods with red and blue lines, respectively, and the combination of the two methods using green lines. It can be seen that combining the two methods results in the total stiffness values ($K_c$) that are not as high as the initial stiffness ($K_i$) and not as low as the total secant stiffness ($K_s$) of the system. Similarly, the total combined force values ($f_c$) are within the range of the total external force (F) and total unbalanced force values ($F + \Delta f_1$). Therefore, the displacements computed by the combined method ($u_{2c}$) are always between the values obtained from the tangent and secant stiffness-based methods ($u_{2t}$ and $u_{2s}$, respectively). Thus, the total forms of the two solution schemes can be directly integrated.

The formulation described in this section is for a force control analysis case. A similar formulation can be derived for a displacement control analysis case. This requires an additional step in which the prescribed displacements are incorporated into the external force vector to reduce the equilibrium equation.

### 2.3. Formulation and solution of the system matrix

For the force-displacement relationship of each substructure module, the degrees of freedom can be partitioned into internal DOFs (indicated by subscripts *n*) and interface DOFs (indicated by subscripts *m*) as follows:

$$\begin{bmatrix} K_{mm} & K_{mn} \\ K_{nm} & K_{nn} \end{bmatrix} \begin{Bmatrix} U_m \\ U_n \end{Bmatrix} = \begin{Bmatrix} P_m \\ P_n \end{Bmatrix} \tag{4}$$

where [K] and {P} are the stiffness matrix and force vector defined according to the nonlinear solution scheme. Eliminating the internal displacements, $\{U_n\}$, from Eq. (4) leads to the following relation:

$$([K_{mm}] - [K_{mn}][K_{nn}]^{-1}[K_{nm}])\{U_m\} = \{P_m\} - [K_{mn}][K_{nn}]^{-1}\{P_n\} \tag{5}$$

Eq. (5) can be written in a format similar to that of the force-displacement equation by defining equivalent forms of the stiffness matrix, $[K_{mm}]_c$, and force vector, $\{P_m\}_c$, as follows:

$$[K_{mm}]_c\{U_m\} = \{P_m\}_c \tag{6}$$

Eq. (6) is the condensed form of Eq. (4) and can be used to calculate interface displacements between the substructures. After determining the interface displacements, the internal displacements for each substructure can be computed as follows:

$$\{U_n\} = [K_{nn}]^{-1}(\{P_n\} - [K_{nm}]\{U_m\}) \tag{7}$$

For analysis tools with accessible source code (Module Type 1), the communication and static condensation functions are implemented in the source code. In each step of the simulation, Cyrus collects the condensed forms of the stiffness matrix and force vector from each numerical module, maps them based on the connectivity of the substructures and a global numbering scheme, and solves for the interface displacements. To satisfy compatibility requirements, the displacements of the common interface DOFs between the substructures are considered to be identical and the displacements for the remaining interface DOFs are approximated using element coupling methods. The framework sends the interface displacements to the related modules so they can determine the internal displacements using Eq. (7). The VecTor suite of programs are integrated into the simulation framework using this approach. VecTor2, VecTor3, VecTor4, and VecTor6 are based on the secant solution scheme; whereas VecTor5 uses a hybridized (tangent-secant) solution scheme.

For analysis software that do not output the stiffness and unbalanced force values or experimental modules (Module Type 2), the multi-platform simulation is performed using the modified Newton Raphson procedure. To estimate the condensed form of the stiffness matrix ($[K]_c$), the simulation framework imposes small displacements to each interface DOF, while restraining other interface DOFs, and collects the computed restoring forces. It can be shown that the stiffness matrix assembled from the restoring forces is equal to the condensed form of the stiffness matrix expressed in Eq. (6). For example, consider a four-DOF substructure module which has two interface DOFs. The force-displacement relationship of the module can be written in a general form as:

$$\begin{Bmatrix} P_1 \\ P_2 \\ P_3 \\ P_4 \end{Bmatrix} = \begin{bmatrix} a & b & c & d \\ e & f & g & h \\ i & j & k & l \\ m & n & o & p \end{bmatrix} \begin{Bmatrix} U_1 \\ U_2 \\ U_3 \\ U_4 \end{Bmatrix} \tag{8}$$

Imposing a unit displacement at the interface DOF 1 ($U_1 = 1$), while restraining the displacement at the interface DOF 2 ($U_2 = 0$), yields the following equation:

$$\begin{Bmatrix} P_1 \\ P_2 \end{Bmatrix} = \begin{Bmatrix} a \\ e \end{Bmatrix} + \begin{bmatrix} c & d \\ g & h \end{bmatrix} \begin{Bmatrix} U_3 \\ U_4 \end{Bmatrix} \tag{9}$$

Also, by eliminating the rows and columns associated with the interface DOFs from Eq. (8), the displacements at the internal DOFs can be found as:

$$\begin{Bmatrix} U_3 \\ U_4 \end{Bmatrix} = \left(\frac{1}{lo - kp}\right)\begin{Bmatrix} pi - ml \\ km - io \end{Bmatrix} \tag{10}$$

Substituting Eq. (10) into Eq. (9), the equivalent forces for the applied displacements can be computed:

$$\begin{Bmatrix} P_1 \\ P_2 \end{Bmatrix} = \begin{Bmatrix} a \\ e \end{Bmatrix} + \left(\frac{1}{lo - kp}\right)\begin{Bmatrix} cip - clm + dkm - dio \\ gip - glm + hkm - hio \end{Bmatrix} \tag{11}$$

Based on Eq. (6), the condensed form of the stiffness matrix at the interface DOFs is as follows:

$$[K]_c = \begin{bmatrix} a\, b \\ e\, f \end{bmatrix} - \left(\frac{1}{lo - kp}\right)\begin{bmatrix} -dkm + clm + dio - cip & -dkn + cln + djo - cjp \\ -hkm + glm + hio - gip & -hkn + gln + hjo - gjp \end{bmatrix} \tag{12}$$

Comparing Eq. (11) and Eq. (12) demonstrates that $P_1$ and $P_2$ are equal to the $K_{11}$ and $K_{21}$ terms of $[K]_c$, respectively. By imposing the unit displacement at DOF 2, while restraining the displacement at DOF 1, the $K_{22}$ and $K_{12}$ terms can also be computed in a similar manner. Therefore, the stiffness matrix assembled from the restoring forces is equal to the condensed form of the stiffness matrix at the interface DOFs.

To minimize the communication time, the framework carries out the stiffness evaluation step only at the beginning of the simulation, estimating the condensed form of the initial stiffness matrix, $[K]_{i,c}$. This stiffness matrix is used throughout the multi-platform simulation. After determining $[K]_{i,c}$, the modified Newton Raphson procedure is employed to satisfy the equilibrium and compatibility requirements between the substructure modules. In each step of the simulation, the framework imposes displacements, determined from the previous load stage, at the interface DOFs on each module ($\{D_m\}$) and receives the restoring forces ($\{P_m\}_r$) obtained from nonlinear analysis or test specimen. Also in order to determine the total unbalanced force vector ($\{P_m\}_u$), the linear elastic force vector ($\{P_m\}_e$) is calculated from the force-displacement relationship at the interface DOFs of each substructure module as follows:

$$\{P_m\}_e = [K]_{i,c}\{D_m\} \tag{13}$$

Using $\{P_m\}_r$ and $\{P_m\}_e$ the framework determines the total unbalanced force vector ($\{P_m\}_u$) at the interface DOFs of each substructure module as follows:

$$\{P_m\}_u = \{P_m\}_e + (\{P_m\}_e - \{P_m\}_r) = 2\{P_m\}_e - \{P_m\}_r \tag{14}$$

Knowing $[K]_{i,c}$ and $\{P_m\}_u$, a similar procedure to that described for the Module Type 1 can be used to integrate the module into the simulation framework and find the interface displacements of next step. The multi-platform simulation procedure is repeated until the interface displacements converge within a predefined error limit.

Although the Module Type 2 integration method is computationally more expensive than the direct combination of the solution schemes (Module Type 1), it is a more generalized approach capable of integrating different programs regardless of their solution schemes. Fig. 2 shows the algorithm of the multi-platform simulation for the Module Type 1 and Module Type 2 substructure modules.

### 2.4. Communication methods

To exchange data among substructures, Cyrus employs two types of communication methods: local method and distributed method. With the local method, anonymous pipes are responsible for sending control commands (e.g., run and pause), whereas the binary files are used to transfer the restoring force, stiffness, and displacement values. The distributed method uses TCP/IP sockets with a recently developed standardized data exchange format, known as UTNP [34,35], to exchange data between multiple computers connected through the Internet network. Other potential analytical and experimental modules can be linked to the UTNP data exchange format, complied as a dynamic-link library (DLL), and use its standard functions to communicate with the simulation framework.

### 2.5. Interface programs

The simulation framework is compatible with two interface programs, NICA [21,35] and NICON [33]. Both programs are connected to the framework through TCP/IP sockets using the UTNP [34,35] data exchange format. NICA allows communication with analysis programs whose source code cannot be modified and only the input and output files containing model information and analysis results are accessible. The communication between NICA and analysis programs is based on named pipes or files. In each step of the simulation, NICA sends the displacements at the interface DOFs to substructure modules, reads the reaction forces, and sends them to the framework. NICON is a generalized controller interface program developed based on the LabView programming software and the National Instrument (NI) hardware. It enables the simulation framework to connect to various types of test configurations for hybrid testing. In each step of the simulation, NICON receives commands from the framework through the Internet network, converts coordinate systems between the FE model and the testing platform, generates analog voltage commands to actuator controllers, and returns measured responses to the framework. For additional information regarding the interface programs refer to [21,33–35].

## 3. Application examples

### 3.1. Wide flange shear wall

Palermo and Vecchio [36] tested a large-scale RC shear wall under lateral cyclic displacements and constant axial load (see Fig. 3). The lateral displacement was applied at the mid-depth of the top slab with two repetitions at each displacement cycle. The self-weight of the top slab contributed an additional 260 kN (58.4 kip) load to the externally imposed axial force of 940 kN (211.3 kip). The large overhanging flanges of the wall with an approximate width to height ratio of 0.75, 50% larger than what can be considered effective in most design code specifications, presents a significant challenge for two-dimensional analysis software. On the other hand, using a detailed three-dimensional analysis software with shear modelling capabilities can be highly time-consuming. Here, the proposed simulation framework is used to integrate two- and three-dimensional analysis tools for practical and effective modelling of this structure.

#### 3.1.1. Single-platform two-dimensional model

For the 2D model, the structure was divided into four regions (web, flanges, top slab, and bottom slab) and meshed with 8-DOF rectangular elements. Regions varied in terms of material properties and mesh size. The full width of the flanges was assumed to be effective in the two-dimensional model. All the nodes located along the bottom row of the bottom slab were fully restrained in both the X and Y translational directions. The external axial load, as well as the self-weight of the top slab, were modelled as vertical loads distributed over all nodes located at the mid-height of the top slab. The lateral load was imposed by controlling the lateral displacement of the node located at the mid-height of the top slab, in 1 mm (0.04 in.) increments, in a reversed cyclic manner. The structure was analyzed using the VecTor2 program, a two-dimensional nonlinear FE analysis program for RC structures capable of considering shear behaviour.

#### 3.1.2. Multi-platform models

Two forms of the multi-platform models were created: Type A and Type B. In both model types, the web which experiences predominantly

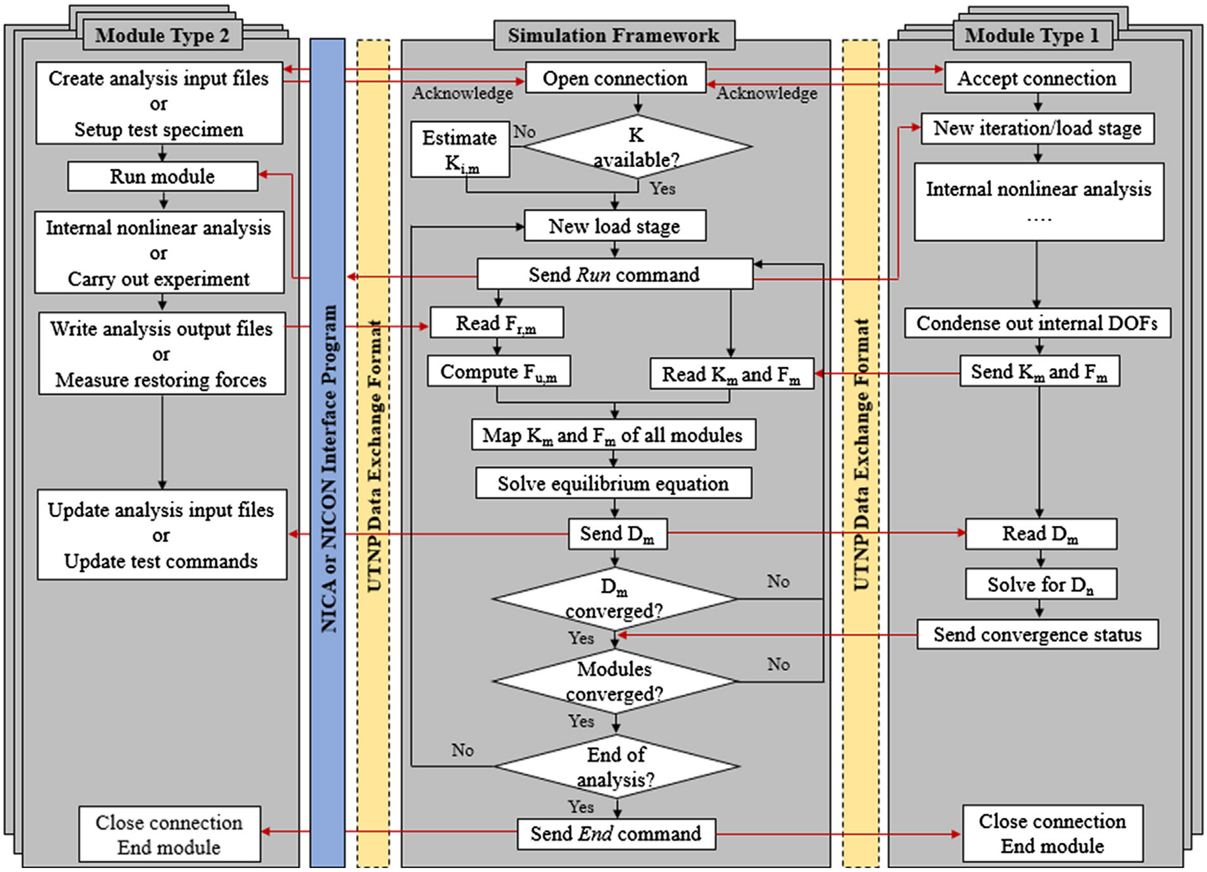


**Fig. 2.** Algorithm of the proposed multi-platform framework.

in-plane behaviour was modelled similar to the two-dimensional model and analyzed with VecTor2. The rest of the structure was modelled with three-dimensional programs enabling the capture of out-of-plane effects. The top and bottom slabs were analyzed with VecTor3. For the Type A model, the flanges were analyzed with VecTor3 using hexahedral elements while for the Type B model they were analyzed with VecTor4 using layered shell elements. Fig. 3 shows different components of the Type B model. Cyrus combined all the substructure modules and performed the multi-platform simulations. Rigid body movement was assumed at the interface between the substructure modules.

### *3.1.3. Comparison of analytical and experimental results*

The calculated push-over and reversed cyclic load-deflection responses for different analysis cases are compared to the experimental results in Fig. 4. The single-platform two-dimensional analysis provided acceptable results; however, it overestimated the post-cracking stiffness and strength. This was mainly attributed to the assumption made for the effective width of the flanges. Compared to the two-dimensional analysis, the multi-platform analysis results provided better agreement with the experimental data. With the Type B model, the multi-layer nature of the shell elements enabled a more accurate analysis of the out-of-plane behaviour of the flanges than that obtained from the Type A model. Compared to the experimentally observed response, all the analysis cases slightly underestimated the initial stiffness of the structure and resulted in a somewhat more dramatic softening effect in the post-peak region.

The computed failure mode in the multi-platform analysis consisted of a shear failure of the web concrete in horizontal planes near the base and crushing of the concrete at the toe (see Fig. 5). This caused high shear stresses on the flange elements near the base at the interface section, resulting in punching of the flanges which also contributed to the failure. The failure mode correlated well with the experimentally observed behaviour. The results illustrated that, unlike the two-dimensional analysis, the multi-platform analysis was able to take into

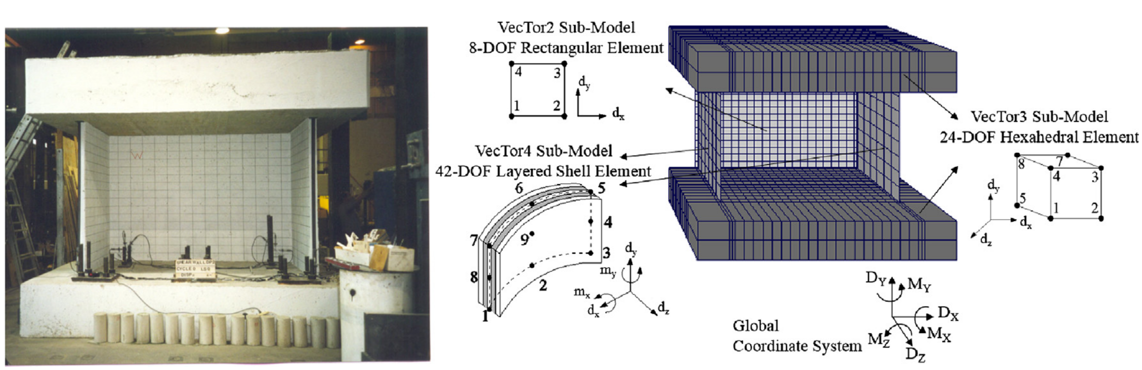


**Fig. 3.** Shear wall test specimen [36] (left) and Type B multi-platform FE model (right).

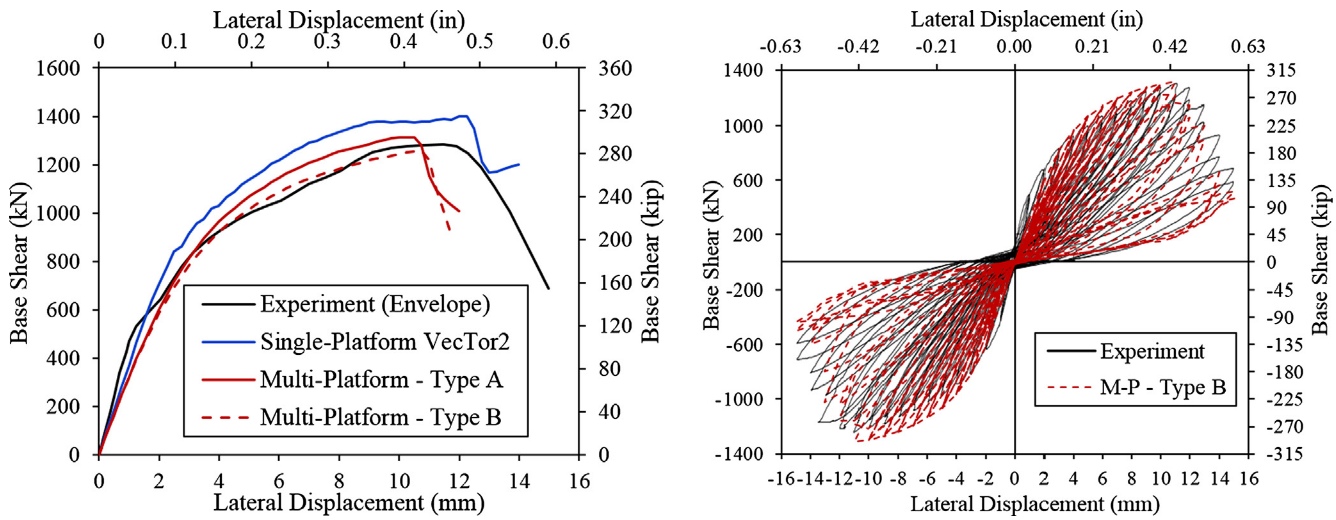


**Fig. 4.** Load-deflection responses of shear wall for push-over and reversed cyclic analyses.

account the three-dimensional effects and capture the observed damage to the flanges.

### 3.2. RC frame with critical joints

Calvi et al. [37] performed a quasi-static cyclic test on a three-storey gravity-based reinforced concrete frame building. The test specimen was constructed with characteristics similar to 1970s typical Italian design practice; namely, smooth bars were used for the reinforcement, the joint panels were constructed without shear reinforcement, and the longitudinal bars were anchored in the exterior joints with short 180 degrees end-hooks. The lateral displacement of the top floor was controlled in a reversed cyclic manner. Meanwhile, a lateral linear force distribution, proportional to the mass and height of each storey, was maintained through the height of the frame. In addition, a gravity load of 73 kN (16.4 kip) was applied at the first and second floor levels and 54.2 kN (12.2 kip) was imposed on the third floor. Based on the test report, the frame experienced a brittle shear failure with most of the damage occurring in the exterior beam-column joints of the first floor.

#### 3.2.1. Single-platform models

The entire frame was modelled using a total of 81 layered beam-column elements with OpenSees. These elements, like most frame-type analysis procedures, are based on the assumption of 'plane sections remain plane' and therefore cannot accurately capture nonlinear shear deformations and stress distributions in disturbed regions such as joint panels. The joint panels of the frame specimen were modelled in two different ways: Type (A) using the nonlinear layered beam-column elements, and Type (B) using the linear elastic beam-column elements. The latter is a more common assumption in modelling frame-type structures as it prevents premature failure at the joint regions. The gravity loads were modelled as nodal and element forces in the vertical direction representing the externally applied loads and self-weight of the structure, respectively. The lateral loads were modelled in a hybrid force-displacement manner; the displacement of the top left corner node was controlled in a reversed cyclic manner with increments of 0.1 mm (0.004 in.), while specific ratios were maintained between the nodal forces located at the left end of each floor. All the nodes located at the base of the columns were fully restrained in the translational and rotational directions.

#### 3.2.2. Multi-platform models

To improve the accuracy of the analysis, the beam-column joints can be modelled in more detail using a local finite element program, VecTor2, while the remainder of the frame is modelled with a global analysis program, OpenSees. It should be noted that modelling the entire frame with VecTor2 is not practical due to its expensive computational analysis procedure. According to the damage levels of the joint panels reported from the experiment, three types of multi-platform models were created. The models varied in terms of the number of joints analyzed in VecTor2. For each model type, the remaining joints were analyzed with both the nonlinear and linear beam-column elements available in OpenSees (Type A and Type B models, respectively). For the OpenSees substructure module, the modelling procedure was similar to that described for the single-platform model. For the VecTor2 substructure module, rectangular and truss elements were used to model the concrete and longitudinal reinforcement, respectively, while the transverse reinforcement was modelled as smeared. In addition, link elements were used between rectangular elements and truss elements to capture potential bond-slip effects. Cyrus combined the OpenSees and VecTor2 substructure modules and coordinated the multi-platform simulation. The interface between the two substructure modules was

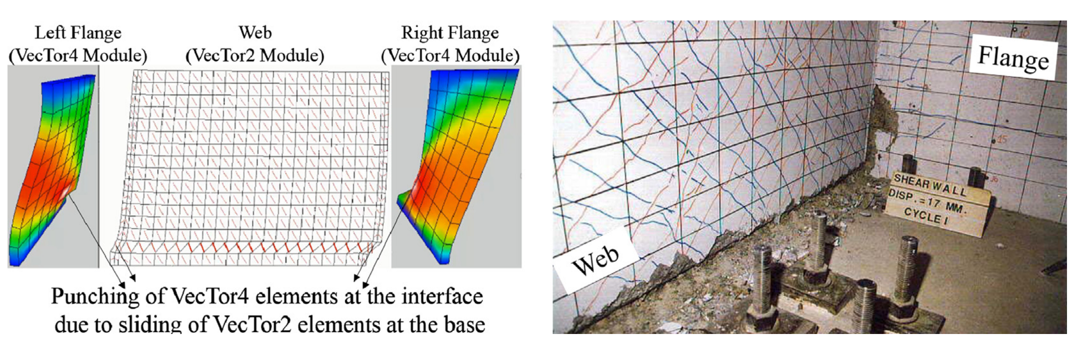


**Fig. 5.** Comparison of computed and observed damage modes of shear wall at ultimate load.

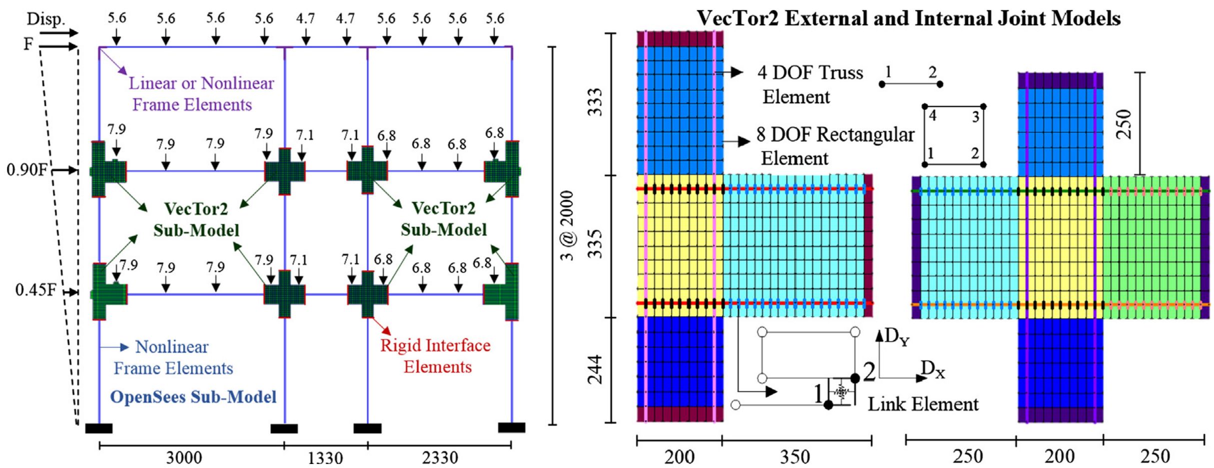


**Fig. 6.** Multi-platform model of RC frame with eight joints modelled with VecTor2 (dimensions in mm & forces in kN, 1 mm = 0.04 in., 1 kN = 0.225 kip).

modelled using a set of rigid beam-column elements. Fig. 6 shows one of the multi-platform models in which eight joints were modelled using the VecTor2 program.

### 3.2.3. Comparison of analytical and experimental results

The computed push-over and reversed cyclic load-deflection responses for different analysis cases are compared to the experimentally observed behaviour in Fig. 7 and Fig. 8, respectively. The single-platform OpenSees analysis responses significantly overestimated the post-cracking stiffness and the ultimate strength. The level of overestimation was higher when linear elastic beam-column elements were used for modelling the joint regions. Discrepancies between the analytical and experimental results were mainly the consequence of the limitations associated with most frame-type and sectional analysis software; namely, (1) inability to accurately consider the shear behaviour, (2) inability to capture highly nonlinear stress distributions at the disturbed regions, and (3) assumption of perfect bond between concrete and reinforcement, particularly for situations where the reinforcement detailing is insufficient. It should be noted that with this case study, the structural response was highly dependent on the behaviour of joint panels. For a typical frame structure, designed according to newer building codes, the level of discrepancies between the results is expected to be lower.

Based on the load-deflection responses presented in Figs. 7 and 8, the multi-platform analyses computed the post-cracking stiffness, ultimate strength, and energy dissipation of loading cycles with much better accuracy compared to the single-platform models. It can be seen that as the number of joints modelled with VecTor2 increased, the correlation between the analytical and experimental results improved. However, the amount of improvement from detailed modelling of the second storey joints was relatively lower demonstrating that non-critical joints can be adequately modelled using layered beam-column elements. There was a tendency to underestimate the energy dissipation which was mostly attributed to neglecting crack shear slip deformations in the concrete cyclic model.

In terms of the damage mode, the multi-platform analyses predicted multiple cracks in the joint zone as well as the formation of a vertical flexural crack at the beam-column interface which resulted in a large amount of slip in the longitudinal reinforcement of the beams. Thereafter, a diagonal shear crack was formed in the joint region which eventually led to the failure of the first storey external joints and a significant reduction in the stiffness of the structure. As shown in Fig. 9, the analytical and experimental crack patterns correlated reasonably well. None of the aforementioned damage mechanisms were captured in the single-platform frame analysis, illustrating the effectiveness of the multi-platform simulation.

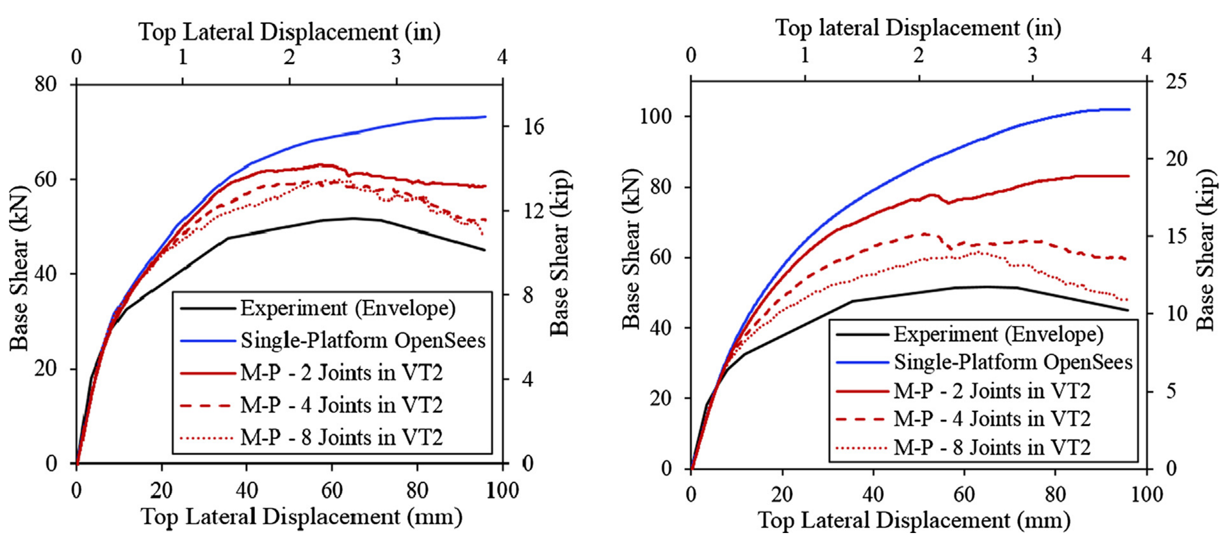


**Fig. 7.** Load-deflection responses of frame for push-over analyses of Type A models (left) and Type B models (right).

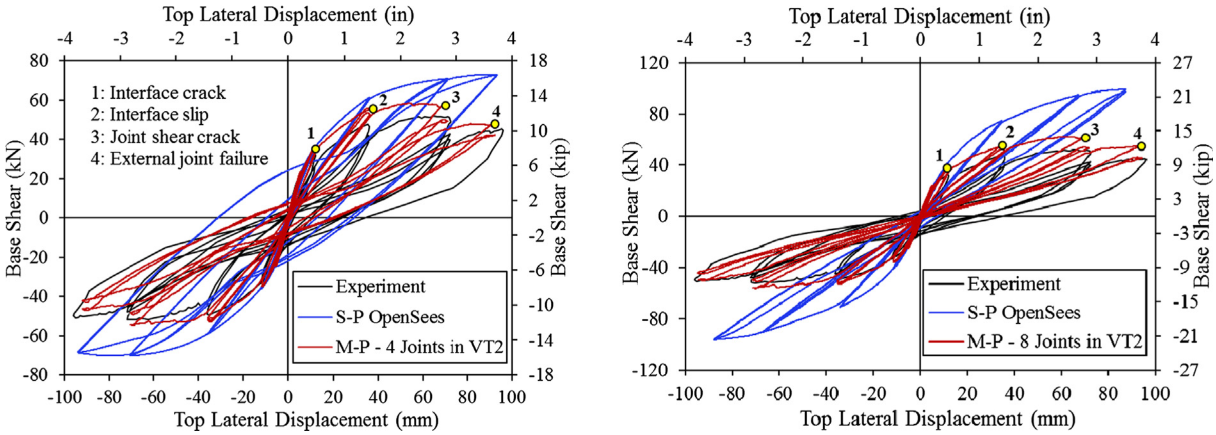


**Fig. 8.** Load-deflection responses of frame for reversed cyclic analyses of Type A models (left) and Type B models (right).

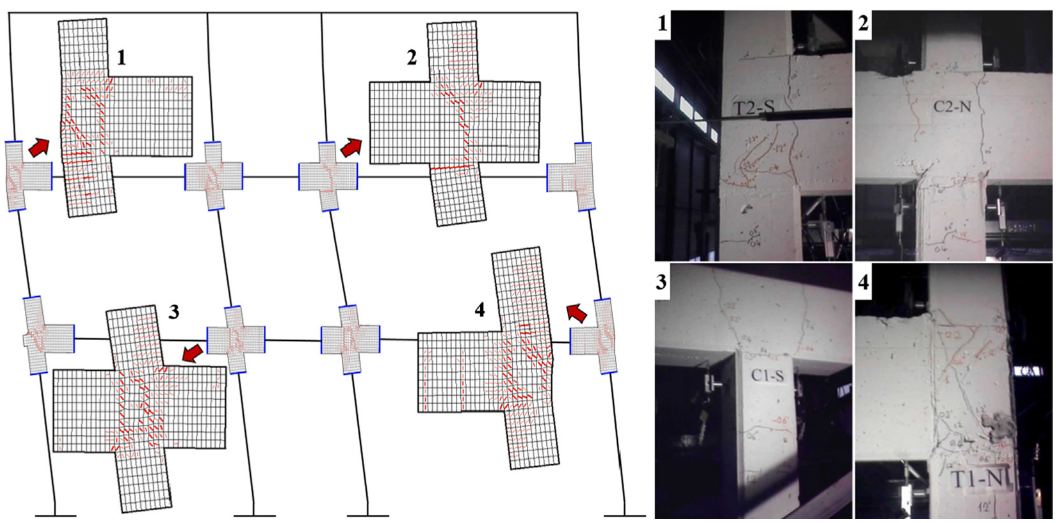


**Fig. 9.** Computed and experimentally reported crack patterns for RC frame.

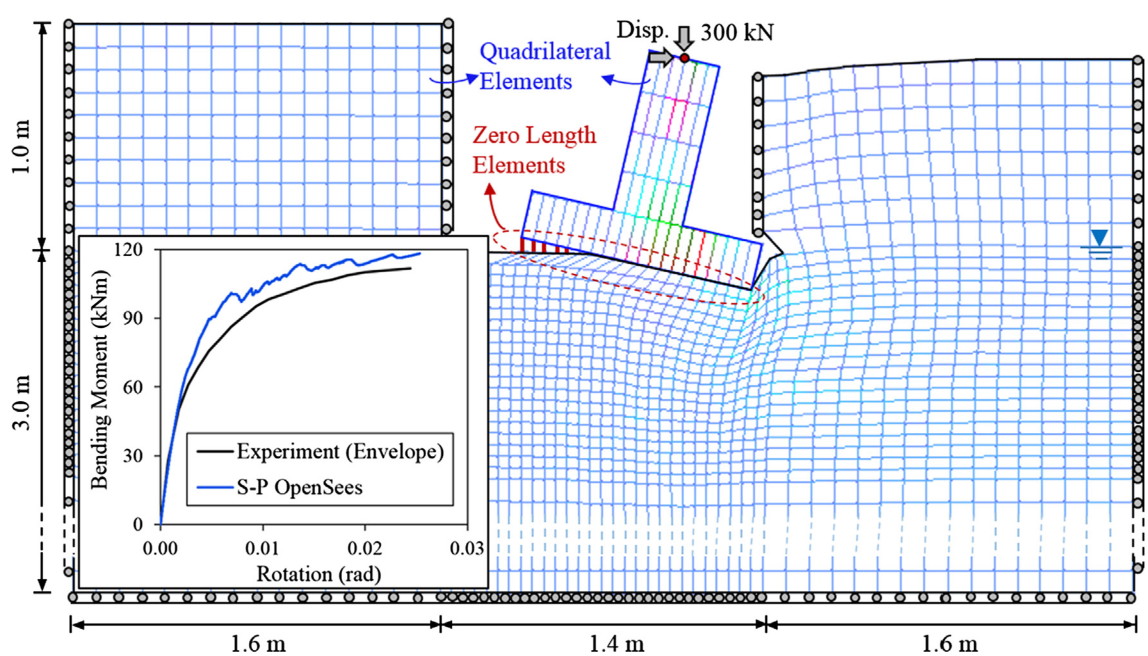


**Fig. 10.** OpenSees model and moment-rotation responses of soil-foundation system tested by Negro et al. [38] (dimensions in m & forces in kN, 1 m = 39.4 in., 1 kN = 0.225 kip).

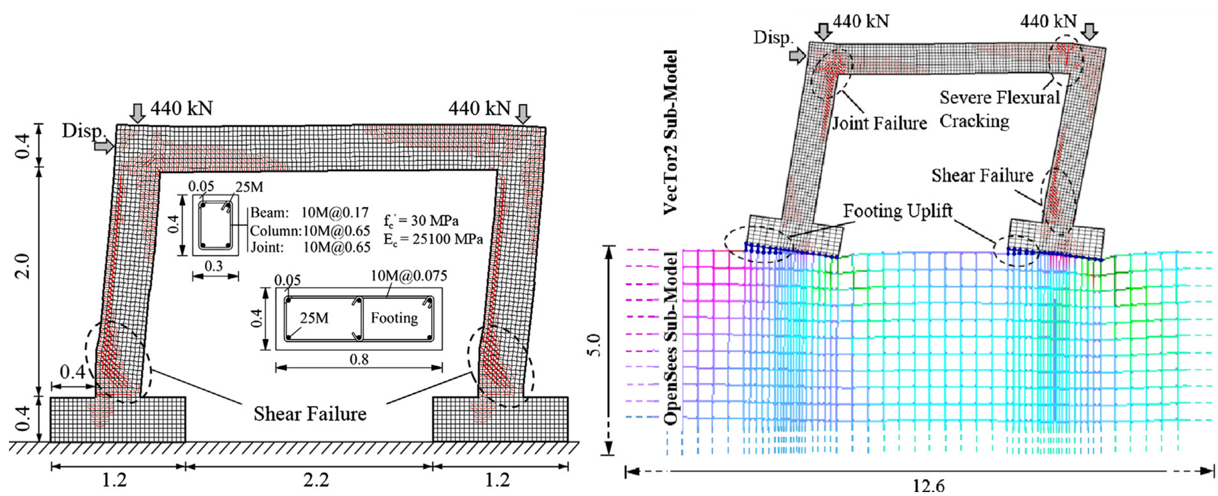


**Fig. 11.** Damage modes for single-platform VecTor2 analysis (left) and multi-platform analysis (right) (dimensions in m & forces in kN, 1 m = 39.4 in., 1 kN = 0.225 kip).

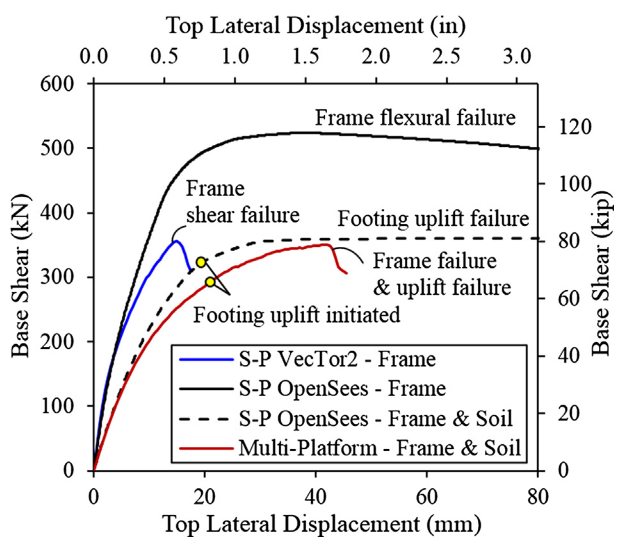


**Fig. 12.** Load-deflection responses of soil-structure system for different modelling approaches.

### 3.3. Soil-structure interaction

#### 3.3.1. Soil-foundation modelling verification

Negro et al. [38] experimentally investigated the behaviour of a large-scale soil-foundation system subjected to a constant vertical load of 300 kN (67.4 kip) and a reversed cyclic lateral loading condition. The foundation was a 1.0 m (39.4 in.) square steel shallow footing constructed on a layer of concrete mortar providing a high lateral friction resistance with the underlying soil. The soil sample was constructed from a high density Ticino sand in a rigid caisson with dimensions of 4.6 m × 4.6 m (181.1 in. × 181.1 in.) in-plane and 4.0 m (157.5 in.) deep. The foundation was embedded 1.0 m (39.4 in.) into the soil sample resulting in a 20 kPa (2.9 psi) overburden.

As shown in Fig. 10, a two-dimensional finite element model of the soil-foundation system was created using OpenSees. The soil was modelled with 2688 four-noded quadrilateral elements assuming a plane strain behaviour. The foundation was modelled using linear elastic quadrilateral elements with a plane stress behaviour. To capture potential uplift of the foundation, the interface of the soil and foundation was modelled using a set of zero length elements. These elements were given a near-zero stiffness value in the vertical direction allowing for the separation of the foundation from the soil. Also, assuming the sliding of the foundation was negligible, a very high stiffness in the tangential direction was assigned to the zero length elements. All the boundary nodes located at the walls and base of the caisson were restrained in the X and Y directions, respectively. The nonlinear behaviour of the sand was modelled using the *PressureDependMultiYield* material. The friction angle at the peak shear strength was assumed to be 32 based on the recommendations provided in the software's manual. The remaining input parameters for the soil model were taken directly from the experimental report.

The bending moment-rotation response obtained from the push-over analysis is compared against the envelopes of the experimentally reported cyclic response in Fig. 10. It can be seen that there was good correlation between the analytical and experimental responses. After the soil became nonlinear, the analysis slightly overestimated the strength and stiffness. This may be due to neglecting the reduction in the shear modulus of the soil after the initial settlement occurred in the test. The analysis computed a considerable amount of footing uplift which was consistent with the experimentally observed behaviour.

#### 3.3.2. Application to soil-structure simulation

To investigate the influence of the foundation rotation on the

**Table 1**
Comparison between analysis times of application examples.

| Structure type | Load type | Disp. inc. (mm) | No. of load stages | Analysis type | Analysis time (min) |
|---|---|---|---|---|---|
| Wide flange shear wall | Reversed cyclic | 1.0 | 960 | S-P (VT2) | 121 |
| | | | | S-P (VT3) | 512 |
| | | | | M-P (VT2, 3, and 4) | 189 |
| Three-storey frame | Reversed cyclic | 0.1 | 17,120 | S-P (OS) | 13 |
| | | | | M-P (OS and VT2) | 80 |
| Soil-structure interaction | Monotonic | 0.1 | 800 | S-P (OS) | 16 |
| | | | | M-P (OS and VT2) | 24 |

behaviour of the structure, a soil-structure system was analyzed using four different modelling approaches. The soil-structure system was a one-storey one-bay RC frame constructed on a sand soil with similar properties to the one tested by Negro et al. [38]. Details of the system are shown in Fig. 11. For the first two modelling approaches, the influence of the soil was neglected and the structure was analyzed with the OpenSees and VecTor2 programs. With the OpenSees model, a frame-type analysis was performed using the layered beam-column elements. With the VecTor2 model, the frame was analyzed in more detail using a combination of rectangular and truss elements. The modelling procedures were similar to those presented in the RC frame with critical joints application example. For the next two modelling approaches, the influence of the soil was taken into account using a single-platform OpenSees model and a multi-platform OpenSees-VecTor2 model. For both analysis cases, the soil model was identical to that described in the Soil-foundation modelling verification section. To capture uplift of the foundation, zero length elements were used at the interface between the soil and the structure. With the single-platform OpenSees model, a set of multi-point constraints were used to transfer the rotation of the layered beam-column elements to the equivalent translational displacements of the zero length elements. With the multi-platform model, the analysis was performed using Cyrus where the VecTor2 frame model and the OpenSees soil model were fully coupled in the X and Y translational directions.

The damage modes and load-deflection responses for different analysis cases are compared in Fig. 11 and Fig. 12, respectively. The OpenSees frame analysis predicted the formation of plastic hinges at the bottom of the columns which led to a ductile flexural failure of the frame. The VecTor2 frame analysis computed a brittle type of failure where the damage mode consisted of large diagonal shear cracks near the base of the columns and vertical cracks along the longitudinal reinforcement layers. Taking into account the soil effects, the single-platform OpenSees analysis predicated an extensive uplift in the footings as the predominant failure mode which occurred well before the structure could reach its load capacity. With the multi-platform analysis a combination of structural and geotechnical damage modes contributed to the final failure; namely, the shear cracks at the left joint panel and at the lower portion of the right column, the flexural crack at the interface of the beam and the right column, and the uplift of the footings. It can be seen that compared to the single-platform analyses, the multi-platform analysis in which both the soil and structure were modelled in a comprehensive manner resulted in a more realistic representation of the damage modes and the load-deflection response of the system.

The computational performance of the single- and multi-platform models are compared in Table 1. All the analyses were performed using a desktop computer with a Core i7 processor. It can be seen that for the wide flange shear wall, the multi-platform analysis is about 2.7 times faster than the three-dimensional single-platform analysis. Two factors that improved the computational performance of the analysis are: (1) modelling the web using a two-dimensional analysis program, and (2) dividing the structure into multiple substructures and analyzing them in a concurrent manner. With the other two application examples, the multi-platform analysis was able to compute the behaviour of the structures in a still reasonable amount of time but with far greater fineness and accuracy. The substructuring technique allows multi-platform analysis to use computing resources more effectively by distributing tasks between different cores of the central processing unit. Analysis of such complex systems with the same level of modelling detail and accuracy may not be feasible using a single-platform software.

## 4. Conclusions

A novel multi-platform simulation framework was introduced for performance assessment of complex reinforced concrete systems. The application of the framework to analysis of shear-critical RC systems was examined in detail. The conclusions of this study can be summarized as follows:

1. The proposed multi-platform modelling approach eliminates assumptions and analysis calibrations commonly used for modelling large complex RC systems. With this approach, instead of using calibrated springs, in which the response often varies from one member to another, local finite element programs are used to model members with complex behaviour in a comprehensive manner.
2. The multi-platform modelling method was able to accurately capture both the component- and system-level behaviour of the test specimens. The method computed damage modes and mechanisms that were difficult to capture with single-platform analysis tools, resulting in more realistic predictions of load-deflection responses and failure modes.
3. Proper selection of the substructure modules requires having a good understanding of the structural behaviour including the mechanical characteristics of the structural elements and an anticipation of the locations of critical regions prior to the analysis.
4. The behaviour of wide-flange shear walls can be greatly influenced by three-dimensional effects resulted from the interaction between the web and the flanges. Two-dimensional analysis methods can lead to unsafe or over-conservative results depending on the assumption made for the effective width of the flanges.
5. Using layered beam-column elements to model frame structures with insufficient joint detailing can lead to significant overestimations of the strength and ductility. The level of overestimation can be more pronounced if rigid elements are used at the joint panels in order to avoid potential premature failure. The analyses also showed that non-critical joints, which are usually located at the upper storeys of the building, can be adequately modelled with nonlinear layered beam-column elements.
6. Ignoring the rotation of shallow foundations of frames may lead to significant underestimations of the inter-storey displacements, and can influence the failure mode and locations of the critical regions. Proper analysis of such systems requires comprehensive modelling of both the structure and the soil domain.
7. Based on the results of the application examples, it can be concluded that with a proper load increment the multi-platform modelling method does not affect the convergence of the nonlinear analysis.

## Acknowledgement

Financial support provided by IC-IMPACTS (the India-Canada Center for Innovative Multidisciplinary Partnerships to Accelerate Community Transformation and Sustainability) is gratefully acknowledged.

## References

[1] Holand I, Bell K. Finite element method in stress analysis. Norway: Tapir Academic Press; 1969. 500 pp.
[2] Mote CD. Global-local finite element. Int J Numer Meth Eng 1971;3(4):565–74.
[3] Noor AK. Global-local methodologies and their application to nonlinear analysis. Finite Elem Anal Des 1986;2(4):333–46.
[4] Hirai I. An exact zooming method. Finite Elem Anal Des 1985;1(1):61–9.
[5] Whitcomb JD. Iterative global/local finite element analysis. Comput Struct 1991;40(4):1027–31.
[6] Mao KM, Sun CT. A refined global-local finite element analysis method. Int J Numer Meth Eng 1991;32(1):29–43.
[7] Cho IH, Hall JF. A parallelized implicit nonlinear FEA program for real scale RC structures under cyclic loading. J Comput Civil Eng 2012;26(3):356–65.
[8] Balay S. Portable, extensible toolkit for scientific computation (PETSc) user's manual. IL, U.S.A.: Argonne National Laboratory, University of Chicago; 2014.
[9] Chen HM, Archer GC. A distributed object-oriented finite-element analysis program architecture. Comput-Aid Civ Infrastruct Eng 2001;16(5):326–36.
[10] Yang YS, Hsieh SH, Hsieh T-J. Improving parallel substructuring efficiency by using a multilevel approach. ASCE J Comput Civ Eng 2012;26(4):457–64.
[11] Li Z, Chen Y, Shi Y. Numerical failure analysis of a continuous reinforced concrete

bridge under strong earthquakes using multi-scale models. Earthq Eng Eng Vibrat 2017;16(2):397–413.
[12] Hallquist JO. LS-DYNA keyword user's manual. California, U.S.A.: Livermore Software Technology Corporation; 2007. 2206 pp.
[13] Yue J-G, Fafitis A, Qian J, Lei T. Application of 1D/3D finite elements coupling for structural nonlinear analysis. J Central South Univ Technol 2011;18(3):889–97.
[14] ABAQUS. ABAQUS/CAE users' manual. Providence, R.I., U.S.A.: Dassault Systèmes Simulia Corp.; 2012. 1137 pp.
[15] Sadeghian V, Kwon O-S, Vecchio FJ. Modelling beam-membrane interface in reinforced concrete frames. ACI Struct J 2018;115(3):826–36.
[16] Yang YS, Hsieh SH, Wang KJ, Wang SJ. Numerical analysis framework for distributed pseudo-dynamic tests. 2nd International conference on structural stability and dynamics, Singapore. 2002.
[17] Takahashi Y, Fenves GL. Software framework for distributed experimental-computational simulation of structural systems. OpenSees developer symposium. Berkeley, CA, U.S.A.: University of California; 2005.
[18] Pan P, Tada M, Nakashima M. Online hybrid test by internet linkage of distributed test-analysis domains. Earthquake Eng Struct Dyn 2005;34(11):1407–25.
[19] Mata P, Barbat AH, Oller S. Two-scale approach for the nonlinear dynamic analysis of RC structures with local non-prismatic parts. Eng Struct 2008;30(12):3667–80.
[20] Chen HM, Lin YC. An internet-based computing framework for the simulation of multi-scale response of structural systems. Struct Eng Mech 2011;37(1):17–37.
[21] Kwon O-S, Elnashai AS, Spencer BF. A framework for distributed analytical and hybrid simulations. Struct Eng Mech 2008;30(3):331–50.
[22] Sadeghian V, Kwon O-S, Vecchio FJ. Small-scale multi-axial hybrid simulation of a shear-critical reinforced concrete frame. Earthq Eng Eng Vibrat J 2017;16(4):727–43.
[23] Sadeghian V, Vecchio FJ. Application of multi-scale modelling on large shear-critical reinforced concrete structural systems repaired with FRP sheets. Innov Corros Mater Sci 2016;6(2):106–14.
[24] Elnashai AS, Papanikolaou V, Lee D. Zeus NL – a system for inelastic analysis of structures user's manual. IL, U.S.A.: Mid-America Earthquake Center, University of Illinois at Urbana-Champaign; 2008. 149 pp.
[25] Mazzoni S, McKenna F, Scott MH, Fenves GL. The OpenSees command language manual, version 2.0. Berkeley, C.A., U.S.A.: Pacific Earthquake Engineering Research Center, University of California; 2007. 465 pp.
[26] Wong PS, Trommels H, Vecchio FJ. VecTor2 and FormWorks user's manual. 2nd ed. Toronto, Canada: Department of Civil Engineering, University of Toronto; 2013. 311 pp.
[27] ElMohandes F, Vecchio FJ. VecTor3 user's manual. Toronto, Canada: Department of Civil Engineering, University of Toronto; 2013. 153 pp.
[28] Hrynyk TD, Vecchio FJ. Capturing out-of-plane shear failures in the analysis of reinforced concrete shells. ASCE J Struct Eng 2015;141(12).
[29] Guner S, Vecchio FJ. Pushover analysis of shear-critical frames: formulations. ACI Struct J 2010;107(1):63–71.
[30] Lulec A. Simplified analytical tools for impact and impulsive loading analysis of reinforced concrete structures. Ph.D. Dissertation Toronto, Canada: Department of Civil Engineering, University of Toronto; 2017. 196 pp.
[31] Vecchio FJ, Collins MP. The modified compression field theory for reinforced concrete elements subjected to shear. ACI Struct J 1986;83(2):219–31.
[32] Sadeghian V, Vecchio FJ. The modified compression field theory: then and now. ACI Struct J, Special Publication (SP-328: Shear in Structural Concrete) 2018:3.1–3.20.
[33] Zhan H, Kwon O-S. Actuator controller interface program for pseudo-dynamic hybrid simulation. World congress on advances in structural engineering and mechanics, Incheon, Korea. 2015.
[34] Huang X, Sadeghian V, Kwon O-S. Development of integrated framework for distributed multi-platform simulation. 6th International conference on advances in experimental structural engineering, University of Illinois at Urbana-Champaign, I.L., U.S.A.. 2015.
[35] Huang X, Kwon O-S. A generalized numerical/experimental distributed simulation framework. J Earthquake Eng 2018. https://doi.org/10.1080/13632469.2018.1423585.
[36] Palermo D, Vecchio FJ. Behavior of 3-D reinforced concrete shear walls. ACI Struct J 2002;99(1):81–9.
[37] Calvi GM, Magenes G, Pampanin S. Experimental test on a three storey RC frame designed for gravity only. 12th European conference on earthquake engineering, London, U.K.. 2002.
[38] Negro P, Paolucci R, Pedretti S, Faccioli E. Large-scale soil-structure interaction experiments on sand under cyclic loading. 12th World conference on earthquake engineering, Auckland New Zealand. 2000.