Engineering Analysis with Boundary Elements 84 (2017) 117–128

Contents lists available at ScienceDirect

Engineering Analysis with Boundary Elements

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

# Improved numerical manifold method (iNMM)—An extra-DOF free and interpolating NMM with continuous nodal stress

Guohua Zhang, Yongtao Yang*, Hao Wang

State Key Laboratory of Geomechanics and Geotechnical Engineering, Institute of Rock and Soil Mechanics, Chinese Academy of Sciences, Wuhan 430071, China

ARTICLE INFO

Keywords:
Numerical manifold method
Partition of unity
Linear dependence
Extra DOFs
Continuous nodal stress
FE-Meshfree element

ABSTRACT

As a partition of unity method (PUM), the numerical manifold method (NMM) is capable of constructing global approximation by simply multiplying PU function with local approximation. In order to enhance accuracy, high order polynomials can be specified as local approximation. This, however, will hinder the engineering application of NMM by its ill conditioning of the global stiffness matrix. In this study, an improved NMM (iNMM) without extra degree of freedoms (DOFs) is developed. Without the extra DOFs, the resulting global stiffness becomes linear independent. In addition, the stresses are continuous at all nodes. Numerical studies show the iNMM's excellent accuracy.



## 1. Introduction

Over the past three decades, the concept of partition of unity (PU) approximations has been established and a number of PU-based methods [1] were developed for solid mechanics, such as the partition of unity method [2–5], the generalized finite element method [6], the extended finite element method (XFEM) [7,8], the numerical manifold method (NMM) [9,10], the phantom-node method [11,12] and many others [13–16].

Since the advent, the NMM has attracted much interest from researchers in computational solid mechanics as it possesses several advantages over the FEM. For example, the local approximation function in NMM can be freely chosen so as to obtain higher resolution of the boundary value problem. In addition, the mathematical mesh in NMM does not have to match the material interface or the fracture face, indicating that NMM can always employ regular mesh to discretize the problem domain. This, however, is nearly impossible for FEM, when dealing with problems with complicated geometric boundaries. According to our experience from isoparametric elements, such as four-node isoparametric quadrilateral element (Quad4) and eight-node isoparametric quadrilateral element (Quad8) [17], regular mesh can generally achieve much better accuracy than distorted mesh. Moreover, NMM is very suitable for simulating problems with moving boundaries, such as crack propagation problems, while in FEM, the mesh has to be ceaselessly regenerated so as to match the evolving fracture face. Due to the attractive advantages, NMM has been successfully used to model static crack propagation problems [18–23], dynamic crack propagation problems [24], contact problems [25] seepage problems [26,27] and wave propagation problems [28].

Within the framework of PU-based method, high-order global approximations can be directly constructed in the NMM by simply adopting high-order polynomial local approximations. Use of smooth polynomial local approximations can achieve the following purposes [29,30]: (1) perform $p$-adaptive analysis without the addition of extra nodes; (2) avoid mesh grading yet obtain a same quality of approximation on a uniform mesh; (3) remove global refinement constraints. However, when both the PU function and the local approximations are simultaneously taken as high-order polynomials, the resulting global stiffness matrix will be "linear dependence" (LD) and special equation solver is needed, because traditional equation solver generally designed for positive definite equations cannot solve singular equations. Here, the LD problem means the global stiffness matrix is still singular even after the basic boundary condition to eliminate the rigid body displacement has been imposed.

The LD problem was first observed by Babuška and Melenk [2,3] when they designed a one-dimensional PUM approximation for the one-dimensional Helmholtz equation. To address LD problem, great efforts have been made in the past years by various means. An et al. [31] proposed an algorithm for predicting the rank deficiency of the stiffness matrix by using the topological information inheriting in the finite element mesh. Griebel and Schweitzer [32] proposed flat-top PU functions to avoid the linear dependence problems. The only problem is the complexity involved in the construction of the flat-top PU functions [29]. Tian et al. [33] carried out numerical experiments among

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

http://dx.doi.org/10.1016/j.enganabound.2017.08.011
Received 19 March 2017; Received in revised form 4 July 2017; Accepted 16 August 2017
Available online 31 August 2017
0955-7997/

several GFEMs to investigate the LD problem. Based on the numerical experiments, they proposed several approaches to eliminate the linear dependence problem, such as suppressing the higher-order degrees of freedom (DOF) and adjustment of the element geometry. However, as discussed in [34], these approaches [33] cannot ensure the removal of the LD problem and are also difficult to be implemented robustly in practice. In [34], Cai et al. developed a PU-based triangular element using a dual local approximation scheme by treating boundary and interior nodes separately. According to their report, the use of dual local approximation scheme can effectively remove the LD problem. Based on overlapping polyhedral covers generated from Voronoi cells, Riker and Holzer [35] proposed a mixed-cell-complex partition of unity method (MCCPUM) to eliminate the LD problem. However, the generation of mixed-cell-complex is very rather complicated and computationally expensive.

In other front, a family of PU-based "FE-Meshfree" elements was proposed in [36–39] which successfully eliminates the LD problem and the shape function possesses the desirable delta property. Although a least square version of point interpolation method (LSPIM) [36,37] or radial point interpolation method (RPIM) [40], which is time-consuming, is used to construct the local approximations of the "FE-Meshfree" elements, extra nodes or DOFs are not needed, because they just use the same mesh as in the FEM, and the total DOFs is the same as FEM. Numerical tests carried out in [36–38] have shown that the "FE-Meshfree" elements are computationally more efficient than FEM. If regular mesh is adopted, accuracy obtained through "FE-Meshfree" elements is much better than that obtained through FEM. If distorted mesh is adopted, FE-Meshfree elements have much better mesh-distortion tolerance than FEM.

Although high-order global approximations can be constructed easily in "FE-Meshfree" elements, the nodal stress is not continuous at nodes, and stress smoothing operation is needed in the post processing stage. To further improve the property of "FE-Meshfree" elements, Yang Zheng et al. [41–48] developed a series of "FE-Meshfree" elements with continuous nodal stress, such as the 'FE-Meshfree' three-node triangular element with continuous nodal stress using radial-polynomial basis functions (Trig3-RPIMcns) [48]. According to their report, Trig3-RPIMcns can obtain better accuracy, higher convergence rate and higher tolerance to mesh distortion than the three-node triangular elements (Trig3) and the four-node quadrilateral element (Quad4) for linear elastic, free vibration and forced vibration problems by simply using the same mesh as in Trig3. Since Trig3-RPIMcns has to deploy conforming mesh to discretize the problem domain, the time spent in mesh generation for problems with complex boundaries is not negligible. If crack propagation is involved, the burden of mesh generation is further amplified. This demerit hinders the applications of Trig3-RPIMcns for practical problems. Since there is no need for NMM to deploy conforming mesh, the mesh generation should be very convenient. Besides, NMM can always adopt regular mesh to discretize the problem domain, and mesh distortion, which results in poor accuracy for FEM, does not exist in NMM. Therefore, developing a method which combines the advantages of both the Trig3-RPIMcns and the NMM is essential.

In this study, an improved version of NMM (iNMM), which synergizes the advantages of both the Trig3-RPIMcns and the numerical manifold method (NMM), is developed for linear elastic problems. The property and performance of the iNMM will be studied in great detail in the rest of this paper. The outline of this paper is as follows: Section 2 briefly introduces the numerical manifold method (NMM); Section 3 presents the formulation of iNMM and the properties of the iNMM shape functions are discussed. Section 4 presents the discrete equations for linear elastic problems in the context of iNMM; Numerical examples and discussions are subsequently presented in Section 5. Some conclusions are drawn in the last section.

## 2. Basic concepts of NMM

The background of NMM has been described in great detail in [49]. Therefore, only the basic concepts are introduced in this section. To illustrate these concepts, an example shown in Fig. 1 is employed.

The core and most innovative feature of the NMM is the adoption of two cover systems, namely the mathematical cover (MC) and the physical cover (PC), from which the nodes and elements are generated.

The MC is the union of a series of user-defined overlapping small domains. Each small domain is called mathematical patch (MP). In Fig. 1, the MC is constructed by regular triangular mesh, and hence each MP is the union of several triangles sharing the same node such as $MP_1$. It is noticed that the MC does not have to match the material boundaries, holes or fracture faces of the problem domain, but have to cover the problem domain completely.

The PC is the union of all the physical patches (PPs). The PPs are generated by intersecting all the MPs with the physical mesh. Here, the physical mesh is the union of all the material interfaces, joints, fractures and domain boundaries, which are used to define the unique problem domain. From a MP, at least one PP can be generated, such as $PP_2$, $PP_3$, $PP_4$, and $PP_5$ (Fig. 1). It is noticed that each PP corresponds to a "NMM node" (also named as "generalized node"), on which the degree of freedoms (DOFs) are defined, such as $GN_2^p$ in Fig. 1. In the rest of this paper, the "NMM node" will be simply called "node" for the purpose of description.

In NMM, the basic units to integrate the weak form of the problem are manifold elements. Each manifold element is the common domains of neighboring PPs, such as $E_1$, which is the common domains of $PP_3$, $PP_4$ and $PP_5$ (Fig. 1).

## 3. Formulation for the iNMM

In order to synergize the advantages of both the Trig3-RPIMcns and the numerical manifold method (NMM), an improved version of NMM (iNMM) is developed. Formulation of the iNMM will be described in great detail in this section.

As a PU-based method, the global approximation of NMM in a manifold element is obtained by multiplying the PU function with the local approximation, and expressed as

$$u^h(\boldsymbol{x}) = w_1(\boldsymbol{x})u_1(\boldsymbol{x}) + w_2(\boldsymbol{x})u_2(\boldsymbol{x}) + w_3(\boldsymbol{x})u_3(\boldsymbol{x}) \tag{1}$$

where $w_i(\boldsymbol{x})$ and $u_i(\boldsymbol{x})$ are the PU function and the local approximation function associated with physical patch $i$ ($PP_i$), respectively.

### 3.1. PU function of the iNMM

The area coordinates are used to construct the PU functions of iNMM. The transformation of the area coordinates is defined as [50]:

$$\begin{bmatrix}1\\x\\y\end{bmatrix}=\begin{bmatrix}1&1&1\\x_1&x_2&x_3\\y_1&y_2&y_3\end{bmatrix}\begin{bmatrix}L_1\\L_2\\L_3\end{bmatrix},\quad \begin{bmatrix}L_1\\L_2\\L_3\end{bmatrix}=\frac{1}{2A}\begin{bmatrix}x_2y_3-x_3y_2 & y_2-y_3 & x_3-x_2\\x_3y_1-x_1y_3 & y_3-y_1 & x_1-x_3\\x_1y_2-x_2y_1 & y_1-y_2 & x_2-x_1\end{bmatrix}\begin{bmatrix}1\\x\\y\end{bmatrix}$$
$$\overset{\text{def}}{=\!=}\frac{1}{2A}\begin{bmatrix}a_1&b_1&c_1\\a_2&b_2&c_2\\a_3&b_3&c_3\end{bmatrix}\begin{bmatrix}1\\x\\y\end{bmatrix}, \tag{2}$$

in which

$$2A=\det\begin{bmatrix}1&1&1\\x_1&x_2&x_3\\y_1&y_2&y_3\end{bmatrix},\quad L_1+L_2+L_3=1. \tag{3}$$

Unlike traditional NMM, which uses the FEM shape functions to construct the PU functions, the PU functions of the iNMM are expressed as [48]