International Journal of High-Rise Buildings
March 2015, Vol 4, No 1, 75-83

International Journal of
High-Rise Buildings
www.ctbuh-korea.org/ijhrb/index.php

Large-scale Seismic Response Analysis of Super-high-rise Steel Building Considering Soil-structure Interaction using K computer

Tomoshi Miyamura[1,2,†], Hiroshi Akiba[3], and Muneo Hori[4]

[1]*Department of Computer Science, College of Engineering, Nihon University, Koriyama 963-8642, Japan*
[2]*Hyogo Earthquake Engineering Research Center, National Research Institute for Earth Science and Disaster Prevention, Miki 673-0515, Japan*
[3]*Allied Engineering Corporation, Tokyo 135-0061, Japan*
[4]*Earthquake Research Institute, The University of Tokyo, Tokyo 113-0032, Japan*

Abstract

In the present study, the preliminary results of a large-scale seismic response analysis of a super-high-rise steel frame considering soil-structure interaction are presented. A seismic response analysis under the excitation of the JR Takatori record of the 1995 Hyogoken-Nanbu earthquake is conducted. Precise meshes of a 31-story super-high-rise steel frame and a soil region, which are constructed completely of hexahedral elements, are generated and combined. The parallel large-scale simulation is performed using K computer, which is one of the fastest supercomputers in the world. The results are visualized using an offline rendering code implemented on K computer, and the feasibility of using a very fine mesh of solid elements is investigated. The computation performance of the analysis code on K computer is also presented.

Keywords: Finite element analysis, Solid element, Super-high-rise steel building, K computer, Parallel computation, Seismic response analysis, Soil-structure interaction

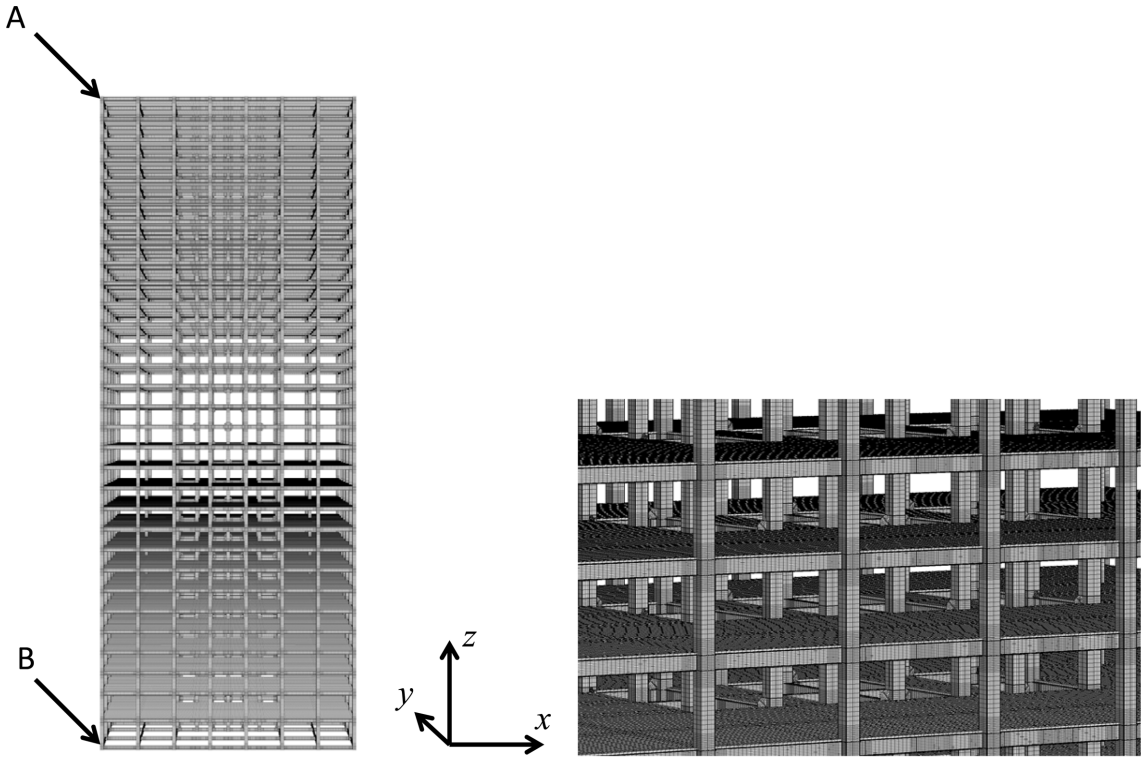
1. Introduction

Recent progress in massively parallel supercomputers enables us to conduct ultra-large-scale structural analyses. At the National Research Institute for Earth Science and Disaster Prevention (NIED), Japan, a parallel finite element software package, E-Simulator, is under development for precisely simulating collapse behaviors of civil and building structures under earthquakes (Hori et al., 2007). In the research group of E-Simulator, a hexahedral mesh with 74 million degrees of freedom (DOFs) of a 31-story super-high-rise steel frame (Fig. 1) was generated, and elastic-plastic seismic response analyses using the mesh were conducted successfully (Ohsaki et al., 2009; Miyamura et al., 2011). The results for the super-high-rise steel frame model were compared with those obtained by a frame model using an adaptive finite element code based on the Timoshenko beam element. (Isobe et al., 2013). Dynamic collapse analyses of a four-story steel frame have also been conducted using E-Simulator, and the code was validated by comparing the results of the simulation with the experimental results obtained through a full-scale shake-table test (Miyamura et al., 2015).

Recently, several studies on the application of parallel computing to the three-dimensional structural analyses of buildings can be found. Mizushima et al. used a detailed finite element model made of shell elements for the nonlinear seismic response analyses of steel frames (Mizushima et al., 2012). They used a dynamic explicit finite element analysis code on the supercomputer, Earth Simulator 2. Mabuchi et al. generated a mesh of a super-high-rise building and a soil region using solid elements and conducted an elastic seismic response analysis (Mabuchi et al., 2008). The number of total degrees of freedom of their model was 1.53 million, and the model was solved by a parallel implicit finite element analysis code in which the balancing domain decomposition method was used as a linear solver. Kanai et al. performed a seismic response analysis of a six-story RC building with surrounding soil using the dynamic explicit finite element analysis code on the supercomputer, Earth Simulator (Kanai et al., 2011).

As part of the next-generation supercomputer, "K computer" project in Japan (AICS, 2014), a research program, the HPCI Strategic Program Field 3, has been started (JAMSTEC, 2014). One research area in the program is earthquake engineering. E-Simulator has been ported and tuned especially for the K computer environment. In the present study, a preliminary simulation of a super-high-rise steel frame considering soil-structure interaction is conducted. Both the super-high-rise steel frame and a soil region are directly modeled using hexahedral solid elements. The seismic response analysis of the model under

[†]Corresponding author: Tomoshi Miyamura
Tel: +81-24-956-8829; Fax: +81-24-956-8863
E-mail: miyamura@cs.ce.nihon-u.ac.jp



(a) Entire frame (b) Close-up view

Figure 1. Mesh of a 31-story super-high-rise steel frame.

the excitation of the JR Takatori record of the 1995 Hyogoken-Nanbu earthquake is performed using E-Simulator on K computer. The remainder of the present paper is organized as follows. In Section 2, a brief overview of E-Simulator is presented. Section 3 describes the analysis model. Section 4 presents the preliminary results of the simulation. Section 5 describes the performance evaluation of the code on K computer. Concluding remarks are presented in Section 6.

2. E-Simulator

E-Simulator is a parallel finite element structural analysis code for virtual shake-table tests of civil and building structures, which enables large-scale analysis to be performed with a very fine mesh of solid elements. The main core of E-Simulator is the commercial software package, ADVENTURECluster (Allied Engineering Corporation, 2014; Akiba et al., 2006), which has been extended from the open-source version, the ADVENTURE system (ADVENTURE project, 2014; Yoshimura et al., 2002).

In ADVENTURECluster/E-Simulator, the Coarse Grid Conjugate Gradient (CGCG) method originally developed by Akiba et al. (Akiba et al., 2010) is used to solve large-scale linear algebraic equations in the implicit finite element method. The CGCG method is based on the hierarchical domain decomposition method, which is a two-level domain decomposition method. A mesh of an analysis domain is first divided into domains called Parts, and each Part is then subdivided into subdomains. In the CGCG method, a preconditioner is constructed by the motions of the decomposed subdomains. This preconditioner performs the same function as the coarse grid correction in the multi-grid method. E-Simulator is ported and tuned especially for the K computer environment. Each processor element (PE) of recent parallel computers including K computer is equipped with (a) multi-core processor(s), and the number of PEs is huge. Therefore, a hybrid parallel implementation technique using a combined MPI and OpenMP programming model is usually used and is adopted in E-Simulator. Using MPI, each Part is assigned to each node of K computer, which consists of eight cores, and each subdomain in a Part is assigned to a thread that exists in each core of the node and is processed in parallel. This multi-thread parallelization is implemented using OpenMP.

In ADVENTURECluster/E-Simulator, basic functions that are necessary for a general-purpose finite element analysis code are implemented. For implicit dynamic analysis, the Hilber-Hughes-Taylor time integration method (a-method) (Hughes, 2000) is implemented. Various finite elements, including an element with incompatible modes, are prepared. Multi-point constraints (MPCs) can be used, and a rigid beam element that connects both translational

and rotational degrees of freedom is also implemented using a set of MPCs. Several million MPCs can be taken into account. Standard inelastic constitutive equations are implemented in ADVENTURECluster. In E-Simulator, inelastic constitutive equations and rupture/fracture models that are particular to civil and building structures are developed and implemented as extended functions.

3. Analysis Model

The mesh of the super-high-rise steel frame shown in Fig. 1 was made for the seismic response analysis in Ref. (Miyamura, 2011) and is used in the present study. The steel frame is a center-core-type 31-story office building that was originally designed as a specimen for E-Simulator. The story height is 5.4 m for the first and second stories, and 4.1 m for the other stories. The total height is 129.7 m, and the size of the plan is 50.4 m × 36.0 m. Buckling-restrained braces as hysteresis passive dampers are located in the core and are modeled by truss elements in Ref. (Miyamura, 2011). However, these braces are omitted in the present study. The mesh has 15,592,786 elements, 24,765,275 nodes, and 74,295,825 DOFs. Plates such as the flanges and webs of beams are divided into at least two layers of solid elements in the thickness direction. Studs connecting the flanges of beams and a slab as well as steel bars in the slab are omitted in the model. The lower surface of the slab is directly connected to the upper surface of the flange. The size of each element in the longitudinal direction of a beam or a column is approximately 70 mm near the connections, whereas a coarser mesh is used for elements located far from the connections.

Fig. 2 shows a schematic diagram of the analysis model of the super-high-rise steel frame combined with the soil region for the seismic response analysis considering soil-structure interaction. The interaction between the soil and super-high-rise steel frame is considered by directly modeling the soil region as a three-dimensional finite element mesh. A mat slab is placed on the soil, and the frame is placed on the mat slab. The mat slab is not embedded in the soil so as to simplify the mesh generation.

Meshes of a mat slab and a soil region that are made of hexahedral solid elements are combined with the mesh of the super-high-rise steel frame shown in Fig. 1. The shapes of the mat slab and soil region are rectangular. The size of the mat slab is 51.0 m × 36.9 m × 3.9 m in the *x*-, *y*-, and *z*-directions, respectively. The size of the soil region is 1,000.0 m × 1,000.0 m × 100.0 m in the *x*-, *y*-, and *z*-directions, respectively. The meshes of the mat slab and soil region are generated by regularly dividing each rectangular solid with hexahedral elements. Each hexahedral element in the meshes has a cubic shape. The edges of the cube for the mat slab are 0.3 m in length, and the edges of the cube for the soil region are 2.0 m in length (Fig. 3(c)).

Figs. 3(a) and 3(b) show the elevation and a close-up view of the mesh, respectively. The number of elements is 28,363,862, and the number of nodes is 37,311,413. The total number of DOFs is 111,934,239. As shown in Fig. 3, the element sizes for the frame, mat slab, and soil region are very different. These meshes are assembled using multi-point constraints (MPCs) that constrain pairs of a master segment and a slave node. The total number of independent MPCs is 2,962,659.

The beams and columns of the frame are steel, and the slabs and mat slab are reinforced concrete. The material properties are shown in Table 1. Steel and reinforced concrete are assumed to be elastic-plastic materials represented by the von Mises yield criterion with kinematic hardening. Soil is assumed to be elastic. The density of each floor slab is increased by an amount equivalent to the floor loads.

4. Results of Preliminary Analysis

A preliminary analysis is conducted in order to demonstrate the feasibility of ultra-large-scale parallel computa-

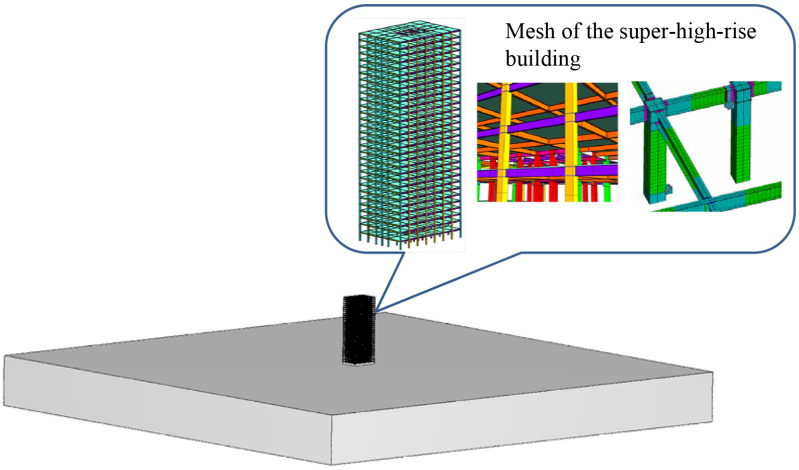
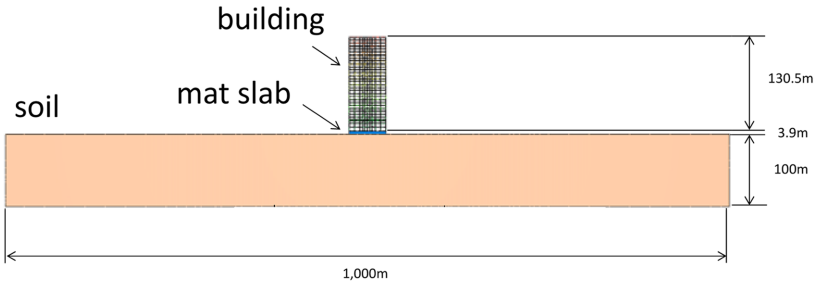
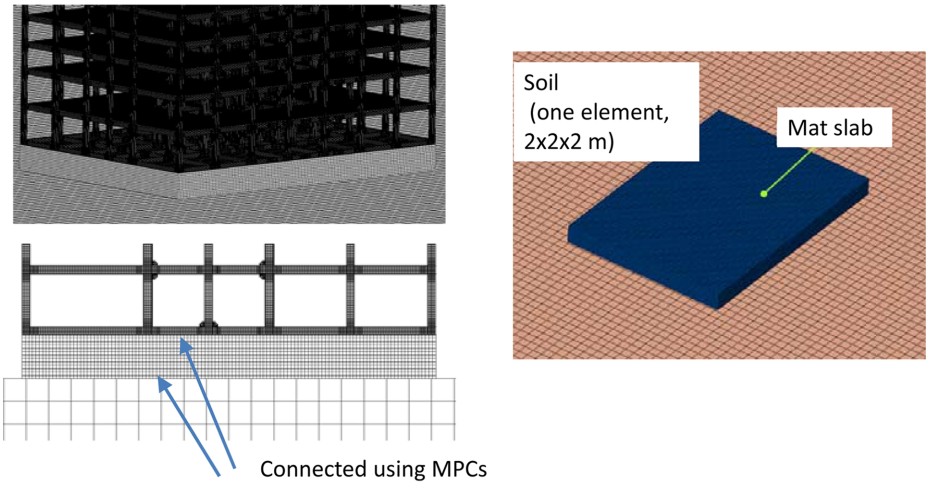


Figure 2. Analysis model.



(a) Elevation



(b) Details (c) Mat slab

Figure 3. Mesh of super-high-rise steel frame, mat slab, and soil region.

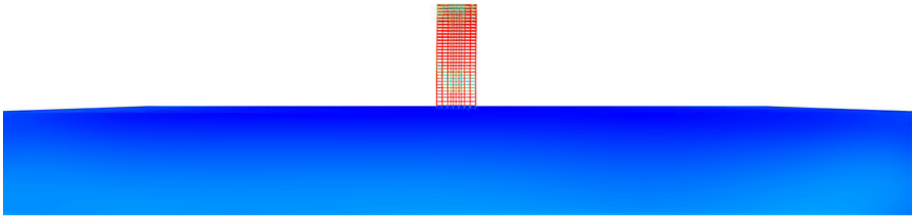
Table 1. Material properties

Steel frame	Young's modulus (kN/mm^2)	205
	Poisson's ratio	0.3
	Density (kg/mm^3)	7.86×10^{-6}
	Yield stress (N/mm^2)	330 (column) and 445 (beam)
	Hardening parameter (kN/mm^2)	0.205 (kinematic hardening)
Slab (reinforced concrete)	Young's modulus (kN/mm^2)	22.7
	Poisson's ratio	0.2
	Density (kg/mm^3)	4.90×10^{-6}
	Yield stress (N/mm^2)	20.0
	Hardening parameter (kN/mm^2)	0.0227 (kinematic hardening)
Soil	Young's modulus (kN/mm^2)	10.0
	Poisson's ratio	0.2
	Density (kg/mm^3)	2.60×10^{-6}

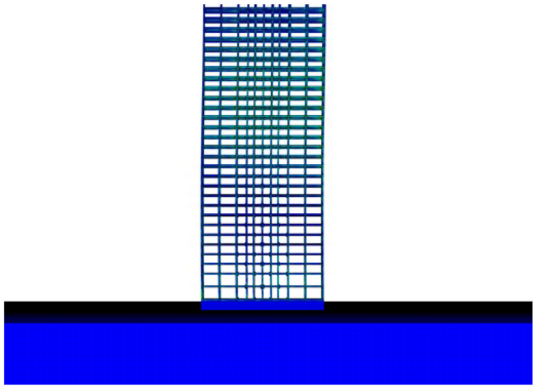
tion for the seismic response analysis of building structures considering soil-structure interaction, and the results are shown in the present section. The seismic response analysis under the excitation of the JR Takatori records of the 1995 Hyogoken-Nanbu earthquake is performed. Tentative boundary conditions are imposed on the mesh described in Section 3, i.e., the time history of the acceleration is prescribed homogeneously on the bottom of the soil region. The NS, EW, and UD components correspond to the *x*-, *y*-, and *z*-directions, respectively. Note that the NS and EW components correspond to the *y*- and *x*-directions, respectively, in Ref. (Miyamura et al., 2011), which is different from the settings of the present study. The free boundary condition is set on the lateral faces and top face of the soil region, although this condition is unnatural and tentative. The dead load due to the gravity is

applied by conducting the static analysis, and the dynamic analysis for simulating the seismic response is then started.

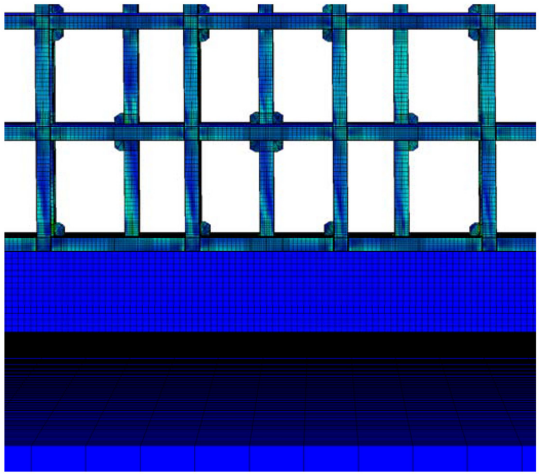
The simulation is performed on K computer. The computation time for 17 s of the seismic response analysis is approximately 18 days using 256 nodes (2,048 cores) of K computer. The time increment is taken to be 0.1 s and is reduced automatically when the Newton-Raphson method, which is used to solve the nonlinear equations, does not converge. Note that the performance evaluation of E-Simulator on K computer is described in Section 5. One of the key technologies for ultra-large-scale parallel finite element analysis is visualization. It is almost impossible to visualize the results of dynamic analyses using a graphic workstation because the amount of data output as the result is enormous and data transfer from a supercomputer to a graphic workstation takes a very long time. In the present study, a parallel offline (server side; software) rendering code (for example, Kawai et al., 2008) developed by Ogino (HDDMPPS project, 2014) is used for the visualization of the results. The code is developed using a visualization library called the VSCG library, which was developed by Wada (Wada et al., 2013; HDDMPPS project, 2014). The VSCG library can generate ultra-precise images without using a general graphics application programming interface (API) such as OpenGL. An example of the ultra-precise image is an image with 10K × 10K pixels. Since this code is also implemented on K compu-



(a) Entire analysis domain (blue: 0.0 Pa, red: 20 MPa).



(b) Lower stories of the building structure (blue: 0.0 Pa, red: 300 MPa).



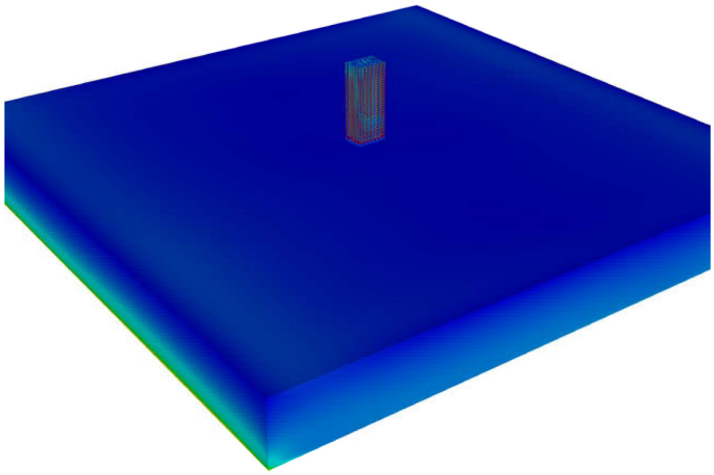
(c) Details of the building structure and mat slab (blue: 0.0 Pa, red: 300 MPa).

Figure 4. Deformed configuration with the contour of the equivalent stress (elevations).

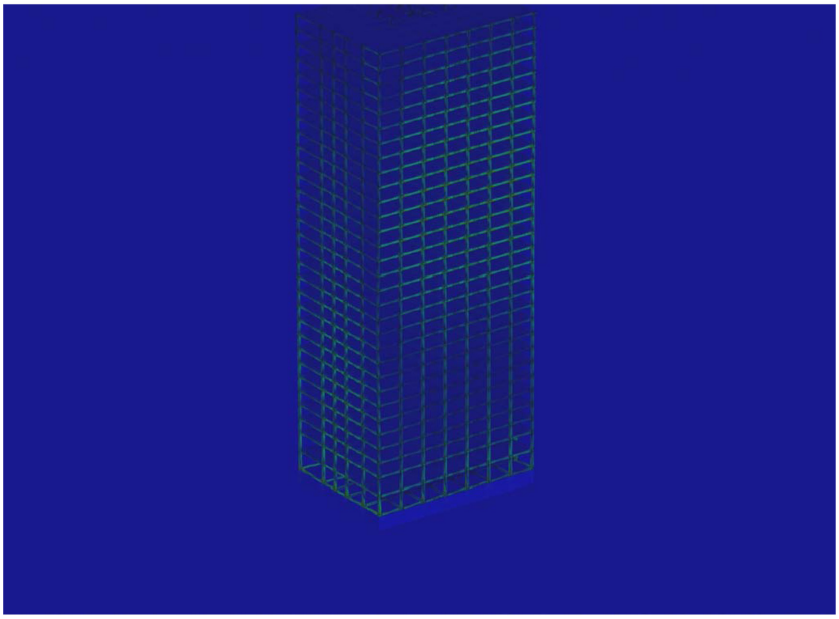
ter, the images for all time steps can be generated by a batch job on K computer and data transfer of the analysis results is not necessary. Only the ultra-precise images are transferred from K computer to a personal computer.

Figs. 4 and 5 show the deformed configuration and color contour of the equivalent stress in the seismic response analysis. Figs. 5(a) and 5(b) show perspective views. Note that a typical magnitude of the equivalent stress in the structure is much larger than that in the soil because the rigidity of the structure is larger than that of the soil. Therefore, appropriate assignment of a color to the magnitude of the stress in the visualization of the color contour is important. In Figs. 4(a) and 5(a), the color range is assigned to the range of 0 to 20 MPa in order to observe the distribution of the equivalent stress in the soil region. In this case, the structure is visualized almost entirely in red. On the other hand, the color range is assigned to the range of 0 to 300 MPa in Figs. 4(b), 4(c), and 5(b) in order to observe the distribution of the stress in the structure. Therefore, the soil region is visualized in blue. Note also that Figs. 4(b) and 4(c) are actually made from the same ultra-precise image, i.e., Fig. 4(c) is generated by simply magnifying the part of the image shown in Fig. 4(b).

Fig. 6 shows the time history of the relative displacements at node A shown in Fig. 1. The reference node to calculate the relative displacements is node B shown in Fig. 1. The time history of the relative displacements for the analysis model without the soil region can be found in Ref. (Miyamura et al., 2011). However, the direction of the input wave is different from that for the present simu-



(a) Entire analysis domain (blue: 0.0 Pa, red: 20 MPa).



(b) Details of the building structure and mat slab (blue: 0.0 Pa, red: 300 MPa).

Figure 5. Deformed configuration with the contour of the equivalent stress (perspective views).

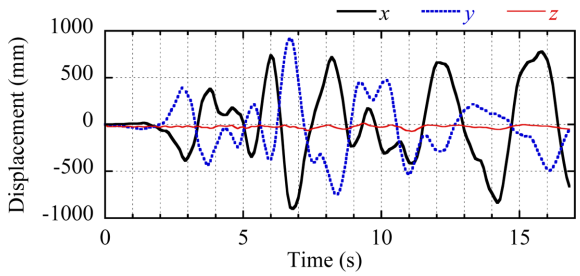


Figure 6. Time history of the relative displacements at node A (Fig. 1).

lation, as described at the beginning of this section. Note also that the analysis model of the super-high-rise steel frame is slightly different from that used in Ref. (Miyamura et al., 2011), i.e., the braces located in the core are omitted, as described in Section 3.

5. Performance Evaluation of E-Simulator on K Computer

The simulation in the present paper is conducted on K computer, which is one of the fastest supercomputers in the world. K computer has more than 80 thousand computation nodes, and each node has eight cores. However, the finite element analysis using the mesh with approximately 100 million DOFs is too small a problem to effectively use all of the nodes in K computer. In addition, it is difficult to tune the analysis code using an unstructured grid, such as a finite element mesh, as compared to numerical methods using a structured grid, such as the finite difference method. The procedures for the coarse grid correction in the CGCG method and for the incorporation of MPCs also affect the parallel performance because a direct solver is used to solve relatively small linear problems in these procedures. Therefore, parallel efficiency only scales up to approximately one thousand computation nodes. Note that the iterative solver does not converge without the coarse grid correction in the analysis of the steel frame that has slender members made of thin plates, and the use of MPCs to assemble meshes of members is necessary in order to reduce the cost of the mesh generation.

A common feature of recent parallel computers, including PC clusters and supercomputers, is that they consist of an enormous number of processing nodes or processor elements (PEs) with multiple cores. Therefore, the combined MPI and OpenMP programming model is often adopted, and E-Simulator adopts this programming model, as described in Section 2. In the present simulation, a thread is assigned to each of the eight cores in a computation node of K computer.

Table 2 shows the computation time and speed-up for the model without the mat slab and soil region. Table 3 shows those for the model with the mat slab and soil region. Note that the speed-up is defined as the computation time divided by the computation time using 120 nodes. The computation time and speedup are obtained for the following two procedures: A) entire procedure for the static analysis including a trial analysis, iterative steps for the Newton-Raphson method, and the output of results, and B) procedure of the linear solver for the first step of the Newton-Raphson method of the static analysis. Note that the domain decomposition of the analysis data, the input of the domain decomposed data, and the contact search to automatically generate the MPCs to connect several members are not included in Procedure A.

For both models, the speedup for Procedure B scales up

Table 2. Computation time and speedup (without mat slab and soil region)

	Number of node	120	240	360	480	600	960
Procedure A[*1]	Computation (elapsed) time (s)	12494	6362	6576	6061	5909	6334
	Speedup	1.000	1.964	1.900	2.062	2.115	1.973
Procedure B[*2]	Computation (elapsed) time (s)	3101	1783	1753	1560	1355	1158
	Speedup	1.000	1.739	1.769	1.988	2.288	2.679

[*1]Entire procedure for the static analysis including trial analysis, iterative steps for the Newton-Raphson method, and output of the results.
[*2]Procedure of the linear solver for the first step of the Newton-Raphson method of the static analysis.

Table 3. Computation time and speedup (with mat slab and soil region)

	Number of node	120	240	360	480	600	960
Procedure A[*1]	Computation (elapsed) time (s)	11669	7725	5812	6027	5919	6073
	Speedup	1.000	1.511	2.008	1.936	1.971	1.921
Procedure B[*2]	Computation (elapsed) time (s)	5119	3476	1879[*3]	2495	2250	2174
	Speedup	1.000	1.473	2.724	2.052	2.275	2.354

[*1]Entire procedure for the static analysis including trial analysis, iterative steps for the Newton-Raphson method, and output of the results.
[*2]Procedure of the linear solver for the first step of the Newton-Raphson method of the static analysis.
[*3]The CG method converges with small number of iterative steps in this particular case.

to 960 nodes. For Procedure A, however, the speedup is saturated for 480 or 360 nodes. One reason for this saturation may be that the output of the results is included in Procedure A. In order to overcome this problem, the amount of output data may be reduced by directly generating images of the results in the simulation code by combining the simulation code and the offline rendering code.

The performance for the model with the mat slab and soil region is worse than that for the model without them although the size of the mesh with them is larger than that without them. A reason may be that the computation cost to consider the numerous MPCs that connect the mat slab and soil region is expensive. In this case, the MPCs are not isolated but rather they have relationships with each other, and the surfaces of the mat slab and soil region to be connected by the MPCs are very wide. Therefore, the computation cost for the projection to incorporate MPCs into the CGCG method becomes very expensive.

6. Concluding Remarks

In the present study, a super-high-rise steel frame, a mat slab, and a soil region are modeled precisely using hexahedral solid elements, and a large-scale finite element mesh of the frame on the soil region is generated. A preliminary seismic response simulation using the mesh is performed on K computer, which is one of the fastest supercomputers in the world, using E-Simulator, which is a parallel finite element analysis code developed at NIED, Japan, for civil and building structures. The preliminary seismic response analysis under the excitation of the JR Takatori records of the 1995 Hyogoken-Nanbu earthquake is performed with tentative boundary conditions. The results of the simulation are visualized using an offline (server side; software) rendering code implemented on K computer. Ultra-precise images of the results are generated by a batch job of the rendering code on K computer. The results of a performance evaluation of E-Simulator on K computer are also presented. The results of the preliminary simulation demonstrate the feasibility of large-scale parallel finite element analysis using solid elements for the seismic response analysis of building structures considering soil-structure interaction.

Acknowledgement

The present study was supported by MEXT of Japan through a grant for research on HPCI Strategic Program Field No. 3. The parallel finite element analysis code, E-Simulator has been developed in the E-Simulator Development Committee in E-Defense at National Research Institute for Earth Science and Disaster Prevention (NIED) in Japan. The authors acknowledge the valuable contributions of the members of the committee. The authors acknowledge the technical support of Mr. Shuhei Takaya at Allied Engineering Corporation. The authors also acknowledge the effort of Prof. Masao Ogino at Nagoya University who conducted the visualization using his offline rendering code. The present study was also supported by JSPS KAKENHI (grant number 24560080).

References

ADVENTURE project. (2014). "ADVENTURE" Accessed on November 16, 2014 http://adventure.sys.t.u-tokyo.ac.jp/

AICS (2014). "Advanced Institute for Computational Science; K computer." Accessed on November 16, 2014 http://www.aics.riken.jp/en/

Akiba, H., et al. (2006). "Large Scale Drop Impact Analysis of Mobile Phone Using ADVC on Blue Gene/L." Proc. of the International Conference on High Performance Computing Networking and Storage. (SC06), Tampa, USA.

Akiba, H., Ohyama, T., and Shibata, Y. (2010). "CGCG Method for Structural Analysis and Its Enhancement." Proc. of ECT2010, Valencia.

Allied Engineering Corporation. (2014). "Parallel Structural Analysis System ADVENTURECluster." Accessed on November 16, 2014 http://www.alde.co.jp/english/index.html

HDDMPPS project. (2014). "LexADV" Accessed on November 16, 2014 http://adventure.sys.t.u-tokyo.ac.jp/lexadv/

Hori, M., Noguchi, H., and Ine, T. (2007). "Project Report of Development of Numerical Shaking Table Coping with E-Defense." *JSCE J. Earthquake Eng.*, 29, pp. 1420~1425. [in Japanese].

Hughes, T. J. R. (2000). The Finite Element Method - Linear Static and Dynamic Finite Element Analysis. Dover Publications, pp. 532~551.

Isobe, D., Han, W. S., and Miyamura, T. (2013). "Verification and Validation of a Seismic Response Analysis Code for Framed Structures Using the ASI-Gauss Technique." *Earthquake Engng. Struct. Dyn.*, 42(12), pp. 1767~1784.

JAMSTEC. (2014). "HPCI SPIRE Field 3." Accessed on November 16, 2014 http://www.jamstec.go.jp/hpci-sp/en/

Kanai, Y., Takeda, S., Narita, Y., Manabe, Y., Niwa, K., and Kasai, Y. (2011). "Analytical Study of Six-Story RC Building Considering Surrounding Soil using the Earth Simulator." Summaries of Technical papers of Annual Meeting Architectural Institute of Japan, B-1, pp. 229~230 [in Japanese].

Kawai, H., Ogino, M., Shioya, R., and Yoshimura, S. (2008). "Vectorization of Polygon Rendering for Off-line Visualization of a Large Scale Structural Analysis with ADVENTURE System on the Earth Simulator." *Journal of the Earth Simulator*, 9, pp. 51~63.

Mabuchi, S. et al. (2008). "Seismic Response Analysis of a System of Soil and Structure Using Parallel Processing." Proceedings of JSCE (Japan Society of Civil Engineers) Annual Meeting 2008, Paper No. 1-566.

Miyamura, T., Ohsaki, M,, Kohiyama, M., Isobe, D., Onda, K., Akiba, H., Hori, M., Kajiwara, K., and Ine, T. (2011). "Large-Scale FE Analysis of Steel Building Frames Using E-Simulator." *Prog. Nucl. Sci. Tech.*, 2, pp. 651~656.

Miyamura, T., Yamashita, T., Akiba, H., and Ohsaki, M. (2015). "Dynamic FE Simulation of Four-story Steel Frame Modeled by Solid Elements and Its Validation Using

Results of Full-scale Shake-table Test." *Earthquake Engng. Struct. Dyn.* (published online, DOI:10.1002/eqe.2526).

Mizushima, Y., Mukai, Y., Ohno, M., and Saruwatari T. (2012). "A Study on Strong Non-Linearity Analysis with Large-Scale and Detailed FE Models -Comparison of Dynamic Responses of Frame and Lumped Mass Models-." Proceedings of International Symposium on Earthquake Engineering, Japan Association for Earthquake Engineering (JAEE), 1, pp. 517~524.

Ohsaki, M., Miyamura, T., Kohiyama, M., Noguchi, H., Akiba, H., Hori, M., Kajiwara, K., and Ine, T. (2009). "High-Precision Finite Element Analysis of Elastoplastic Dynamic Responses of Super-Highrise Steel Frames." *Earthquake Engng. Struct. Dyn.*, 38, pp. 635~654.

Wada, Y., Kawai, H., and Shioya, R. (2013). "Development of High Resolution Visualization Library for Very Large Scale Analysis." Proc. of the Conference on Computational Engineering and Science, 18, Tokyo, Japan. [in Japanese].

Yoshimura, S., Shioya, R., Noguchi, H., and Miyamura, T. (2002). "Advanced General-Purpose Computational Mechanics System for Large Scale Analysis and Design." *J. Comput. Appl. Math.*, 149, pp. 279~296.