Finite Elements in Analysis and Design 105 (2015) 44–55

Contents lists available at ScienceDirect

Finite Elements in Analysis and Design

journal homepage: www.elsevier.com/locate/finel

Ultra-large scale fracture mechanics analysis using a parallel finite element method with submodel technique

Koichiro Arai [a], Kaworu Yodo [b], Hiroshi Okada [a,*], Tomonori Yamada [c], Hiroshi Kawai [d], Shinobu Yoshimura [c]

[a] Tokyo University of Science, Japan
[b] Insight, Inc., Japan
[c] The University of Tokyo, Japan
[d] Tokyo University of Science, Suwa, Japan

ARTICLE INFO

Article history:
Received 8 February 2015
Received in revised form
24 May 2015
Accepted 14 July 2015
Available online 1 August 2015

Keywords:
Fracture mechanics analysis
Parallel finite element method
Large scale analysis
Crack

ABSTRACT

A software system for performing large-scale finite element fracture mechanics analyses is presented in this paper. The method is based on a submodel technique in which the large-scale finite element mesh is divided into two parts. One is the submodel which only represents the crack and its vicinity. The other is the global model for the whole structure. They are generated separately and connected. It is assumed that we use a supercomputer at a remote site and a PC/Workstation at a local site. The local PC/Workstation is used to perform pre/post-processings. The remote supercomputer is used to carry out the large-scale finite element analyses. In a crack propagation analysis, a number of remeshing steps are involved. We have developed a software system that the remeshings are performed only for the submodel. The global model stays the same during the analysis. Also, we minimize the amount of data which are transferred between the computers at the remote and the local site. In this paper, we present 100 million node finite element fracture mechanics analysis on a section model of nuclear pressure vessel, demonstrating the capabilities of present software system.

1. Introduction

It is important to assure the safe operations and the structural integrities of social infrastructures such as power plants and of transportation systems such as railway structures, aircrafts, ships, etc. The authors consider that the demands on the safety assurances in modern social and engineering environment have kept growing. To assure the safe operations, inspections and maintenances on the structures are generally performed. Parts and structures that are found to have structural damages such as cracks, corrosions, etc. are replaced. There are rules and codes to evaluate the structural integrities. For example, JSME (Japan Society of Mechanical Engineers) have published a code “The rules on fitness-for-service for nuclear power plants” [1].

Recently, some attempts to apply large scale finite element analyses to the problems of structural integrity/safety of power plant structures are seen in literature. Rupp et al. [2] performed a thermal analysis for the pressure water reactor (PWR) vessel structure. In their analysis, the PWR pressure vessel structure was modeled by the linear tetrahedral finite elements in detail. They focused the temperature distributions of the bolts, connecting the baffle and the vessel structure, which are much smaller than the pressure vessel structure. Quinay et al. [3] presented a multiscale fault-building structure interaction analysis. In their analysis, an interaction between the building structure of nuclear power plant and wave propagation through the crust was modeled by using a parallel finite element method. There is a large difference between the length scales of the building structure and of the crust. Methodologies to perform large scale finite element analysis on parallel computers have been developed by researchers. The early development of parallel finite element implementation can be seen in the article of Yagawa et al. [4]. They presented the domain decomposition method (DDM) on Transputer computers. DDM has widely been adopted to perform the parallel finite element analysis. For example, Yoshimura et al. [5] have developed a software system “ADVENTURE” based on DDM. The methodology has been developed further to perform the large scale fluid-structure interaction analysis [6]. Recently, Yusa et al. [7,8] proposed a submodeling technique based on an iterative solution algorithm. The crack and its vicinity are modeled by the submodel which is embedded in the global model. The equilibrium at the interface between them is

* Correspondence to: Department of Mechanical Engineering, Faculty of Science and Technology, Tokyo University of Science, 2641 Yamazaki, Noda 278-8510, Japan. Tel./fax: +81 4 7122 1808.
E-mail address: hokada@rs.noda.tus.ac.jp (H. Okada).

http://dx.doi.org/10.1016/j.finel.2015.07.006
0168-874X/

obtained iteratively. Linear elastic as well as large strain elastic–plastic analyses were presented.

The authors think that the size of engineering finite element analyses in terms of the numbers of nodes and elements continue to grow due to the enhancements of computer hardware and software technologies. Thus, local structural damages such as cracks can be included in the analysis models of large and complex engineering structure and the structural integrity analysis can be performed without a large degree of simplification on the model, crack geometries, etc. The purpose of present research is to develop a software technology that is suited for performing the crack and crack propagation analysis of large and complex engineering structure such as nuclear pressure vessels.

To perform such analyses, we adopt the parallel finite element program ADVENTURE_Solid [9] to carry out large scale finite element computations on massively parallel supercomputers. However, it is not easy to generate a finite element mesh with cracks in general. The finite element model generation has been recognized as the bottleneck process in the fracture mechanics analysis. The difficulty may be more pronounced when the finite element model is large and complex.

Methodologies that can eliminate or reduce the needs for meshing have been proposed by the researchers. eXtended finite element method (X-FEM) was proposed (see Belytschko et al. [10], Moës et al. [11], Grabouil et al. [12] and Pathak et al. [13]). The applications of the generalized finite element method (G-FEM) to crack problems were presented by Duarte et al. [14] and Pereir et al. [15]. X-FEM and G-FEM are based on the partition of unity concept and they eliminate the needs to explicitly model the crack by the finite element mesh. The cracks are expressed by the so-called enrich/analytical functions and are inserted in the finite element mesh without the crack. S-version finite element method (S-FEM) that superposes two or more finite element models was adopted to perform the fracture mechanics analysis by Okada et al. [16] and Kamaya et al. [17]. S-FEM was originally proposed by Fish [18]. In S-FEM fracture mechanics analysis, a local finite element model contains the crack and it is superposed on the global finite element model without the crack. The local model containing the crack is generated independently of the global finite element model. Thus, the model generation processes are simplified compared with the case that the finite element model explicitly expresses the crack.

Another way to reduce the meshing effort is to use the automatic meshing technology with the tetrahedral finite elements. For crack problems, Okada et al. [19] and Kaneko et al. [20] have presented the automatic meshing techniques based on the Delaunay tessellation technique. Technostar Co. Ltd. released the finite element pre-processing software that had a crack mesh option [21]. The computations of the crack parameters by the virtual crack closure-integral method (VCCM), J-integral and interaction integral method were made possible by Okada et al. [22,23] and Daimon et al. [24], respectively, even if the model was consisting of the tetrahedral finite elements only. Crack/crack propagation analyses using the straight forward finite element method can be found in the literature. The mixed element approach or submodeling technique is commonly adopted to perform crack/crack propagation analyses (see, for example, Ural et al. [25], Wawrzynek et al. [26], Bremberg et al. [27,28] Schöllmann et al. [29] and Buchholz et al. [30]). Such techniques may have been developed to place the hexahedral finite elements at the vicinity of crack front so that proven numerical techniques to compute the crack parameters can be adopted (see Shivakumar et al. [31], Okada et al. [32], Nikishkov et al. [33], Banks-Sills [34]).

Present authors have sought a way to perform the structural integrity analyses on large scale and complex structure with assuming the existence of cracks (for example, see Yodo et al. [35,36]). Yodo et al. used the massively parallel supercomputer "Earth Simulator" and demonstrated the performance of ADVENTURE_Solid. However, the supercomputers are, most of time, owned and operated by organizations such as university's information technology center and are designed mainly for performing large scale computation. Thus, data preparations and pre/prost processings for the large scale finite element analyses such as generating the finite element mesh, specifying the boundary conditions, computing the stress intensity factors, etc. should be performed on a local PC/Workstation, while the large scale finite element analysis should be carried out by the supercomputer. Therefore, the mesh data, the result files, etc. need to be transferred between the remote supercomputer and the local PC/Workstation through the Internet. The sizes of data files may be very large because of those of the finite element analyses and their transfers between the computers may take a significant amount of time. In this paper, a computational strategy to perform large scale fracture mechanics analyses for real structure with structural

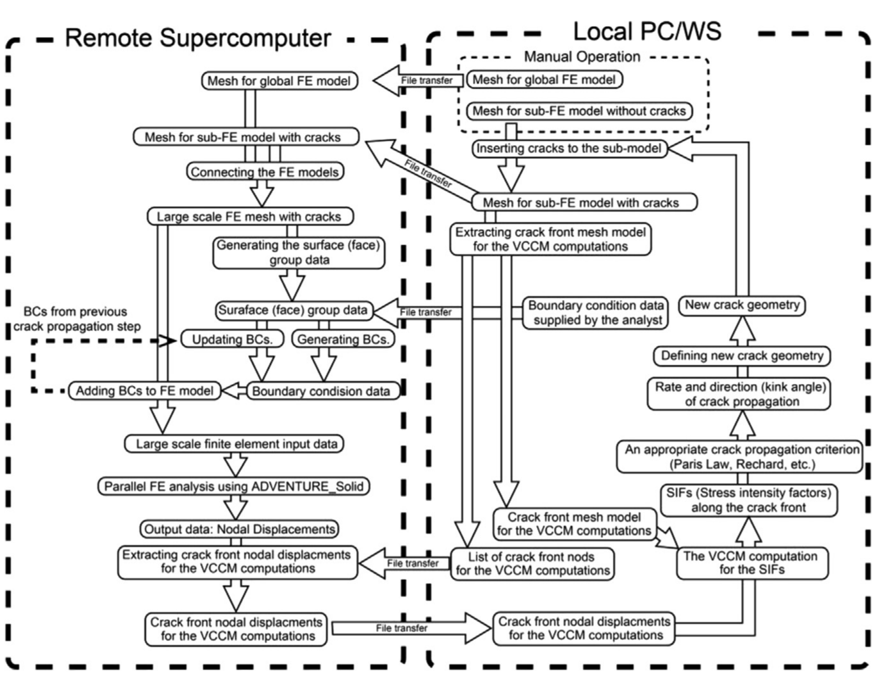

Fig. 1. The analysis flow of crack propagation analysis using the submodel approach.