# Seismic response analysis of super-high-rise steel building considering soil-structure interaction using fine mesh of solid elements

**\* Tomoshi Miyamura[1, 3], Shuhei Takaya[2], Takuzo Yamashita[3], Hiroshi Akiba[2],**

**and Muneo Hori[4]**

[1]Department of Computer Science, College of Engineering, Nihon University, 1 Nakagawara, Tokusada, Tamura-machi, Koriyama 963-8642, Japan
[2]Allied Engineering Corporation, Toyosu Front, 3-2-20 Toyosu, Koto-ku, Tokyo 135-0061, Japan
[3]Hyogo Earthquake Engineering Research Center, National Research Institute for Earth Science and Disaster Prevention, 1501-21 Nishikameya, Mitsuda, Shijimi, Miki 673-0515, Japan
[4]Earthquake Research Institute, The University of Tokyo, 1-1-1 Yayoi, Bunkyo-ku, Tokyo 113-0032, Japan

*Corresponding author: miyamura@cs.ce.nihon-u.ac.jp

A parallel finite element analysis software package, the E-Simulator is under development at NIED (National Research Institute for Earth Science and Disaster Prevention), Japan. A mesh that consists of hexahedral elements with 74 M DOFs of a 31 story super-high-rise steel frame was made, and the precise elastic-plastic seismic response analysis was conducted successfully using the E-Simulator.
In the project of developing the K Computer, the HPCI Strategic Program (Field 3) was started, and a preliminary simulation of the super-high-rise steel frame considering soil-structure interaction has been conducted in the program. The soil region is modeled directly as a hexahedral mesh and is assumed to be elastic. In this study, the seismic response analysis of the model under the JR Takatori record of the 1995 Hyogoken-Nanbu earthquake is performed on the K computer using the E-Simulator. Computation performance of the analysis code on the K computer is also presented.

**Keywords:** Finite element analysis, Solid element, Super-high-rise steel building, K-Computer, Parallel Computation, Seismic response analysis