Finite Elements in Analysis and Design 88 (2014) 16–24

Contents lists available at ScienceDirect

Finite Elements in Analysis and Design

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

Large increment method for elastic and elastoplastic analysis of plates

Hong-xue Jia [a,*], Xi-la Liu [b]

[a] Department of Engineering Mechanics, School of Naval Architecture, Ocean And Civil Engineering, Shanghai Jiao Tong University, Shanghai 200240, PR China

[b] Department of Civil Engineering, School of Naval Architecture, Ocean and Civil Engineering, Shanghai Jiao Tong University, Shanghai 200240, PR China

ARTICLE INFO

Article history:
Received 26 August 2013
Received in revised form
6 May 2014
Accepted 7 May 2014
Available online 4 June 2014

Keywords:
Displacement-based finite element method
Material nonlinear analysis
Error accumulation
Large Increment Method
Mindlin–Reissner plate theory

ABSTRACT

The displacement-based finite element method (FEM) has become a method of choice for material nonlinear analysis of plates. For material nonlinear problems, the displacement-based FEM relies upon a step-by-step incremental approach and the repetitive computation of the systematic stiffness matrix. These shortcomings lead to the error accumulation and huge computational consumption, which encourage the reconsideration of force-based methods for elastoplastic problems. In this paper, a force-based Large Increment Method (LIM) is employed for the elastoplastic analysis of plates using a force-based 4-node quadrilateral plate element which is based on Mindlin–Reissner plate theory. The consistent elastoplastic flexibility matrix of plate element is derived and implemented to solve elastoplastic plate problems. Two numerical examples are presented to illustrate the mesh convergence of the plate element by solving the linear elastic thin and moderately thick plate problems by comparing with the analytical solutions and displacement-based plate elements. Two simple elastoplastic plate problems are presented to illustrate the accuracy and the computational efficiency of LIM by comparing with the results from the FEM software ABAQUS.

1. Introduction

Over the last few decades, the displacement-based finite element method (FEM) has been proved to be a highly powerful tool for analyzing a wide variety of physical problems in many areas of engineering. In these fields, two distinct finite element methods named the displacement-based FEM and the force-based FEM have been developed for analyzing very complicated nonlinear problems in structural mechanics and solid mechanics. In recent decades, the displacement-based FEM has dominated current practice for the numerical simulations of nonlinear analysis of structures due to its straightforward implementation and easy computer automation. However, many shortcomings of the FEM have been observed in the analysis of the material nonlinear problems and can be summarized as follows: (a) the error accumulation, which mainly comes from the step-by-step solution procedure, (b) huge computing time consumption, which is due to obtain accurate results with many small enough steps and an iterative algorithm to solve the nonlinear system of equations. At each step, the linearized constitutive relation may vary from the previous step, which will lead to a different system stiffness matrix. Consequently, the different system stiffness matrix has to be handled again and again during the step-by-step process, and (c) inaccuracy of stress, the stresses are calculated indirectly using the differentiation from the displacement, which may cause numerical errors in stress predictions. These shortcomings lead to the reconsideration of the force-based approaches for the material nonlinear problems. It is well known that, the main unknowns in the traditional force-based method are the redundant forces. However, this approach is not easy to implement in computer software because there are difficulties in the selection of the statically determinate structures and constructing the self-stress matrix, and an improper selection may lead to the programming difficulty and computational instability. A number of novel schemes have extended the force method for linear and nonlinear problems [1–5]. The main purpose of these researchers is to overcome the shortcomings of the traditional force-based method and the displacement-based FEM. By far, many force-based finite element methods still have to linearize the constitutive model and the error accumulation problem is not suppressed in essence.

As a novel iterative displacement-based method, the large time increment method (LATIN) is proposed for the material nonlinear problem by Ladevèze [6]. In recent years, the LATIN has been extended to solve many elastoplastic material [7,8], frictional contact [9], nonlinear multiscale [10] and parallel computing problems [11]. In LATIN, the governing equations are divided into

* Corresponding author. Tel.: +8615821795650.
E-mail address: jiahongxue211@gmail.com (H.-x. Jia).

http://dx.doi.org/10.1016/j.finel.2014.05.006
0168-874X/

two groups: the global equilibrium and compatibility equations, and the local constitutive equations. This method relies on a step-by-step solution process at the system analysis stage and nonlinear iterative procedure at the local stage using different search direction. It should firstly assume the material behavior to form the initial the system stiffness matrix for getting the initial solution, which can lead to error accumulation. The large increment method (LIM), which is first proposed by Zhang and Liu [4] for material nonlinear problem, is a novel iterative force-based FEM. In LIM, all the independent internal element forces are adopted as system unknowns and the basic determinate structure is no needed, so the LIM becomes a systematic method and it is easy to implement in the computer program. Like the LATIN, the governing equations of LIM are also divided into two groups: the global equilibrium and compatibility equations, and the local constitutive equations. In the case of small deformation, the linear equilibrium equations and the compatibility equations are treated in the global stage and are only solved one time, which will lead to less time consumption. The nonlinear constitutive relation is treated in the local element stage separately. By separating the governing equation into two stages, the constitutive model need not be linearized. Hence the step-by-step solution process is replaced by an overall iterative procedure, and the error accumulation can be avoided. In other words, the loading process often can be handled in one large increment for monotonic loading or a few large increments for cyclic loading. In recent years, LIM has been extended for solving elastic-perfectly plastic analysis of plane frame structures under monotonic and cyclic loading and 2D continuum elastoplastic problems [12–14]. The accuracy and efficiency of LIM have been proved. For the parallel computing in the time and spatial domain of LIM, it has great potential for solving large scale structural problems with significant computational saving [15].

In this paper, LIM is employed to analyze the elastic and elastoplastic plate problems using a force-based 4-node quadrilateral plate element (PQ2) [16]. Meanwhile, the consistent elastoplastic flexibility matrix of plate element is derived and implemented to solve elastoplastic plate problems. The PQ2 element has no extra spurious energy modes and can avoid shear locking for solving the thin plate. Two linear elastic plate bending problems are presented to further illustrate the mesh convergence and accuracy of the PQ2 element. Two simple elastoplastic plate problems are presented to study the accuracy and the computational efficiency of LIM by comparing with the results from the FEM software ABAQUS. All the numerical results show that the performance of the LIM is as good as the commercial FEM software in linear elastic and elastoplastic problems.

The outline of the paper is as follows: In Section 2, the basic equations of LIM for plate element is presented. The consistent elastoplastic flexibility matrix is given in Section 3. In Section 4 we present the basic theory of LIM. Numerical examples are provided in Section 5 and some conclusions are stated in Section 6.

Only small deformation and small displacement are considered in this paper.

2. The governing equations of plate element

In LIM, the physical domain of the structure is Ω with boundary S. The boundary is divided into two parts: the traction boundary S_σ and the displacement boundary S_u, where $S_\sigma \cap S_u = 0$. The stress resultant $\boldsymbol{M}$ and displacement $\boldsymbol{u}$ at each point in any element can be given by the element generalized force variables and element nodal displacement variable, respectively. The stress resultant fields of the plate element are written as

$$\boldsymbol{M} = \mathbf{Z}\boldsymbol{F}^e \tag{1}$$

where $\boldsymbol{M}$ is the stress resultant vector of plate element, $\mathbf{Z}$ is the shape function of generalized force parameter vector, and $\boldsymbol{F}^e$ denotes the elemental generalized force parameters vector.

The displacement fields in element are described by

$$u = N\boldsymbol{d}^e \tag{2}$$

and the strain vector in element is given by

$$\hat{\boldsymbol{\varepsilon}} = \mathrm{L}u = \mathrm{LN}\,\boldsymbol{d}^e = \boldsymbol{B}\boldsymbol{d}^e \tag{3}$$

where $\boldsymbol{u}$ is the displacement vector, $\boldsymbol{N}$ is the shape function of the node displacement vector, $\boldsymbol{d}^e$ denotes the nodal displacement vector of an element, $\hat{\boldsymbol{\varepsilon}}$ is the strain vector, and $\boldsymbol{L}$ is the differential operator.

Using the principle of virtual work, one finds

$$\int_{\Omega^e} \boldsymbol{M}\cdot\delta\hat{\boldsymbol{\varepsilon}}\,\mathrm{d}\Omega = \int_{\Omega^e} \boldsymbol{b}\cdot\delta\boldsymbol{u}\,\mathrm{d}\Omega + \int_{S_\sigma^e} \boldsymbol{t}\cdot\delta\boldsymbol{u}\,\mathrm{d}S \tag{4}$$

where $\delta\hat{\boldsymbol{\varepsilon}}$, $\boldsymbol{b}$, $\delta\boldsymbol{u}$, $\boldsymbol{t}$ and S_σ^e denote the virtual strain, the body force vector, the virtual displacement, the traction force vector and the traction boundary of the element, respectively.

Substituting the Eqs. (1) and (3) into the Eq. (4), the Eq. (4) can be rewritten as

$$\delta\boldsymbol{d}^{eT}\left(\int_{\Omega^e}\boldsymbol{B}^T\mathbf{Z}\,\mathrm{d}\Omega\right)\boldsymbol{F}^e = \delta\boldsymbol{d}^{eT}\left(\int_{\Omega^e}\boldsymbol{N}^{\mathrm{T}}\boldsymbol{b}\,\mathrm{d}\Omega + \int_{S_\sigma^e}\boldsymbol{N}^{\mathrm{T}}\boldsymbol{t}\,\mathrm{d}S\right) \tag{5}$$

and the element equilibrium equations can be expressed as

$$\boldsymbol{C}^e\boldsymbol{F}^e = \boldsymbol{P}^e \tag{6}$$

where

$$\begin{cases} \boldsymbol{C}^e = \int_{\Omega^e}\boldsymbol{B}^{\mathrm{T}}\mathbf{Z}\,\mathrm{d}\Omega \\ \boldsymbol{P}^e = \int_{\Omega^e}\boldsymbol{N}^{\mathrm{T}}\boldsymbol{b}\,\mathrm{d}\Omega + \int_{S_\sigma^e}\boldsymbol{N}^{\mathrm{T}}\boldsymbol{t}\,\mathrm{d}S \end{cases} \tag{7}$$

In Eq. (7), $\boldsymbol{C}^e$ is the element equilibrium matrix, $\boldsymbol{P}^e$ is the element equivalent nodal force vector.

The equilibrium equations of the system which are obtained by assembling the element equilibrium equations are given as

$$\boldsymbol{C}\boldsymbol{F} = \boldsymbol{P} \tag{8}$$

where $\boldsymbol{C}$ is an $m\times n$ equilibrium matrix of the structure, and it is assembled from the element equilibrium matrix. For the statically indeterminate structures, the system equilibrium matrix $\boldsymbol{C}$ is an $m\times n$ non-square matrix with $m<n$. $\boldsymbol{F}$ is an $n\times 1$ generalized inner force vector of the structure, and $\boldsymbol{P}$ is an $m\times 1$ node load vector.

Similarly, using the principle of complementary virtual work, one finds

$$\int_{\Omega^e}\delta\boldsymbol{M}^T(\hat{\boldsymbol{\varepsilon}} - \boldsymbol{B}\boldsymbol{d})d\Omega = 0 \tag{9}$$

Substituting Eqs. (1) and (3) into Eq. (9), the assembled compatibility equations of the system can be expressed as

$$\boldsymbol{C}^T\boldsymbol{D} = \boldsymbol{\delta} \tag{10}$$

where $\boldsymbol{D}$ is the nodal displacement vector of the structure and $\boldsymbol{\delta}$ is the generalized deformation vector of the structure.

Then, the element constitutive equation can be written as

$$\boldsymbol{\delta}^e = \Phi^e(\boldsymbol{F}^e, \kappa) \tag{11}$$

where

$$\begin{cases} \boldsymbol{\delta}^e = \int_{\Omega^e}\boldsymbol{Z}^T\hat{\boldsymbol{\varepsilon}}\,\mathrm{d}\Omega \\ \boldsymbol{\Phi}^e(\boldsymbol{F}^e,\kappa) = \int_{\Omega^e}\boldsymbol{Z}^T\boldsymbol{\phi}(\boldsymbol{M},\kappa)\mathbf{Z}\,\mathrm{d}\Omega \end{cases} \tag{12}$$