DEVELOPMENT OF EXTREMELY LARGE-SCALE ANALYSIS SOFTWARE (E-SIMULATOR: VIRTUAL SHAKING TABLE) FOR EARTHQUAKE DYNAMIC COLLAPSE ANALYSIS OF STRUCTURES

Tatsuhiko Ine[1], Muneo Hori[2],Hirohisa Noguchi[3],Makoto Ohsaki[4], Tomoshi Miyamura[5], Kenji Oguni[6], Masayuki Kohiyama[7], Koichi Kajiwara[8] and Hiroshi Akiba[9]

[1] *Invited research fellow, National Research Institute for Earth Science and Disaster Prevention, Tsukuba, Japan*
[2] *Professor, Earthquake Research Institute, The University of Tokyo, Tokyo, Japan*
[3] *Professor, Dept. of System Design Eng., Keio University, Yokohama, Japan*
[4] *Associate Professor, Dept. of Architecture and Architectural Eng., Kyoto University, Kyoto, Japan*
[5] *Lecturer, Dept. of Computer Science, College of Eng., Nihon University, Koriyama, Japan*
[6] *Associate Professor, Earthquake Research Institute, The University of Tokyo, Tokyo, Japan*
[7] *Associate Professor, Dept. of System Design Eng., Keio University, Yokohama, Japan*
[8] *Senior Researcher, National Research Institute for Earth Science and Disaster Prevention, Japan*
[9] *President, Allied Engineering Corporation, Tokyo, Japan*
Email: ine@bosai.go.jp, hori@eri.u-tokyo.ac.jp, noguchi@sd.keio.ac.jp, ohsaki@archi.kyoto-u.ac.jp, miyamura@cs.ce.nihon-u.ac.jp, oguni@eri.u-tokyo.ac.jp, kohiyama@sd.keio.ac.jp, kaji@bosai.go.jp, akiba@alde.co.jp

ABSTRACT :

This report presents a research project of developing “E-Simulator: virtual shaking table”, which is a simulation software based on a finite element method to analyze non-linear seismic response of constructions, as a part of E-Defense project. The key features of E-Simulator are 1) extremely large-scale numerical computation that realizes the finest temporal and spatial resolution of the structure response and 2) simulation of collapsing processes of structures that is comparable with the 3D full-scale model earthquake tests of E-Defense. This report explains the objectives of the E-Simulator and its two features. It also introduces the current state of E-Simulator project which includes the earthquake dynamic collapse analysis of 31-story steel framed structure and particle discretization scheme FEM for the RC structure.

KEYWORDS: Extremely Large-Scale Analysis, Earthquake Dynamic Collapse, FEM, E-Defense, E-Simulator

1. INTRODUCTION

As shown in Fig.1, the final goal of E-Simulator project is to develop a software environment that simulates global and local seismic responses of a city through integrated techniques such as structures, fracture models, numerical and multi-scale analyses. In doing this, we take full advantage of recent development of computer science and high-performance computing in computational mechanics to design an optimal quakeproof countermeasures. A general-purpose parallel Finite Element (FE)-analysis software based on the domain decomposition method, which is a platform of the E-Simulator, is applied to simulate dynamic responses of large-scale steel frames.
This report presents a research project of developing E-Simulator, a simulation software based on a finite element method to analyze non-linear seismic response of structures, as a part of E-Defense project. The key features of E-Simulator are 1) large-scale numerical computation that realizes finest temporal and spatial resolution of the structure response and 2) simulation of collapsing processes of

structures that is comparable with the 3D full-scale model earthquake tests of E-Defense. This report explains the objective of the E-Simulator and its two features. It also introduces the current state of E-Simulator.
As a part of this project, numerical results on dynamic collapse analysis of truncated 5-story framed models are shown to demonstrate that both global and local behaviors can be simulated by a high-precision FE-analysis. Eigenvalue analysis is also carried out for a 31-story frame to demonstrate the applicability of FE-analysis to a structure with more than 70 million DOFs. Furthermore, particle discretization scheme (PDS)-FEM is developed to be incorporated into E-Simulator for RC structure. And it will be planned to simulate 3D full-scale RC bridge pier model of E-Defense.

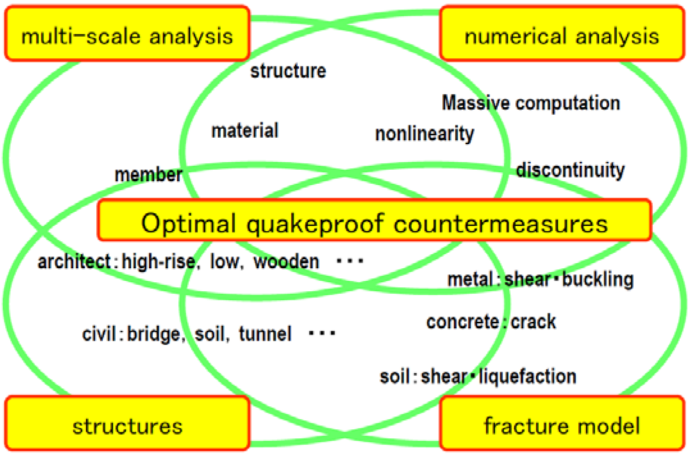


Figure 1. Overview of E-Simulator: integrated urban quakeproof techniques.

2. THE 2nd E-SIMULATOR PLATFORM

2.1. ADVC as the 2nd E-Simulator platform

Since 2001, the E-Simulator project has started at the Hyogo Earthquake Engineering Research Center that facilitates the world's largest "3D Full-scale Earthquake Testing Facility (E-Defense)" [Hori et al. 2007, E-Defense]. The 1st E-Simulator was developed as a seismic collapse simulator only for reinforced concrete constructions. This is nicknamed "JANIS" [E-Defense] which is an open source code of JAVA language by NIED. The 2nd E-Simulator project has been using ADVC [Akiba et al. 2006, ADVETURE] as a platform since 2006. ADVC is a commercial finite element package that is specially tuned for parallel computation on 64-bit architecture CPU's, and uses domain decomposition technique for parallel implementation. Its code is based on the ADVENTURE system. It can solve dynamic nonlinear problems with more than 10 million DOFs for a structure discretized by three-dimensional solid elements.

2.2. Overview of analytical method and hardware

ADVC's analysis function is highly enhanced. ADVC utilizes the Coarse Grid (CG) based Conjugate Gradient (CG) method (CGCG) developed by Suzuki et al. [Suzuki et al. 2002] to solve the stiffness equation on parallel environment. Contrary to other iterative sub-structuring methods, which commonly utilize the direct method to solve each subspace, CGCG method uses the parallel CG method on the entire solution space. A coarse grid motion of the sub-domains is superimposed into the CG iteration as a precondition operator. Performance and parallel efficiency depend essentially on the performance of the precondition operator. A rougher estimation of the global course grid motion results in higher parallel efficiency, since the simple parallel CG method without precondition has the highest parallel efficiency in the framework of preconditioned CG method. For this reason and based on our experience, the rigid body motion is chosen for the coarse grid motion in CGCG method.

Moreover, for contact analysis in general, structural analysis needs algorithms that can deal with MPC for constructing various elements and also for analyzing the assembled models. In ADVC, the subspace of the DOFs of the MPC is constructed in the global solution space, and then CGCG method is applied to this subspace. In dynamic analysis, implicit integration scheme based on the Newmark-β method is used with $\beta = 0.25$, $\gamma = 0.5$ (called trapezoidal rule), where CGCG method is applied on each time step. To build large FE-analysis models and to visualize large analysis results, the ADVC preprocessor and postprocessor are used, respectively.

In what follows, we use the ADVC for elasto-plastic dynamic analysis of steel frames. By making an analysis model with fine meshing, a complicated sequence of local buckling can be simulated for a steel frame. In the following examples, HP blade server BL465 equipped with 2.6 GHz dual core Opteron processor and PC2-5300 DDR2 DIMM 16GB memory is used for computation. Different numbers of cores and processes are taken for the numerical examples below.

3. SEISMIC ANALYSIS OF 31-STORY SUPER-HIGHRISE STEEL FRAMED MODEL

3.1. Design of 31-story super-highrise steel framed structure model

A 31-story steel building frame as shown in Fig. 2 has been designed as a specimen of E-simulator [Ohsaki et al. 2008]. The frame is a center-core-type 31-story office building. The story height is 5.4 m for 1st and 2nd stories, and 4.1 m for the other stories. The total height is 129.7 m, and the size of the framing plan is 50.4 m × 36.0 m. The buckling-restrained braces as hysteresis passive dampers are located in the building core as indicated by dotted lines in Fig. 2.

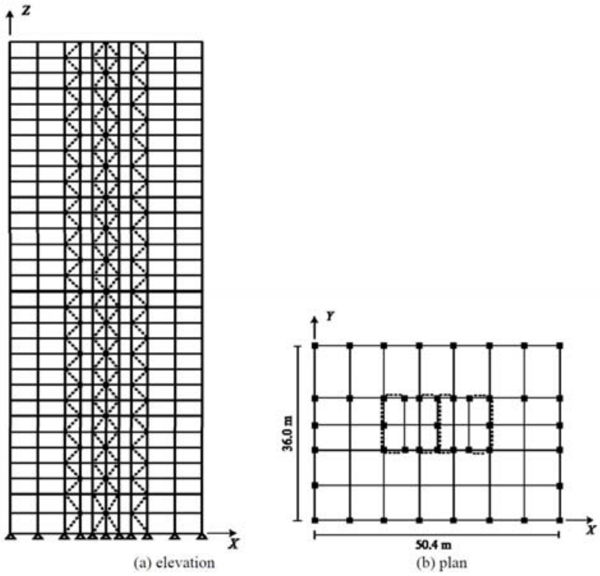


Figure 2. Locations of beams, columns and braces in typical elevation view and floor plan.

In order to simplify the model, the braces are neglected, or the stiffness of brace is replaced by additional stiffness of the beam and column in the following examples. Note that the main purpose of this paper is to demonstrate the applicability of high-precision FE-analysis using solid elements to simulate elasto-plastic dynamic responses of super-highrise steel frames. The frame is made of steel beam, column members and reinforced concrete floor slabs. The elastic modulus (E), yield stress and Poisson's ratio of the steel material are 205 kN/mm^2, 330 N/mm^2 and 0.3, respectively, and the isotropic plastic hardening modulus E/1000 is used. The RC floor slab is also assumed to be made of von-Mises (elasto-plastic) material with isotropic hardening rule, where the RC elastic modulus (Ec) is 22.7 kN/mm^2, RC Poisson's ratio is 0.2, the RC yield stress is 20 N/mm^2, and the RC isotropic hardening plastic modulus is Ec/1000. The mass density of steel is 7.86×10^3 kg/m^3, whereas the mass density of RC floor slab is increased by the density equivalent to the floor loads as shown in Table 1. Note that the thickness of the RC floor slab is 0.1275 m, and the area covered by the RC floor slab is 1645.92 m^2 for each floor. The foundation beams in 1st floor are assumed elastic and have the same sections as those in the 2nd floor; however, the elastic modulus (E) is 5.5 times as large as the standard value to represent the stiffness of the underground structure. The nodes in each column-base are connected by rigid beams to a node at the center of the column, which is pin-supported.

Table 1. Mass density of RC floor slab and the total floor load.

Floor	Mass density (× 10^3 kg/m^3)	Load (kN)
2	5.28	10858
3	5.09	10468
4–31	4.90	10077
32 (roof)	6.33	13018

3.2. Eigenvalue analysis of 31-story framed structure model

Eigenvalue analysis is carried out for the 31-story frame model shown in Fig. 1. To reduce the cost of mesh generation, the properties of the 11th to 31st stories, where no significant plastification is expected, are assumed to be the same as those of the 10th story. The elastic modulus, however is modified to approximate the original model in Fig. 1. Furthermore, the elastic modulus of beams, columns, RC floor slabs and foundation beams are scaled by 2.0 to consider the initial stiffness of the braces. The numbers of elements, nodes, and DOFs are 15,635,158, 24,783,633, and 74,350,770, respectively.

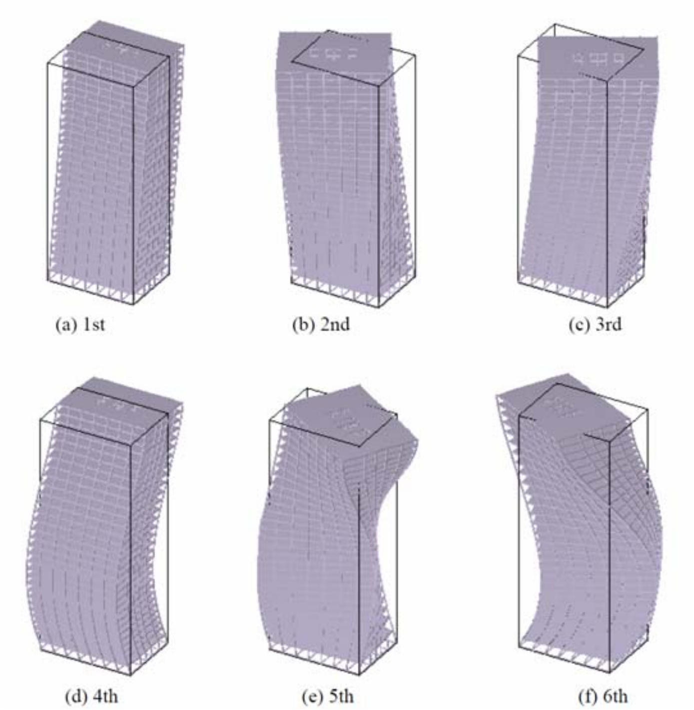


Figure 3. Eigenmodes of the 31-story framed model.

The six lowest natural frequencies and periods are listed in Table 2, and the corresponding eigenmodes are shown in Fig. 3. The natural periods of the 31-story model are larger than those of the 5-story model. The elapsed CPU time for eigenvalue analysis is 92,087 secs. (25.6 hrs.) with 32 cores. The 1st mode is a translational vibration in Y-direction, and the 2nd and 3rd modes are the mixture of torsion and X-directional translation. In this way, the eigenvalue analysis has been successfully carried out by the high-resolution FE-model. Obviously, the three-dimensional torsional vibration cannot be detected by the conventional method of dynamic analysis of lumped-mass model. Therefore, it is important to investigate the dynamic properties of super-highrise steel framed models based on three-dimensional effects.

Table 2. Natural frequencies and periods of the 31-story model

Mode	Frequency (Hz)	Period (sec.)
1	0.3704	2.6998
2	0.4235	2.3613
3	0.4596	2.1758
4	1.1035	0.9062
5	1.2346	0.8100
6	1.3392	0.7467

3.3. Seismic dynamic collapse analysis of 5-story framed structure model

3.3.1 Eigenvalue analysis of 5-story framed model

The next problem which we investigated was a truncated 5-story framed model of 31-story steel framed model with regard from the 1st to 5th story region. The mass above the 6th floor is represented by a lumped mass of 3.655 × 107 kg at the center of the 23rd floor. The fine mesh is used for the column-bases which are supported by foundation beams at the 1st floor. The total numbers of nodes, DOFs, and elements are 4,471,144, 13,413,300, and 2,849,802, respectively.

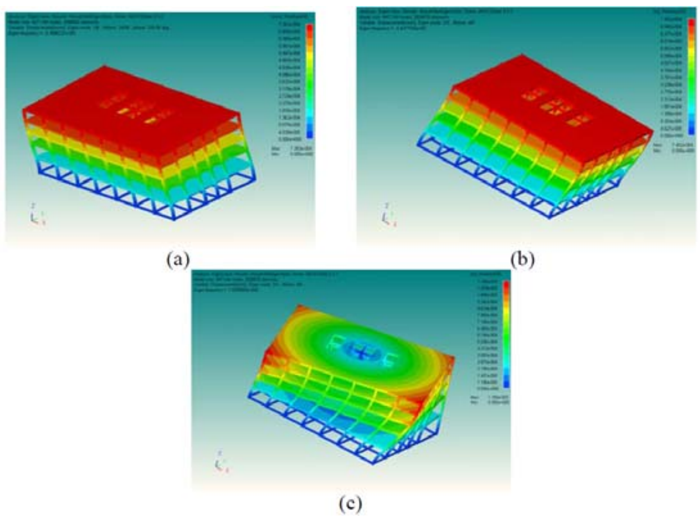


Figure 4. Three lowest eigenmodes of the 5-story model.

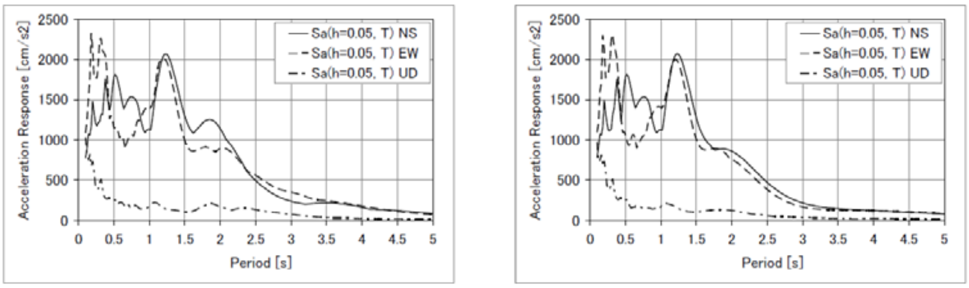
Table 3. Natural frequencies and periods of the 5-story model.

Mode	Frequency (Hz)	Period (sec.)
1	0.5369	1.8625
2	0.6327	1.5805
3	2.6081	0.3834
4	4.5625	0.2192
5	5.5357	0.1806
6	5.7173	0.1749

To simplify the model, the braces are neglected and the stiffness of braces is replaced by additional stiffness of the beams and columns etc., The elastic modulus of all the beams, columns, RC floor slabs and foundation beams are two times as large as those standard values defined in Section 3.1 to maintain the initial stiffness of the buckling-restrained braces. The six lowest natural frequencies and natural periods are listed in Table 3, and the three lowest eigenmodes are shown in Figs. 4(a)-(c). The 1st and 2nd modes are translational vibration in X- and Y-directions, respectively, and the 3rd mode is a torsional mode. The elapsed CPU time for eigenvalue analysis is 25,585 secs. (7.106hrs) with 16 cores of HP blade server BL465.

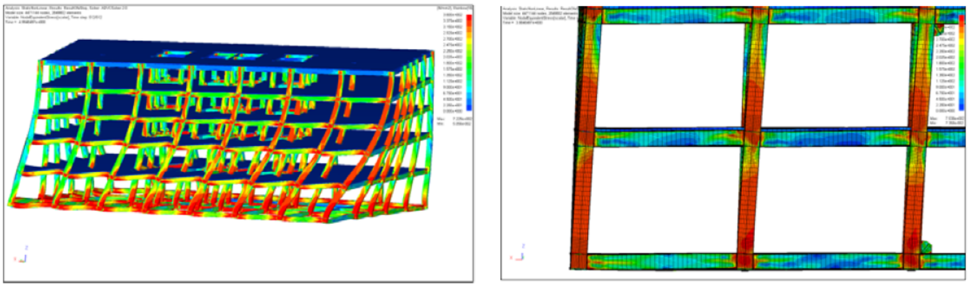
3.3.2 Dynamic collapse analysis of 5-story framed model

Before dynamic collapse analysis is carried out, the application of gravity load was conducted by static analysis. Then elasto-plastic dynamic response analysis is carried out for such a 5-story framed model under scaled recorded seismic motion. The input motions in X-, Y- and Z-directions are the EW-, NS- and UD-components, respectively, of the 1995 Hyogoken-Nanbu Earthquake Takatori wave scaled by 2.0 [Nakamura et al. 1995].



(a) duration: 40 sec. (b) duration: 5 sec.

Figure 5. Acceleration response spectra of the Takatori wave.



(a) (b)

Figure 6. Distribution of von-Mises stress at the maximum deformation response

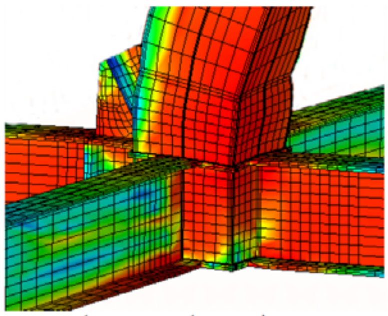
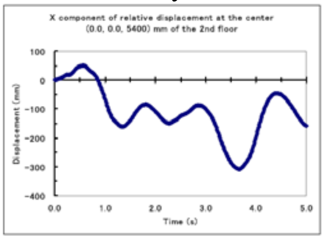
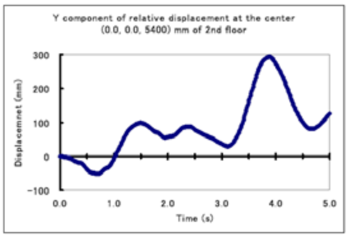


Figure 7. Deformation near the building core at the maximum deformation response

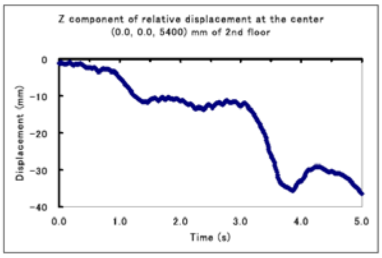
The prescribed acceleration is applied at the pin-support of each column-base. The elapsed CPU time for one step with 32 cores of HP blade server BL465 is 4.99 sec.; i.e., more than 5 months will be needed for computation for the duration of 40 seconds. Fig. 5(a) shows the acceleration response spectra for the three components of Takatori wave of full duration (40 sec.). The response spectra for the waves of duration 5 seconds from 2 sec. to 7 sec. of the Takatori wave are also shown in Fig. 5(b). As will be seen from these two figures that analysis of 5 seconds will be enough to predict the maximum response under Takatori wave. Since the fundamental natural period of the 5-story frame after plastification will be larger than 2 sec., and the response acceleration will be less than 150 cm/s^2, the input wave of 5 sec. is scaled by 2.0 to simulate dynamic collapse behavior of the frame using implicit integration scheme. The stiffness-proportional damping is used with damping ratio 0.02 for the 1st mode. The elapsed CPU time for the analysis of 511 steps is 551.5 hrs. (23 days). Note again that the purpose of this research is to demonstrate whether local and global collapse behaviors under seismic excitations can be simulated by high-precision FE-analysis [Fig. 6-8]. Therefore, the details of response quantities such as maximum stress, equivalent plastic strain, etc., are not discussed in the following investigation of the analytical results.



(a) *X*-direction



(b) *Y*-direction



(c) *Z*-direction

Figure 8. Relative displacements at the center of 2nd floor.

4. ANALYSIS OF REINFORCED CONCRETE STRUCTURE (PDS-FEM)

One of the features of E-Simulator is the collapse process simulation.In the computational solid mechanics, the simulations of the brittle and ductile fracture including the singularity causing the discontinuity of stresses, displacements and strains are the classical mechanics problems where various analytical techniques such as XFEM, discontinuous Galerkin method, mesh-free finite element method and joint element method are proposed [Hori et al. 2007]. From the mathematical point of view, however, it is very difficult to simulate the fracture mechanics problems. When premising smooth displacement functions, it is unreasonable for smooth displacement function to deal with the discontinuity and singularity even if various ideas are taken. Actually for such many proposed analytical techniques, calculation cost and elapsed CPU time are so huge.

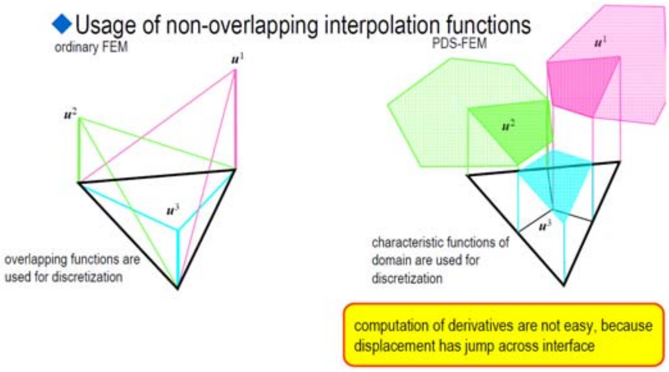


Figure 9. Comparison between ordinary linear FEM and PDS-FEM in terms of two shape functions

The E-Simulator can solve the above mentioned problems using PDS-FEM [Hori et al. 2005] that adopts a new particle discretization scheme. The method deals with a constant function as base functions of an element, so the discretization function represents discontinuities in the element. Because the derivatives of discontinuous discretization function can be calculated exactly, the mathematical singularity can be obtained easily. Actually, it is obvious that the linear finite element method based on PDS-FEM has the same accuracy as the ordinary linear FE analysis. This method can locate the discontinuous region such as cracks and shear strain band to the inter-element boundary, because that discretized displacement function is discontinuous. Therefore, it is easy to calculate the occurrence and the evolution of the crack. In future, PDS-FEM will be incorporated into ADVC and will be able to simulate a fracture process more effectively than ordinary FE analysis.

5. CONCLUDING REMARKS

The platform of the E-Simulator, a general-purpose parallel FE-analysis software based on the domain decomposition method, is applied to simulate dynamic collapse behavior of large-scale steel frames. Numerical results on a truncated framed model of the 31-story frame are shown. For verification purpose of the mesh generation, analytical algorithm and software, elasto-plastic dynamic analyses are carried out only for the 5-story models without buckling-restrained braces, Also, eigenvalue analyses are carried out for framed models including the 31-story framed model.
The numerical examples show that local buckling can be successfully simulated by high-precision FE-model. In this way, the local elasto-plastic responses will be simulated by FE-analysis without resorting to assumptions on nonlinear member behavior such as plastic hinge beam element model, etc.
Furthermore, PDS-FEM is proposed for the seismic collapse RC analysis. In future, PDS-FEM will be incorporated into ADVC and will be able to simulate a fracture process more effectively than ordinary FE analysis.

ACKNOWLEDGEMENTS
This study is a part or E-Defense Seismic Experimental study and Simulation System Construction Project conducted by The National Research Institute for Earth Science and Disaster Prevention (NIED) and its E-Simulator Product Committee (Leader: Hirohisa Noguchi, Keio University). The authors acknowledge the valuable contribution from the committee members and the Director Dr. Masayoshi Nakashima at the Hyogo Earthquake Engineering Research Center who gave a great financial support.

REFERENCES
1. Hori M, Noguchi H, Ine T, Project report of development of numerical shaking table coping with E-Defense, JSCE Journal of Earthquake Engineering 2007; 29:1420-1425.(in Japanese)
2. Hyogo Earthquake Engineering Research Center (E-Defense), http://www.bosai.go.jp/hyogo/ehyogo/index.html
3. Akiba H et al., Large scale drop impact analysis of mobile phone using ADVC on Blue Gene/L, Proc. International Conference for High Performance Computing Networking and Storage (SC06), Tampa, 2006.
4. ADVENTURE project website, http://adventure.sys.t.u-tokyo.ac.jp
5. Suzuki M, Ohyama T, Akiba H, Yoshimura S, Noguchi H. Development of fast and robust parallel CGCG solver for large scale finite element analyses, Transactions of the Japan Society of Mechanical Engineers, Series A, 2002; 68:1010-1017.. (in Japanese)
6. Ohsaki M et al., High-precision finite element analysis of elastoplastic dynamic responses of super-highrise steel frames, submitted to Earthquake Engngineering and Structural Dynamics, 2008
7. Nakamura Y, Uehan F, Inoue H. Waveform and its analysis of the 1995 Hyogo-Ken-Nanbu Earthquake information, Report No. 23d, Railway Technical Research Institute, 1995. (in Japanese).
8. Hori M, Oguni M and Sakaguchi H, Proposal of FEM implemented with particle discretization for analysis of failure phenomena, Journal of the Mechanics and Physics of Solids, 53, 3, 681-703, 2005