Transactions of the 15th International Conference on
Structural Mechanics in Reactor Technology (SMiRT-15)
Seoul, Korea, August 15-20, 1999

Elastic-Plastic Analysis of Nuclear Structures with Millions of Dofs Using Hierarchical Domain Decomposition Method

Tomoshi Miyamura[1], Hirohisa Noguchi[2], Ryuji Shioya[3], Shinobu Yoshimura[1] and Genki Yagawa[1]

1) University of Tokyo, Japan
2) Keio University, Japan
3) Kyushu University, Japan

ABSTRACT

The hierarchical domain decomposition method (HDDM) is applied to large scale elastic-plastic finite element analyses. The HDDM is a solver for large scale algebraic equations in the finite element method, and is suitable for various types of parallel environments. In this paper, the static analysis based on the incremental method is considered. The elastic-plastic analysis of a nuclear structure with 1.3 M DOFs is presented as an illustrative example. It is first confirmed that CG iteration in each incremental or Newton-Raphson iteration step is converged, and then that the iteration of the Newton-Raphson method is converged. Computation time of an incremental step and successive Newton-Raphson iteration steps is 7053 seconds by using 256 PEs of HITACH SR2201.

INTRODUCTION

Recently, massively parallel processors are the main stream of high performance computers for scientific and engineering computations. Virtual parallel processors such as workstation (WS) / personal computer (PC) cluster are also becoming popular. To efficiently utilize these parallel computers, new software has to be developed.

The hierarchical domain decomposition method (HDDM) is a solver of large scale algebraic equations in the finite element method, which is suitable for various types of parallel environments[1]. In the present study, the HDDM is applied to large scale elastic-plastic analyses. The domain decomposition method (DDM) has already been applied to the elastic-plastic analysis [2, 3]. A target of our study is to analyze the nonlinear structures with 1-10 millions DOFs by using an implicit solver.

A nuclear structure model with 1.3 M DOFs is analyzed as an illustrative example. Pre- and post-processors such as automatic mesh generator, boundary condition attachment tool, domain decomposer, visualization tool etc., are also introduced, and the analysis procedure is illustrated.

HIERARCHICAL DOMAIN DECOMPOSITION METHOD

In the domain decomposition methods (DDM), an analysis model is subdivided into subdomains. Although there are several kinds of DDM, a kind of substructure method[4] is adopted in this study. This method is simply called the DDM here.

The DOFs in the internal nodes of each subdomain, and those in the nodes on the internal boundaries between subdomains are called the internal DOFs and the interface DOFs, respectively. First, the stiffness matrix for each subdomain is partitioned as follows:

$$\mathbf{K}^k = \begin{bmatrix} \mathbf{K}^k_{II} & \mathbf{K}^k_{IB} \\ \mathbf{K}^k_{BI} & \mathbf{K}^k_{BB} \end{bmatrix}, \tag{1}$$

where k denotes subdomain number, I shows the part concerning with the internal DOFs, B shows the part concerning with the interface DOFs. Equilibrium equations for the internal DOFs are as follows:

$$\mathbf{K}^k_{II}\mathbf{u}^k_I + \mathbf{K}^k_{IB}\mathbf{u}^k_B = \mathbf{p}^k_I , \tag{2}$$

and those for the interface DOFs are represented as follows:

$$\sum_{k=1}^{NDOM} \mathbf{K}^k_{BI}\mathbf{u}^k_I + \sum_{k=1}^{NDOM} \mathbf{K}^k_{BB}\mathbf{u}^k_B = \sum_{k=1}^{NDOM} \mathbf{p}^k_B , \tag{3}$$

where $\mathbf{u}$ denotes the nodal displacement vector, $\mathbf{p}$ denotes the equivalent nodal external force vector, and *NDOM* shows a number of subdomains. Equation (2) can be solved independently for $\mathbf{u}^k_I$ in each subdomain as follows:

$$\mathbf{u}^k_I = \left(\mathbf{K}^k_{II}\right)^{-1}\left(\mathbf{p}^k_I - \mathbf{K}^k_{IB}\mathbf{u}^k_B\right). \tag{4}$$

Therefore, $\mathbf{u}^k_I$ is statically condensed by substituting from Eq. (4) into Eq. (3). The interface DOFs, i. e., $\mathbf{u}^k_B$ is solved with an interative solver such as the conjugated gradient (CG) method.

Yagawa and Shioya proposed a hierarchical technique to implement the DDM[1]. This technique is called the hierarchical domain decomposition method (HDDM). In this method, a group of processor elements (PEs) is subdivided into three groups: one Grand Parent PE (Grand), several Parent PEs (Parent), and many Child PEs (Child/Children).

An analysis model is subdivided into several "parts" whose number is the same as the number of Parents. Each Parent stores the data for a part that include nodal coordinates, connectivities of elements and material properties. Stresses and displacements also have to be stored when solving nonlinear analyses.

The static condensation of each subdomain (i. e., the elimination of $\mathbf{u}^k_I$ using Eq. (4)) is done by Children. Each Child receives the analysis data from a Parent, and sends back the results to the Parent. Children are dynamically allocated to Parents that contain the subdomains whose static condensation is not finished. Therefore, the dynamic workload balancing is attained.

When $\mathbf{K}^k$ and $\left(\mathbf{K}^k_{II}\right)^{-1}$ are stored in Children's memory, computation speed is improved. In this case, hybrid workload balancing is necessary, i. e., dynamic workload balancing is adopted only in the first CG step, and after that Children

are statically assigned to Parents that have subdomain data for $\mathbf{K}^k$ and $(\mathbf{K}_{II}^k)^{-1}$ stored in the Children.

Grand checks the status of each Parent and arranges Children. It also manages the whole computation. After the static condensation processes of all the subdomains are finished, Grand and Parents update the solution vector based on the CG method.

In static nonlinear analyses of solid, incremental and the Newton-Raphson iteration loops are necessary. These loops are placed outside the CG loop. Figure 1 illustrates the schematic flow of the nonlinear FE analysis with HDDM.

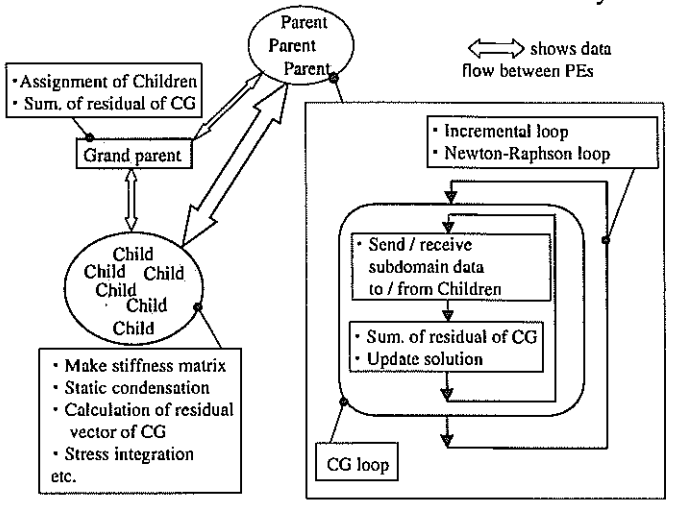

Fig. 1 Schematic flow of HDDM in nonlinear FEM

ELASTIC-PLASTIC ANALYSIS

In static elastic-plastic analysis, an incremental method combined with the Newton-Raphson method is employed. The elastic predictor-radial corrector (radial return) method is employed as a stress integration scheme. By using this method, errors of integrated stresses are small even when large increment is applied. Full Newton-Raphson iteration is performed with the use of the tangent stiffness that is consistent with the stress integration scheme[5].

CONVERGENCE CRITERIONS OF CG AND NEWTON-RAPHSON METHOD

A norm of a vector is defined as a maximum absolute component of the vector. ε_{CG0} is the norm of the residual of CG in the first CG step of the first incremental step. ε_{CG} is the norm of the residual of CG at an iterative step of CG.

The following convergence criterion for CG is employed:

$$\frac{\varepsilon_{CG}}{\varepsilon_{CG0}} < \mathrm{TOL}_{CG}, \tag{5}$$

where TOL_{CG} is a tolerance.

The norm of the out-of-balance force of Newton-Raphson method for the interface DOFs and that for the internal DOFs are denoted by $\varepsilon_{NR\text{-}INN}$ and $\varepsilon_{NR\text{-}INN}$, respectively. When TOL_{CG} in Eq. (16) is relaxed, $\varepsilon_{NR\text{-}INB}$ converges to the order of TOL_{CG}, whereas $\varepsilon_{NR\text{-}INN}$ converges to the order of numerical error. Therefore,

the following convergence criterion for the Newton-Raphson method is adopted:

$$\frac{\varepsilon_{NR\text{-}INN}}{\varepsilon_{CG0}} < TOL_{NR} \ , \qquad (6)$$

where TOL_{NR} is a tolerance. Since $\varepsilon_{NR\text{-}INB}$ can not be smaller than TOL_{CG}, TOL_{NR} is set equal to TOL_{CG}.

HTTR MODEL

Figure 2 shows the analysis model of a support structure of the high temperature engineering test reactor (HTTR)[6]. In this model, the geometry is simplified, i. e., the real structure has more holes. A basic coarse mesh with hexahedral elements is automatically generated by the intelligent local approach[7]. Then, each element in the coarse mesh is recursively subdivided into eight hexahedral elements. This subdivision is repeated three times, and finally the 1.31 M DOFs model shown in Fig. 2 is obtained. The number of elements is 407,040, and that of nodes is 437,639.

The structure is made of carbon, therefore, the creep analysis in high temperature considering thermal strain is necessary from a practical viewpoint[6]. In this study, however, the material of the structure is assumed to be steel, and the elastic-plastic analysis is performed. As shown in Fig. 2, the right-hand surface is fixed, while uniform displacements are prescribed in the left-hand surface.

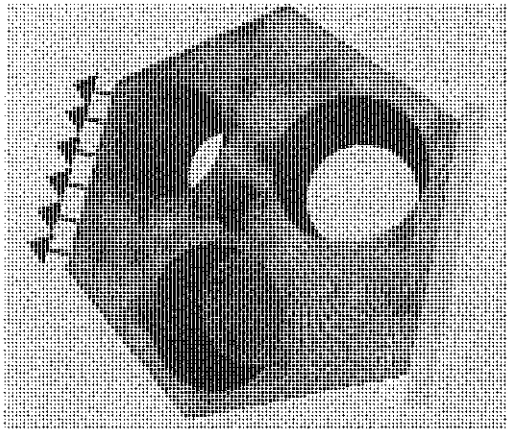

Fig. 2 1.3 M DOFs HTTR model

PROCEDURE OF LARGE SCALE PARALLEL FEM

In the large scale finite element analysis, a series of processes including pre-processes and post-processes should be automated. For example, an input data file of 1 M DOFs model is over several tens M Bytes, and it is impossible to open the file by an editor. Figure 3 shows the schematic flow of the analysis.

The boundary conditions shown in Fig. 2 are imposed by a tool developed by our group. This tool first extracts surface patches, and subdivides these patches into several groups by comparing angles between the patches. The group data are output in HTML and VRML as shown in Fig. 4. Then, a user specifies the group to which boundary conditions are imposed.

The analysis model with the boundary conditions is hierarchically decomposed. This domain decomposition is carried out by using a parallel domain decomposer that is also developed by our group[8]. This system utilize METIS and PARMETIS [9, 10].

Each process above has to be checked by visualizing the results. When models with several millions of DOFs are analyzed, a visualization process can be performed on a single processor. Figures 3 to 5 are depicted by using VRML on a single processor. However, a parallel visualization system is needed when an ultra large scale problem, i. e., over ten millions DOFs, is analyzed. Our group is now developing such a system [11], and the following Figs 8 and 9 are depicted by the system.

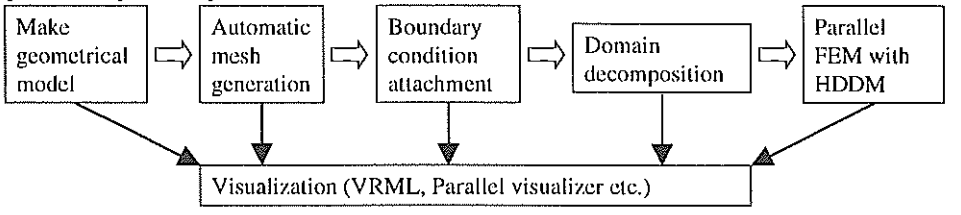

Fig. 3 Analysis procedure of parallel FEM based on HDDM

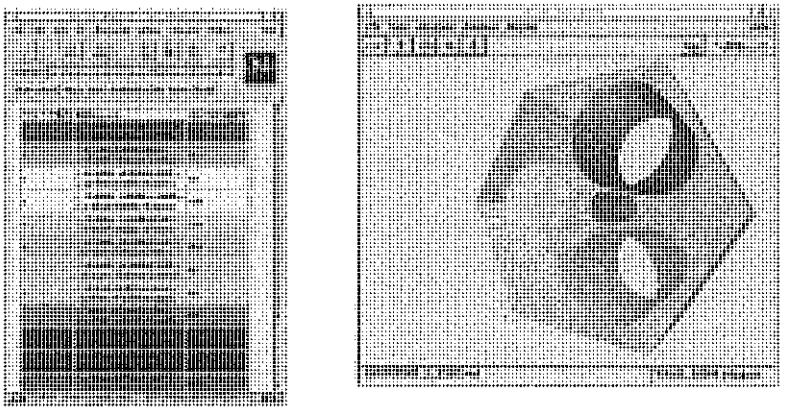

Fig. 4 Boundary condition attachment tool

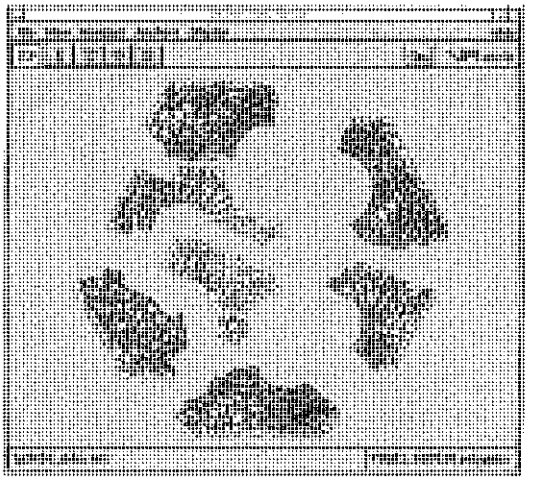

Fig. 5 Hierarchical domain decomposition of HTTR model

ANALYSIS OF HTTR MODEL

The von Mises yield criterion with isotropic hardening, linear hardening function, and the associated flow rule are assumed in the elastic-plastic constitutive equations. As a preliminary analysis, a smaller model with 0.175 M DOFs is analyzed. This model is created by recursively subdividing elements in the coarse mesh twice. The tolerances in Eqs (5) and (6) are set as $TOL_{CG} = 10^{-15}$ and $TOL_{NR} = 10^{-15}$, respectively. A PC cluster consisting of 16 PCs with DEC alpha 21164 533 MHz processors (OS : Linux) is utilized. Figure 6 illustrates the convergence of the CG and the Newton-Rahson method. $\varepsilon_{CG}/\varepsilon_{CG0}$, $\varepsilon_{NR\text{-}INN}/\varepsilon_{CG0}$ and $\varepsilon_{NR\text{-}INB}/\varepsilon_{CG0}$ are plotted in the same figure. The horizontal axis shows the accumulated number of the CG iteration. The quadratic convergence of $\varepsilon_{NR\text{-}INN}/\varepsilon_{CG0}$ or $\varepsilon_{NR\text{-}INB}/\varepsilon_{CG0}$ is observed. Total computation time until the Newton-Raphson method converges is 5676 seconds.

Then, the elastic-plastic analysis of the 1.31 M DOFs model is performed. The elastic analysis takes about 20 minutes using 256 PEs of HITACHI SR2201 in the University of Tokyo. The maximum elapsed time of a batch job for 256 PEs on this computer is limited to two hours, and the Newton-Raphson method did not converge within a batch job. To finish the computation in a batch job, the convergence criteria are slightly relaxed, i. e., $TOL_{CG} = 10^{-12}$ and $TOL_{NR} = 10^{-12}$. Figure 7 shows the convergence of the CG and the Newton-Raphson method. Quadratic convergence of the latter is attained except for the last step in which $\varepsilon_{NR\text{-}INB}/\varepsilon_{CG0}$ does not become smaller than the relaxed TOL_{CG}. On the other hand, $\varepsilon_{NR\text{-}INN}/\varepsilon_{CG0}$ fully converges. Total computation time until the Newton-Raphson method converges is 7053 seconds.

Figure 8 shows a distribution of the equivalent stresses and deformation. Due to the high resolution of the analysis, one can see details of the distribution. Figure 9 shows the plastic region. Figures 8 and 9 are depicted using the parallel visualization system [11].

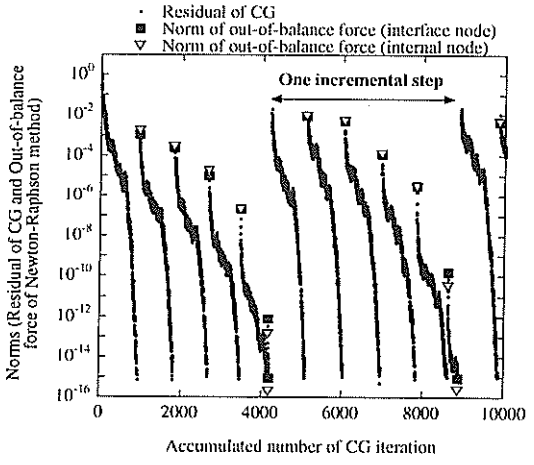

Fig. 6 Convergence of CG and Newton-Raphson method (0.175 M DOFs model)

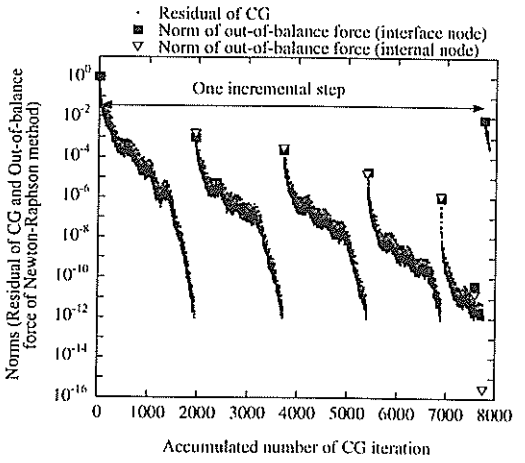

Fig. 7 Convergence of CG and Newton-Raphson method (1.31 M DOFs model)

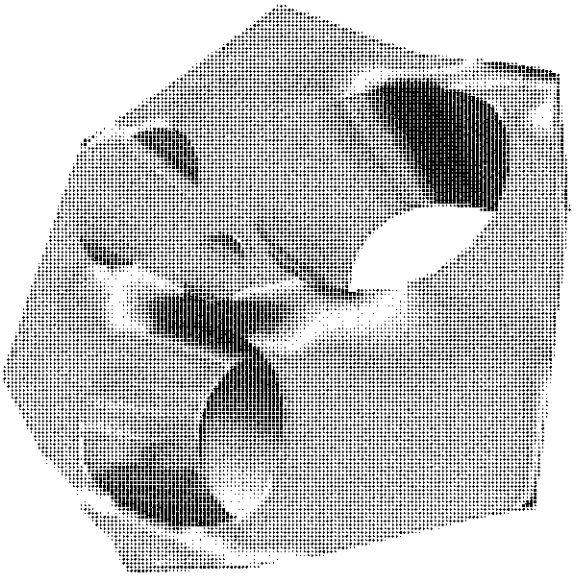

Fig. 8 Distribution of equivalent stress and deformation

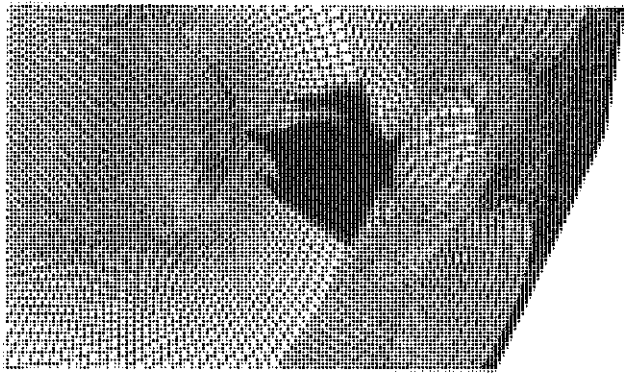

Fig. 9 Plastic region

CONCLUDING REMARKS

The hierarchical domain decomposition method is applied to the elastic-plastic analysis. A structure of HTTR that is represented by a mesh with 1.3 M DOFs is analyzed by the developed system. The analysis procedure of the parallel FEM including pre- and post-processes is presented. Results of the example show that the static elastic-plastic analysis of several millions DOFs is possible within practical computation time by using a currently available parallel environment.

ACKNOWLEDGEMENT

This study is a part of ADVENTURE project[12] sponsored by the Japan Society

for the Promotion of Science. Pre- and post-processing tools, i. e., automatic mesh generator, BC attachment tool, parallel domain decomposer and parallel visualizer are developed by Dr. Y. Wada (Univ. of Tokyo), Mr. H. Nitta (Allied Eng.), Mr. H. Takubo (Univ. of Tokyo) and Mr. S. Shoui, respectively, who are members of the project.

REFERENCES

1 Yagawa, G. and Shioya, R., "Parallel Finite Elements on a Massively Parallel Computer with Domain Decomposition," *Comp. Sys. in Eng.*, 4, pp 495-503, 1993

2 Soneda, N., Yagawa, G. and Yoshimura, S., "Elastic-Plastic Finite Element Analysis using Domain Decomposition Method,", *Proc. SMiRT-11*, Tokyo, Japan B, pp 243-248, 1991

3 Yagawa, G. and Uchida, Y., "Elastic-Plastic Finite Element Analysis Using EWS Network," *Trans. of Japan Soc. of Mech. Eng.*, Vol. 60, No. 580, A, pp 2737-2742, 1994 (in Japanese)

4 Yagawa, G. and Soneda, N., *Parallel Computing*, Baifu-kan, pp 27-73, 1991 (in Japanese)

5 Bathe, K. J., *Finite Element Procedure*, Prentice Hall, 1996

6 Arai, T. et al., "Experiment and Analysis on Elastic Deformation Properties of Graphite and Carbon Materials of HTTR Core Components," *JAERI-M report*, Japan Atomic Energy Research Institue, 92-085, 1992 (in Japanese)

7 Wada, Y. , Yoshimura, S. and Yagawa, G., "Hexahedral Mesh Generation of Nuclear Structures Using Intelligent Local Approach," *15th Int. Conf. on Structural Mechanics in Reactor Technology*, Seoul, Aug. 1999

8 Takubo, H., Yoshimura, S. and Yagawa, G., "Domain Decomposer of Ultra-large Scale Finite Element Mesh in Parallel/Distributed Environment," *Proc. of 11 th JSME CMD Meeting*, No. 98-2, 575-576, 1998 (in Japanese)

9 Karypis, G and Kumar, V., "Multilevel k-way Partitioning Scheme for Irregular Graphs," *Tech. Rep. TR 95-064*, Dept. of Comp. Sci., Univ. of Minnesota, 1995

10 Karypis, G and Kumar, V., "A Coarse-grain Parallel Multilevel k-way Partitioning Algorithm, *Proc. of 8 th SIAM Conf. on Prallel Processing for Sci. Comp.*, 1997

11 Yoshimura, S., Shoui, S, Yagawa, G. and Akiba, H., "User Friendly and Parallel Visualization System of Finite Element Solutions of Ten Million DOFs," *Proc. of Conf. Comp. Eng. Sci.*, JSCES, Vol. 3, No. 1, 71-72, 1998 (in Japanese)

12 http://adventure.q.t.u-tokyo.ac.jp/