A MULTI-SCALE STRUCTURAL MODEL FOR COUPLING 1D WITH 3D ELEMENTS

Jianguang Yue[1], Apostolos Fafitis[2], Jiang Qian[1]
[1]State Key Laboratory of Disaster Reduction in Civil Engineering,
Tongji University, Shanghai 200092, China
[2]Department of Civil Engineering, Arizona State University, Tempe, AZ 85287, USA

Abstract: In most framed structures the nonlinearities and the damage are localized, extending over a limited length of the structural member. Therefore the segments of a member that have entered the nonlinear range may need to be analyzed by three-dimensional (3D) elements whereas the rest of the member can be modeled by simpler one-dimensional (1D) beam elements with fewer degrees of freedom. The proposed Multi-scale model couples the small scale solid 3D elements with the large scale 1D beam elements. The mixed dimensional element coupling based on the kinematic coupling technique of ABAQUS software is implemented for the analysis of the solid and the beam elements. The analysis results are compared with test results of a reinforced concrete pipe column and a structure consisting of reinforced concrete columns and a steel space truss subjected to static and dynamic loading. The same structures are analyzed using 3D solid elements for the entire structure. A comparison of the accuracy and the computational effort indicates that by the proposed Multi-scale method the accuracy is almost the same but the computational effort is significantly reduced.

Keywords： multi-scale model; mixed dimensional elements; kinematic coupling; nonlinear analysis.

1. INTRODUCTION

Structural engineering has different scale regions, such as length and time scale, external forces, and physical actions. In length scale, the structure damage is always at local level, more attentions should be paid on the local detailed characteristics, while most structure analysis is carried in one scale which cannot reflect the damage in the member scale clearly[1, 2]. It is essential to develop a numerical method coupling different scales for the local damage and global behavior analysis.

Different dimensional elements would represent different length scales. It would to combine the lower dimensional elements type with higher ones in a single model to simulate the structure behavior in different spatial scales. The kinematic coupling is a most basic and widely used method of coupling of Finite Element with Finite Element. The constraints imposed by kinematic coupling are usually calculated as a function of the nodal coordinates [3]. This method avoids the use of multi-points constraints or Lagrange multipliers [2].

In this paper, a multi-scale structure model using kinematic coupling technique of ABAQUS software is presented. Taken a reinforced concrete pipe column test and a composite structure test as examples, the results of solid-element-column model, multi-scale element model and tests are compared from each other. Thus the application of multi-scale element model in structure engineering is investigated in different ways from member to structure and static case to dynamic case.

2. MULTI-SCALE MODELING

For different purposes, structural analysis can be carried out at different scale levels which are represented by different dimensional elements. According with normal damages of frame structure, the major damaged positions such as the joints as shown in Fig.1 can be determined as material scale. Other positions, such as beam or column member, can be determined as member scale.

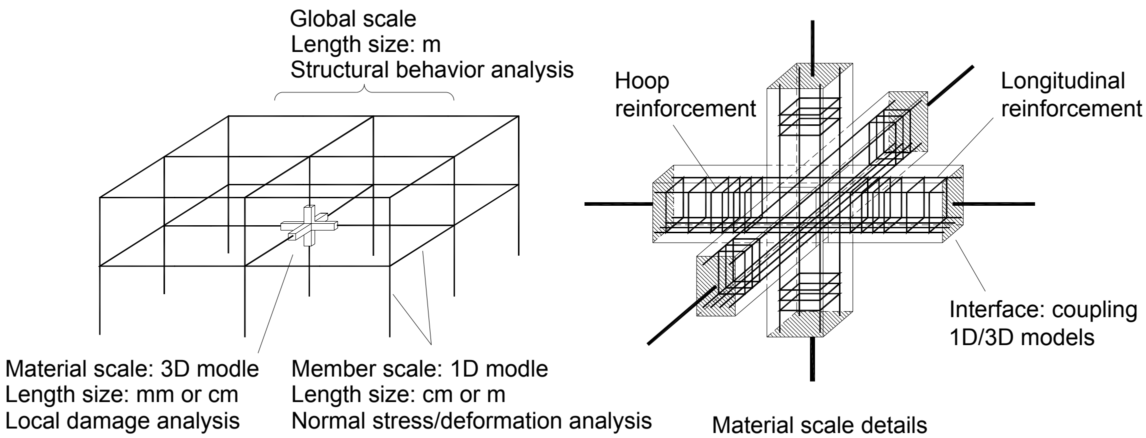


Fig.1 Multi-scale model

Material scale can be simulated with solid elements and a complex material constitutive, which can reflect detailed mechanical behavior. Member scale is simulated with beam elements or shell elements and a normal material constitutive for its material behavior, which can reflect normal mechanical behavior. Different mixed dimensional elements are coupled together using “kinematic coupling” technique [3].

3. MEMBER MULTI-SCALE MODEL

Rong and Tu [4, 5] had done a quasi-static test research for a reinforced concrete pipe column, see Fig.2 and 3 respectively. As the top displacement increasing, the concrete cracked and damaged at column bottom and the reinforcement fielded, see Fig.4.

Fig.2 Column test model

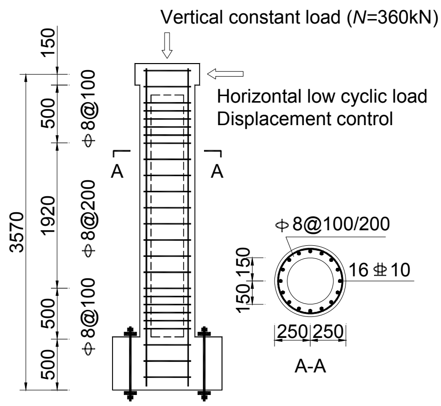


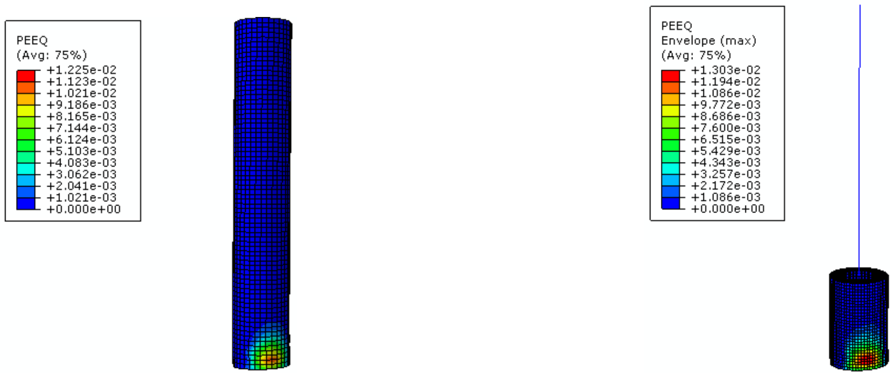
Fig.3 Column test details

Fig.4 Column damage

A solid-element-column model (Model 1) is used to compare with the multi-scale model (Model 2). The concrete material behavior of solid element is simulated with the plastic damage model of ABAQUS software. In Model 1, the concrete and reinforcement of column are simulated by the C3D8R elements and T3D2 elements of ABAQUS. In Model 2, the concrete and reinforcement simulation of the region from the column bottom to the height 750mm are same with the solid column model; concrete in other region is simulated with B31 elements; 1D and 3D elements are coupled together using kinematic technique. Fig.5 shows the contours of concrete max equivalent plastic strain of two models.

Fig.6 shows the load-displacement curves of test and two finite element models, which reflect the column whole mechanical behavior in member scale. For verifying the local simulation accuracy of the multi-scale model, the concrete max equivalent plastic strains and max reiforement stresses of two finite element models are compared in Fig.7 and 8 respectively.

The CPU calculation times of Model 1 and Model 2 are 4851 seconds and 786 seconds respectively. The computational efficiency of Model 2 is approximately six times to Model 1. Those results indicate that the multi-scale model can reflect the mechanical behavior in member scale and local damage in material scale well with efficiency computer calculation.



(1) Model1 (2) Model2

Fig.5 Concrete equivalent plastic strain of two models

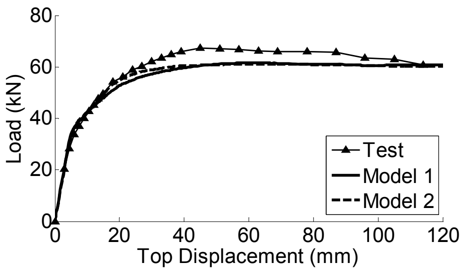


Fig.6 Load-displacement curves

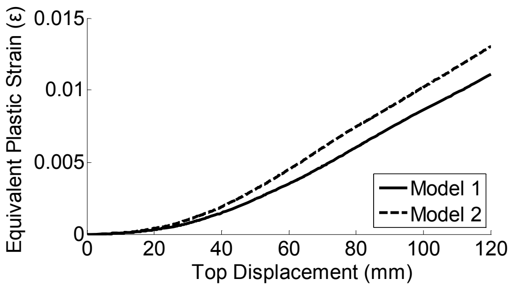


Fig.7 Strain-displacement curves

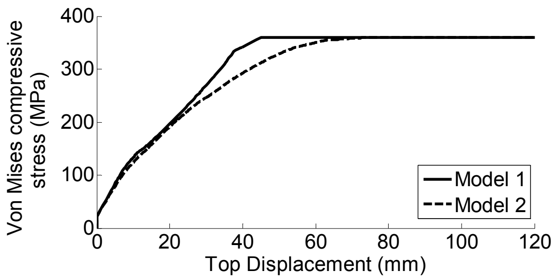


Fig.8 Stress-displacement curves

4. STRUCTURE MULTI-SCALE MODEL

Bai and Li [6, 7] had done a pseudo-static and a pseudo-dynamic 1:8 scale model test from a real structure, see Fig.9 and 10 respectively. Fig.11 shows the plane and profile drawings of the test.

Fig.9 Real structure

In the pseudo-dynamic test, El Centro (1940 NS) waves were input at the loading points (see Fig.10 and 11 respectively), which the peak acceleration were 50 cm/sec^2, 100 cm/sec^2, 200 cm/sec^2, 400 cm/sec^2, 600 cm/sec^2 and 800 cm/sec^2. In the pseudo-static test, the test loading was controlled by the top displacements.

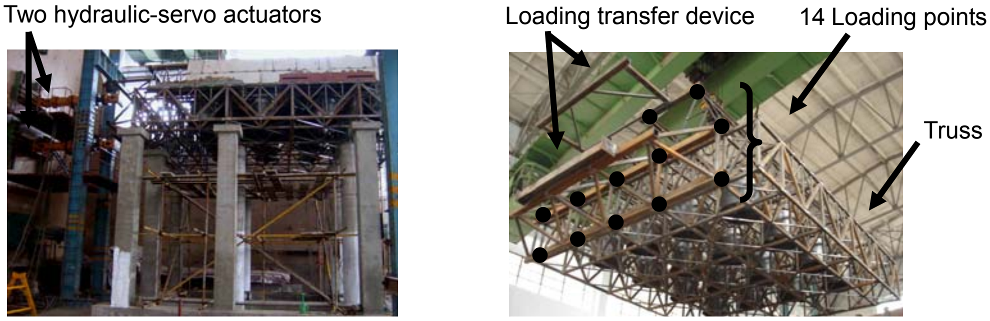


Fig.10 Test model and loading transfer device

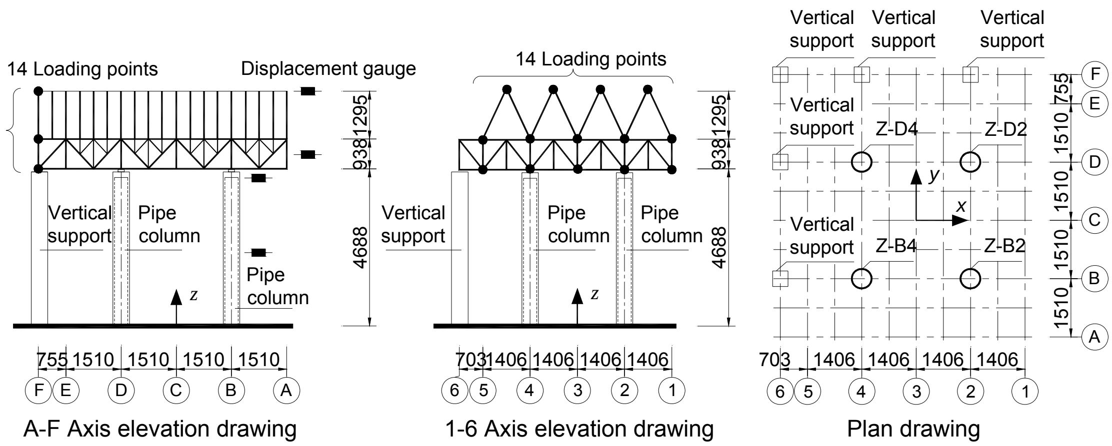


Fig.11 Plane and profile drawing

The test is simulated also by two finite element models. The reinforcement material behaviors were simulated with ideal plastic model. The material behavior simulation of concrete was same with the column test. In structure solid-element-column model (Model 1), the columns are completely used C3D8R elements for concrete and T3D2 elements for reinforcement in ABAQUS. Steel truss was simulated with B31 elements. Concrete slab was simulated with S4R elements in ABAQUS. In structure multi-scale model (Model 2), concrete and reinforcement in the region from column bottom to the height 1100mm were simulated with C3D8R elements and T3D2 elements. Steel truss was simulated with B31 elements. Concrete in the other region was simulated with the B31 elements. 1D and 3D elements were coupled by kinematic coupling technique.

The models dynamic properties listed in Table 1 indicate the frequencies calculated by two models agree well with the measured. Top displacement and base shear are compared with the experiment as list in Table 2. Table 3 shows the CPU calculation time for the two finite element models.

Mode	Measured frequency(Hz)	Model 1		Model 2	
		frequency (Hz)	Modal shape	frequency (Hz)	Modal shape
1	3.20	3.06	Torsion	3.17	Torsion
2	4.59	3.43	Translation	3.43	Translation

Table 1 Comparison of dynamic response

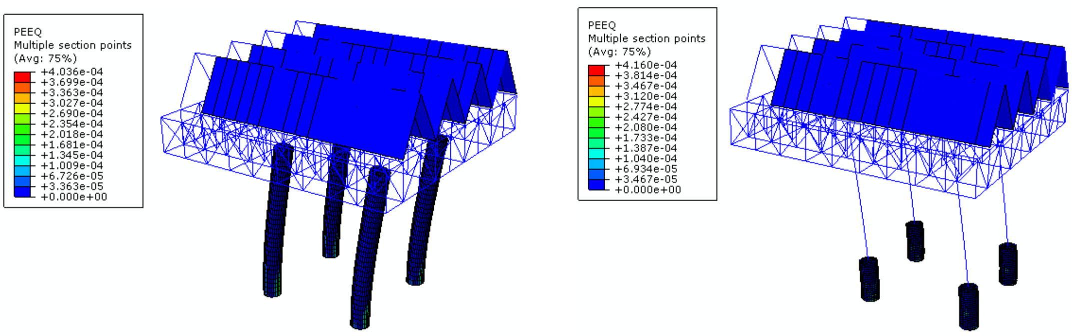
Peak acceleration (cm/sec^2)	Top displacement(mm)			Base shear(kN)		
	Test	Model 1	Model 2	Test	Model 1	Model 2
50	1.73	1.49	1.40	11.55	10.94	8.27
100	2.20	2.89	2.80	16.36	18.25	15.70
200	4.93	5.68	5.60	29.33	29.73	24.08
400	16.80	16.73	17.26	48.04	53.53	50.47
600	22.20	25.04	24.83	65.57	72.11	69.21
800	33.23	33.36	34.00	102.2	92.47	90.17

Table 2 Comparison of structure displacement and base shear

Peak accelaration (cm/sec2)		50	100	200	400	600	800
CPU Time (second)	Model 1	2367	2500	4257	13801	21976	27713
	Model 2	571	751	790	1501	1636	2120

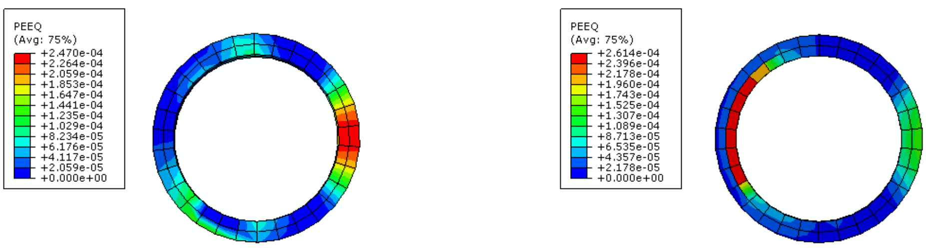
Table 3 Dynamic calculation efficiency comparison

Under the peak acceleration 800 cm/sec^2 waves, the concrete max equivalent plastic strains of columns, and distributions of the column cross section are shown in Fig.12 and 13 respectively. And the column concrete max equivalent plastic strains vs the structure top displacements and the reinforcement max stresses vs the structure top displacements are shown in Fig.14 and 15 respectively.



(1) Model 1 (2) Model 2

Fig.12 Concrete max equivalent plastic strain



(1) Model 1 (2) Model 2

Fig.13 Cross section concrete max equivalent plastic strain distribution

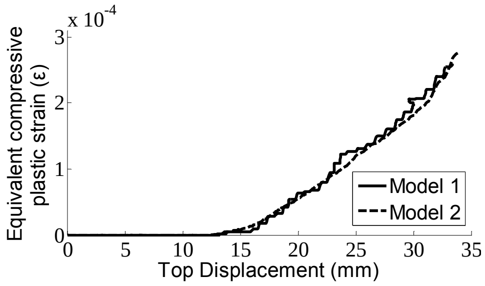


Fig.14 Equivalent plastic strain-displacement curves

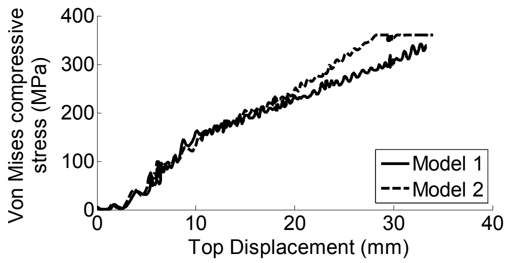


Fig.15 Von Mises stress-displacement curves

In the static case, the load-displacement curves of the two finite element models and test (see Fig.16). Fig.17 and 18 respectively shows the concrete max equivalent plastic strains and max reiforement stresses of two finite element models. Under static actions, the CPU calculation times of Model 1 and Model 2 are 2940 seconds and 152 seconds respectively. The computational efficiency of Model 2 is approximately twenty times of Model 1.

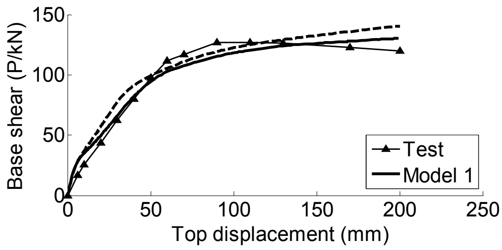


Fig.16 Load-displacement curves

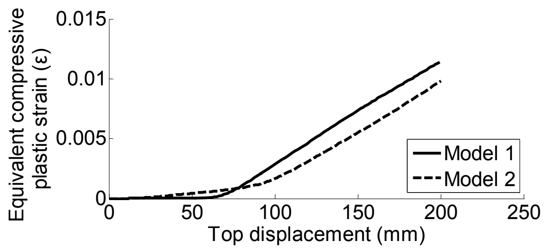


Fig.17 Displacement- strain curves

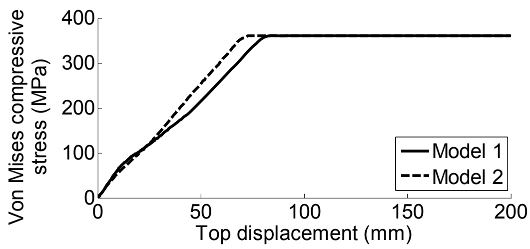


Fig.18 Displacement- stress curves

Those results indicate that the multi-scale structure model can well reflect the structure behavior at global scale and local damage at material scale with efficient compute calculation under dynamic or static actions.

5. CONCLUSIONS

A modeling method of multi-scale finite element model is studied in this paper using the kinematic coupling technique of ABAQUS software. Taken a reinforced concrete pipe column test and a composite structure test as examples, the application of multi-scale element model in structural engineering is investigated in static nonlinear analyses and dynamic nonlinear analyses. Under static action and dynamic action, the multi-scale finite element models of the column and the structure can reflect the mechanic behavior well in member scale and global scale, and predict major failure features and detail damages in the material scale, what are all similar to the test and has a good computational efficiency.

6. ACKNOWLEDGEMENTS

Financial supports from the National Basic Research Program of China (Grant No. 2007CB714202) and China Scholarship Council are gratefully acknowledged. The authors particularly thank Pro. Bai(Xi’an University of Architecture and Technology) for providing the test results.

REFERENCES

[1] Li ZX, Zhou TQ, Chan THT, Yu Y. Multi-scale numerical analysis on dynamic response and local damage in long-span bridges. Engineering Structures 2007;29(7):1507-1524.
[2] Mata P, Barbat AH, Oller S. Two-scale approach for the nonlinear dynamic analysis of RC structures with local non-prismatic parts. Engineering Structures 2008;30(12):3667-3680.
[3] ABAQUS Analysis User's Manual. ABAQUS, Inc., 2003.
[4] Rong Z. An experimental research on seismic behavior of great size thin wall reinforced concrete columns. M.D. Thesis, Xi'an University of Architecture and Technology, Xi’an, 2006.
[5] Tu F. Experimental study and nonlinear finite analysis on seismic behavior of reinforced concrete pipe columns with large size and thin wall. M.D. Thesis, Xi'an University of Architecture and Technology, Xi’an, 2006.
[6] Bai GL, Zhu LH, Zhao CL, Li HX, Li XW. Model test on behavior of direct air cooled condenser support platform under load and erathquake action. Journal of Building Structures 2008;29(5):8.
[7] Li HX. Study on the support system for air cooled condenser of big power plant. Ph.D. Thesis, Xi'an University of Architecture and Technology, Xi’an, 2006.

Corresponding author: Jianguang Yue
Email: 79jgyue@tongji.edu.cn