Engineering Analysis with Boundary Elements 50 (2015) 188–197

Contents lists available at ScienceDirect

Engineering Analysis with Boundary Elements

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

An ultra-accurate hybrid smoothed finite element method for piezoelectric problem

Eric Li [a], Z.C. He [a,*], L. Chen [b], Bing Li [c], Xu Xu [d], G.R. Liu [e]

[a] State Key Laboratory of Advanced Design and Manufacturing for Vehicle Body, Hunan University, Changsha 410082, P.R. China
[b] Department of Materials Science and Engineering, Penn State University, University Park, PA 16802, USA
[c] LTCS and Department of Mechanics and Aerospace Engineering, College of Engineering, Peking University, Beijing 100871, P.R. China
[d] College of Mathematics, Jilin University, 2699 Qianjin Street, Changchun 130012, PR China
[e] School of Aerospace Systems, University of Cincinnati, Cincinnati, OH 45221-0070, USA

ARTICLE INFO

Article history:
Received 22 January 2014
Received in revised form
6 June 2014
Accepted 18 August 2014
Available online 18 September 2014

Keywords:
Numerical methods
Hybrid smoothed finite element method (HS-FEM)
Meshfree method
Piezoelectricity

ABSTRACT

An ultra-accurate hybrid smoothed finite element method (HS-FEM) is presented for the analysis of piezoelectric structures, in which the electrostatic equations governing piezoelectric problem are solved numerically with simplest triangular elements in 2D and tetrahedral elements in 3D. In the present method, the strain field is assumed to be the weighted average between compatible strains from finite element method (FEM) and smoothed strains from node-based smoothed finite element method (NS-FEM). Numerical results demonstrate that the proposed method possesses a novel bound solution in terms of strain energy and eigenfrequencies, which is very important for safety and reliability assessments of piezoelectric structural properties. In addition, the numerical results obtained from HS-FEM are much more accurate than the standard finite element method using the same of nodes. Furthermore, the computational efficiency of HS-FEM is much better than the FEM.

1. Introduction

Smart structures and materials have experienced tremendous growth over the past decade in the field, and they have been widely used in various applications such as sensors, actuators, transducers or active damping devices with the control systems. These applications are not only available in micro–electro-mechanical systems of sub-millimeter length scales but also in the design of smart electromechanical structures of large scales [1]. Piezoelectric material is one of such smart material. The transfers between electrical and mechanical energy is known as the piezoelectric effects. The direct piezoelectric effect can be seen when it is deformed to generate charge. On the other hand, the converse piezoelectric effect can be observed when stress or strain is produced in piezoelectric material with applied electric field.

Further development of piezoelectric actuators or generators demands robust, stable and accurate numerical methods. The analytical solutions are only available for simple case such as the analysis of smart beams with embedded or surface-distributed piezoelectric sensors and actuators [2–4]. Currently, the finite element method (FEM) is probably the most popular numerical method in the simulation of piezoelectric problem. A 3D finite element framework is developed by Duan et al. [5] with fully coupling among the piezoelectric coupled stator, contact interface and rotor to study the steady state transient performance of ultrasonic motor. Sze et al. has derived a four-node plane, a nine-node plane and a four-node axisymmetric stabilized elements for piezoelectric analysis [6]. Based on the classical plate theory, Lam and Liu have built up finite element model [7,8] for the active vibration control of beams and plates containing distributed sensors and actuators subjected to both mechanical and electrical loadings. Furthermore, Nguyen et al. have developed extended finite element method for dynamic fracture of piezoelectric materials [9].

Although the FEM has achieved remarkable progress in the simulation of piezoelectric problem, there are certain inherent drawbacks that limits the application of FEM [10]. The first issue is overestimation of stiffness matrix due to the 'overly-stiff' phenomenon of a fully compatible FEM model of assumed displacement based on the Galerkin weak form, which can cause 'locking' behavior and poor accuracy in stress solution. The second issue is that the FEM is limited by the rigid reliance on the elements. The third issue is mesh generation. Although the triangular elements in 2D and tetrahedral elements in 3D are very easy to generate for

* Corresponding author.
E-mail address: acessg2012@gmail.com (Z.C. He).

http://dx.doi.org/10.1016/j.enganabound.2014.08.005
0955-7997/

problems with complex geometry automatically using commercial software, the FEM dislikes the triangular elements or tetrahedral elements due to poor accuracy for these kinds of elements [10].

In recent years, strain smoothing techniques have been applied by Chen et al. [11] to stabilize the solutions of nodal integrated meshfree methods and also in the natural element method [11]. Using the generalized gradient smoothing technique, Liu and coworkers have developed a series of smoothed finite element methods (SFEM), which can effectively soften the stiffness of the model and hence make these methods possess a number of attractive features [12–15]. Currently, the SFEM has been extended to solve many engineering problems such as fracture mechanics [16], plate [17] and vibration [18] problems.

With the node-based strain smoothing operation, the node based smoothed point interpolation method (NS-PIM) and a node-based smoothed finite element method NS-FEM [19,20] have been formulated in the frame of meshfree and FEM, respectively. It has been found that the NS-PIM and NS-FEM work well with linear (triangular for 2D and tetrahedral for 3D) elements, are free from volumetric locking, and especially can provide upper bound solutions in energy norm for elasticity problems [21,22]. On the other hand, due to 'stiffer' stiffness matrix in FEM model, the displacement-based fully compatible FEM gives a lower bound in energy norm for the exact solution to elasticity problems [23–25]. Therefore, the certified solutions with both upper and lower bounds can be achieved using both NS-FEM and FEM. The important point is that FEM and NS-FEM play complementary roles in the numerical analysis for solution bounds. Based on this idea, Xu et al [26] has developed hybrid smoothed finite element method (HS-FEM) by combining the good features of two methods into a single numerical method. This newly developed method is able to provide a new means to obtain even nearly exact solutions.

Lured by excellence feature of HS-FEM, the HS-FEM is further extended to analyze static and frequency analyses of piezoelectric structures. A parameter α is equipped in the HS-FEM for ultra-accurate solutions. With adjustment of α value, a solution that is as close as possible to the exact solution can be obtained using a finite number of triangular elements or tetrahedral elements. Additionally, both the lower and upper bound of the exact solution in terms of strain energy and eigenfrequencies are obtained.

The outline of this paper is as follows: in Section 2, the Galerkin finite element method formulation for piezoelectric problem is presented. The concept of HS-FEM is illustrated in Section 3. In Section 4, the detailed procedure of determination of α value is presented. In Section 5, intensive numerical examples including 2D and 3D are examined to study the accuracy, convergence rate and efficiency of the present method. Finally, Section 6 concludes this work.

2. Formulation of the piezoelectric problem using finite element method

The governing equation in piezoelectric problem can be written in the following form:

$$div\boldsymbol{\sigma}+\mathbf{b}=0 \tag{1}$$

$$div\mathbf{D}+q_s=0 \tag{2}$$

where $\boldsymbol{\sigma}$ is the Cauchy stress tensor, $\mathbf{b}$ denotes the vector of body force applied in the problem domain, $\mathbf{D}$ represents the eclectic displacement and q_s is the free point charge density.

If inertia effect is considered in piezoelectric problem, the strong form of governing equation can be expressed:

$$\nabla\boldsymbol{\sigma}+\mathbf{b}=\rho\ddot{\mathbf{u}}+g\dot{\mathbf{u}} \tag{3}$$

where ρ is the density of the mass, and g is a set of viscosity.

The mechanical and electric boundary conditions are given as

$$\phi=\phi_\Gamma \quad \mathbf{u}=\mathbf{u}_\Gamma \quad \text{on } \Gamma_u \quad \text{essential boundary condition} \tag{4}$$

$$\sigma\bullet\mathbf{n}=\mathbf{t}_\Gamma \quad \mathbf{D}\bullet\mathbf{n}=q_\Gamma \quad \text{on } \Gamma_t \quad \text{natural boundary condition} \tag{5}$$

where ϕ_Γ and $\mathbf{u}_\Gamma$ represent the prescribed electrical potential and the vector of the prescribed displacement; q_Γ and $\mathbf{t}_\Gamma$ denote the surface charge and the vector of prescribed tractions; and n_j is the surface outward normal of the boundary.

The general functional L is determined by a summation of the kinetic energy, strain energy, dielectric energy and potential energy arising from external work

$$L=\int_\Omega\left[\frac{1}{2}\rho\dot{\mathbf{u}}^T\dot{\mathbf{u}}-\frac{1}{2}\boldsymbol{\varepsilon}^T\boldsymbol{\sigma}+\frac{1}{2}\mathbf{D}^T\mathbf{E}+\mathbf{u}^T\mathbf{f}_s-\phi\mathbf{q}_s\right]d\Omega+\int_V\mathbf{u}^T\mathbf{f}_b dV+\sum\mathbf{u}^T\mathbf{F}_P-\sum\phi\mathbf{Q}_p \tag{6}$$

where $\mathbf{u}$ and $\dot{\mathbf{u}}$ are the vectors of mechanical displacements and velocity; $\mathbf{F}_P$, $\mathbf{f}_b$ and $\mathbf{f}_s$ are the vectors of point, mechanical body, surface forces; $\mathbf{q}_s$ and $\mathbf{Q}_p$ are the vectors of surface and point charges respectively; ϕ is electric potentials; $\boldsymbol{\sigma}$, $\boldsymbol{\varepsilon}$, $\mathbf{D}$, $\mathbf{E}$ are vectors of mechanical stresses, mechanical strains, electric displacements and electric fields respectively. For the linear electroelastic problem, the constitutive equations have the following form:

$$\{\boldsymbol{\sigma}\}=[\mathbf{c}^E]\{\boldsymbol{\varepsilon}\}-[\mathbf{e}]^T\{\mathbf{E}\} \tag{7}$$

$$\{\mathbf{D}\}=[\mathbf{e}]\{\boldsymbol{\varepsilon}\}+[\boldsymbol{\kappa}^s]\{\mathbf{E}\} \tag{8}$$

where $\mathbf{c}^E$ is the elastic matrix for constant electric field, $\boldsymbol{\kappa}^s$ is the dielectric constant matrix for constant mechanical strain and $\mathbf{e}$ is the piezoelectric matrix.

The strain $\boldsymbol{\varepsilon}$ and the electric field $\mathbf{E}$ are, respectively, derived from the displacement $\mathbf{u}$ and the electric potential φ, and could be expressed in the following vector form:

$$\boldsymbol{\varepsilon}=\nabla_s\mathbf{u} \tag{9}$$

$$\mathbf{E}=-\text{grad}\varphi \tag{10}$$

where ∇_s is the symmetric gradient operator,

$$\nabla_s=\begin{bmatrix}\frac{\partial}{\partial x} & 0 & \frac{\partial}{\partial y}\\ 0 & \frac{\partial}{\partial y} & \frac{\partial}{\partial x}\end{bmatrix}^T \quad 2D \tag{11}$$

$$\nabla_s=\begin{bmatrix}\frac{\partial}{\partial x} & 0 & 0 & \frac{\partial}{\partial y} & 0 & \frac{\partial}{\partial z}\\ 0 & \frac{\partial}{\partial y} & 0 & \frac{\partial}{\partial x} & \frac{\partial}{\partial z} & 0\\ 0 & 0 & \frac{\partial}{\partial z} & 0 & \frac{\partial}{\partial y} & \frac{\partial}{\partial x}\end{bmatrix}^T \quad 3D \tag{12}$$

using the FEM approximation method, the mechanical displacements and electrical potentials are similarly interpolated with approximate shape functions in the following expressions:

$$\mathbf{u}(\mathbf{x})=\sum_{i\in n_n^e}\mathbf{N}_i(\mathbf{x})\mathbf{d}_i \tag{13}$$

$$\phi(x)=\sum_{i\in n_n^e}\mathbf{N}_i(\mathbf{x})\phi_i \tag{14}$$

where n_n^e is the number of the nodal variables of the element, $\mathbf{d}_i$ is the nodal displacement vector, ϕ_i is the nodal electric potential and $\mathbf{N}_i(\mathbf{x})$ is the shape function matrix.

With the help of Eqs. (13) and (14), the linear strain $\boldsymbol{\varepsilon}$ and electric field $\mathbf{E}$ are derived from

$$\boldsymbol{\varepsilon}=\nabla_s\mathbf{u}=\sum_{i\in n_n^e}\mathbf{B}_i^u\mathbf{d}_I \tag{15}$$

$$\mathbf{E}=-\text{grad}\phi=-\sum_{i\in n_n^e}\mathbf{B}_i^\phi\phi_I \tag{16}$$