Computers and Structures 191 (2017) 180–192

Contents lists available at ScienceDirect

Computers and Structures

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

A four-node tetrahedral element with continuous nodal stress

Yongtao Yang [a,*], Guanhua Sun [a], Hong Zheng [b]

[a] State Key Laboratory of Geomechanics and Geotechnical Engineering, Institute of Rock and Soil Mechanics, Chinese Academy of Sciences, Wuhan, China
[b] Key Laboratory of Urban Security and Disaster Engineering, Ministry of Education, Beijing University of Technology, Beijing 100124, China

ARTICLE INFO

Article history:
Received 13 June 2017
Accepted 3 July 2017

Keywords:
Partition of unity
FE-Meshfree element
Tetrahedral element
Tetr4-CNS
Mesh distortion

ABSTRACT

Formulation of a partition of unity (PU) based four-node tetrahedral element with continuous nodal stress (Tetr4-CNS) and its applications to the analysis of linear elasticity problems in three-dimension are presented in this paper. By simply using the same mesh as the classical tetrahedral element (Tetr4), Tetr4-CNS element is able to obtain continuous nodal stress without recourse to stress smoothing operation in the post-processing process, and to construct high order global approximation without adding extra nodes or nodal DOFs. Moreover, it is free from the linear dependence problem which cripples many of the PU-based methods. A series of numerical tests are carried out to evaluate the performance of the Tetr4-CNS element. The numerical results show that accuracy through the proposed element is superior to that through Tetr4 element and eight-node hexahedral element (Hexa8). More importantly, the proposed element has excellent mesh distortion tolerant capabilities.

1. Introduction

In simulating engineering problems, such as dam engineering, tunneling and underground space, it is of great importance to obtain the displacement and stress fields around the concerned areas. If the displacement or stress in these areas exceeds allowable value, then reinforcement measures should be carried out. Accurate prediction of the displacement and stress fields within the problem domain using analytical or semi-analytical method is only available for problems with simple geometry and boundary conditions. When it comes to deal with engineering problems with complex geometry and boundary conditions, numerical approaches have shown to be more suitable.

Although the finite element method (FEM) [1,2] has been one of the most popular numerical methods and successfully used in many fields of engineering, the method however is not free from drawbacks. The shape function for FEM with standard DOF is only C^0 continuous, thus the nodal gradient fields, e.g., the stress field, is discontinuous across element boundaries, and stress smoothing operation is needed in the post-processing process. Moreover, results obtained from some classic isoparametric elements, such as parabolic quadrilateral element (Quad8) are very sensitive to mesh distortions [3]. Since the FEM has to deploy conforming mesh to discretize the domain of problem, distorted mesh cannot be avoided in solving geometric nonlinear problems and crack propagation problems. Recent effort on reducing mesh distortion in solving crack propagation problems can be found in [4,5].

Alternatively, a series of other numerical methods have been proposed to overcome the difficulty encountered in FEM, such as the class of meshfree methods [6–10], the extended finite element method (XFEM) [11], the generalized finite element method (GFEM) [12], the numerical manifold method [13–19] and the isogeometric analysis [20,21]. Each numerical method has its own advantages and disadvantages. For example, the meshfree methods have advantages in solving crack propagation problems [22] and impact-induced failure [23], because they do not need a mesh to discretize the computational domain and are therefore immunized from mesh distortions. However, shape function for some of the meshfree methods may not possess the much desired Kronecker-delta property, resulting in a more complex manner to impose the essential boundary conditions than the FEM. Besides, high computational cost and complex process in constructing the shape function will deteriorate the stability and efficiency of numerical integration [24].

Recently, a partition of unity (PU) based "FE-Meshfree" 3-node triangular element with continuous nodal stress (Trig3-CNS) [25] was proposed for the analysis of mechanical stress of two-dimensional problems. Subsequently, the Trig3-CNS element was extended to study free vibration problems [26] and enriched by crack-tip functions for dealing with crack propagation problems [27]. By using the same mesh as the classical 3-node triangular element (Trig3), Trig3-CNS element is capable of obtaining continuous nodal stress without recourse to stress smoothing operation

* Corresponding author.
E-mail address: scuhhc@126.com (Y. Yang).

http://dx.doi.org/10.1016/j.compstruc.2017.07.001
0045-7949/

during the post-processing process, and to construct high order global approximation without adding extra nodes or nodal DOFs. Moreover, it is free from the "linear dependence" (LD) problem [12,28] which cripples many of the PU-based methods, such as the GFEM [12] and the NMM [17]. Here, the "linear dependence" (LD) problem means after applying the basic boundary condition to eliminate the rigid body displacement, the global stiffness matrix is still singular.

Apart from Trig3-CNS element, two PU-based "FE-Meshfree" 4-node quadrilateral elements with continuous nodal stress were also proposed and applied for two dimensional solid problems [29–31]. In other front, based on the "twice-interpolation" procedure, Zheng et al. [24] and Bui et al. [32] also proposed 3-node triangular element and 4-node quadrilateral element with continuous nodal stress.

From the viewpoint of practical applications, only a few problems could be simplified into 2D models [33,34]. In most cases, engineers and researchers have to deal with 3D problems. Therefore, there are natural demands in developed effective numerical methods which are capable of accurately simulating 3D problems [35].

However, solving 3D problems is usually much more difficult due to the complexity of the geometry. When the FEM is used, meshing a complicated 3D domain with quality elements can be quite a difficult task to any analyst. The four-node tetrahedral element (Tetr4) is often used for 3D problems, because of its simplicity in formulation and implementation. More importantly, most commercial FEM software use tetrahedral elements for adaptive analyses of 3D problems, due to the simple fact that tetrahedral meshes can be most automatically generated and refined for complicated geometrical domains [36]. The Tetr4 element is thus clearly superior at least for two counts: simplicity and adaptation. However, the Tetr4 element also possesses some crucial shortcomings for problems of solid mechanics, such as poor accuracy. In order to overcome these disadvantages, some new elements were proposed in [35–37].

Inspired by the advantages of Trig3-CNS element [25] in solving 2D linear elastic problems, in this paper a partition-of-unity (PU) [28] based 4-node tetrahedral element with continuous nodal stress (Tetr4-CNS) is developed for linear elastic problems in 3D. Like Trig3-CNS element, Tetr4-CNS element is also capable of obtaining continuous nodal stress without smoothing operation, and constructing high order global approximation without adding extra nodes or nodal DOFs by simply using the same mesh as the Tetr4 element. Moreover, it is free from the LD problem.

The outline of this paper is as follows: Section 2 introduces the partition of unity method (PUM) briefly; Section 3 presents the formulation of Tetr4-CNS element in great detail; Section 4 gives the basic equations used in linear elastic analysis. Linear dependence test is carried out in Section 5, while numerical tests about five typical example problems are carried out in Section 6. Some conclusions are drawn in the last section.

2. Partition of unity method

As mentioned in the introduction section, formulations of the proposed Tetr4-CNS element are constructed based on the partition of unity (PU). Therefore, the Tetr4-CNS element can be considered as a partition of unity method (PUM). In PUM, the global approximation functions are constructed by using a set of non-negative weight functions (PU functions) multiplied with local approximations. According to Ref. [28], the summation of weight functions should be equal to one, namely,

$$\sum_{i=1}^{n} w_i(\boldsymbol{x}) \equiv 1 \tag{1}$$

where $w_i(\boldsymbol{x})$ is the weight function corresponding to node i, $\boldsymbol{x} = (x, y, z)$ represents the coordinates of an arbitrary point, n is the total number of the nodes in the computational domain V. It is noticed that, in the early version of PUM, the requirement for $w_i(\boldsymbol{x}) \geqslant 0$ is discarded, which is slightly different from the standard statements of the partition of unity theorem [38,17].

The global approximation can then be expressed in the following form:

$$u^h(\boldsymbol{x}) = \sum_{i=1}^{n} w_i(\boldsymbol{x}) u_i(\boldsymbol{x}) \tag{2}$$

where $u_i(\boldsymbol{x})$ is the local approximation associated with the node i. Here, $u_i(\boldsymbol{x})$ can be defined by using the known information about the boundary value problem. For example, when dealing with crack problems by XFEM [11] or NMM [17], asymptotic crack-tip functions capable of capturing the singular stress field near the crack tips are used to define the local approximations of the nodes near the crack tips. In this study, polynomial functions are employed to construct the local approximations of the Tetr4-CNS element, since only linear elastic continuous problems are considered.

3. Formulation of Tetr4-CNS element

The present Tera4-CNS element can be considered as a development of the Trig3-CNS [25] element and other "FE-Meshfree" elements [39–42]. Before discussing the global approximation of Tera4-CNS element, the definitions of nodal support and element support are given firstly.

In defining the support of a given node, such as node i, as shown in Fig. 1, the *first order nodal connectivity* which includes the nodes of all the elements connected to node i is usually considered. Such a support is called *first order nodal support*. As can be seen in Fig. 1, the node set $\{i\,1\,2\,3\,4\,5\,6\,7\,8\}$ is the *first order nodal support* of node i. Similarly, the *second order nodal support* is defined based on the *second order nodal connectivity* by including the nodes of all the elements connected to the nodes in the *first order support*. The element support, $\hat{\Omega}$, for a given element, such as i-j-k-l in Fig. 2 is therefore the union of the four nodal supports corresponding to nodes i, j, k and l, namely, $\hat{\Omega} = \bigcup_{i=1}^{4} \Omega_i$. Here, Ω_i is the nodal support of node i.

Now we consider a tetrahedral element defined by four nodes $\{P_1 P_2 P_3 P_4\}$ and introduce an arbitrary point $P(\boldsymbol{x})$ with the coordinates $\boldsymbol{x} = (x, y, z)$. According to Eq. (2), the Tera4-CNS global approximation, $u^h(\boldsymbol{x})$, in this element can be obtained through

$$u^h(\boldsymbol{x}) = \sum_{i=1}^{4} w_i(\boldsymbol{x}) u_i(\boldsymbol{x}) \tag{3}$$

in which, $w_i(\boldsymbol{x})$ and $u_i(\boldsymbol{x})$ are the weight function and the local approximation associated with node i, respectively.

3.1. Construction of weight functions

The formulation for coordinate transformation is represented as [1]

$$x = \sum_{i=1}^{4} \tilde{N}_i(\xi, \eta, \zeta) x_i \tag{4}$$

$$y = \sum_{i=1}^{4} \tilde{N}_i(\xi, \eta, \zeta) y_i \tag{5}$$

$$z = \sum_{i=1}^{4} \tilde{N}_i(\xi, \eta, \zeta) z_i \tag{6}$$