Ninth Canadian Conference on
Earthquake Engineering
Ottawa, Ontario, Canada
26-29 June 2007

SEISMIC SIMULATION OF COMPLEX STRUCTURES USING DISTRIBUTED ONLINE HYBRID TEST SYSTEM

T. Wang[1], P. Pan[2], N. Yoshitake[3] and M. Nakashima[4]

ABSTRACT

Seismic simulation of a complex structure is conducted by a distributed online hybrid test system called the Peer-to-Peer (P2P) Internet online hybrid test system. In this system, each substructure is encapsulated, and only the displacements and forces at the boundaries are exchanged between substructures via standard Input/Output (I/O) interfaces. This arrangement makes it feasible to incorporate multiple finite element (FEM) programs into this system. Seismic behavior of an SRC (Steel encased Reinforced Concrete) structure with a steel tower on the top is obtained using this system. The entire structure is divided into three substructures, i.e., the SRC frame, the first story of the tower, and the upper part of the tower. The SRC frame is analyzed by OpenSEES, which has a great capacity to analyze composite SRC members. The first story of the tower is taken as the experimental part, because it sustains the largest deformations and subsequent structural damage. The upper part of tower is analyzed by ABAQUS, which is strong at simulating steel braced frames involving large geometric nonlinearity. The results demonstrate that nonlinearities of each numerical substructure are accurately simulated by the respective FEM program.

Introduction

The online hybrid test (also called the pseudo dynamic test) is an effective experimental method for the examination of the seismic response of structures (for example, Takanashi 1975). It solves the equations of motion in a computer domain by using the restoring forces obtained from the associated test. When employing the substructuring technique, the online hybrid test becomes a more powerful tool for testing large-scale structures (for example, Nakashima 1988). It treats parts of the structure numerically and others by test, thus making use of the benefits of both the analysis and test.

The Peer-to-Peer (P2P) Internet online hybrid test system (Pan 2006) is a system developed recently along this line. This system has already been demonstrated valid by a simple model with a nine DOF structure. In that test, however, the numerical substructures were assumed to be linear elastic and implemented by a handmade source code.

[1]JSPS Postdoctoral fellow, Disaster Prevention Research Institute, Kyoto University, Gokasho, Uji, Kyoto 611-0011, Japan
[2]Postdoctoral fellow, Disaster Prevention Research Institute, Kyoto University, Gokasho, Uji, Kyoto 611-0011, Japan
[3]Graduate student, Graduate School of Engineering, Disaster Prevention Research Institute, Kyoto University, Gokasho, Uji, Kyoto 611-0011, Japan
[4]Professor, Disaster Prevention Research Institute, Kyoto University, Gokasho, Uji, Kyoto 611-0011, Japan

The P2P system needs further examination and/or improvement. First, convergence of the proposed iterative procedure needs further investigation when nonlinear numerical substructures are included, because it may need a larger number of iterations, which would make the system inefficient. Second, numerical substructures may have to be more accurate, which requires more sophisticated numerical models. It has already been demonstrated that finite element models can improve the accuracy greatly (for example, Pinto 2004, Takahashi 2006). The previous application, however, was restricted to the use of a single FEM program. It would enhance the accuracy of the numerical substructures more significantly if multiple FEM programs, each of which has its own features and advantages, were used simultaneously. In the proposed P2P system, each substructure is treated as independently as possible, so that it is possible to combine different FEM programs without touching upon the inside of the programs.

In this study, the seismic behavior of a complex structure, a steel encased reinforced concrete structure (SRC) with a steel transmission tower placed on the top, is investigated by the P2P Internet online hybrid test system. The natural periods of the SRC part and the steel tower are so close that deformations of the tower may be amplified during an earthquake, thus making the responses highly nonlinear. The first story of the tower, the weakest portion, is taken out for the experiment. The SRC part and the upper part of tower are treated numerically by different FEM programs, i.e. OpenSEES and ABAQUS, respectively. This test shall calibrate the effectiveness of the P2P system in terms of its feasibility in nonlinear response simulation and its versatility in combining multiple FEM programs for the simulation of numerical substructures.

Summary of P2P Internet Online Hybrid Test

The theoretical basis and detailed implementation of the proposed P2P system can be found in the previous study (Pan 2006). The system design and the test scheme are summarized as follows.

System design

In the proposed P2P Internet online hybrid test system, the simulated structure is divided into multiple substructures, as shown in Fig.1. All substructures are equally treated and geographically distributed to various laboratories. A center part called “Coordinator” is devised to achieve the compatibility and equilibrium on the boundaries between the substructures. The boundary displacements and the corresponding forces are exchanged between each substructure and the “Coordinator” via an I/O interface. By this way, each substructure is implemented as a highly encapsulated “Partner”, and can be treated as either an experimental part or an analytical part.

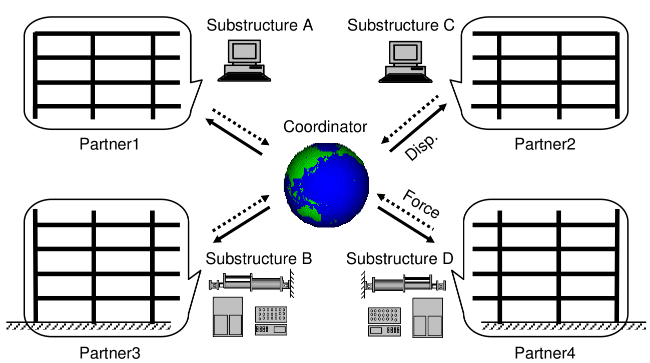


Figure 1. Concept of P2P Internet online hybrid test system (Pan 2006).

Implementation of the P2P system is explained by an example in which the structure consists of one tested substructure and one numerical substructure, with only one boundary existing between the two substructures. At the beginning of one step, the “Coordinator” sends one trial displacement, which could be the displacement of the previous step, to both substructures. Compatibility between the two substructures is satisfied automatically. Then the substructures are analyzed independently either by numerical simulation or physical test, and the boundary forces are sent back to the “Coordinator”, where the equilibrium on the boundary is examined. If the boundary is balanced, the current step is completed, and the analysis proceeds to the next step. Otherwise, the “Coordinator” calculates a new trial displacement based on the imbalanced force at the boundary. The above procedure is repeated until the equilibrium is satisfied. To realize this system, however, two problems are to be resolved. On the one hand, an effective iterative procedure is required for the “Coordinator” to determine and modify the boundary displacement. On the other hand, iteration should be avoided for physical tests.

Application of Quasi-Newton Method

In the “Coordinator”, the only information required to determine the next trial displacement is the imbalanced force at the boundary. The natural way is to use the tangential stiffness at the boundary. However, it is not feasible to obtain such a stiffness, because measurement of tested substructures has limited resolution. One alternative is to use the secant stiffness, which can be updated by the quasi-Newton method with the gradient information of previous steps. In the quasi-Newton method, the initial value of the stiffness is not necessarily close to the tangential stiffness. Any positive-definite matrix is acceptable. The updated trial displacement can be calculated by the common equation solution procedure using the imbalanced force and the updated secant stiffness. Note that the quasi-Newton method has a super-linear convergence speed. It can determine the trial displacement effectively and systematically. In this P2P system, a commonly used Broyden-Fletcher-Goldfarb-Shanno (BFGS) method is adopted.

Two-round Quasi-Newton Procedure

A testing scheme featuring a two-round quasi-Newton procedure is devised to avoid iteration for the tested substructures. The procedure, shown in Fig.2, is in essence a predicting and correcting scheme, each corresponding to one round quasi-Newton procedure.

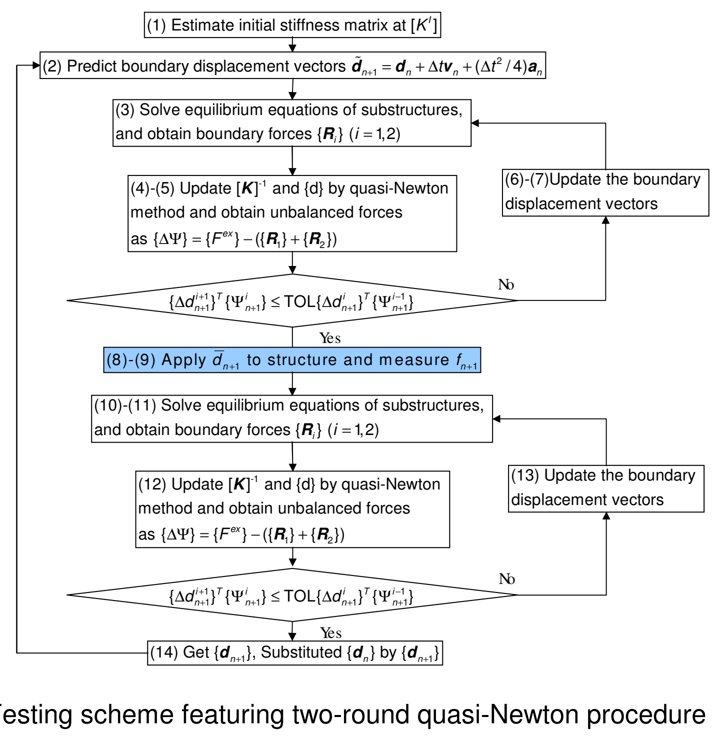


Figure 2. Testing scheme featuring two-round quasi-Newton procedure (Pan 2006).

In one step of analysis, the tested substructures are always assumed to be linear elastic, and associated nonlinearities are taken into consideration only by one time loading between the predicting and correcting quasi-Newton procedures. The error introduced by this linear assumption will not affect the responses greatly if major parameters of the system, i.e. the assumed linear stiffness and the time interval, are carefully selected (Yoshitake 2006).

Seismic Simulation of SRC Structure using P2P Internet Online Hybrid Test System

Description of SRC Structure with Steel Tower on Top

Shown in Fig.3 is a bird's-eye view of the structure, which consists of an SRC structure and a braced steel tower. The structure length is 73 m in the X-direction. The width is 40 m in the Y-direction. The total height of the entire structure is 80 m, in which the SRC structure is 38 m and the steel tower is 42 m. The SRC structure is a seven-story frame with a penthouse. The SRC frame consists of ten planar frames in the X-direction, and five identical frames in the Y-direction. Columns and beams are made of SRC where a cold-formed steel pipe and H-shaped steel are used as the bone of the column and beam, respectively. Two pieces of shear walls with a thickness of 200 mm are set in each frame in the Y-direction. Concrete slab is placed on each story with the thickness of 150 mm. The SRC frame is rather rigid with a natural period of about 0.6 sec. The steel tower is an eleven story braced frame. Braces exist only in the first nine stories, and all braces have a pipe-shaped cross-section. The tower consists of six columns, two large pipes extended from the SRC columns and four smaller pipes built on independent foundations. All of the beams of the steel tower are made of H-shaped steel. Concrete slab exists only on the second floor for placing special facilities. The thickness of the slab is 150 mm. When the base is fixed, the period of the steel tower is 0.7 sec. The weight of this structure is mostly concentrated on the SRC frame, 220 MN in this case, while the steel tower is relatively light, 5 MN in weight. The weight of each floor of the SRC frame is 29 MN; the weight of the roof is 35 MN; the weight of the penthouse is 4.1 MN. Most of the weight of the tower (3 MN) is concentrated on the first story where special facilities are located. All other stories have the same weight of 200 kN.

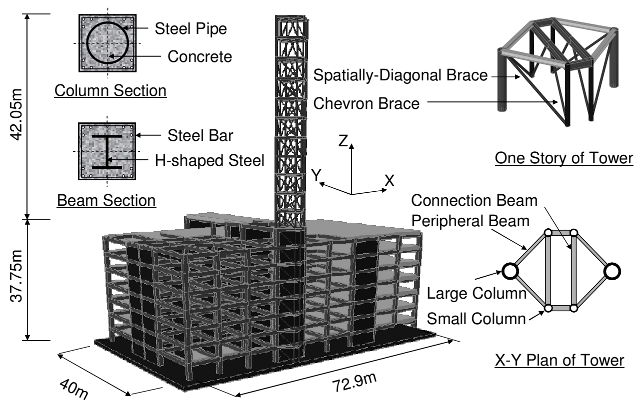


Figure 3. SRC structure with a steel tower.

Structure Simplification and Finite Element Modeling for Numerical Substructures

This spatial structure is simplified to a planer structure in the X direction, in which the response dominates. The natural periods of the SRC frame and the steel tower are kept unchanged. The simplified planer model is shown in Fig.4. For this particular structure, most of the mass on the tower is concentrated at the first story of tower, while the stiffness of this story is almost identical to the upper stories. This would naturally cause large deformations in the first story. Therefore, it is found reasonable to take the first story of the tower out for the physical test. Furthermore, the remaining two parts, i.e. the SRC frame and the upper part of tower, are analyzed separately.

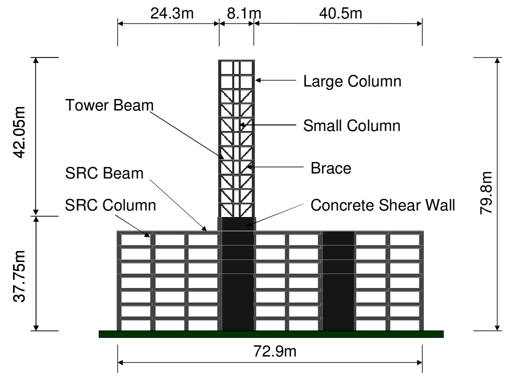


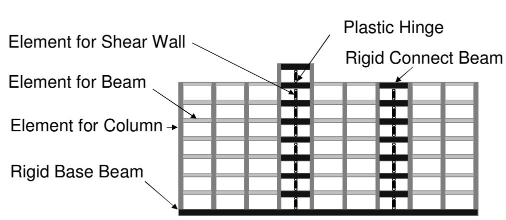
Figure 4. Simplified planar structure.

The SRC frame and the upper part of the tower are analyzed by different FEM codes, namely OpenSEES for the SRC frame and ABAQUS for the upper part of the tower. Use of two separate FEM codes is deemed unique, intending to make the best use of the strengths of individual codes. Note that OpenSEES is excellent in fiber beam-column elements, which are very suitable for the SRC member models, while ABAQUS is very strong in handling geometrical nonlinearities so that the nonlinear behavior of the steel tower would be simulated accurately.

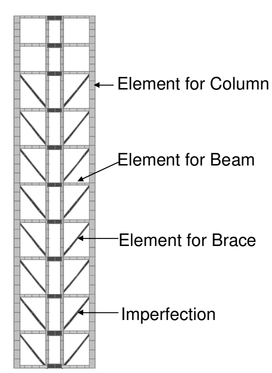
The boundaries between the substructures, however, had to be somewhat simplified because of the limitation of the loading facilities. Two horizontal displacements, one between the upper part and the first story of tower, the other between the first story of tower and the SRC frame, are controlled by the “Coordinator” program. Note that there is no mass on the boundary of the upper part of the tower.

Each SRC member, a beam or a column, is modeled by one fiber beam-column element. Five integration points are inserted along the element. The bases of the columns are fixed on the ground. The diaphragm effect of the concrete slabs is considered by restricting the nodes on each story to move together in the horizontal direction. The shear wall of each story is simulated by a beam-column element with concentrated plasticity. A plastic hinge, which is to represent a nonlinear shear force-story drift angle relationship, is inserted at each end of one element. The beam-column element representing the shear wall is placed at the middle point of one span, and is connected with the surrounding frame by rigid beams, as shown in Fig.5 (a). This FEM model has 177 elements and 178 DOFs in total.

The FEM model of the upper part of tower is built by ABAQUS, as shown in Fig.5 (b). The Euler-Bernoulli beam element is used to represent each column, beam, and brace. Both material and geometric nonlinearity are considered. Initial imperfection (1/750) is imposed at the middle point of each brace to reproduce buckling. This FEM model contains 334 elements and 867 DOFs in total.



(a) FEM model for SRC frame



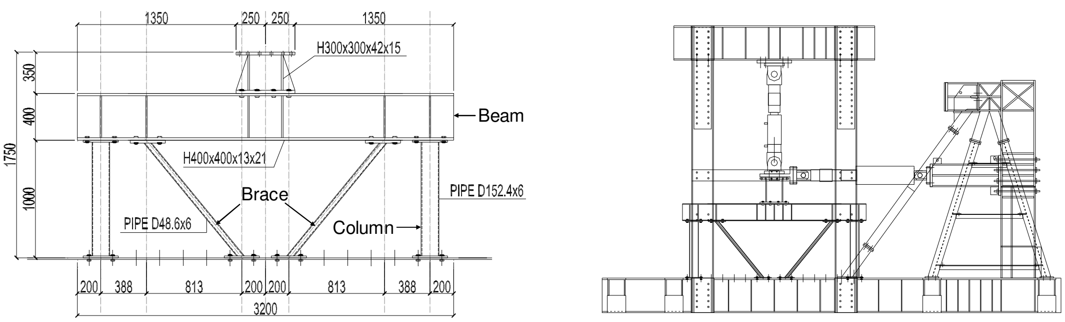
(b) FEM model for upper part of tower

Figure 5. FEM models for numerical substructures.

Test Specimen and Loading System

The conventional online hybrid test procedure is adopted for the test. One dynamic degree of freedom, the relative horizontal displacement between the top and the bottom of the base-tower, is considered. The base-tower is scaled to one-quarter of the prototype, as shown in Fig.6 (a). Although some differences in the configuration between the prototype and specimen do exist, similitude is maintained by proper selection of the force scale ratio. Based on a preliminary numerical analysis, the force scale ratio of 1/11 is adopted.

Shown in Fig.6 (b) is the test setup, which includes the reaction frame, two jacks, and test specimen. The specimen is securely fastened by high-tension bolts to the foundation beam at the bottom and to the jacks at the top. The out-of-plane deformation of the specimen is restricted at the top of the specimen where two jacks are attached. The details of this system can be found elsewhere (Nakashima 1995).



(a) Scaled specimen (b) Test setup

Figure 6. Specimen design and installation.

The specimen is loaded through an external control by the displacement measured directly from the specimen, because the deformation of the reaction frame is not negligible. The external control scheme adopts an iterative procedure to shoot the target displacement. A demonstration test shows that it takes on the average about five iterations in one step of loading, and the control precision is about 0.05 mm, somewhat larger than the inherent control precision of the jack, which is 0.04 mm. Because of the limitation of the controlling precision of the available loading system, a smaller displacement increment than the controlling precision, 0.05 mm, cannot be achieved. Therefore, it is reasonable to jump over the steps for a displacement increment not greater than the controlling precision, and the restoring force of this step is updated by using the prescribed initial stiffness. This scheme is also called “virtual loading scheme”. In this study, 0.2 mm is set as the virtual loading tolerance, which has been demonstrated to be accurate by numerical simulation.

Distributed Online Hybrid Test Environment

In this online hybrid test, the substructures: the SRC frame, the first story of tower and the upper part of the tower, are distributed to three geographically different locations, namely Katsura campus of Kyoto University, a structural laboratory at Uji campus of Kyoto University, and an office at Uji campus of Kyoto University. Constitution of the distributed system is shown in Fig.7. The experimental substructure is located in the laboratory at Uji campus. The PC in the laboratory is used to solve the dynamics of the first story of tower. Numerical simulation of the SRC frame is carried out by a computer at the office of Uji campus. The upper part of the tower is simulated numerically at the office of Katsura campus. The “Coordinator” program is running on a computer at the office of Uji campus. All computers belong to a network called “Kuins III”, but on different subnets, which are protected by strict firewalls. A computer, located at the office of Uji campus but belonging to another network called “Kuins II”, is set outside all firewalls and used to run the proxy program. All communication between the substructures and the “Coordinator” is transferred by a proxy program.

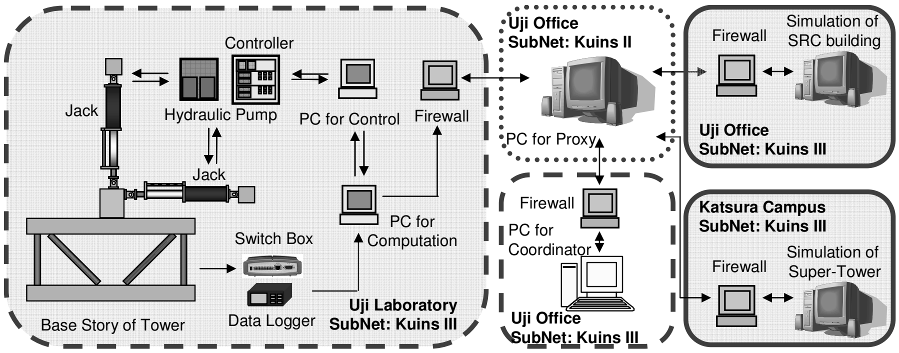
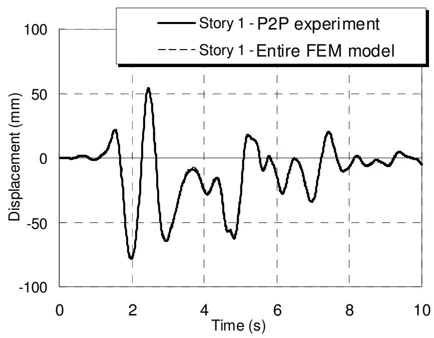


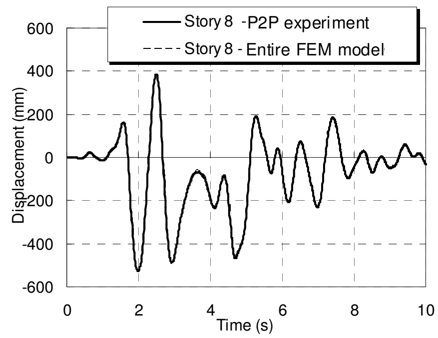
Figure 7. Distributed online hybrid test system.

Results of Test

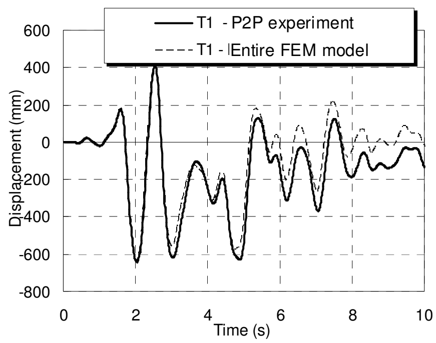
Earthquake responses of the structure were simulated for 10 seconds. The fault-normal ground motion recorded by Japan Meteorological Agency (JMA), Kobe, 1995, was adopted and enlarged by two times to examine responses involving very large nonlinearity. The responses of the first story and the top story of the SRC frame and the tower are shown in Fig.8 (a) to (d). These results are compared with those obtained by an FEM analysis (OpenSEES) applied to the overall structure. The responses of the SRC frame are very close between the overall FEM model and the P2P test, as shown in Fig. 8 (a) and (b). The difference of the peak values is 0.3 mm and 2.2 mm for the first and eighth stories, respectively. Comparing the maximum values for the two stories, 78 mm and 529 mm, the difference can be ignored. The same responses of the SRC frame come from the same FEM models used by the overall simulation and the P2P test. On the other hand, significant difference can be found in the tower part, as shown in Fig. 8 (c) and (d), which stems from the different modeling in overall simulation and the P2P test. In the overall simulation, the braces of the tower are simulated by “truss” elements, which cannot handle buckling behavior of the braces. This compromise was adopted, because dynamic simulation using OpenSEES has difficulties to converge when both material and geometric nonlinearities are significant. Contrast to the overall FEM model, the P2P test adopts ABAQUS and physical test for the upper part and the first story of tower, respectively, and buckling behavior of the braces can be simulated accurately. Comparison of the hysteresis behaviors of the first story of the tower for the overall FEM model and the P2P test indicates that buckling does not occur in the overall FEM model, while significant instable behavior is notable in the P2P test, as show in the rectangle of Fig.9. These responses indicate that the P2P test is valid, and the response obtained by the P2P test is more accurate because buckling behavior is simulated for the tower part.



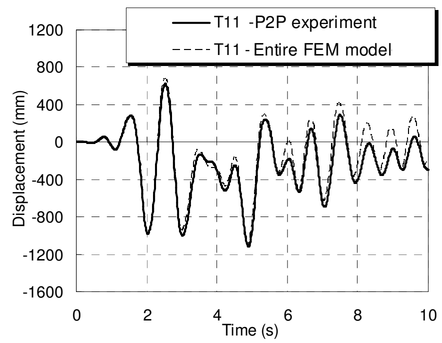
(a) First story of SRC frame



(b) Top story of SRC frame



(c) First story of tower



(d) Top story of tower

Figure 8. Response comparison.

The maximum story drift angle of each story is shown in Fig.10, which shows that significant plastification occurs in the lower stories of the SRC frame. Deformations of the tower concentrated on the first story, while the upper part of the tower is not so large.

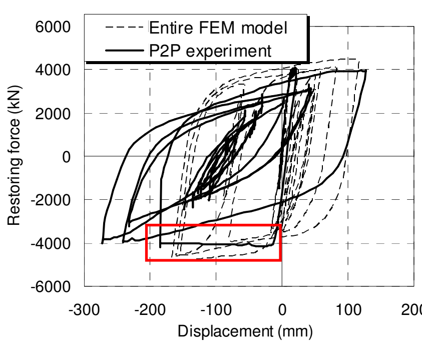


Figure 9. Hysteretic behavior comparison.

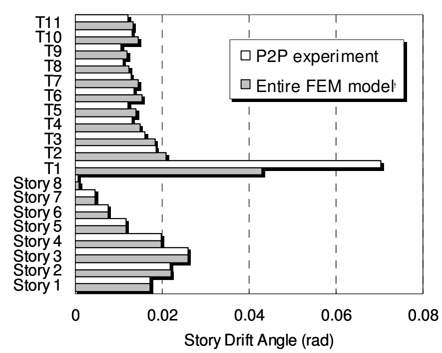
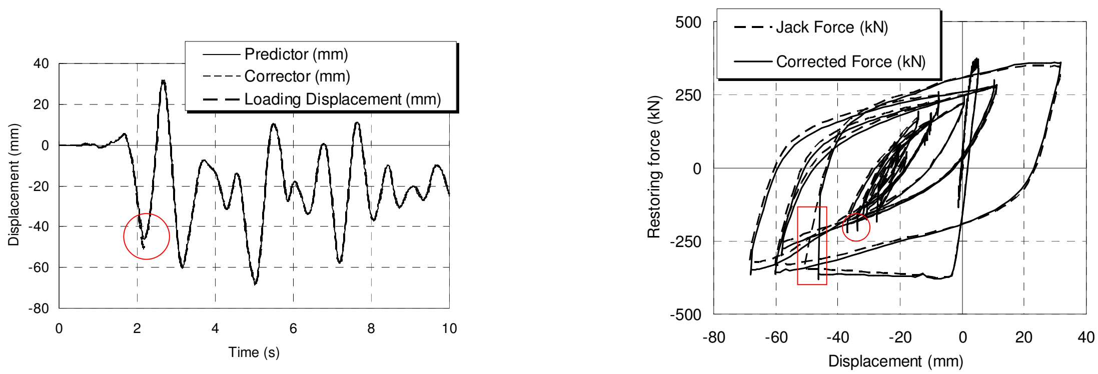


Figure 10. Maximum story drift angle.

The P2P Internet online hybrid test system employs a predicting-correcting procedure based on a linear assumption for the tested substructures. To evaluate the effectiveness of the predicting-correcting procedure and the external loading control, the predicted displacement, the loaded displacement, and the corrected displacement are compared in Fig.11 (a). The maximum difference between the predicted displacement and corrected displacement is 0.13 mm. Compared with the maximum displacement of 70 mm, it is very small, thereby demonstrating that prediction is pretty good, and not much correction effort is needed. The maximum error between the predicted and loaded displacements is 4.62 mm (shown by a circle) and takes place only at the instant of brace buckling. In this occasion, this displacement is very difficult for the loading system to control because of a sharp stiffness change. In other steps, loading error is limited in a range of 0.05 mm. Two relationships, i.e. the jack force vs. the specimen displacement, and the corrected force vs. the corrected displacement, are compared in Fig.11 (b). The steep peak on the curve of corrected force (shown by a circle) represents one time of virtual loading in which linear stiffness is used for the correction. Note that the difference highlighted by a rectangle corresponds to the control error when buckling occurs.



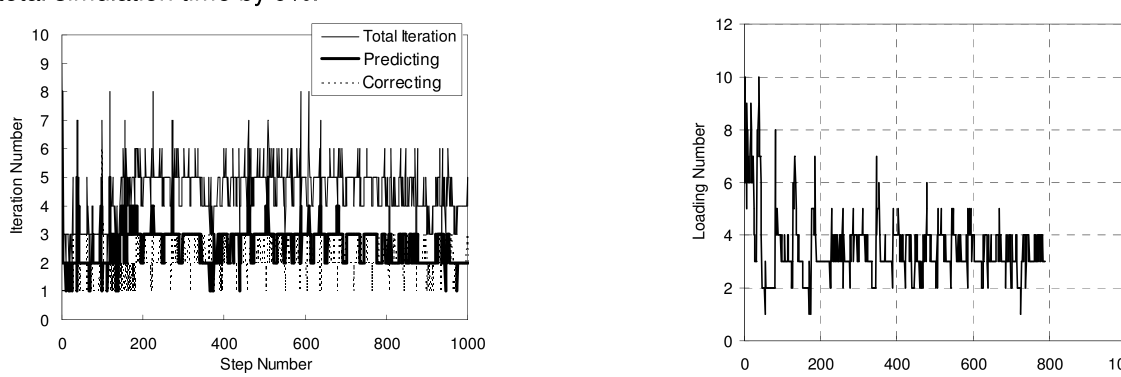
(a) Predictor, corrector and loading displacement

(b) Force from jack and corrected force

Figure 11. Effectiveness of two-round quasi-Newton procedure and external load control.

The entire simulation took 12.4 hours. OpenSEES took 7 hours (25.2 sec/step), ABAQUS took 5.3 hours (19.1 sec/step), and the physical loading took 4.7 hours (16.9 sec/step). In this simulation, OpenSEES simulation and ABAQUS simulation are processed in parallel, while the physical loading is implemented independently. Time efficiency is improved by this parallel mechanism, and the simulation time is reduced by 27% [= (7 + 5.3 + 4.7-12.4) / 17]. The number of iterations of each step is shown in Fig.12 (a). On the average, one step required about 4.5 iterations: 2.6 iterations for the predicting procedure, and 1.9 iterations for the correcting procedure. Comparing with the previous study (Pan 2006), 3 iterations in one step on the average, the increase of the number of iterations is not significant. This is interpreted such that the time interval is so small that the degree of nonlinearity of each substructure in one step is not significant.

The external control took several times of load increments in one step to achieve the target displacement. The number of load increments is shown in Fig.12 (b). At the beginning, it took many times because of the insensitiveness of the measuring device for controlling small displacements. On the average, one step of loading took 3.5 times of load increments. Because of the virtual loading, 211 steps of loading were skipped. Therefore, the virtual loading scheme in this test reduced the loading time by 20% and shortened the total simulation time by 9%.



(a) Number of iterations of Coordinator in each step

(b) Number of load increment in each step

Figure 12. Efficiency of P2P Internet online hybrid test system.

Conclusions

In this study, the seismic behavior of an SRC structure with a steel tower is simulated using the concept of substructuring into many portions, either physically testing or numerically simulating individual portions at different locations, and connecting them by Internet to obtain the earthquake response of the entire structure. Two FEM source codes are incorporated into the test system, in which only the standard I/O interfaces are employed. The test was successful without any malfunction during the test. The major findings of this study are summarized below.

(1) The P2P Internet online hybrid test system is demonstrated to be able to simulate complex structures. Different FEM source codes, i.e. ABAQUS and OpenSEES, are successfully incorporated into the system. By treating each substructure through the most suitable approach, the accuracy of the seismic simulation can be improved.

(2) The P2P online hybrid test took 12.5 hours. This occurs because of the iterative procedure employed in the concerned online hybrid test system and the time-consuming nature of the FEM programs. The number of iterations increases slightly, although larger than the previous study in which all numerical substructures were assumed linear.

(3) The external control is proved accurate, and the virtual loading scheme is able to reduce the total simulation time by 9%. The external control precision is 0.05 mm, and the virtual loading tolerance is set to be 0.2 mm. Both are demonstrated small enough to ensure accurate simulation.

(4) For this particular structure, significant deformations occurred in the tower, with concentration at the weakest first story. This was because the natural periods of the main structure and the tower were close to each other. Careful consideration into the tower response is deemed important for these types of structures.

References

Nakashima M, *et al.*, 1988. Feasibility of pseudo dynamic test using substructuring techniques, *Proceedings of 9th World Conference on Earthquake Engineering*, Tokyo 4, 47-52.

Nakashima M, Akazawa T, Igarashi H., 1995. Pseudo-dynamic testing using conventional testing devices, *Journal of Earthquake Engineering and Structural Dynamics* 24, 1409-1422.

Pan P, Tomofuji H, Wang T, Nakashima M, Ohsaki M, Mosalam KM., 2006. Development of peer- to-peer (P2P) Internet online hybrid test system, *Journal of Earthquake Engineering and Structural Dynamics* 35, 867-890.

Pinto AV, Pegon P, Magonette G, Tsionis G., 2004. Pseudo-dynamic testing of bridges using non-linear substructuring, *Journal of Earthquake Engineering and Structural Dynamics* 33, 1125-1146.

Takahashi Y, Fenves GL,2006. Software framework for distributed experimental-computational simulation of structural systems. *Journal of Earthquake Engineering and Structural Dynamics* 35, 267-291.

Takanashi K, Udagawa K, Seki M, Okada T, Tanaka H, 1975. Nonlinear earthquake response analysis of structures by a computer actuator online system. Part 1: details of the system. *Transactions of the Architectural Institute of Japan* 229, 77-83 (in Japanese).

Yoshitake N, Wang T, Pan P, Nakashima M, 2006. A distributed online hybrid test using commercial finite element program, part 1: Accuracy characteristic of proposed system. *Proceedings of Technical Papers of Kinki Branch*, 2006.