

## TIME DOMAIN RESPONSE ANALYSIS FOR ASSEMBLY BY INTEGRATING COMPONENTS

Norihiro Nakajima<sup>1</sup>, Akemi Nishida<sup>2</sup>, Yoshiaki Kawakami<sup>3</sup>, Yoshio Suzuki<sup>2</sup>, Keisuke Matsukawa<sup>4</sup>,  
Masami Oshima<sup>5</sup> and Hisao Izuchi<sup>4</sup>

<sup>1</sup>Deputy Director, Japan Atomic Energy Agency, JAPAN

<sup>2</sup>Senior Researcher, Japan Atomic Energy Agency, JAPAN

<sup>3</sup>Technical Engineer, Japan Atomic Energy Agency, JAPAN

<sup>4</sup>Technical Expert, Chiyoda Cooperation, JAPAN

<sup>5</sup>Technical Specialist, Chiyoda Cooperation, JAPAN

### ABSTRACT

The Center of Computational Science and e-systems in Japan Atomic Energy Agency has been researching and developing a digital shaking table, which is based on a finite element analysis for structure of assembly. The digital shaking table is introduced to carry out numerical experiments for the so called “structure” of a petroleum plant. In numerical experiments, “structure” was precisely modelled as it is designed and meshed into fine finite elements. The components of “structure” were meshed one by one, and the code of a finite element analysis for structure of assembly gathered every meshed components to run time domain response analysis. Four waves are applied to the analysis to determine its behaviour. Four waves are namely as El Centro, Taft, Hachinohe, and Geiyo. The results of experiments are discussed by comparing accumulating data in the past. It is concluded to reconfirm the methodology of gathering meshed components and a finite element analysis for structure of assembly with the “structure”.

This research used computational resources of the K computer provided by the RIKEN Advanced Institute for Computational Science through the HPCI System Research project (Project ID:hp120291, hp130029, hp140206).

### INTRODUCTION

The Center of Computational Science and e-systems (CCSE) in Japan Atomic Energy Agency (JAEA) has been researching and developing a digital shaking table as shown in Figure 1, to carry out simulations of various evaluations and analyses. For example, a nuclear facility loaded by an earthquake during operation, which is neither not realistic nor desirable to experiment. See, for example, Nakajima et al. (2004) and Nishida et al. (2006). The objective of a digital shaking table is to provide a better and more logical circumstance to analysis of nuclear facilities’ structural robustness.

Figure 1 shows a human-machine interface of the suggested digital shaking table, which consists of a user interface and a toolbox containing databases and simulation capabilities. In Figure 1, so called a “structure” of a petroleum plant is appeared in the view of the digital shaking table. In the user interface, the user can designate a seismic response analysis, a structural analysis, natural frequency analysis and so on by the selection of various codes, such as FIESTA (Finite Element Analysis for Structure of Assembly), Midas Gen, NASTRAN, ABAQUS, ANSYS and FINAS. See Tani et al. (2007), Midas\_Gen, Russell and Cohn (2012), Fish and Belytschko (2008), CAD/CAE committee (2005) and FINAS. Also, free wares such as Adventure and FrontISTR are users’ choice. See Adventure and FrontISTR. The load as exogenous factors can be given by either observed or simulated data, through the interface and toolbox of the digital shaking table.

In this paper, the digital shaking table is operated by assembly structural analysis (ASA) code, the FIESTA package, which executes FE (Finite Element) analysis by gathering components. See Nakajima et al. (2013, 2014).

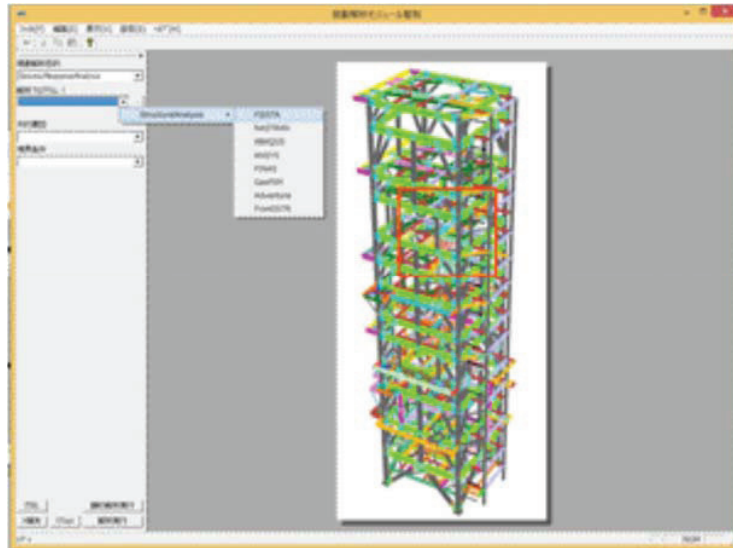


Figure 1. Analysis by ASA on Simulation in a digital shaking table.

#### **FIESTA (Finite Element Analysis for Structure of Assembly)**

FIESTA is a structural analysis code for assembly structure, based upon the finite element method (see, for example, Zienkiewicz and Taylor (2000)). General speaking in a structural analysis based upon the finite element method, a model of assembly like in Figure 2 can be modelled into 4 models as illustrated in Figure 3. An assembly may simplify to just a beam with one dimensional elements as shown in Figure 3 (a). It could be a tube with either the thin wall like (b) or thick wall as (c). These models can be created by a professional analyst, but a designer may prefer as is model shown in (d). Of course, a professional analyst may generate a like as is model of (d) neglecting bolts and nuts. When the analysis performed with the models like (a), (b) and (c), there is no consideration necessary for an assembly. Models, (a), (b) and (c) are just a uniform structure and a continuous model for FE analysis. Model (d) can be just a uniform structure and a continuous model. Assuming two flanges faces are just contacted by tighten with eight bolts, which means not a solid structure, model (d) is an assembly. A very simplified image of difference between a uniform structure and an assembly in FE meshed mode is illustrated in Figure 4.

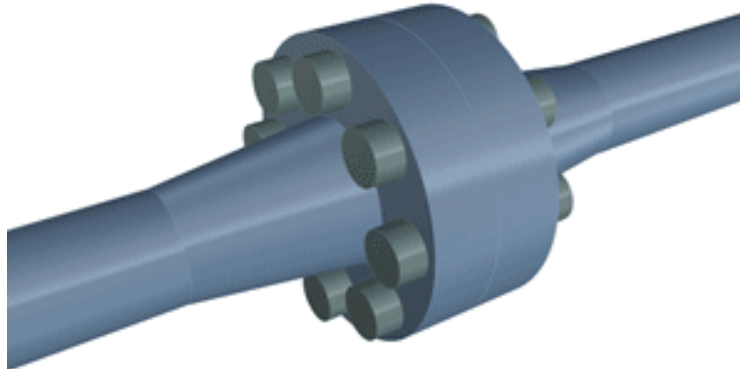


Figure 2. A model of assembly.

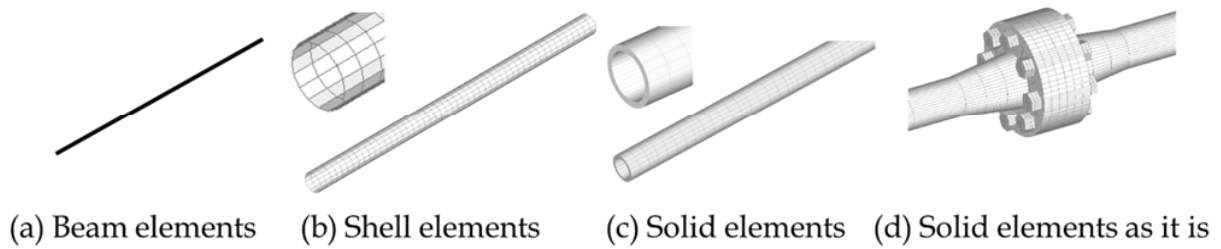


Figure 3. FE analytical models for an assembly.

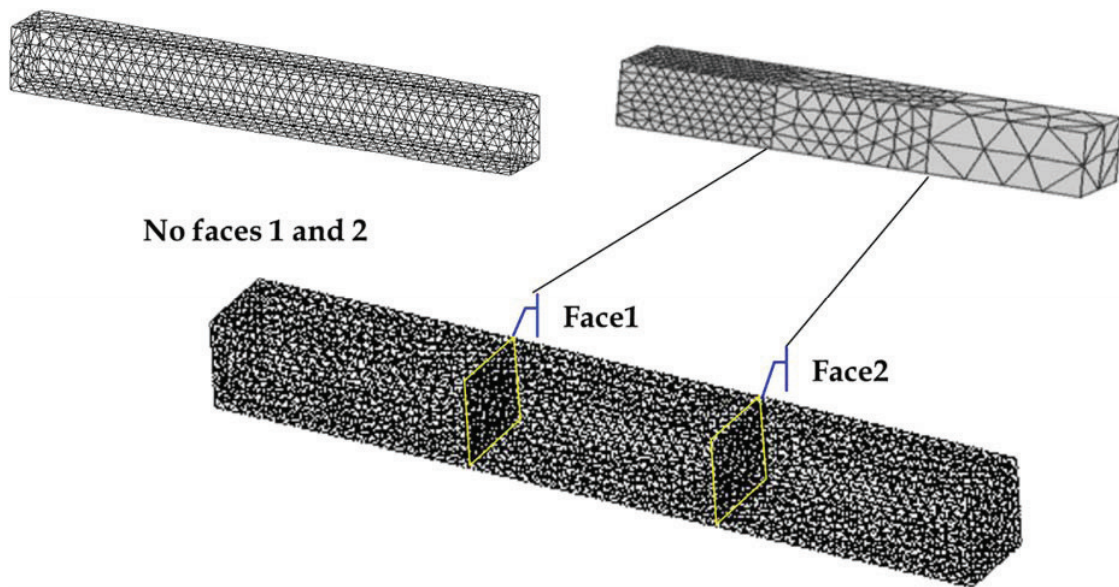


Figure 4. A simplified image of difference between a uniform structure and an assembly in FE meshed model.

To concern the condition among components in assembly, FIESTA is step wisely developed. Two steps have been reported in the past. The level 1 is to connect FE meshes between components by penalty method as shown in Figure 5. See Nakajima et al. (2013). By using penalty method, it can treat the condition of attached faces among components. The level 2 is to merge nodal points of FE meshes between components as shown in Figure 6. See Nakajima et al. (2013). This helps to establish a meshed model for an assembly, which has thousands of components. Due to the limitation of hardware resources, a complex structure like an assembly always meet a trouble to create a continuous FE meshed model. When the faces among components are needed to be concerns the condition of interaction among components, the nodes on the faces are only related to each other. When a continuous model is required, the nodes on the faces are merged to be a unique nodal point, otherwise, they have unique nodal number with the same coordinate. In this paper, the level 2 methodology is examined with industrial data and reconfirmed practicality.

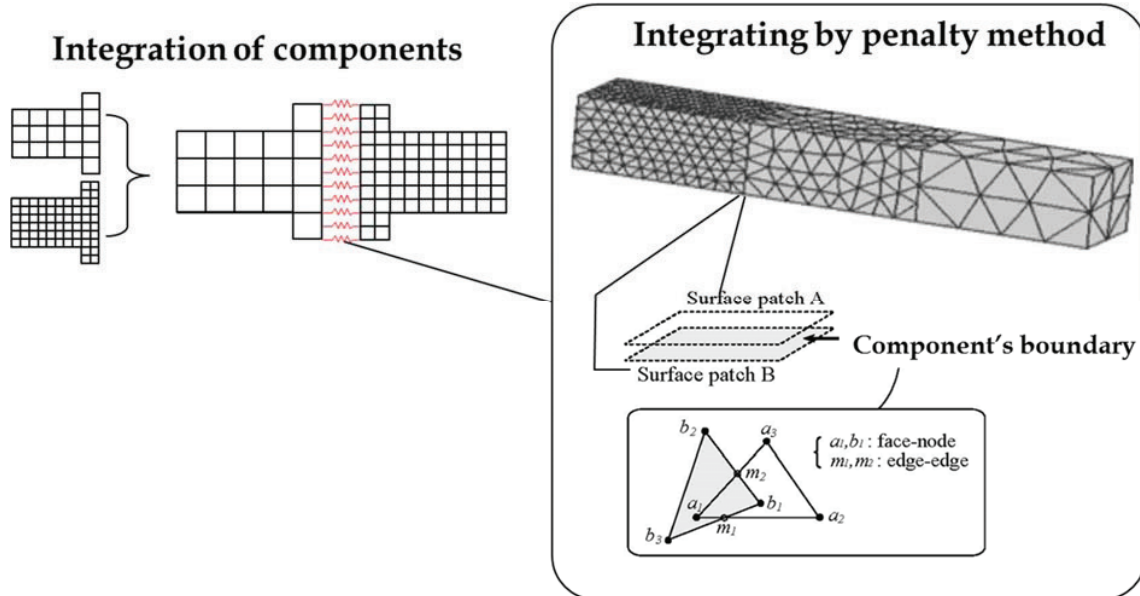


Figure 5. FE analytical models (Level 1) for an assembly.

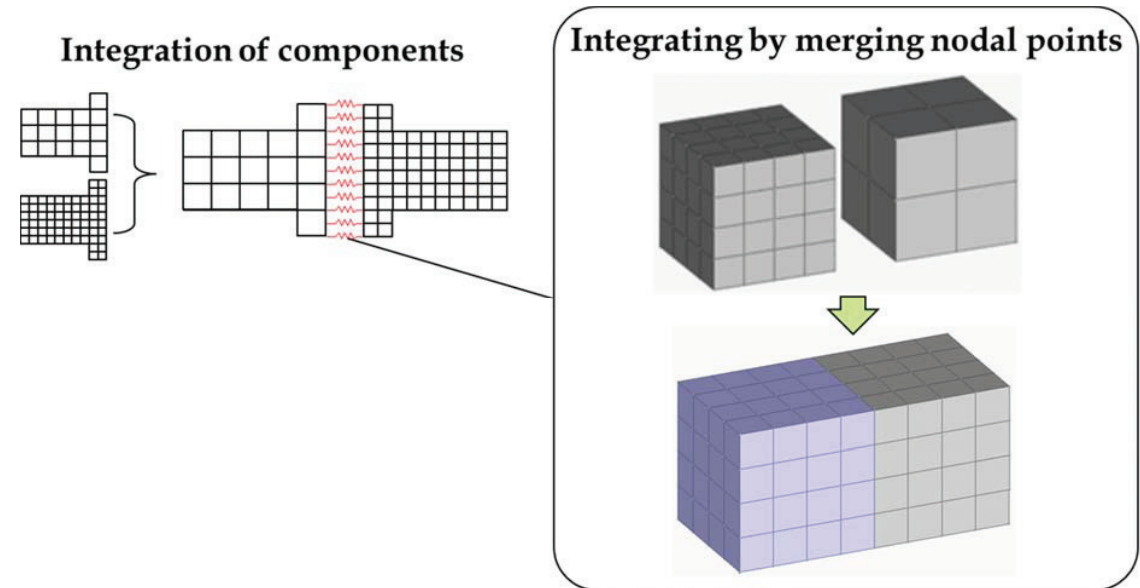


Figure 6. FE analytical models (Level 2) for an assembly.

FIESTA merges meshed components to analyse an assembly structure. The way to merge components is started to load two components data from data base of the digital shaking table. Second, two data of geometric model are checked whether surfaces and edges are matched, in topologically then geometrically. After confirmed a well condition among components, meshed data are merged by elements. Figure 7 shows an example of a merging result.

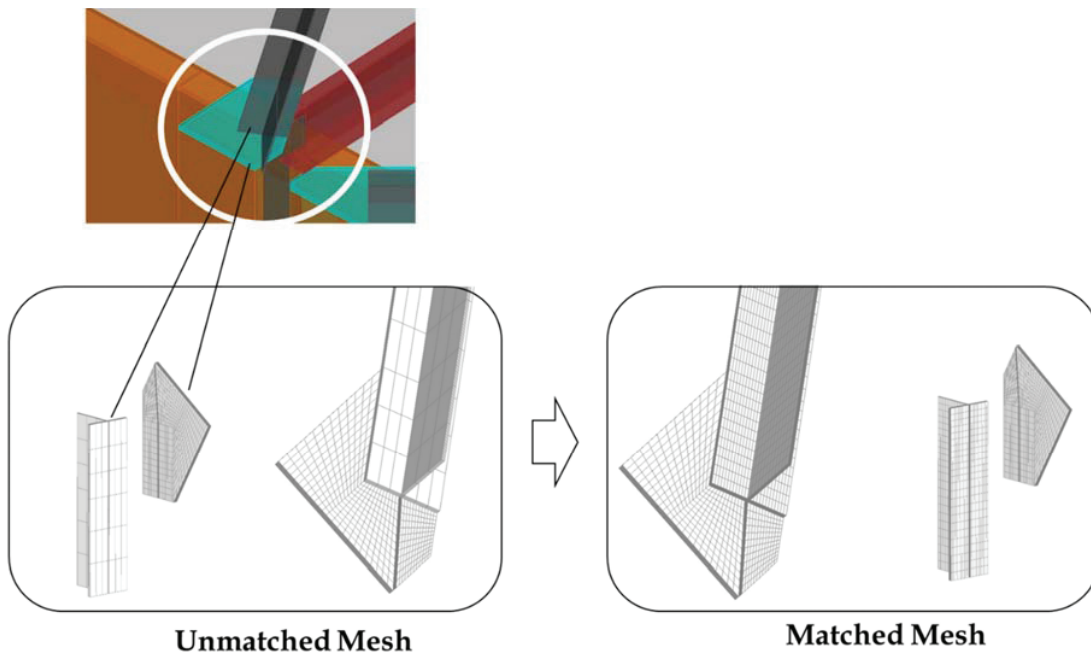


Figure 7. An attached Surface creation and mesh matching in practical.

## NUMERICAL EXPERIMENTS

In FE analysis, modelling starts from creating a design object by using a geometric modeller. Generally, a design object is created a component by a component. For FE analysis, usually a component becomes an analytical object, or components are merged to an object to be calculated. In the practical process of the second step of the modeling, the geometric model is manually divided into parts so-called a substructure in this paper, for a mapped mesh generator or used as input to an automatic mesh generator. One may apply a fully automatic mesh generator instead of manually dividing into substructures. In analysis of design, many points and profiles are exactly defined to analyze the design specification. From this point of view, a substructure model must be provided for detailed analyses. A substructure is a solid primitive of a geometric model and has the same topology as a hexahedron type of the finite element. The substructure model is now discretized into finite elements. For the preparation for a time domain response analysis, an analytical model is created by inputting material properties and boundary conditions, such as constraint conditions and so on from the feature-based model, which is defined at designing process. The load conditions are given by using a digital shaking table as exogenous factors, which are namely as El Centro, Taft, Hachinohe, and Geiyo. Hachinohe is happened at north-east of Japan in 1968. Geiyo is also happened in Japan at west side when 2001.

In this experiment, a geometric model is introduced in Figure 8 shown a bird's-eye view of "structure" of a petroleum plant and few floors of "structure". Each component are modelled and assembled to a floor. A floor is assembled with columns to become "structure". FIESTA' s input data are meshed components and build up the meshed object for analysis. Since feature based modelling is introduced in FIESTA, boundary conditions are already set for the meshed data. See for example, Nakajima et al., (1992).

Figure 9 demonstrates an attached Surface creation and mesh matching in practical. Meshed components are gathered and integrated by the suggested method to become a well continuous meshed model for FE analysis.

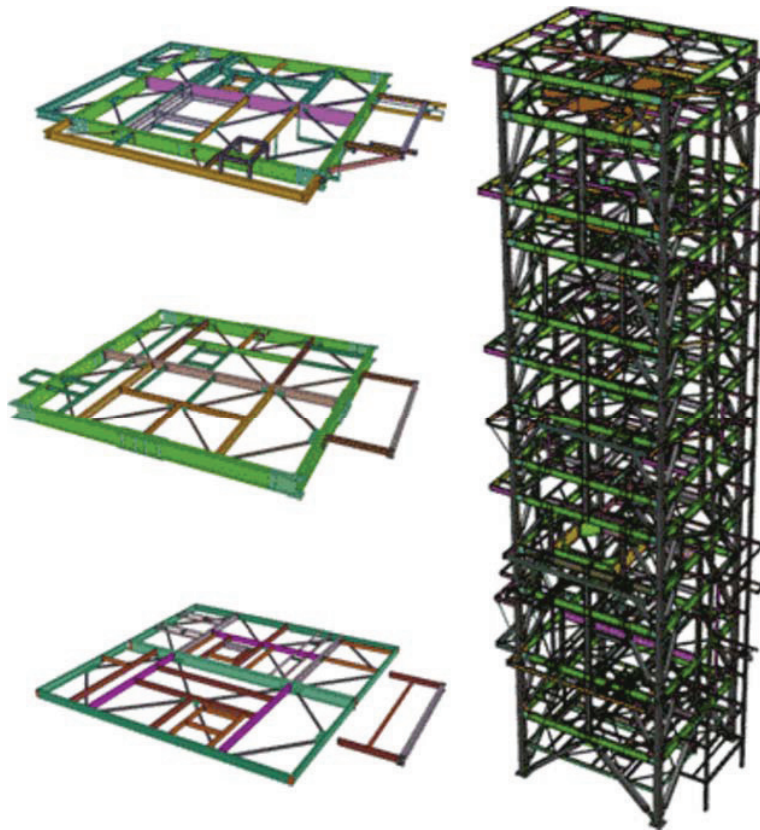


Figure 8. A bird's-eye view for “Structure” of a petroleum plant.

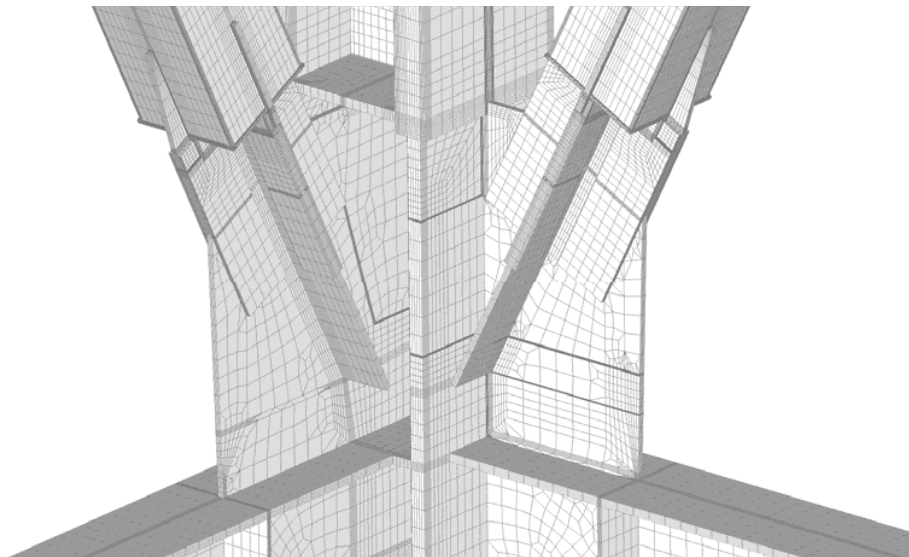


Figure 9. An attached Surface creation and mesh matching in practical.

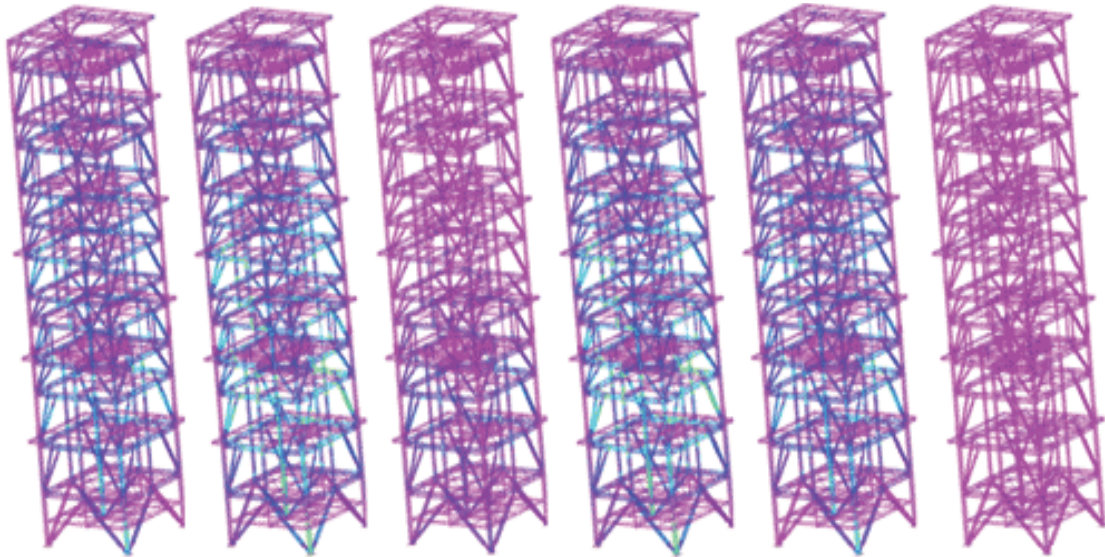


Figure 10. A time domain response analysis with El Centro.

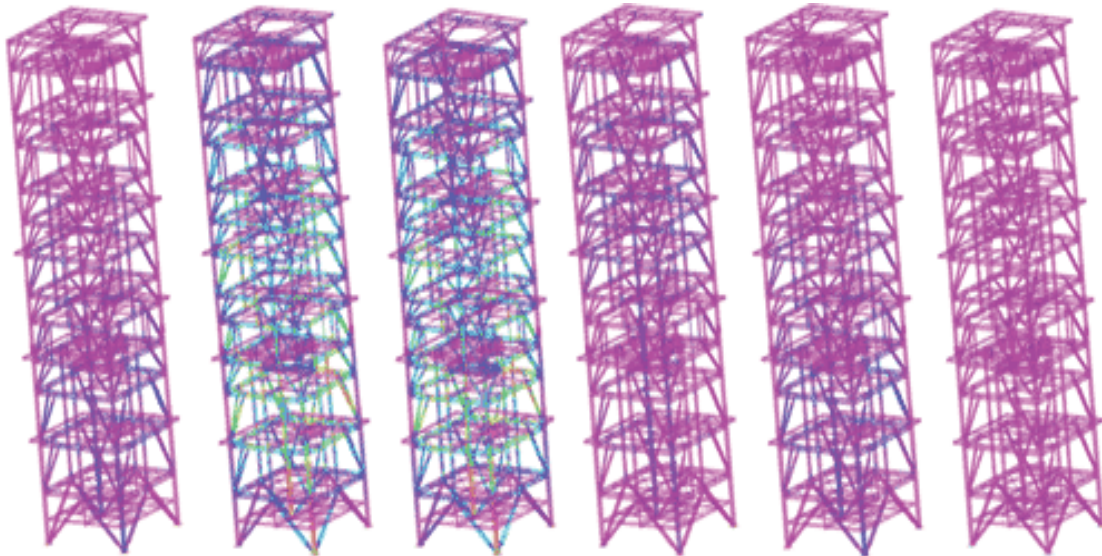


Figure 11. A time domain response analysis with Taft.

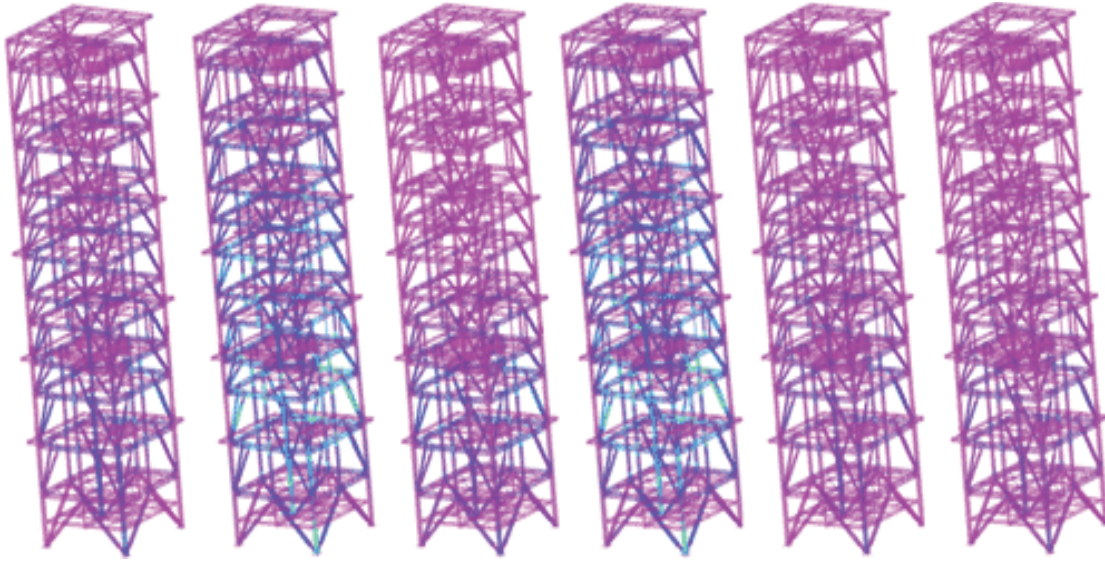


Figure 12. A time domain response analysis with Hachinohe.

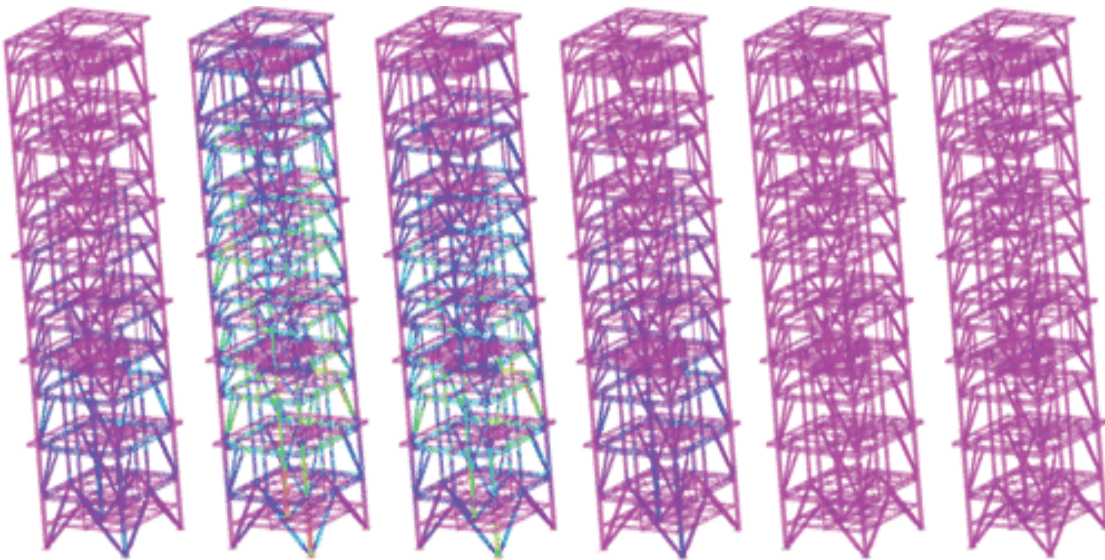


Figure 13. A time domain response analysis with Geiyo.

## RESULTS AND DISCUSSION

Time domain response analyses had been carried out. Four waves are applied for verification and validation. The results of experiments are compared to the accumulated data in the past. As the data of simulations are

macro model analysis, the comparison among simulation data is done by macro values. As FIESTA can analyse the object in the overlooking and detailed point of view, deformations and stress are compared in macroscopic data. Deformations in time domain are well matched for each waves and also stress on certain observation points are corresponding.

## CONCLUSION

Verification and validation for FIESTA as a code of finite element analysis for structure of assembly is carried out and confirmed. A technique for connecting components was also verified and validated through time domain response analyses. The code and technique were applied to the actual problem with a practical design object. The code was improved and tested throughout. The resulting code was applied to actual models to simulate practical problems in time response analyses, and the results indicated good modeling power and simulation.

The developed procedure is intended to correct for inconsistent data among components that share a surface. FE discretization data obtained from this procedure showed the consistency of the approach.

One method of assembly by structural analysis was suggested for the procedure, although the simulation has a certain limitation to assembly analysis. However, it has introduced an easier way of the analysis to structures that require enormous amounts of components to model.

## ACKNOWLEDGMENTS

This research and development was partially supported by the 2012–2014 budget of HPCI strategic program of the Ministry of Education, Culture, Sports, Science and Technology of Japan.

This research used computational resources of the K computer provided by the RIKEN Advanced Institute for Computational Science through the HPCI System Research project (Project ID:hp120291, hp130029, hp140206).

The authors specially appreciate to Hitachi Industry & Control Solutions, Ltd., Research Organization for Information Science and Technology, and JSD Inc., for their supports in the development of software. Especially, M Mr. Osamu TSURUTA of Hitachi Industry & Control Solutions, Ltd., also Dr. Tatsuo OKADA of Research Organization for Information Science and Technology for their commitments in developing software and perpetrating data.

## REFERENCES

- Adventure, <http://sys.t.u-tokyo.ac.jp/jp>  
CAD/CAE committee, 2005, Finite Element Analysis Software ANSYS – Introduction to engineering analysis, Ohm  
FINAS, <http://www.engineering-eye.com/finas/finas/>  
FrontISTR, <http://www.multi.k.u-tokyo.ac.jp/lectures/>  
Fish, J., and Belytschko, T., 2008, A First Course in Finite Elements (ABAQUS Student Edition), Maruzen (Translated to Japanese edition by Yamada, K)  
Midas\_Gen, <http://en.midasuser.com/products/products.asp?nCat=353&idx=29235>  
Nakajima, N., et al., 1992, “Feature-based heuristics for finite-element meshing using quadtrees and octrees”, Computer Aided Design, Vol.24, No.12, P677 – 690  
Nakajima, N., et al., 2004, “Towards to realize a Quake –proof Information Control and Management System for Nuclear Power Plant,” Proceedings of 23rd the Japan Society for Simulation Technology, Tokyo, 117-120  
Nakajima, N., et al., 2013, “Assembly Structural Analysis System,” Transactions, SMiRT-22, San Francisco, California, USA - August 18-23, Division III

- Nakajima, N., et al., 2013, "Structural Analysis for Assembly by Integrating Parts" Proceedings of the 2014 22nd International Conference on Nuclear Engineering, ICONE22, Prague, Czech Republic, ICONE22 2014-30251, July 7-11, 2014
- Nishida, A. et al., 2006, "Numerical Simulation System Three-Dimensional Virtual Vibration Testbed," Trans. Atomic Energy Society Japan, 6(3), 376-382
- Russell, J., and Cohn, R., 2012, "NASTRAN", Book on demand Ltd.
- Tani, M., Nakajima, N., Nishida, A., Suzuki, Y., Matsubara, H., Araya, F., Kushida, N., Hazama, O., Kondoh, M., and Kawasaki, K., 2007, "A methodology of structural analysis for nuclear power plant size of assembly", Proc. of M&C + SNA 2007(CD-ROM) , American Nuclear Society, 1-13
- Zienkiewicz, O.C. and Taylor, R. L. (2000). The finite element method, 5th ed., Butterworth Heinemann