COMPDYN 2015
5th ECCOMAS Thematic Conference on
Computational Methods in Structural Dynamics and Earthquake Engineering
M. Papadrakakis, V. Papadopoulos, V. Plevris (eds.)
Crete Island, Greece, 25–27 May 2015

AN INTEGRATED FRAMEWORK FOR THE ANALYSIS OF MIXED-TYPE REINFORCED CONCRETE STRUCTURES

Vahid Sadeghian[1], Oh-Sung Kwon[1] and Frank Vecchio[1]

[1]University of Toronto Department of Civil Engineering
35 St. George St., M5S 1A4, Toronto, ON, Canada
vahid.sadeghian@utoronto.ca, os.kwon@utoronto.ca, fjv@ecf.utoronto.ca

Keywords: Reinforced concrete, Nonlinear analysis, Integrated simulation, Substructuring technique

Abstract. *The substructuring concept has been used in different forms such as in hybrid simulation and parallel computing methods, all contributing significantly to experimental testing and numerical analysis research programs. In this paper, unlike previous studies, the main goal is to use the substructuring technique to develop a framework which expands the capabilities of current nonlinear programs, enabling detailed analysis of mixed-type reinforced concrete (RC) systems in an integrated fashion, to a degree of accuracy unattained before. To achieve this goal, the primary objective of the proposed framework is to combine different VecTor programs which are among the most advanced nonlinear analysis tools for RC structures, while fully considering the interaction between substructures. This can greatly extend the application of conventional stand-alone nonlinear analysis. For instance, integrating VecTor2 (membrane software) and VecTor5 (frame software) provides a unique and effective solution technique for detailed analysis of disturbed regions in RC frames which is not available in any other stand-alone frame type analysis program. The flexible and object-oriented architecture of the framework facilitates inclusion of new analysis software. The application and effectiveness of the proposed framework are illustrated by modelling and analysis of two experimental case studies.*

1. INTRODUCTION

Nonlinear finite element analysis (NLFEA) of reinforced concrete structures has seen tremendous advancement over the past few decades. There are some situations where using advanced NLFEA tools is essential due to complexity of the analysis or the required level of accuracy. For instance, if the structure is subjected to unintended or extreme loads, if the codes or standards used for its design are deemed to be deficient today, if the structure was incorrectly designed or constructed in the first place, or if the structure is damaged and requires rehabilitation, then a more comprehensive analysis may be warranted. Especially considering the ageing infrastructure, the importance of having an advanced analysis tool is apparent.

Over the past decades, the VecTor programs, a suite of NLFEA software, have been developed on the theoretical basis of the Disturbed Stress Field Model (DSFM) [1]. The computational approach of the programs is based on a secant stiffness formulation using a total-load iterative process. Several experimental programs with a wide range of specimens and loading conditions have been conducted to examine the accuracy of the software [2, 3]. Moreover, analyzing real-world structures including frames, slabs, shear walls, silos, bridges, offshore platforms, and nuclear containment structures have been demonstrated the value of the VecTor programs in evaluating the complex nonlinear behaviour of reinforced concrete structures [4, 5]. Analysis application and element types for each VecTor program are presented in Table 1.

Program	Structure Type	Elements Type	Available DOFs
VecTor2	2D Membrane	Rectangular; Quadrilateral; Triangular; Truss; Link; Contact	Dx and Dy
VecTor3	3D Solid	Hexahedral; Wedge; Truss; Link; Contact	Dx, Dy and Dz
VecTor4	Plates and Shells	Heterosis; Truss	Dx, Dy, Dz, R1 and R2
VecTor5	Plane Frames	Frame	Dx and Dy
VecTor6	Axisymmetric Solids	Rectangular Torus; Triangular Torus; Ring	Dx and Dy

Note: Dx, Dy and Dz are translational DOFs in the X, Y and Z directions, respectively and R1 and R2 are rotational DOFs in the local directions.

Table 1: The analysis application and element types for each VecTor program

In many finite element analysis software, it has become necessary to combine lower-dimensional elements with higher-dimensional elements to provide both global and local assessment of the behaviour of the complex structural systems. In the VecTor analysis programs, each program is specialized for one particular type of problem; for example, VecTor2 can only model 2D planar continuums and VecTor5 can only model 2D plane frames. There are many applications where several types of structural components act together; for example, in a typical building, a moment resisting frame (2D frame) may act compositely with floor slabs (3D plate) and shear walls (2D continuum). Analysis capabilities for such systems or any other mixed-type structures are not available in the VecTor programs. In addition, modelling the entire structural system in detail is not practical due to computational time and memory storage limitations.

The most common approach to analyze integrated structural systems and overcome the above-mentioned limitations is the global-local method. In this approach, the structure is analyzed in two separate steps. In the first step, a global analysis of the entire structure is performed to find the force distribution and reactions. In the second step, using the global analysis results, the critical components of the structure are analyzed in detail. However, determining the critical regions that needs to be modelled requires expert users with a very good understanding of the force flow in the structure. Also, defining the right boundary conditions for each component is a challenging task. Even if the critical components are chosen and modelled properly, force redistribution due to stiffness changes in the system raises questions about the accuracy of the method.

One effective approach to accurately simulate the response of integrated systems is to develop a multi-scale framework which can combine the best features of different analysis software. UI-SimCor [6] is an example of such framework, which is targeted toward hybrid (experiment-analysis) simulation. In the multi-scale framework approach, each component, based on its mechanical characteristics, can be modelled using the most suitable analysis tool. The components are connected to each other through an integrated framework which coordinates the analysis of the whole system. The coupled nature of the simulation allows one to fully consider the interaction between the substructures, resulting in a more accurate analysis of the system. The framework provides a multi-platform analysis environment which includes a broad range of analysis methods, element types, material models, and load options. Moreover, it enables the use of the parallel processing technique to avoid computational time and memory storage limitations associated with sequential single-platform analyses. Developing such framework for the VecTor programs can greatly extend the application of the software to assess the behaviour of complex integrated reinforced concrete structures to a degree that was difficult or impossible before.

In this paper, the proposed multi-scale simulation framework is presented. The effectiveness of the proposed framework are illustrated with two experimental case studies.

2. PROPOSED INTEGRATED FRAMEWORK

The proposed integrated framework, named as Cyrus, intends to incorporate single-platform NLFEA tools and provide a unique simulation environment which helps to eliminate the limitations associated with analysis of mixed-type reinforced concrete structures. The architecture of the framework is described in detail in subsequent sections.

2.1 Overview of the Framework

Cyrus is written in the C++ programming language using Microsoft Foundation Classes and Open Graphics Library. The architecture of the framework is based on object-oriented methodology, making it suitable to adapt to other analysis tools. Cyrus consists of three main parts: solution algorithms, inter-process communication (IPC) methods, and graphical user interfaces (GUI). Detailed descriptions of each part are provided in the following sections. The framework is linked with advanced C/C++ libraries such as MKL [7] and PARDISO [8] enabling high-performance, memory efficient sparse matrix calculation and fast multi-threaded partial factorization computation which is used in the static condensation phase of the substructuring technique.

By running the simulation, Cyrus connects to all the components. The first stage of the analysis is the initial static equilibrium. In this stage, depending on the solution algorithm, Cyrus collects full or condensed forms of the stiffness and force values of all the substructures, maps them based on an internal numbering format, and solves for the displacements. This satisfies equilibrium for all DOFs of the mixed-type system including the boundaries between the substructures. Then, the framework sends computed displacements to each substructure along with a proceed command to continue their analyses. The boundary displacements between connecting substructures are taken to be identical so that compatibility is satisfied in the system.

For dynamic analysis, after imposing the static loads in the first load stage, Cyrus collects mass and stiffness values of all the components, performs an eigenvalue analysis, and sends the computed Rayleigh damping coefficients to the substructures. In this procedure, the equation of motion is formulated for each substructure separately using numerical integration methods available in the VecTor programs. Based on a total-load secant stiffness formulation of the integration methods developed by Saatci [9], equivalent stiffness matrix and force vector of dynamic effects are calculated in each component and sent to Cyrus. These values will be added to the corresponding static values and will be used to compute displacements.

In each load stage, when a substructure reaches convergence, Cyrus temporarily holds the analysis of that specific substructure and uses its converged stiffness and force values for the analysis of the rest of the system. At the end of the analysis, all the substructures are disconnected from the framework. Figure 1 demonstrates the overall architecture of the proposed framework and its application on a shear wall-frame system. The dotted line steps are only preformed in the dynamic analysis.

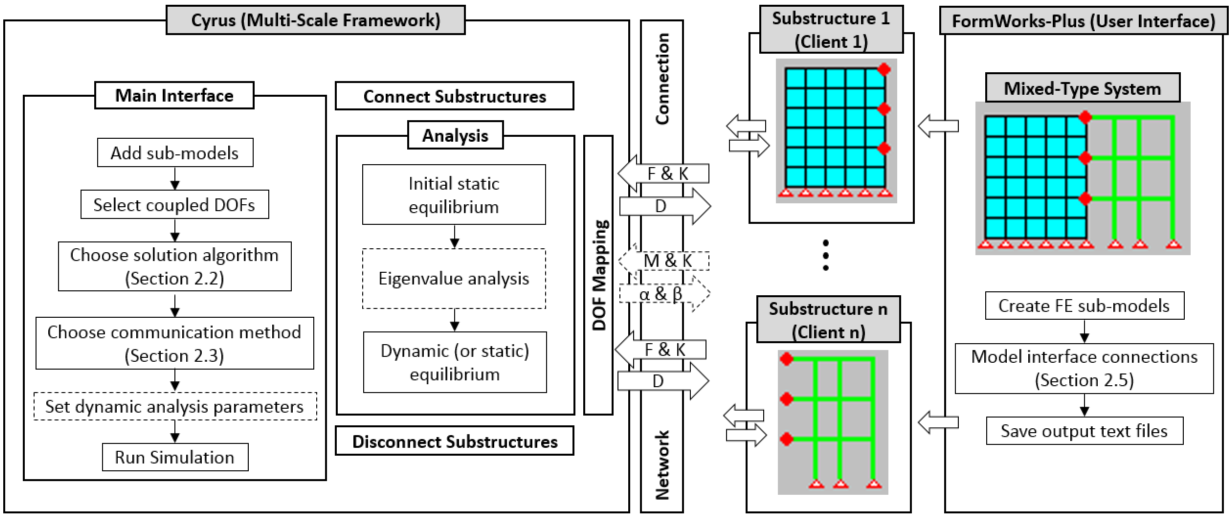


Figure 1. Overall architecture of the proposed multi-scale framework

2.2 Solution Methods

To consider geometry and material nonlinearity of structures, analysis software use different types of iterative solution methods. The majority of these methods can be categorized into two groups: Tangent-Based methods and Secant-Based methods. In the Tangent-Based methods, the displacements are computed from tangent stiffness and unbalanced force values. The unbalanced forces are calculated by subtracting the applied external loads from sectional forces. Since finding the stiffness matrix in each iteration is computationally expensive, some methods use initial stiffness instead of tangent stiffness in their procedure. In the Secant-Based methods, unlike Tangent-Based methods, the force vector is constant and equal to the applied forces through the iterations while the nonlinear behaviour of the structure is taken into account by updating the secant stiffness values. This section intends to demonstrate that, by using reasonably small load steps, the Tangent-Based methods and Secant-Based methods are equivalent; therefore, analysis tools with different solution techniques can be combined.

In the Incremental Tangent method, as shown in Figure 2(a), the next step displacement (u_{n+1}) is estimated from the current displacement (u_n) based on:

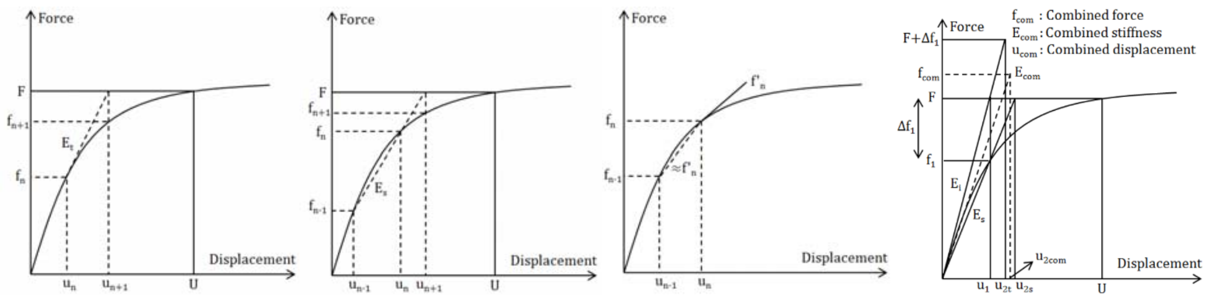
$$u_{n+1} = u_n + \frac{F - f_n}{E_{tn}} \tag{1}$$

where F is the external applied force; f_n and E_{tn} are the internal force and tangent stiffness corresponding to the u_n displacement, respectively. In the Incremental Secant method, however, the next step displacement is computed from the intersection of applied force and incremental secant line (E_s) (Figure 2(b)). The equation of the incremental secant line can be described as:

$$E_s(u) = \frac{f_n - f_{n-1}}{u_n - u_{n-1}} u + \frac{u_n f_{n-1} - u_{n-1} f_n}{u_n - u_{n-1}} \tag{2}$$

From $E_s(u_{n+1}) = F$, the next step displacement is computed as:

$$u_{n+1} = \frac{F(u_n - u_{n-1}) + u_{n-1} f_n - u_n f_{n-1}}{f_n - f_{n-1}} \tag{3}$$



(a) (b) (c) (d)

Figure 2. (a) Tangent method; (b) Secant method; (c) Tangent stiffness approximation; (d) Combined method

As illustrated in Figure 2(c), for small load steps one can approximate the tangent stiffness at u_n with $\frac{f_n - f_{n-1}}{u_n - u_{n-1}}$. Substituting this equation into the Tangent method equation gives the following:

$$u_{n+1} \approx \frac{(F - f_n)(u_n - u_{n-1})}{f_n - f_{n-1}} + u_n \tag{4}$$

Expanding this equation results in a similar equation as the one described in the Secant method.

This also can be illustrated graphically. As shown in Figure 2(d), which presents the total form of the Tangent and Secant solutions, combing the two methods results in stiffness values which are not as high as the initial stiffness and not as low as the secant stiffness of the system. Similarly, the combined force values are within the range of the external force and total unbalanced force values. Therefore, the computed displacements from the combined method are always between the values obtained from the Tangent method and Secant method. Assuming small load steps, it can be concluded that the solution of the combined method is equivalent to the solution of Tangent and Secant methods.

All the VecTor programs are formulated based on the Secant solution method except VecTor5 (frame analysis) which uses the Tangent solution method (Newton-Raphson method). The combined solution method analysis results are examined using VecTor2 and VecTor5 programs in the application example section of this paper.

In order to solve the equilibrium equation and find displacements at coupled DOFs, two different solution techniques were implemented in Cyrus: Centralized solution and Distributed solution. While both methods give exact the same results, depending on the number of substructures, amount of communication data, and number of available nodes, their performance can be significantly different.

In the Centralized solution method, substructures compute the stiffness and force values and send them to Cyrus in sparse format (non-zero values). Cyrus generates the global stiffness matrix and force vector and solves for the displacements of all DOFs of the system. This method is used in most previous simulation frameworks. However, for large systems, transferring the stiffness and force values of all the DOFs requires an excessive amount of communication which can significantly affect the analysis time. In addition, storing all the information of the entire system on as single computing node is not memory efficient.

To avoid the aforementioned problems, a new solution technique, Distributed solution method, is implemented in Cyrus, which is not available in any other similar framework. In this method, unlike the Centralized method, only the condensed form of the stiffness matrix and force vector are sent to Cyrus. The simulation framework is responsible for solving equilibrium equations for the coupled DOFs while each substructure solves for the displacements of the internal DOFs. Therefore, the solution step of the analysis is distributed between the framework and components.

2.3 Communication Procedures

To exchange data among substructures, Cyrus is enhanced with several Inter-Process Communication (IPC) methods. These techniques enable transferring data between two or more threads of one or more processors. Processors can be located on multiple computers connected by a network. Figure 3 illustrates the overall architecture of the different communication methods implemented in the simulation framework.

The first communication method available in Cyrus is pipe communication which can be used to redirect input and output of substructures and send them control commands (run and pause). To exchange data in both directions, Cyrus creates two anonymous pipes per substructure. It writes data to one pipe using its write handle, while the substructure reads the data from that pipe using its read handle. Similarly, the substructure writes data to the other pipe and Cyrus reads from it. Pipes have limited size which depends on the system properties. If the size of data is larger than the pipe size, the program transfers data in multiple byte blocks.

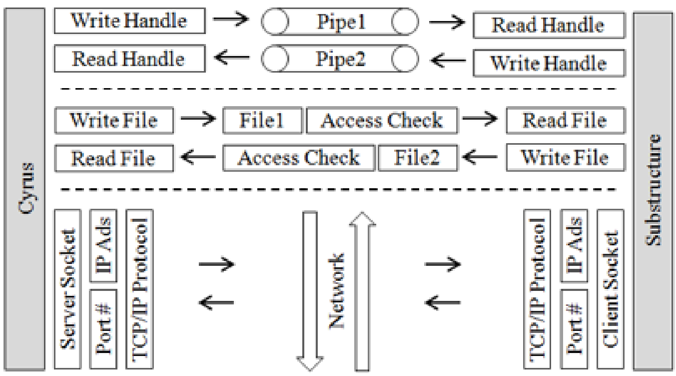


Figure 3. Different communication methods in the multi-scale framework

For large structural systems, using the pipe option to transfer data in multiple byte blocks can be time consuming. To improve the communication performance, binary files are also used in conjunction with pipes. In this configuration, pipes are responsible for sending control commands (run and pause) while binary files are used to send stiffness and force values. Unlike pipes, there is no limited size for files; therefore, all the data can be transferred between server and client in a single file instead of multiple byte blocks.

Both pipes and files are local communication methods and cannot be used over a network. To perform geographically distributed simulation or connect multiple distinct computing nodes to increase the computational performance and storage capacity of the system, a socket communication method is implemented in the framework. The type of socket used is the stream socket, also known as connection-oriented socket, which uses TCP (Transmission Control Protocol) internet protocol for data exchange. In this architecture, Cyrus, the server program, creates sockets on the startup and puts them in the listening state. Then, the sockets wait for initiatives from substructures which are the client programs. Each socket is characterized by a socket address which includes an IP address and a port number.

2.4 Graphical User Interface

Cyrus is compatible with a generalized graphical user interface program, FormWorks-Plus [10], which enables the user to conveniently generate finite element mesh for substructures and model the interface between the components. In addition, Cyrus is enhanced with a graphical user interface which facilitates the process of selecting sub-models, choosing coupled DOFs, and specifying proper analysis options. These developments make the mixed-type modelling process more transparent for engineers, contributing to acceptance and utilization of the simulation framework.

2.5 Interface of Multi-Scale Models

One of the challenges in combining different analysis tools is the modelling of the connecting sections at the interface of sub-models. Each FE program has a different element library and number of DOFs. There are many instances where the rotation at the interface nodes of one program must be transferred to the equivalent displacements for another program which only supports translational DOFs. The connection between the VecTor2 membrane elements and VecTor5 frame elements is an example of this type of mixed-dimensional problem. The common approach to connect the two programs is to use Rigid Constraints at the connecting section. This enables the analysis to transfer rotation from the frame sub-model to equivalent translational displacements in the membrane sub-model based on the plane sections remain plane assumption, while satisfying the compatibility and equilibrium in the multi-scale analysis. However, a set of transverse rigid members at the connection acts as a very strong stirrup which does not allow transverse expansion at the interface and adds additional stiffness to the structure which may affect the response of the system.

To overcome the above-mentioned limitations and model the interface section more accurately, a new interface element, F2M element, is proposed. The F2M element is a two nodded semi-deformable element which can fully transfer translational and rotational displacements at the interface. It does not add any additional stiffness to the system and allows lateral expansion at the connecting section. The stiffness matrix of the proposed element was set up such that it has high stiffness values in the transverse and rotational directions while zero stiffness in the axial direction. Similar to the rigid elements, F2M elements are defined along the membrane elements at the interface. A master node connects each set of F2M elements to the corresponding frame sub-model. From the shear stress distribution at the master node of the frame sub-model, the equivalent axial forces at the F2M element nodes are computed. To transfer shear between the two sub-models, the computed equivalent forces are applied in the opposite direction on corresponding nodes of connecting membrane elements.

3. APPLICATION EXAMPLES

The application of the newly developed multi-scale framework is demonstrated through modeling and analysis of two different case studies. All the analyses were done according to the default material models and analysis parameters. No “fine tuning” of the analysis was done in an attempt to obtain a better fit to experimental results.

3.1 Beam-Column Joint

Shiohara and Kusuhara [11] conducted an experimental test program to investigate the response of six half-scale reinforced concrete beam-column joints under quasi-statically reversed cyclic load. Specimen A2 was selected for modeling and analysis in this section. The study intended to compare the analysis results against the experimental results and illustrate the improvements in the response of the multi-scale analysis over the stand-alone analysis.

The test setup and experimental crack pattern are illustrated in Figure 4. Also, the cross-sectional dimensions and reinforcement layout are shown in Figure 5. The loading conditions considered in the experimental program were a constant axial force of 216 kN and a horizontal reversed cyclic load in a displacement controlled manner. The loads were applied at the top of the column.

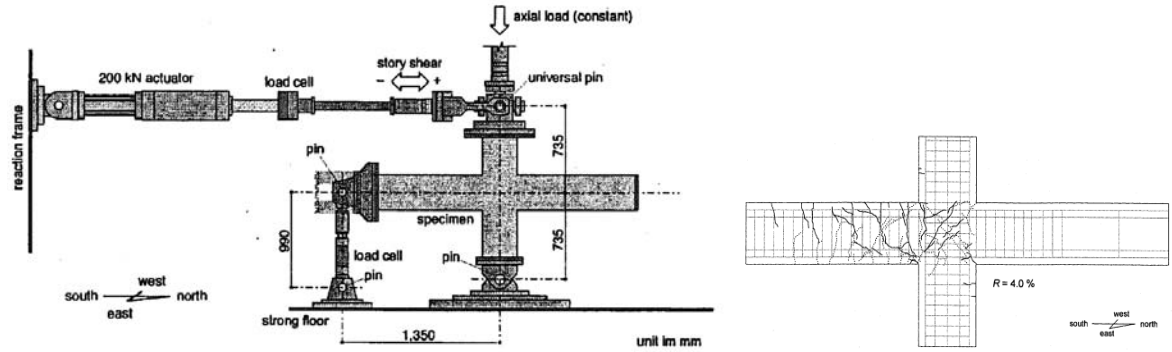


Figure 4. Test setup and experimental crack pattern [11]

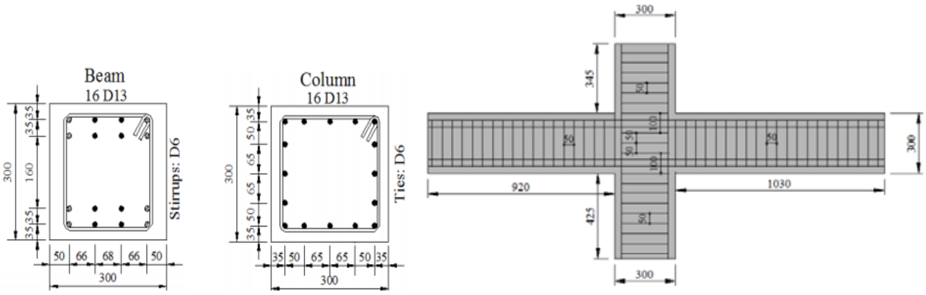


Figure 5. Cross-sectional dimensions and reinforcement layout

A frame model of the entire structure (joint, beams, and columns) was created and analyzed using the frame analysis program, VecTor5 (Figure 6). Based on the analysis results, VecTor5 calculated the overall response of the structure reasonably well. However, the load capacity after the first two cycles was considerably underestimated. Also, the pinching effect in the load-deflection response was not captured accurately and was underestimated. These issues are not exclusively due to the limitations associated with VecTor5 program. Most frame analysis programs, including VecTor5, assume plane section remains plane and perfect bond between reinforcement and concrete in their analysis procedure. However, according to the experimental crack pattern, the test specimen experienced major cracks in the joint panel zone which is considered to be a disturbed region. Also, there were noticeable slips between the reinforcement and concrete at the joint region and extension of the left beam.

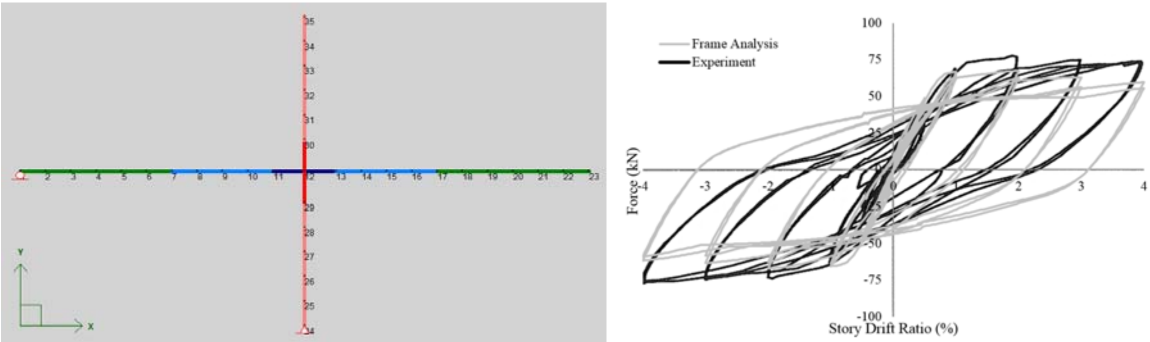


Figure 6. Stand-Alone frame model and load-deflection analysis results

To overcome the limitations associated with VecTor5 program and most frame type analyses, the critical part of the structure can be modelled using VecTor2 which is a more detailed analysis software. The critical zone consists of the joint region and an extension of connecting members in each direction. For the 2D membrane sub-model, rectangular and truss elements were used to model concrete and longitudinal reinforcement, respectively. The transverse reinforcement was modelled as smeared. In addition, link elements were used between rectangular elements and truss elements to capture any possible slip between concrete and longitudinal reinforcing bars. Newly developed F2M elements were used to connect the two finite element sub-models (VecTor2 and VecTor5). Cyrus combined the two sub-models and coordinated the multi-scale simulation.

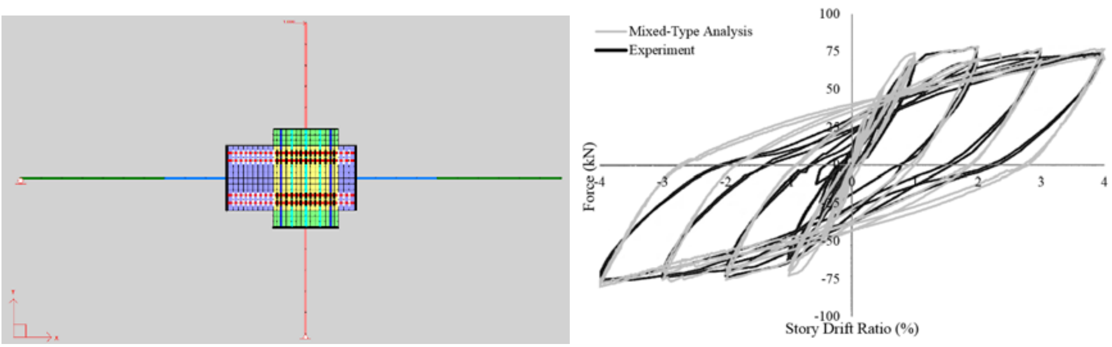


Figure 7. Mixed-Type model and load-deflection analysis results

The experimental and mixed-type analysis load-deflection results are presented in Figure 7. It can be seen that the mixed-type analysis predicted the peak loads and pinching effects with better accuracy than the stand-alone frame analysis. The reason is that unlike frame analysis software, VecTor2 is applicable for the joint panel zone where the strain distribution is significantly nonlinear. In addition, the VecTor2 sub-model was able to compute the slip in the critical part of the structure and consider it in the analysis.

3.2 Three Story Reinforced Concrete Frame

Calvi et al. [12] performed quasi-static cyclic test on a three-story 2/3-scaled reinforced concrete building frame designed only for gravity loads based on typical Italian construction practice common between the 1950s and 1970s (Figure 8). To be consistent with the old design practice, smooth bars were used for the reinforcements and the joint regions were constructed without any transverse reinforcement. Also, instead of bending the longitudinal bars in the exterior joints, they were anchored with end-hooks. The lateral loads were applied in a hybrid force-displacement control manner; the displacement at the top floor was increased in reversed cyclic regime while maintaining a linear force distribution along the height of the structure. In addition, a gravity load of 73 kN was applied on the first and second floors and 54.2 kN on the third floor.

The poor detailing of the reinforcements resulted in a brittle failure mode with most of the damage concentrated in the exterior beam-column joint regions of the first floor (Figure 9). The failure mechanism consisted of the shear cracks in the joint region along with the formation of a wide flexural crack at the interface of the beam due to the slip of smooth bars.

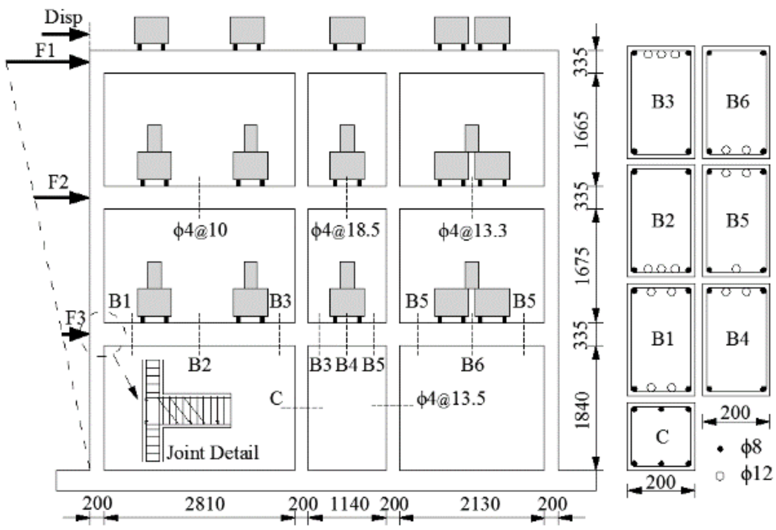


Figure 8. Reinforced concrete frame tested by Calvi et al. [12]

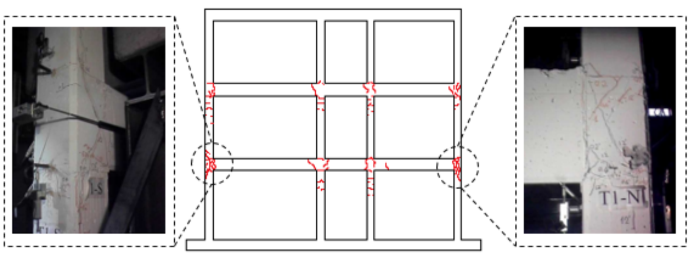


Figure 9. Experimentally observed crack pattern at ultimate load [12]

A frame model of the entire structure was analyzed using VecTor5. Six DOF layered frame elements with member lengths in the range of half of the cross section depth were used. Since like most other sectional analysis procedures, VecTor5 is unable to analyze the disturbed regions, the amount of reinforcement was increased by a factor of two in the beam-column joint regions to avoid artificial damage.

The calculated push-over load-deflection responses are compared to the experimentally observed behaviour in Figure 11. The stand-alone VecTor5 analysis response agreed reasonably well with the experimental results up to the point where joints began to crack. However, beyond this point, the analysis began to overestimate the strength and stiffness, resulting in much higher failure load than the experiment. This is a consequence of the aforementioned limitations associated with most frame analysis software including VecTor5.

The proposed mixed-type analysis procedure can be used to extend frame analysis methods application from a global analysis tool to include local behaviour of critical parts of the structure. Here, based on the stand-alone frame analysis results and experimental crack pattern presented in Figure 9, the external joints in the first floor and an extension of connecting members in each direction (equal to the member height) were selected as critical parts of the structure. These regions were modelled using more detailed analysis software, VecTor2, while the rest of the structure was modelled using frame analysis software VecTor5 (Figure 10). The multi-scale framework, Cyrus, was used to combine the two sub-models and coordinate the mixed-type analysis.

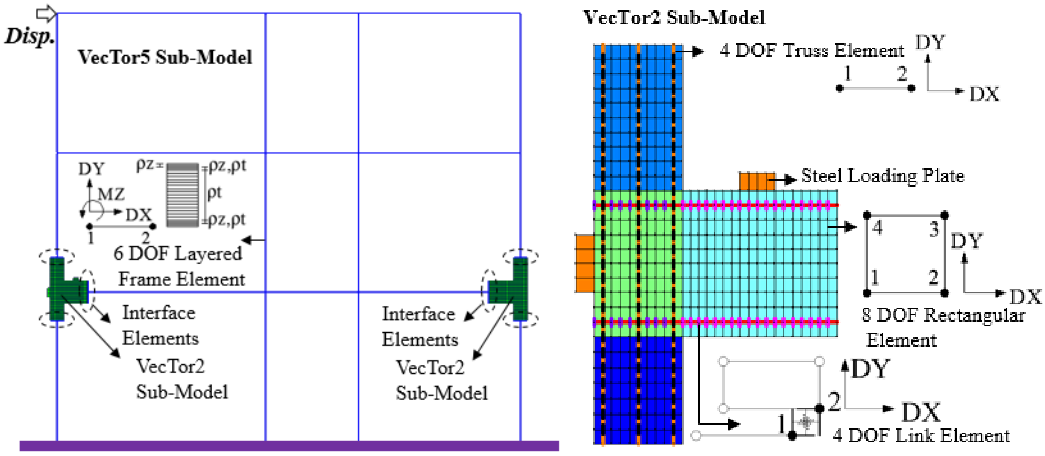


Figure 10. Mixed-type FE model of reinforced concrete frame

Based on the load-deflection results, the mixed-type analysis computed the peak load and stiffness with better accuracy compared to the stand-alone analysis. The mixed-type analysis predicted multiple cracks in the joint panel zone followed by formation of a wide flexural crack at the beam-column interface (Figure 11). Thereafter, a large amount of slip was computed in the longitudinal reinforcement of the beams at the interface section, resulting in significant reduction in the stiffness of the system. The computed crack pattern and bond slip effects agreed well with the experimentally observed behaviour. The analysis also showed the post-peak decay in strength due to the local failure in the joints. It must be noted that none of these mechanisms were captured in the stand-alone frame analysis.

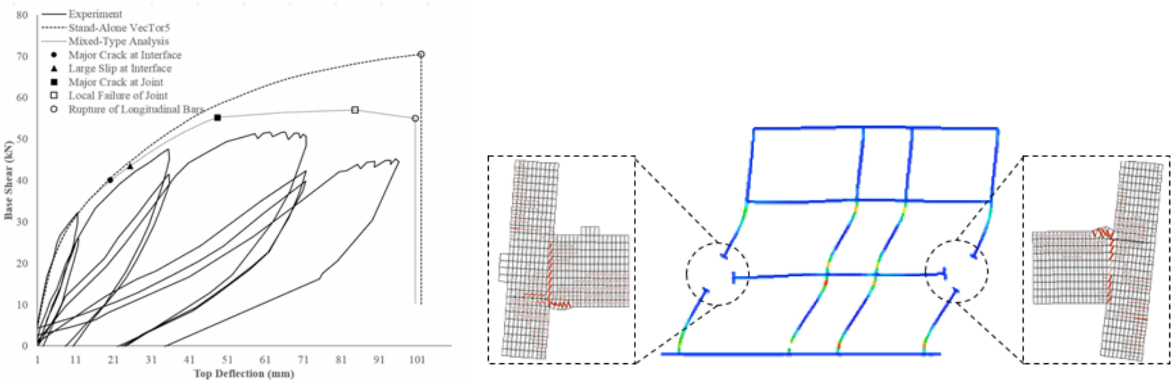


Figure 11. Load-deflection responses and mixed-type analysis crack pattern and deflected shape

4. SUMMARY AND CONCLUSIONS

A multi-scale analysis framework is presented which can integrate different finite element analysis software, enabling accurate simulation of complex mixed-type reinforced concrete systems. The primary goal of the framework is to combine different VecTor analysis programs, while fully considering the coupled nature of the simulation, although other commonly used software (e.g., OpenSees) can also be combined. The proposed framework greatly expands the software application and provides a unique analysis environment where well-recognized behaviour models for reinforced concrete (e.g., DSFM) can be applied to complicated structures. The object-oriented architecture of the framework allows easy inclusion of new analysis tools.

Using substructuring techniques and static condensation procedure, two solution algorithms are developed and implemented into the program to consider the interaction between components. The combined tangent-secant solution method is verified theoretically and numerically, illustrating that analysis tools with different solution algorithms can be integrated. The framework is implemented with a wide range of communication methods including binary file, pipe, and TCP/IP socket. A graphical user interface is implemented into the framework, facilitating the mixed-type structure modelling process. A new interface element is developed to address problems associated with traditional mixed-dimensional coupling approach.

To demonstrate the new capabilities of the framework, two application examples are presented. In these examples, the frame type analysis is enhanced with an effective solution technique for detailed analysis of disturbed regions such as beam-column joints and bond effects between reinforcement and concrete. In both application examples, the mixed-type analysis predicted the behaviour of the structures with a level of accuracy which was impossible or impractical to achieve with most stand-alone analysis tools.

The development of the framework is still in progress. In the near future, the program will have a feature to integrate experimental specimen such that pseudo-dynamic hybrid simulation is possible using the framework. In addition, by using a standardized communication protocol and data exchange format, which is under development at University of Toronto, the framework will be inter-operable with several other analysis tools.

ACKNOWLEDGMENTS

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

REFERENCES

[1] F.J. Vecchio, Disturbed stress field model for reinforced concrete: Formulation. *J. Struct. Eng.*, 126(9), 1070–1077, 2000.

[2] D. Palermo, F.J. Vecchio, Behaviour of 3-D reinforced concrete shear walls. *ACI Struct. J.*, 99(1), 81-89, 2002.

[3] F.J. Vecchio, W. Shim, Experimental and analytical re-examination of classic concrete beam tests. *J. Struct. Eng.*, 130(3), 460-469, 2004.

[4] M.P. Collins, F.J. Vecchio, R.G. Selby, The failure of an offshore platform. *Can. J. Civil Eng.*, 24(3), 460-470, 1997.

[5] F.J. Vecchio, Contribution of NLFEA to the evaluation of two structural concrete failures. *J. Perform. Constr. Fac.*, 16(3), 110-115, 2002.

[6] O. Kwon, A.S. Elnashai, B.F. Spencer. A framework for distributed analytical and hybrid simulations, *Structural Engineering and Mechanics*, 30(3), 331–350.

[7] E. Indrupska, Intel math kernel library for windows user's guide. 2014.

[8] O. Schenk, K. Gartner, PARDISO user guide version 5.0.0. 2014.

[9] S. Saatci, Behaviour and modelling of reinforced concrete structures subjected to impact loads. Ph.D. thesis, Dept. of Civil Engineering, Univ. of Toronto, 288, 2007.

[10] V. Sadeghian, FormWorks-Plus: improved pre-processor for VecTor analysis software. Master thesis, Dept. of Civil Engineering, Univ. of Toronto, 144, 2012.

[11] H. Shiohara, F. Kusuhara, Benchmark test for validation of mathematical models for non-linear and cyclic behaviour of RC beam-column joints. Report by Dept. of Architecture, School of Engineering, University of Tokyo, 2007.

[12] G.M. Calvi, G. Magenes, S. Pampanin, Experimental test on a three storey RC frame designed for gravity only. Proc., *12th ECEE*, London, Paper n. 727, 2002.