E-SIMULATOR VIRTUAL SHAKING-TABLE TEST

Comparative Study on Elastoplastic Dynamic Responses of Super-Highrise Steel Frame Between Virtual Shaking-Table Test Using E-Simulator and Beam Element Analysis

Masayuki Kohiyama [a], Tomoshi Miyamura [b, g], Daigoro Isobe [c], Makoto Ohsaki [d], Kunizo Onda [e], Muneo Hori [f], Hiroshi Akiba [e], Koichi Kajiwara [g] and Tatsuhiko Ine [g]

[a] Keio University, Dept. of System Design Engineering, Japan
[b] Nihon University, Dept. of Computer Science, College of Engineering, Japan
[c] University of Tsukuba, Dept. of Engineering Mechanics and Energy, Japan
[d] Hiroshima University, Dept. of Architecture, Graduate School of Engineering, Japan
[e] Allied Engineering Corporation, Japan
[f] The University of Tokyo, Earthquake Research Institute, Japan
[g] National Research Institute for Earth Science and Disaster Prevention, Hyogo Earthquake Engineering Research Center, Japan

INTRODUCTION

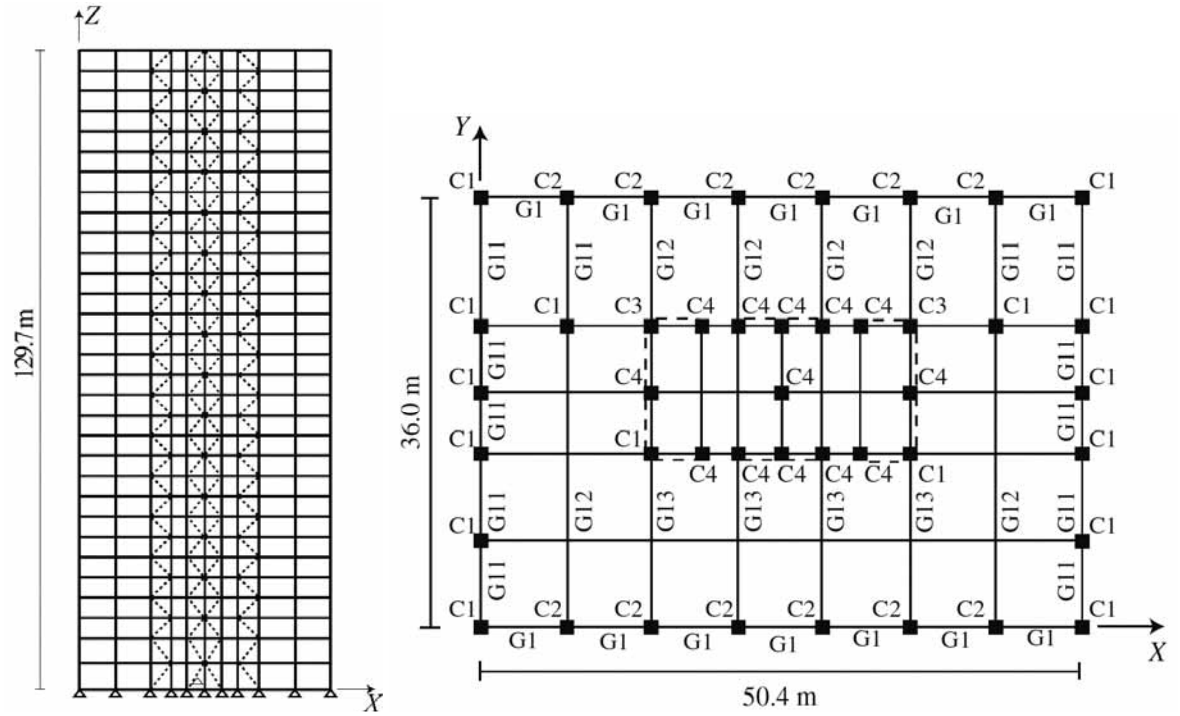
The project of the E-Simulator [1] is under way at the Hyogo Earthquake Engineering Research Center of the National Research Institute for Earth Science and Disaster Prevention (E-Defense NIED), Japan, which facilitates the world’s largest shaking table. The E-Simulator is a parallel finite element (FE) analysis software package that uses ADVENTURECluster [2] as a platform, and constitutive laws and laws of fracture of materials have been implemented to analyze civil and building structures.
A virtual shaking-table test of a 31-story high-rise office building is carried out using the E-Simulator in order to demonstrate its capability to analyze dynamic response of a large-scale structural model. The preliminary study using a model with a simplified mesh has been reported by Ohsaki et al. [3]. In this study, a more detailed model is constructed and we report the comparison results between the FE-analysis using the E-Simulator and the beam element analysis.

1 A VIRTUAL SHAKING-TABLE TEST OF A 31-STORY STEEL BUILDING FRAME

1.1 Analysis model

A virtual shaking-table test of a 31-story high-rise office building is conducted, which has been designed for this study. The story heights are 5.4 m for the 1st and 2nd stories and 4.1 m for the others. The total height is 129.7 m, and the size of the plan is 50.4 m × 36.0 m as shown in *Fig. 1*. The structural system of the building is a steel moment-resisting frame with a center core, which is surrounded by buckling-restrained braces of hysteretic passive dampers. The slab is allocated except an elevator shaft in the center core.
All the members and floor slabs except the braces are modeled by 8-node hexahedral solid elements; the braces are modeled by truss elements connecting gusset plates. All the plate members, such as webs and flanges, are discretized into two-layer elements. Regarding a direction perpendicular to the thickness direction of the plate, the size of elements is about 70 mm near the beam-to-column connections. Studs connecting a flange and a slab are neglected in the model and element nodes are shared by the upper face of the flange and the lower face of the slab. The bottom of each column of the 1st story is connected to the elastic base beams, which have the same shape as beams under the 2nd floor but have the modulus of elasticity 5.5 times as large as those.
The model has 15,592,786 hexahedral solids, 78,686 rigid beams, 372 truss elements, 1,503,130 slave points on stick contact surfaces, 24,765,275 nodes and 74.29 million degrees of freedom. The steel material of the model has the mass density of 7.86×10^3 kg/m^3, the modulus of elasticity of 205 GPa, the Poisson ratio of 0.3, the yield stress of 330 MPa and the hardening ratio of 1/1000 for the bilinear model with kinematic hardening. For the reinforced concrete slabs, the material is assumed to be an elastic body with the modulus of elasticity of 22.7 GPa and the Poisson ratio of

0.3. The dead loads due to non-structural components and live loads are considered by increasing the mass density of the slabs.



(a) Vertical section (b) Floor plan

Fig. 1. Typical vertical section and floor plan

1.2 Result of a virtual shaking-table test

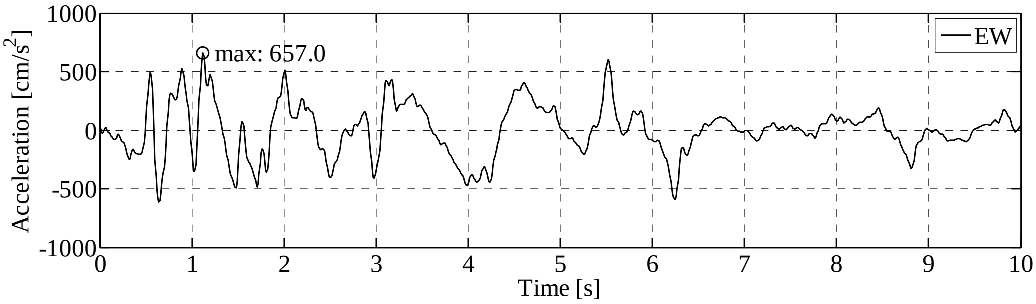
The eigenvalue analysis has been conducted. The natural frequencies up to the 6th mode are 0.3074, 0.3484, 0.3822, 0.9688, 1.0518 and 1.1763 Hz. The damping is modeled to be proportional to the initial stiffness with the damping factor of 2% for the fundamental frequency, i.e. the coefficient to the initial stiffness matrix is 0.02071. The massively parallel computer T2K in the University of Tokyo, of which CPU is AMD Quad Core Opteron 2.3 GHz, is utilized and 192 cores (24 node × 8 core/node) are used in total for the computation.

A 10-second segment of the JR Takatori record of the 1995 Hyogo-ken Nanbu Earthquake that includes principal strong motion is used as input ground motion (*Fig. 2*). After the static analysis of self-weight loading, the EW- and NS-components are applied to the longitudinal (*X*-axis) and the lateral (*Y*-axis) directions, respectively, and the UD-component to the vertical (*Z*-axis) direction.

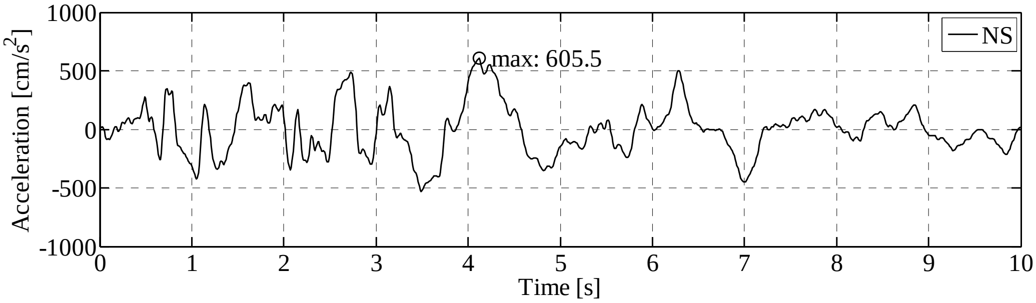
Considering elastoplastic large deformation, a time-history analysis is carried out to examine seismic response up to the 10 s. The average elapsed time for one step is 12,312 s when 192 cores of T2K are used. *Fig. 3* shows the deformation of *(a)* the whole building and *(b)* the stories with large stress around the 22nd floor at a time of 4.99 s; the deformation is scaled by a factor of 20 and the color contour of the equivalent stress is overlaid.

Fig. 4 shows the deformation of *(a)* the whole building and *(b)* the stories near the 1st floor at a time of 6.21 s, at which the displacement of the top of the corner column of the 31st story is a maximum. In *Fig. 4*, the deformation is scaled by a factor of 10 and the colors indicate equivalent plastic strain. It is observed that the lower parts of the first-story columns are yielding.

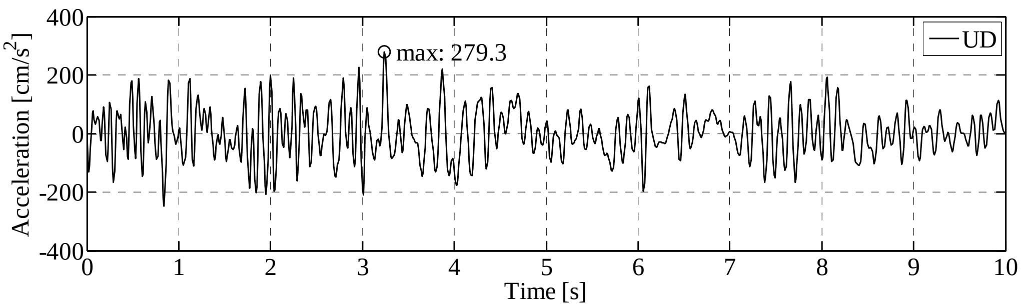
Fig. 5 shows the time history of the *Z*-component of the nodal stress in the center of *(a)* the northward surface and *(b)* the westward surface at the bottom of the X1-Y6 (the north-west corner, *i.e.* the upper-left corner in *Fig. 1*) column on the first story. The average value of the *Z*-component stress is positive because of the own weight of the building, but negative values are also observed. This is possibly due to bending deformation of the column along with the story deformation and due to the tensile sectional force in the column caused by the overturning motion of the whole building. The time history of the relative displacement responses is shown in the next section.



(a) EW-component

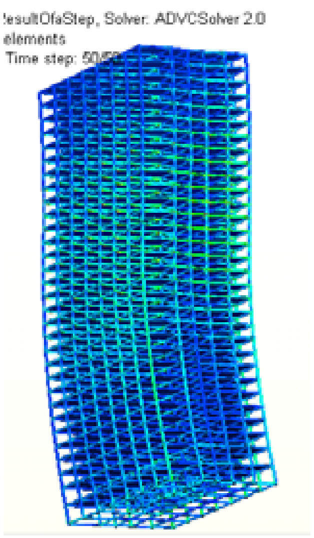


(b) NS-component

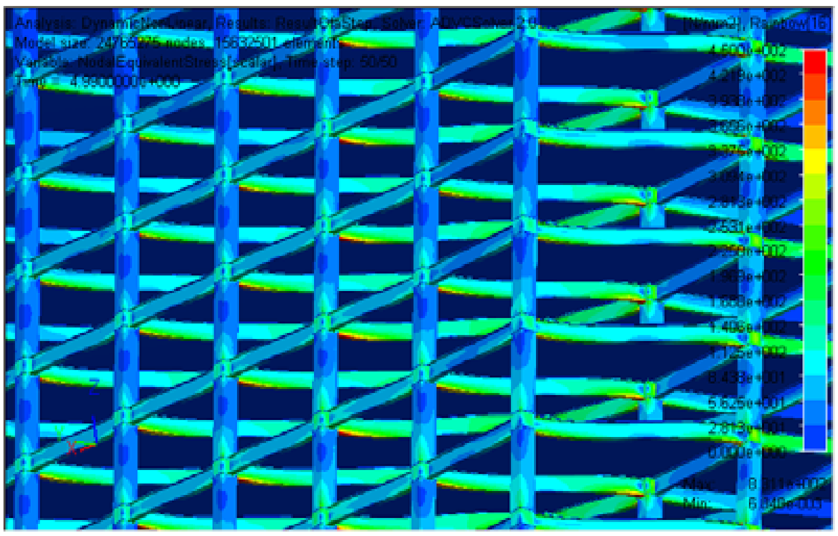


(c) UD-component

Fig. 2. Input ground motion

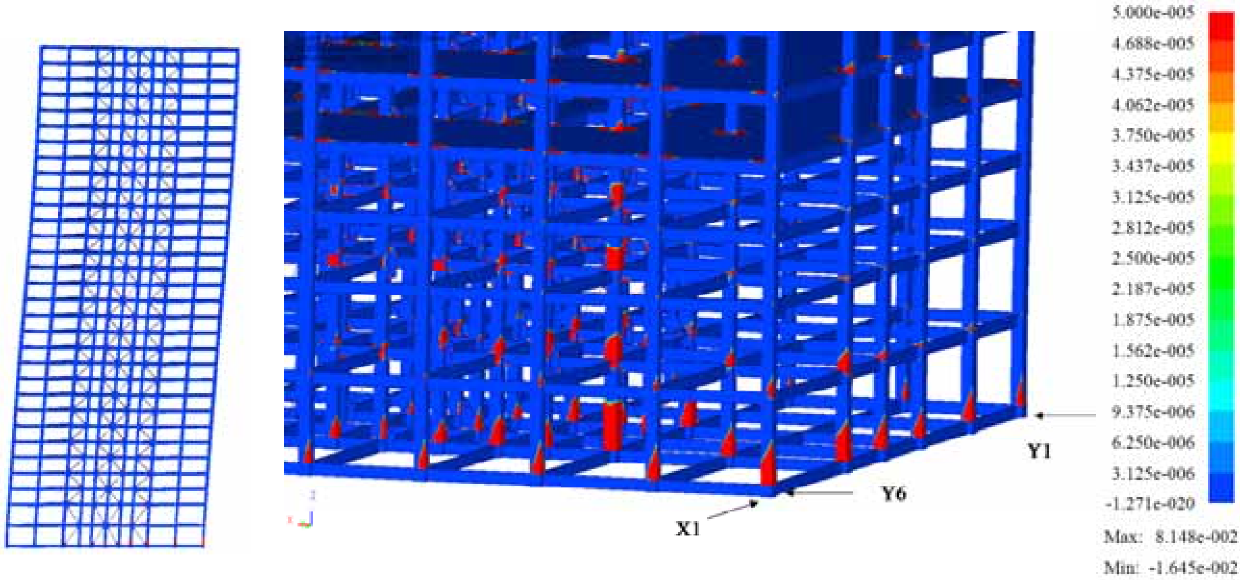


(a) The whole building



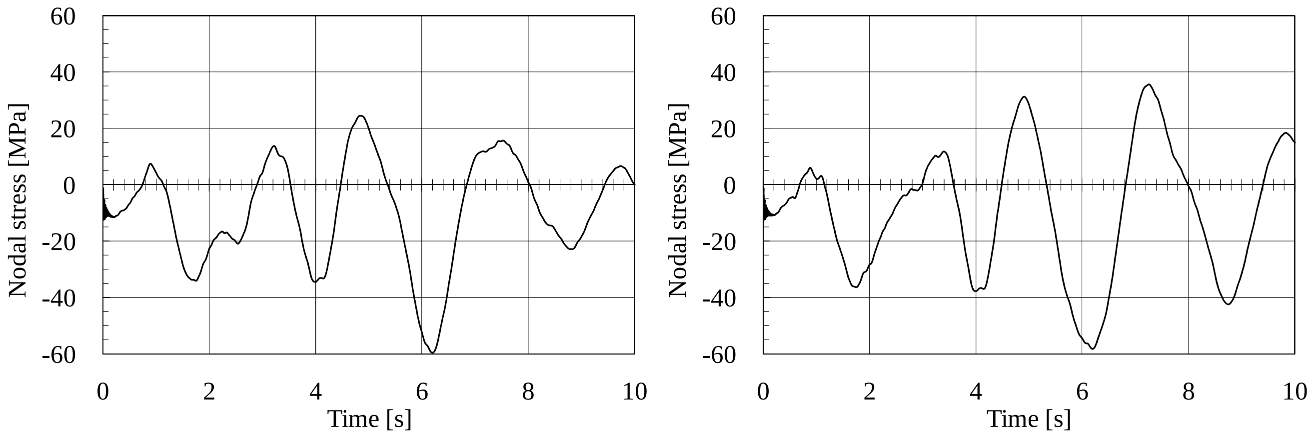
(b) The stories with large stress around the 22nd floor

Fig. 3. Deformation (scaled by a factor of 20) and distribution of equivalent stress at a time of 4.99 s



(a) The whole building *(b)* Stories near the 1st floor

Fig. 4. Deformation (scaled by a factor of 10) and distribution of equivalent plastic strain at a time of 6.21 s



(a) Northward surface *(b)* Westward surface

Fig. 5. Time history of the *Z*-component of the nodal stress in the center of *(a)* the northward surface and *(b)* the westward surface at the bottom of the north-west-corner column on the first story

2 BEAM ELEMENT ANALISYS

2.1 ASI-Gauss technique

To compare with the result of the FE-analysis with solid elements using the E-Simulator, the beam element model is next analyzed. The ASI-Gauss technique [4], which can reduce calculation cost by adaptively shifting numerical integration points based on material property, is employed for a elastoplastic seismic response analysis. The beam element model has 90,648 degrees of freedom. The elapsed time is 15,358 s for the computation of the 10-second input wave when two cores of a CPU of Intel Core2 1.86 GHz are used. The material properties of the model are the same as described in Section 1.1. However, undamped vibration is assumed for simplicity.

2.2 Result of an analysis based on ASI-Gauss technique

Fig. 6 depicts the deformation (not scaled) at a time of 5.2 s and the colors show the values of the yield function; the red color indicates yielding. The lower parts of the columns of the core in the 1st story and braces of truss elements exhibit yielding as observed in the E-Simulator model.

the time history of the relative displacement of the bottom of the corner column (the centroid of the section) at X1-Y1 (the lower-left corner in *Fig. 1*) on the 23rd and the 32nd (roof) floors are shown in *Fig. 7* and *Fig. 8*, respectively. The relative displacement of the beam element model is slightly larger than that of the E-Simulator model. This is likely due to the assumption of undamped vibration in the beam element analysis. Other possible reasons are model difference in slabs and an error due to modeling of a plastic region with a plastic hinge.

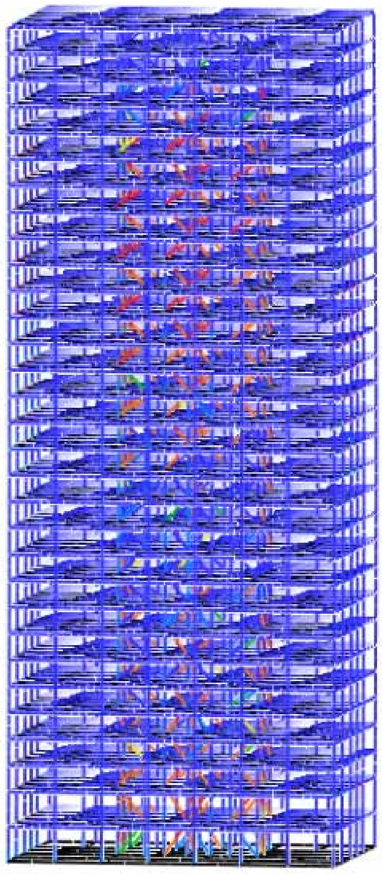
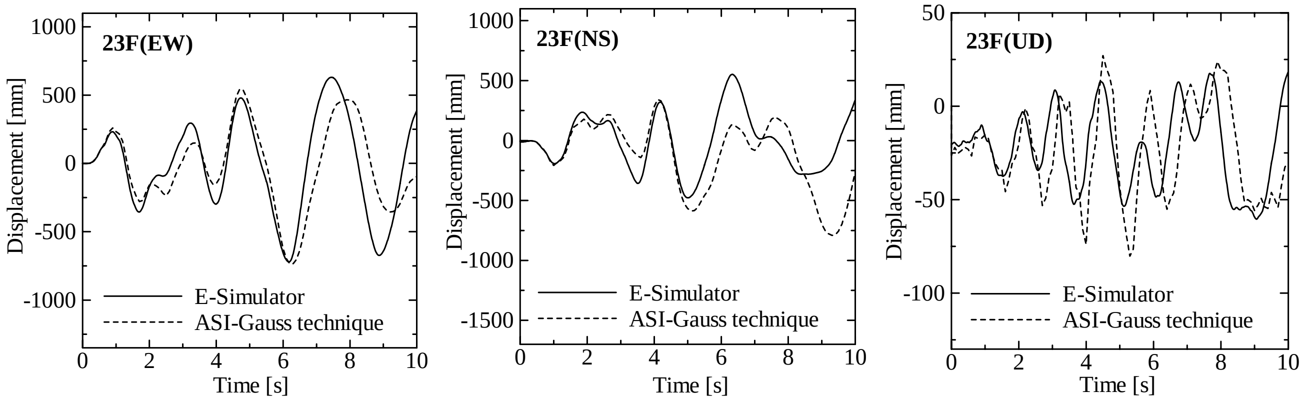
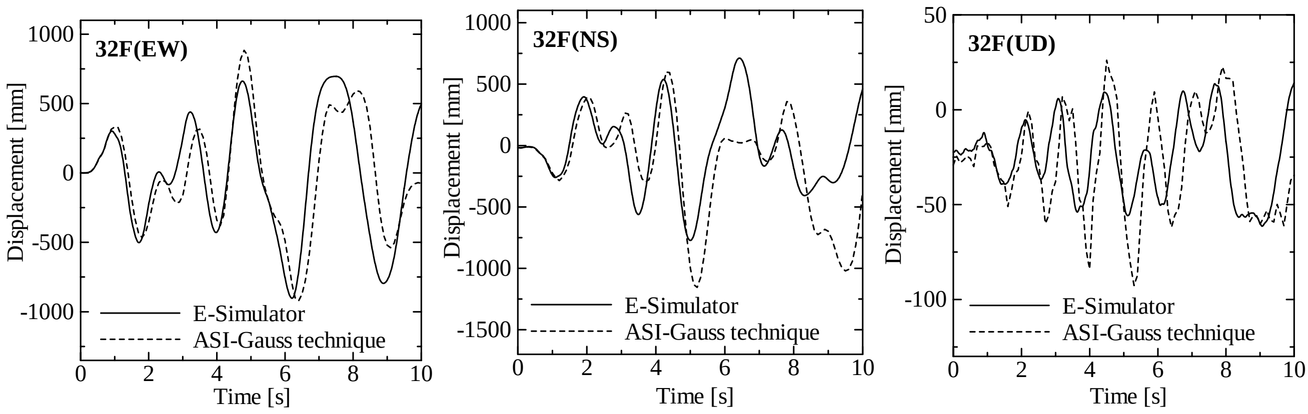


Fig. 6. Deformation (not scaled) and distribution of the plastic region analyzed by ASI-Gauss technique



(a) EW-component *(b)* NS-component *(c)* UD-component

Fig. 7. Comparison of the time history of relative displacements on the 23rd floor



(a) EW-component *(b)* NS-component *(c)* UD-component

Fig. 8. Comparison of the time history of relative displacements on the 32nd (roof) floor

3 CONCLUSIONS

A virtual shaking-table test of a 31-story high-rise office building has been carried out using the E-Simulator. The model consists of solid elements and the result of the E-Simulator analysis was compared with that of the beam element analysis based on the ASI-Gauss technique. Consequently, the response difference between the E-Simulator models and the beam element possibly arose from model differences in damping, slabs and modeling error of plastic hinge of the beam element model. In the future, it is expected to evaluate the accuracy of a plastic hinge in a macro model based on a virtual experiment using the E-Simulator.

4 ACKNOWLEDGMENTS

This study is part of the E-Defense Seismic Experimental Study and Simulation System Construction Project conducted by the E-Simulator Production Committee (Leader: Prof. Muneo Hori, The University of Tokyo). The authors acknowledge valuable contributions from the committee members, and financial support from NIED. The contributions of Mr. Kiyoshi Yuyama from Allied Engineering Corporation with regard to computation and mesh generation and of Mr. Yuta Arakaki from the University of Tsukuba with regard to analysis based on the ASI-Gauss technique are also gratefully acknowledged.

REFERENCES

[1] Hori, M, Noguchi, H, Ine, T, “Project report of development of numerical shaking table coping with E-Defense”, *Journal of Earthquake Engineering* (JSCE), Vol. 29, pp. 1420–1425 (in Japanese), 2007.

[2] Allied Engineering Corporation, http://www.alde.co.jp/english/advc/index.html, as of 2011/2/13.

[3] Ohsaki, M, Miyamura, T, Kohiyama, M, Hori, M, Noguchi, H, Akiba, H, Kajiwara, K, Ine, T, “High-precision finite element analysis of elastoplastic dynamic responses of super-highrise steel frames”, *Earthquake Engineering and Structural Dynamics*, Vol. 38, No. 5, pp. 635-654, 2009.

[4] Lynn, KM, Isobe, D, “Finite Element Code for Impact Collapse Problems of Framed Structures”, *International Journal for Numerical Methods in Engineering*, Vol. 69, No. 12, pp. 2538-2563, 2007.