Comput. Methods Appl. Mech. Engrg. 253 (2013) 1–14

Contents lists available at SciVerse ScienceDirect

Comput. Methods Appl. Mech. Engrg.

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

Adaptive finite elements using hierarchical mesh and its application to crack propagation analysis

Kohei Murotani [a,*], Genki Yagawa [b], Jae Boong Choi [c]

[a] Department of Systems Innovations, School of Engineering, The University of Tokyo, 7-3-1 Hongo Bunkyo-ku, Tokyo 113-0032, Japan
[b] Center for Computational Mechanics Research, Toyo University, 2-36-5 Hakusan Bunkyo-ku, Tokyo 112-0001, Japan
[c] School of Mechanical Engineering, Sungkyunkwan University, Suwon, Kyonggi 440-320, Republic of Korea

ARTICLE INFO

Article history:
Received 31 March 2012
Received in revised form 3 July 2012
Accepted 31 July 2012
Available online 23 August 2012

Keywords:
Adaptive meshing
Hierarchical mesh
Level-of-detail approximations
Fracture mechanics
Crack propagation
Finite element method

ABSTRACT

In the present paper, a high-speed adaptive meshing using a hierarchical mesh is studied. We use the level-of-detail approximations of the mesh data structure used often in the computer graphics field. To achieve the correct level of detail, we employ the hierarchical regional partitions. Meshes using this data structure can be realized in a very quick manner. Finally, we apply the level of detail approximation to the adaptive analyses of crack propagation, demonstrating the efficiency of the present method.

1. Introduction

It is well known that the data handling for computer graphics field is usually very time-consuming. To speed up the rendering, a 3D object is often treated with a set of level-of-detail (LOD) approximations, where a detailed mesh is used when the object is close to the viewer and coarser approximations when the object is away [1,2]. The LOD approximations need be pre-computed by mesh simplification methods [3–8]. For efficient data storage and transmission, mesh compression schemes [9,10] have also been developed. Hoppe [11,12] studied the progressive mesh (PM) representation, a new mesh format that provides a unified solution to these problems.

In the finite element analysis, on the other hand, time dependent problems such as the dynamic analysis are often more time-consuming than the rendering of the 3D detailed objects. Therefore, to reduce the calculation time, the adaptive analyses have been developed and applied to the large-scale complex problems. The adaptive analyses have been studied with advanced mesh generations, mesh update and parallel search techniques [13,14]. Also, the adaptive mesh approaches have been studied based on the Enhanced-Discretization Interface-Capturing Technique (EDICT) with the two-levels of meshes or a combination of coarse and fine meshes [15–17], which have been applied to compressible flows with shocks [15] and free-surface flows [17].

The present paper applies the LOD approximations to the adaptive analysis [18–20] of the FEM, which consists of the posterior error estimation and the re-meshing based on the result of the above error estimation. These steps are repeated many times together with the adaptive calculation, where we utilize an efficient re-meshing method that employs the above progressive mesh (PM) representation, which has been used in the rendering techniques to reduce the computer cost. First, we create a fine mesh for an entire domain using a meshing technique such as the Delaunay triangulation, which is called here as the mother mesh. Second, the progressive meshing is performed on this mother mesh, which results in a hierarchical mesh. Third, an adaptive FEM and a posterior error analysis are performed based on the hierarchical mesh. Although the CPU time required for preprocessing is considerable, the adaptive meshing process is so fast. Thus, the present method is very efficient in many classes of problems, particularly for problems of slow convergence and long time-evolution.

Here, we take a 2D elasticity problem with the Zienkiewicz–Zhu's method [19] as the posterior error estimation, where we call the zero-dimensional simplex a "vertex", the one-dimensional simplex an "edge", the two-dimensional simplex or the so-called triangle an "element", and the triangulated body of an entire region a "mesh". In addition, we call the element where linear shape functions are embedded a "1st-order element" and the element where quadratic or cubic shape functions are embedded "2nd-order

* Corresponding author. Tel.: +81 3 5841 8922.
E-mail addresses: muro@save.sys.t.u-tokyo.ac.jp (K. Murotani), yagawag@gmail.com (G. Yagawa), boong33@skku.edu (J.B. Choi).

0045-7825/$ - see front matter © 2012 Elsevier B.V. All rights reserved.
http://dx.doi.org/10.1016/j.cma.2012.07.024

element" or "3rd-order element", respectively. It is noted that the 1st-order element has three nodes, the 2nd-order element six nodes, and the 3rd-order element ten nodes, respectively.

As is well-known, it has been an issue of fracture mechanics to solve exactly the crack propagation problems in mixed-mode condition [21–24]. First of all, the crack propagation analyses require high accuracy around crack tips. If a standard finite element method is used, the fine meshes with high-quality elements are needed around the crack tips. This is very difficult for the propagating cracks due to the high calculation cost. Recently, the extended finite element method (XFEM) [25–27] and the mesh-free [28–33] method, which can perform high accuracy analyses without relying on meshes are researched in order to avoid the complex remeshing problems.

The XFEM is a method to model discontinuities and singularities independently of the mesh using enrich functions. In the crack propagation problems using the XFEM, even if relatively coarse elements are arranged around the crack tips, the high accuracy analyses are performed. Additionally, the efficient analyses are performed because the models of the cracks are completed by extending simplify the crack lines or surfaces. Among the mesh-free methods, there are the methods using only nodal data such as Smoothed Particle Hydrodynamics: SPH [28,29] and so on and those using both nodal data and background cell for integration such as Element Free Galerkin Method: EFGM [30–33] and so on. Recently, the method coupling the XFEM and the EFGM [21,34,35] has been developed. The XFEM and the mesh-free method have been developed in order to avoid the complex remeshing problems.

On the other hand, if the high-quality, fast and automatic adaptive remeshing can be performed, the adaptive finite element analyses are performed with the same level of accuracy and calculation cost as the XFEM and the EFGM. Therefore, the hierarchical mesh is applied here to the adaptive crack propagation problems.

In the following section, we describe an adaptive analysis in general and Section 3 covers the generation and the properties of the hierarchical mesh. In Section 4, we discuss an issue of the hierarchical mesh in the case of the adaptive analysis with a solution to it. Some numerical results on the crack propagation problems are given in Section 5 and we conclude the present paper in Section 6.

2. Adaptive analysis

Fig. 1(a) shows the flowchart of the traditional adaptive analysis method. If the value of the posterior error is larger than that of the maximum permissible error, re-meshing based on the posterior error is performed. If this is not the case, the calculation is finished. On the other hand, the present method is characterized by the new preprocessing: the hierarchical mesh generation, as shown in Fig. 1(b).

2.1. Posterior error estimation

First, a scalar quantity $\|E\|$ in each element is defined as

$$\|E\|^2 \equiv \int_{\Omega^e} \sigma^{\mathrm{T}} \sigma \, \mathrm{d}\Omega, \tag{1}$$

where $\int_{\Omega^e}()\mathrm{d}\Omega$ is the integral over the element region Ω^e, and σ is the stress.

Next, an index of error is defined as

$$\|e\|^2 \equiv \int_{\Omega^e} (\sigma - \hat{\sigma})^{\mathrm{T}} (\sigma - \hat{\sigma}) \mathrm{d}\Omega, \tag{2}$$

in which σ is assumed to be the exact solution of the stress, and $\hat{\sigma}$ the FEM solution. Then, the error norm of the entire region is approximated as

$$\|e\|^2_{total} \approx \sum_{m=1}^{M} \|e\|^2_m, \tag{3}$$

where M is the number of elements in the whole domain. As we cannot obtain the exact solution rarely, the error norm is approximated by substituting $\tilde{\sigma}$ instead of the exact solution σ, where $\tilde{\sigma}$ is chosen to be closer to the exact solution than the FEM solution $\hat{\sigma}$. In practice, $\tilde{\sigma}$ is determined using the one-order-higher shape function rather than using $\hat{\sigma}$ (see SubSection 2.2 for details). Thus, we have

$$\|e\|^2 = \int_{\Omega^e} (\tilde{\sigma} - \hat{\sigma})^{\mathrm{T}} (\tilde{\sigma} - \hat{\sigma}) \mathrm{d}\Omega. \tag{4}$$

The relative error norm of the entire region of the analysis model is defined as

$$\eta = \frac{\|e\|_{total}}{\|E\|_{total}}, \tag{5}$$

where

$$\|E\|^2_{total} = \sum_{m=1}^{M} \|E\|^2_m = \sum_{m=1}^{M} \int_{\Omega^e} \hat{\sigma}^{\mathrm{T}} \hat{\sigma} \mathrm{d}\Omega. \tag{6}$$

We take η as an acceptance criterion for the re-meshing. If η is larger than a suitable value for the maximum permissible relative error $\bar{\eta}$, the re-meshing is performed, where the error of the entire region $\bar{\eta}\|E\|_{total}$ is distributed uniformly in all the elements of the region to be solved. Thus, the maximum permissible relative error of each element $\bar{e}_{max}$ is given as

$$\bar{e}_{max} = \bar{\eta} \frac{\|E\|_{total}}{\sqrt{M}}. \tag{7}$$

Since the error norm of each element converges with the order of the degree of the shape function, a new element size h^{new} could be defined as

$$h^{new} = \frac{h^m}{\xi_m^{1/p}}, \tag{8}$$

where h^m is the current element size,p the degree of the shape function, and

$$\xi_m = \frac{\|e\|_m}{\bar{e}_{max}}. \tag{9}$$

The bounds of h^{new} are generally determined as

$$h^{min} \leqslant h^{new} \leqslant h^{max}, \tag{10}$$

where h^{max} and h^{min} are the upper and lower bounds for the element size, respectively.

2.2. Zienkiewicz–Zhu's method [19,20]

As is well known, $\hat{\sigma}$ is represented as

$$\hat{\sigma} = DBu, \tag{11}$$

where u is the nodal displacement vector, D the stress–strain matrix, and B the strain–node displacement matrix. On the other hand, $\tilde{\sigma}$ is determined using the one-order-higher shape function rather than using $\hat{\sigma}$ as

$$\tilde{\sigma} = N\bar{\sigma}, \tag{12}$$

where $\bar{\sigma}$ is a variable and N the shape function of the FEM.

If $\bar{\sigma}$ is determined by minimizing the error norm (4), a regular linear system

$$\left(\int_{\Omega} N^{\mathrm{T}} N \mathrm{d}\Omega\right) \bar{\sigma} = \int_{\Omega} \left(N^{\mathrm{T}} DBu\right) \mathrm{d}\Omega \tag{13}$$